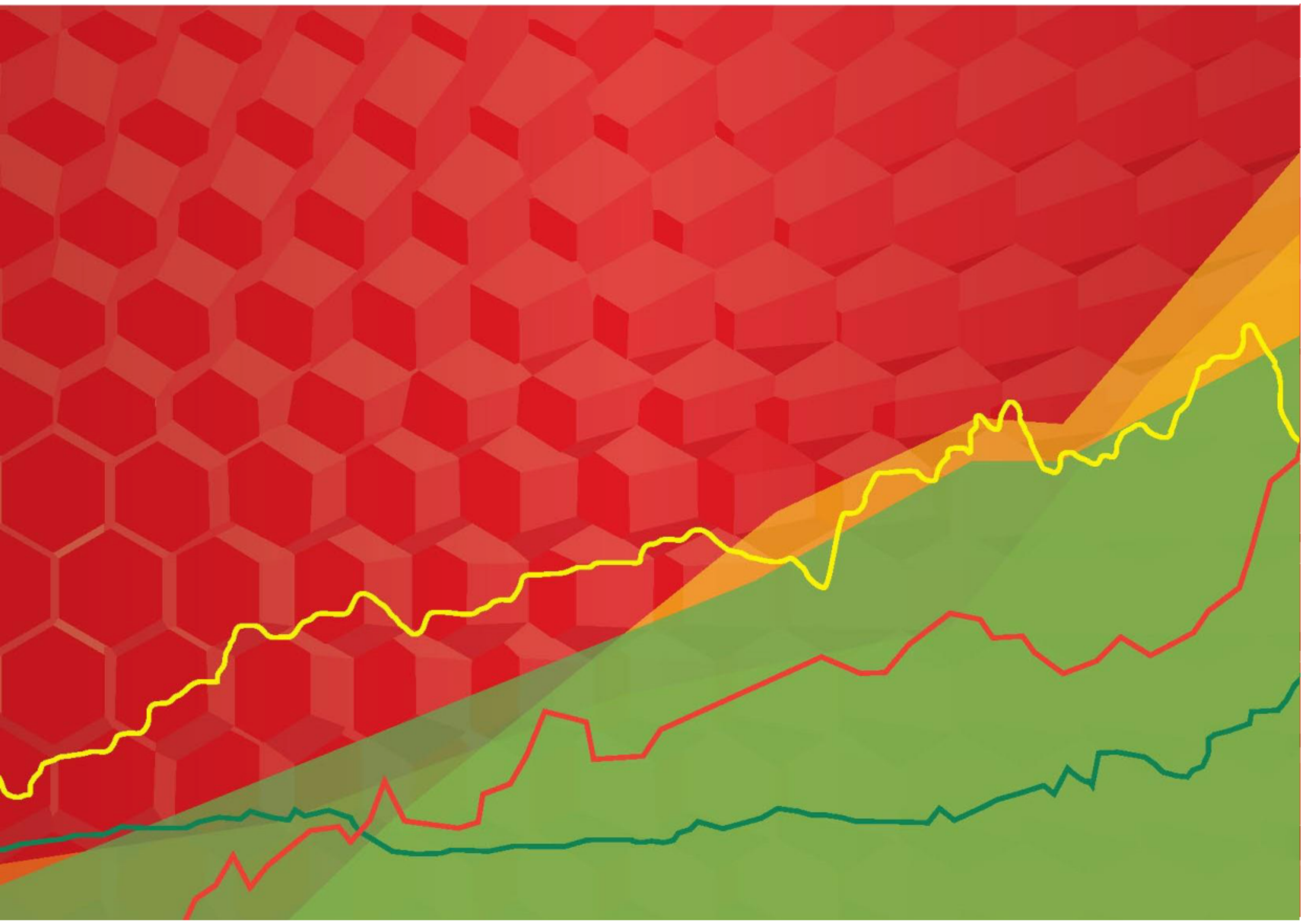
T/HIS

T/HIS Manual

from Oasys Ltd



Version 21.0

For help and support from Oasys Ltd please contact:

UK
Tel: +44 121 213 3399
Email: dyna.support@arup.com
China
Tel: +86 21 3118 8875
Email: china.support@arup.com
India
Tel: +91 40 69019723 / 98
Email: india.support@arup.com
USA West
Tel: +1 415 940 0959
Email: us.support@arup.com

Web: www.arup.com/dyna

or contact your local Oasys Ltd distributor.

THIS 21.0

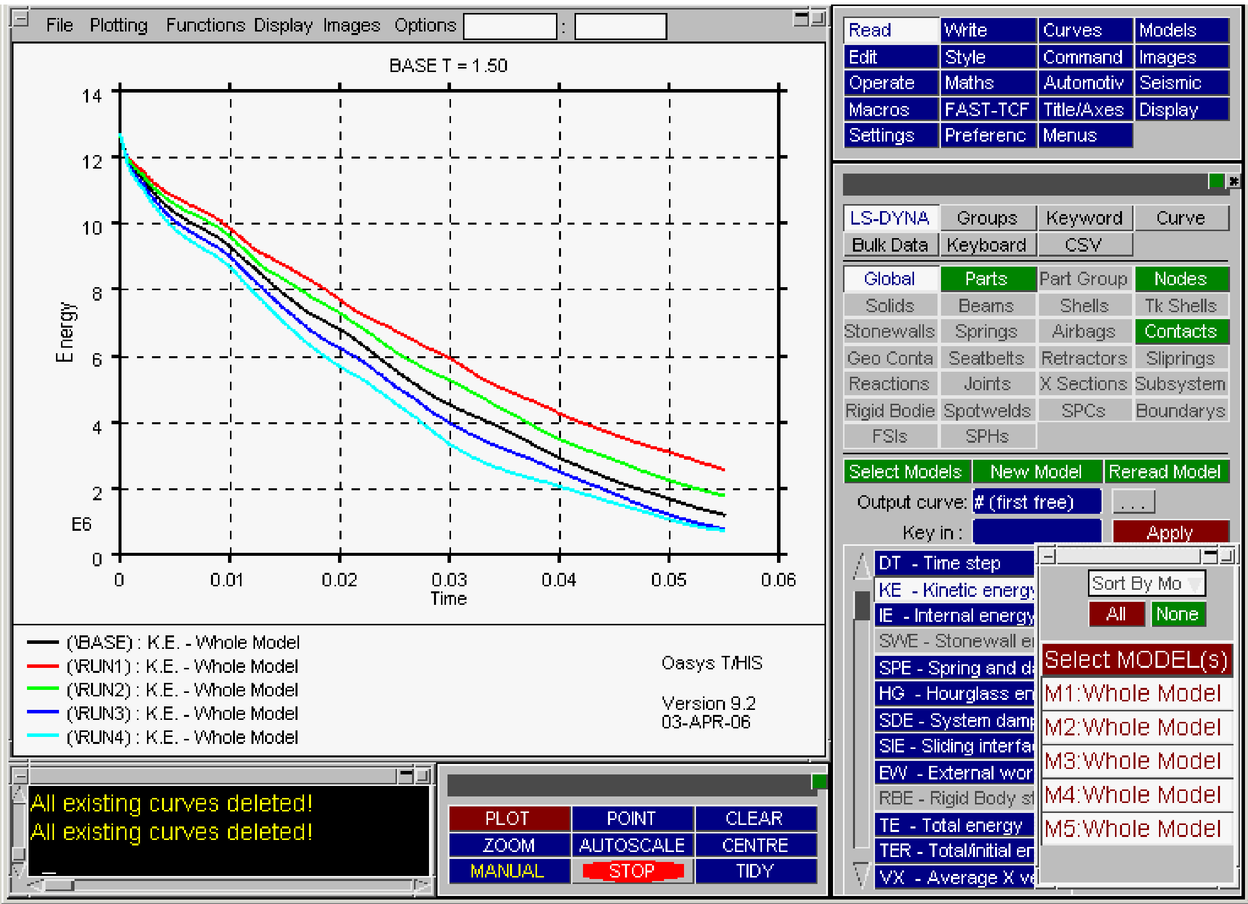
Table of Contents

1. Title

T/HIS Software Manual
from Oasys Ltd

T/HIS

X/Y Data Processing for LS-DYNA



Oasys Limited | 8 Fitzroy Street | London | W1T 4BJ

2. Preamble

2.1. Text Conventions Used in this Manual

Text conventions used in this manual

Typefaces

Three different typefaces are used in this manual:

Manual text	This typeface is used for text in this manual.
`Computer type`	This one is used to show what the computer types. It is also used for equations, keywords (eg `*PART`) etc.
`Operator type`	This one is used to show what you must type.
Button text	This one is used for screen menu buttons (eg **APPLY**)

Notation

Triangular, round and square brackets have been used as follows:

- Triangular

  To show generic items, and special keys. For example: <list of integers> <filename> <data component><return> <control Z> <escape>

- Round

  To show optional items during input. For example: <command> (<optional command>) (<optional number>)

  And also to show defaults when the computer prompts you, e.g.:

  Give new value (10) :

  Give model number (12) :

- Square

  To show advisory information at computer prompts, e.g.:

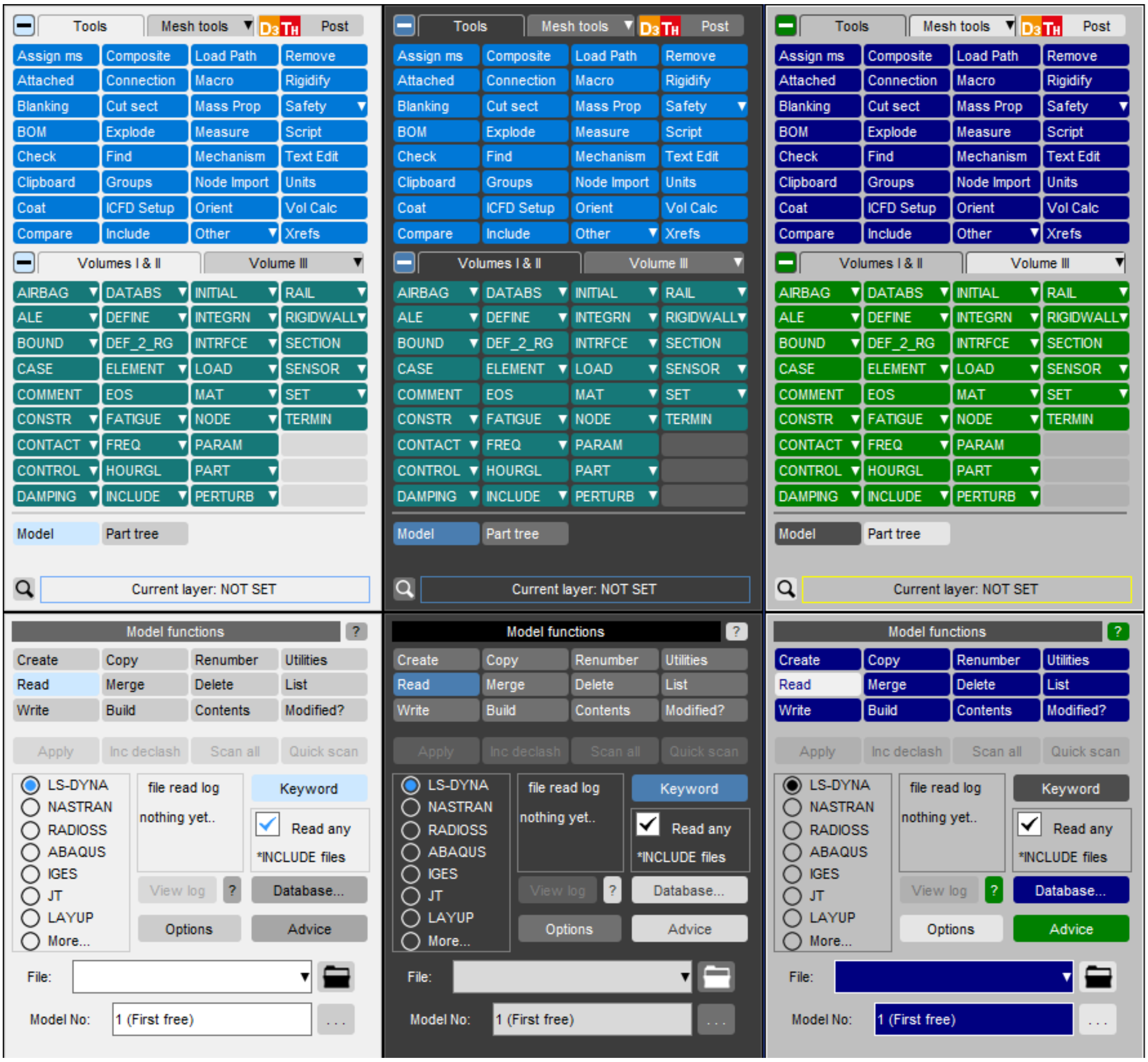
  Give filename: [.key] :

  THIS >>> [H for Help] :

3. Themes for the Graphical User Interface

Themes for the Graphical User Interface

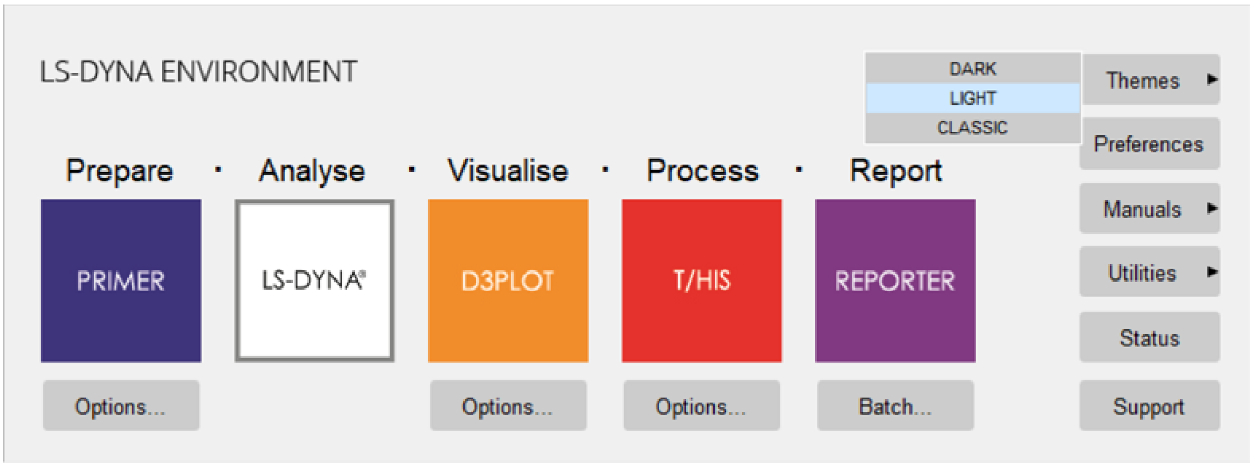
In addition to our Classic GUI theme, beginning in Oasys Suite 17.0, users can select either a Light or Dark theme. Both of these provide a more modern look and feel for the software, as well as offering different colour and contrast options for comfort and accessibility.



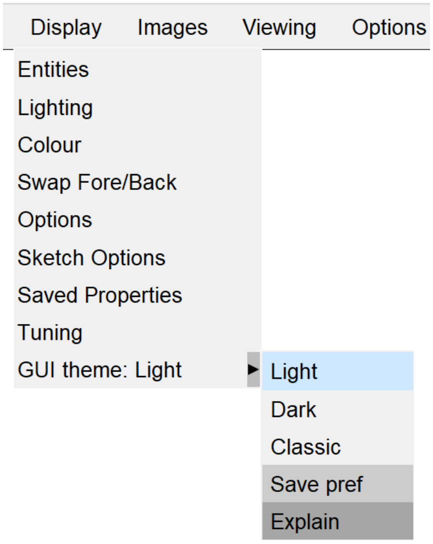
3.1. Setting the Theme

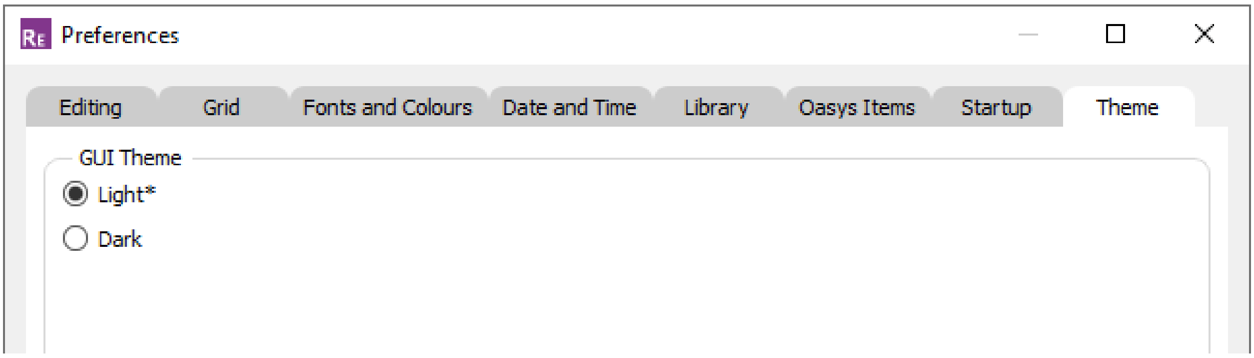
Setting the theme

The default software theme in Oasys Suite 21.0 is Light. This can be changed from the Oasys SHELL by choosing from the **Themes** pop-up. This automatically saves the selected theme as your preference for all programs.



The theme can also be set for individual programs from the **Display** menu in PRIMER, D3PLOT and T/HIS or the **Preferences** menu (**File**->**Preferences...**) in REPORTER. This choice is not automatically retained after exiting the program, so you must select a theme, then select **Save pref** to ensure a theme is used for all future sessions.




Preferences
Editing
Grid
Fonts and Colours
Date and Time
Library
Oasys Items
Startup
Theme
GUI Theme
Light*
Dark

4. Introduction

Introduction

T/HIS is an x/y plotting program, specifically written to perform two functions:

1. To produce time-history plots from transient analyses, such as those performed using LS-DYNA.
2. To plot any form of x/y data that is produced either by a program or by directly typing in values.

T/HIS is a graphically driven, interactive program. Input and manipulation of data is through a graphical user interface on systems capable of running X-Windows applications; selections are made through "pressing buttons" using a mouse. On machines not capable of running X-Windows it is also possible to use T/HIS in a "command line" mode of operation; instructions are entered through the keyboard to perform the required operations.

4.1. Program Limits

Program Limits

There are a number of limits in T/HIS of which the user should be aware. These are listed below:

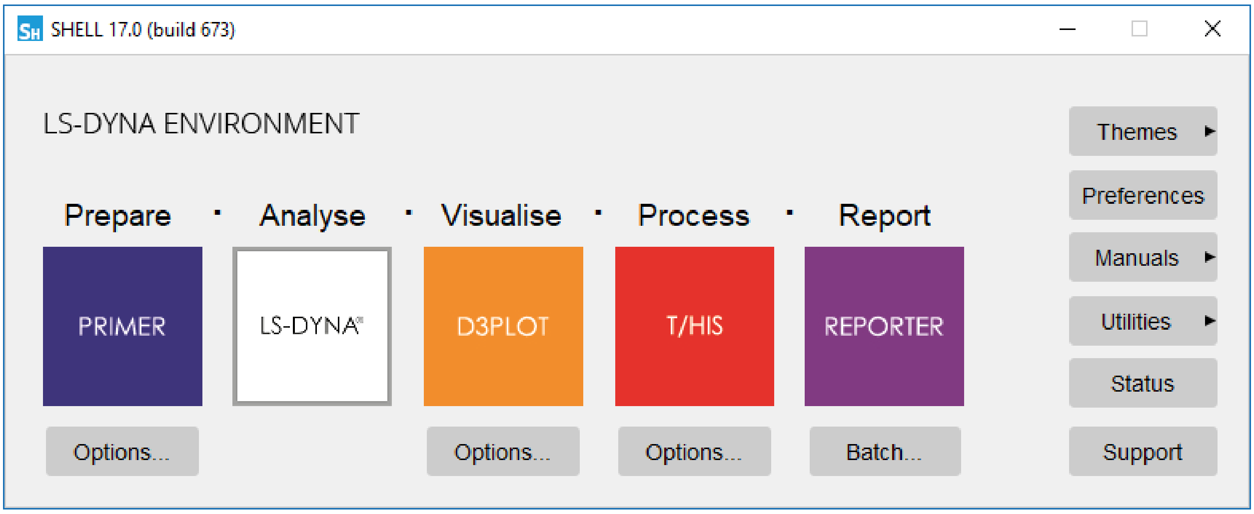
Number of graphs	T/HIS can have a maximum of 32 graphs
Number of curves	The number of curves is unlimited
Number of points	The number of points that can be defined per curve is unlimited.
Time-history blocks	In the interface to the LS-DYNA time-history **`(.thf)`** file there is a limit of 100,000 items in each of the node, solid, beam, shell and thick shell time-history blocks: thus 500,000 items overall. In the interface to the LS-DYNA extra time-history **`(.xtf)`** file up to 100,000 nodal reactions (or groups of reactions) may be processed.
Number of colours	By default, T/HIS curves wrap around the following six colours in order: WHITE RED GREEN BLUE CYAN MAGENTA However, a further 24 predefined colours are available if required and 6 user defined ones can be created.
Title	The title can contain up to 80 characters.
Labels	Labels for axes and lines can contain up to 80 characters.

4.2. Running T/HIS

Running T/HIS

Starting the code

For users on a device with a window manager T/HIS is run from the **T/HIS** button in the SHELL:



If your system has been customised locally you may have to use some other command or icon: consult your system manager in this case.

Graphics Driver

T/HIS 9.3 onwards uses a OpenGL graphics driver.

T/HIS uses 32 bit (single-precision) numbers to store and plot data.

"Batch" Mode

T/HIS can run in "batch" mode where the main application window is not displayed on the screen. "Batch" mode is available on all platforms.

To start T/HIS in batch mode use the command line option "-batch".

```
e.g. this14_64.exe -tcf=script.inp -batch
```

When running in "batch" mode T/HIS will automatically exit at the end of the script regardless of whether or not "-exit" is specified.

NOTE : All image, postscript and PDF outputs require a DISPLAY on UNIX / LINUX systems. If you are running T/HIS in "batch" mode as part of a automatic post processing script then T/HIS must have a X Windows DISPLAY even though the main window is not displayed. If the machine you are using is a server or part of a cluster without an X-Server then T/HIS can be used with the Xvfb software.

Selecting a device when a window manager is not running

If you are running on a non-window device, for example a Tektronix display or emulator, you may not be able to use screen menus. Instead you will have to run in "command-line" mode.

It is very unlikely that a user on a modern workstation will see these options, since the machine will have a window manager and will be running in "screen menu" mode. If they do appear it suggests that the machine and/or software are wrongly set up: see below for suggested remedies.

If T/HIS will not start in screen-menu mode

You may be running on a device with a window manager, but still only get the command-line prompt (and probably no menu driven _93 shell either).

This is almost certainly because of one or both of the following setup errors:

(1) The DISPLAY environment variable has not been set up, or has been set incorrectly. This tells the X11 window manager where to place windows, and it must be set to point to your screen. Its generic setup string is:

`setenv DISPLAY <hostname>:<display number>` (C shell syntax)

Where <hostname> is your machine's name or internet address, for example:

`setenv DISPLAY :0`	(Default display :0 on this machine)
`setenv DISPLAY tigger:0`	(Default display :0 on machine "tigger")
`setenv DISPLAY 69.177.15.2:0`	(Default display :0, address 69.177.15.2)

You may have to use the raw network address if the machine name has not been added to your **/etc/hosts** file, or possibly the "yellow pages" server hosts file.

(2) Your machine (strictly the X11 "server") has not been told to accept window manager requests from remote machines. This is usually the case when you are trying to display from a remote machine over a network, and you get the message similar to:

```
Xlib: connection to "<hostname>" refused by server

Xlib: Client is not authorised to connect to server
```

In this case go to a window with a Unix prompt on your machine, and type:

```
xhost +
```

Which tells your window manager to accept requests from any remote client. It will produce a confirmatory message, which will be something like:

```
access control disabled, clients can connect from any host
```

If T/HIS still fails to work then please contact your system manager, or contact Oasys Ltd for advice and help.

Command Line Mode

Command line mode is the main method of data input on non X-Windows devices. Command line mode is also available within the X-Windows screen interface and is accessed through the dialogue window. In command line mode the user will be presented with a prompt which also indicates which level of the menu structure the user is at. For example:

```
Defaults >
```

In response to the prompt a valid option must be given. These are usually a two or three letter abbreviation of a command; for example **PL** is the command to plot a graph. A list of the commands available is provided by typing **M** (for Menu). In addition to commands specific to one menu there are a number of commands which have the same effect throughout T/HIS.

Q	- (Quit) Abort and return to current menu
!	- Go up a level in the menu structure
/	- Return to the top level menu
;	- Equivalent to a **<carriage return>** in a string of commands
M	- Lists menu.

Several commands can be strung together on one line, separated by spaces, for example:

```
/DE GR ON
```

Numeric data can also be included in the command line if required, for example:

`/OP ADX #1 7.2 #`

Commands can be in upper or lower case.

As well as menu level commands you will be asked questions such as:

`THF file to read (filename_1)?`

The default response, if one exists, is given in parentheses.

4.3. Command Line Options

Command Line Options

Instead of starting T/HIS using the Command shell it is also possible to start T/HIS from the command line with a number of optional input parameters. Starting T/HIS from the command line offers a number of advantages.

- Faster start-up is possible by pre-selecting the device type.
- The input filename can be specified and opened automatically.
- Faster start-up is possible by pre-selecting the device type

Argument format:

<application name> (<arg 1>) (<arg n>) (<input filename>)

T/HIS 21.0 can be started with a number of optional command line options

Graphics device type	**-d=<device type>** eg **-d=default**	Valid device types are: **opengl** — OpenGL **tty** — No windows **default** — OpenGL
Command file name	**-cf=<filename>** eg **-cf=run_1.tcf**	Any valid T/HIS command file filename
FAST-TCF input file	**-tcf=<filename>** eg **-tcf=run_1.inp**	Any valid T/HIS FAST-TCF command file filename
Settings file	**-set=<filename>** eg **-set=this001.set**	Any valid T/HIS settings file
JavaScript	**-js=<filename>** eg **-js=sort_curve.js**	Any valid T/HIS JavaScript file
JavaScript Arguments	**-js_arg=<argument>** eg **-js_arg=abc**	Any valid string. The arguments can be accessed in the script by using the global **arguments** array. Multiple arguments can be given to a

		script by using more than one – `js_arg` command line argument.
LS-DYNA Model All the files associated with the model are opened and the contents scanned.	**<filename>** eg **run_1.thf**	Any filename from the analysis This should be the last argument on the command line.
LS-DYNA Model list Specify a file containing a list of models for T/HIS to automatically open.	**-model_list=<filename>** eg **-model_List=job_list**	The model list file should contain the full pathname of one file from each model that T/HIS should open. Each file should be on a seperate line and it should be the first item on each line (other items seperated with commas can be specified on the same line for use with REPORTER).
Model Database file Specify the name of the default model database file.	**-mdb=<filename>** eg **-mdb=database.xml**	The model database file is an XML format file that contains information on where models are located along with a brief description of each model The model database can be used to easily select multiple models..
T/HIS curve file Specify a T/HIS curve file containing one or more curves for T/HIS to automatically open.	**-cur=<filename>** or **-curve=<filename>** eg **-cur=test.cur**	
T/HIS curve file list Specify a file containing a list of curve files for T/HIS	**-curve_list=<filename>** or **-curve=<filename>** eg **-cur=test.cur**	The curve list file should contain the full pathname of each cure file that you want T/HIS to open. Each file should be on a separate line.

to automatically open.		
T/HIS bulk data file Specify a T/HIS BDF file containing one or more curves for T/HIS to automatically open.	**-bdf=<filename>** eg **-bdf=test.cur**	
Automatically maximises the T/HIS window so that it occupies the full screen.	**-maximise**	
Read THF file	**-thf=<yes/no>**	
Read XTF file	**-xtf=<yes/no>**	
Read LSDA (binout) file	**-lsda=<yes/no>**	
Read ASCII files	**-ascii=<yes/no>**	
Specifying a custom "oa_pref" file. This causes an extra, optional "oa_pref" file to be read.	**-pref=<filename>**	<filename> must be a valid "oa_pref" file. If it has no path prefixed, the file is assumed to be in the OA_INSTALL directory. Any legal filename may be used.
Use ELOUT instead of ELOUTDET	**-use_elout=<yes/no>**	By default T/HIS uses the ELOUTDET part of the LSDA file in preference to ELOUT if the LSDA file contains both. This option can be used to force T/HIS to use the ELOUT data when reading Shell and ThickShell data as the ELOUT data can be in the global

		coordinate system instead of the element local coordinate system.
Write out data in the ISO-MME format (See [WRITE Options])	**`-write_iso_mme`**	This option should be used in conjunction with the – **`iso_output_dir`** and **`-iso_config`** options. A model to extract the data from also needs to be specified. As an example: **`this.exe -write_iso_mme -iso_output_dir=<directory> -iso_config=<filename> <model_filename>`**
Specify the output directory to write data to for the – **`write_iso_mme`** option.	**`-iso_output_dir=<directory>`**	
Specify the configuration file to use for the – **`write_iso_mme`** option.	**`-iso_config=<filename>`**	
Specify a directory for T/HIS to start in.	**`-start_in=<directory>`**	Any valid directory
Set the width of the T/HIS graph window (in pixels)	**`-xres=<size>`** **`eg -xres=800`**	
Set the height of the T/HIS graph window (in pixels)	**`-yres=<size>`** **`eg -yres=600`**	
Run T/HIS without the console window.	**`-noconsole`**	Windows only.
Run T/HIS in "batch" mode	**`-batch`**	For this option to work you must also specify a command file " –

where the main application window is not displayed on the screen.		**cf=filename** " and the name of the PTF file to open. This option will automatically set " –**exit** " so that D3PLOT terminates after playing the command file.
Redirect output from the console window to a file on Windows. To redirect output on Unix/Linux use the shell redirection options (typically > for **<stdout>**, **&** for **<stderr>**)	**-eo=<filename>** **-eo** **-eo=default**	**-eo=<filename>** is designed for the user to suppress the console and redirect logfile output to the specified filename. In order to permit multiple sessions to coexist on the same machine the process ID will be appended to the **<name>** part of the filename to give **<name>_pid.<ext>**. If plain "**-eo**" or "**-eo=default**" are found then filename generation is automatic, and the first valid of: **%TEMP%\this_log_<pid>.txt** **%TMP%\this_log_<pid>.txt** **%HOMESHARE%\this_log_<pid>.txt** **%USERPROFILE%\this_log_<pid>.txt** will be used.
Read/Write checkpoint files Start writing the checkpoint files upon T/HIS startup Read checkpoint files and Show checkpoint playback panel upon T/HIS startup. Directory path to write checkpoint files	**write_checkpoint_files=< *TRUE/FALSE* >** **show_checkpoint_files=< *TRUE/FALSE* >** **checkpoint_dir=< *directory* >**	TRUE/FALSE, turn on/off the writing of the checkpoint files (default is FALSE) TRUE/FALSE, turn off the initial checkpoint files panel (default is FALSE) If the writing of the checkpoint files is OFF, the reading will also be OFF. **<directory>** must be a valid directory name on your system. If the value is **<none>** then the checkpoint files are not recorded for the T/HIS session.

Stop and exit after command file	**`-exit`**	

Some examples for T/HIS might be:

`pathname/this12.exe -d=x run_2.thf`	(Use device X, open a **`.thf`** file)
`pathname/this12.exe -d=tty cf=batch.tcf -exit`	(No graphics, run command file)

Note that no spaces should be left in the syntax <arg>=<value>.

For example: " **`-d = x`** " is illegal.

Correct syntax is: " **`-d=x`** "

5. Using Screen Menus

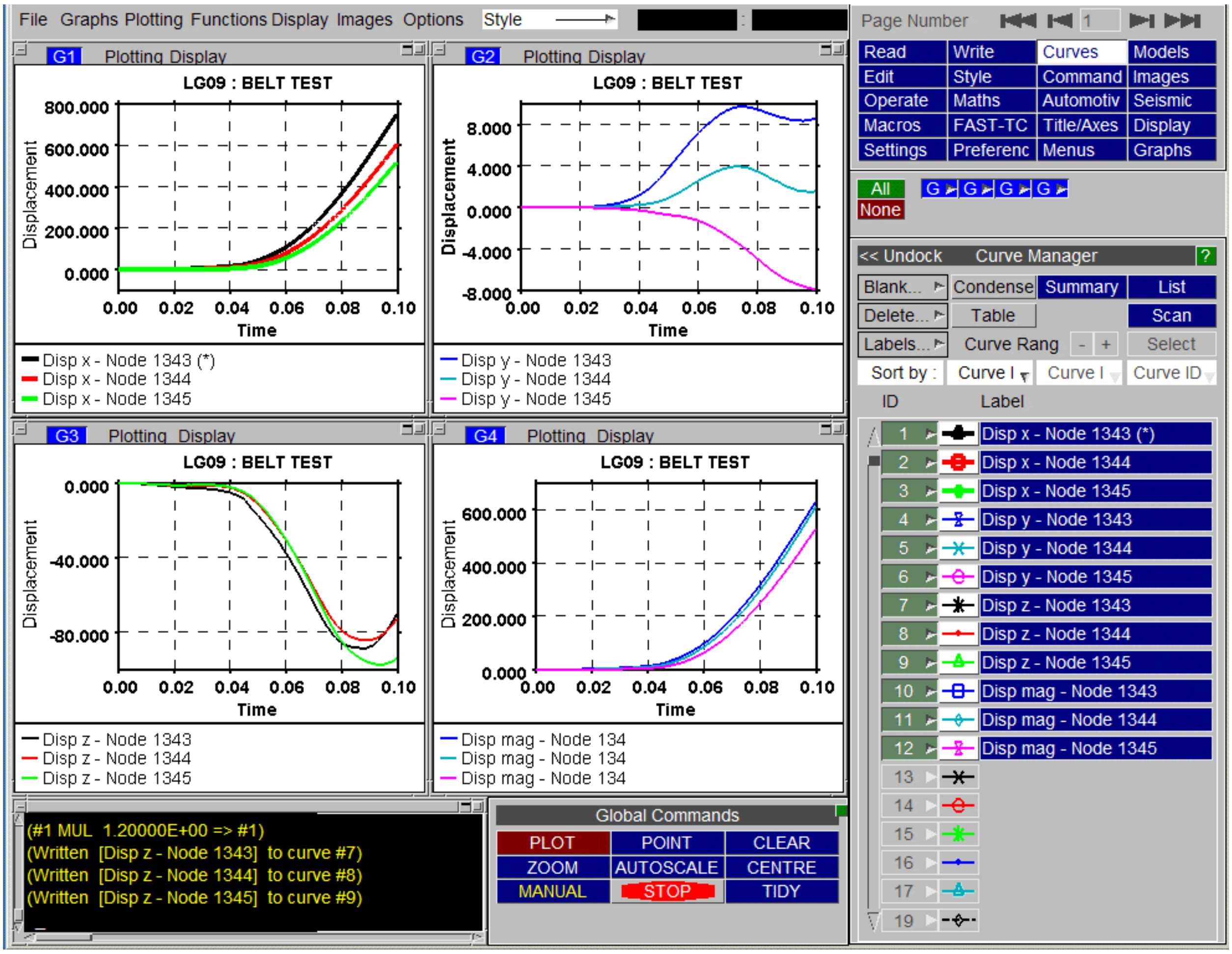
Using Screen Menus

Versions of T/HIS prior to release 6.1 only had a "command-line" interface. This has been preserved for backwards compatibility, but a "screen-menu" interface has been added which allows you to drive the program almost entirely with the mouse.

5.1. Basic Screen Menu Layout

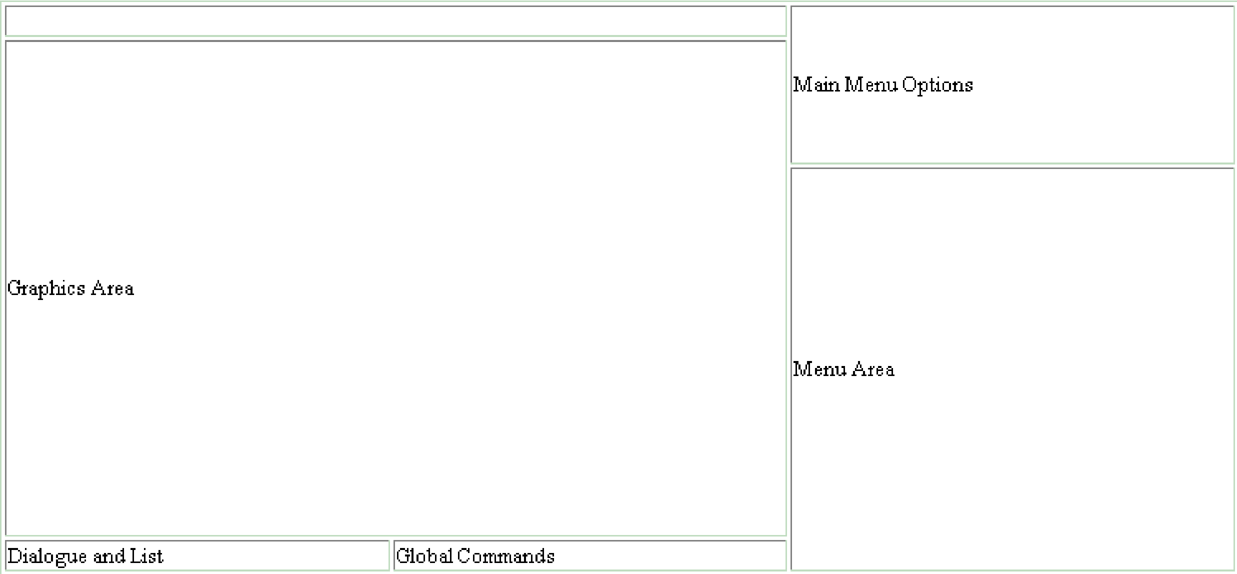
Basic screen menu layout

T/HIS runs within a single window, owned by the window manager, which has several sub-windows inside it. A typical T/HIS session will look like this:



The various sub-windows always exist within the master window, and may be moved and resized at will inside it. They will keep their relative size and position as the master window is changed in size and/or shape, and will reappear after the main window is de-iconised.

The default layout of the main sub-windows is as follows:



These windows cannot be dismissed. A brief description of their functions is:

Main Menu Options	Provides access to the majority of the commands and options available in T/HIS through a series of sub menus (see [Main Menu](#)).
Graphics area	Is where graphs are drawn. In T/HIS 21.0 this area can contain a maximum of 32 graphs (see Graphs and Pages). Alternatively if graphs have been organised into pages (see Page Layout) then this area will display a single page of graphs.
Dialogue & list	Allows "command-line" input and output, also provides a listing area for messages.
Menu Area	Displays the commands and options associated the current selection from the main menu options.
Global Commands	Gives access to commonly used commands (see Global Commands and Pages).

While you are free to reposition these master windows it is recommended that you keep to this default layout. This is because when further sub-windows appear their position and size is designed assuming this layout, and aims to obscure as little useful information as possible.

5.2. Mouse and Keyboard Usage for Screen Menu Interface

Mouse and keyboard usage for screen-menu interface

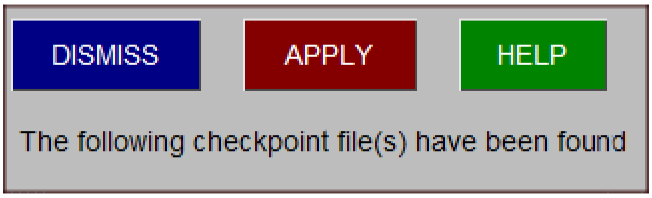
All screen-menu operations are driven with the left mouse button, with the following exceptions:

(a) Text in the dialogue area and text boxes requires keyboard entry.
(b) Text strings saved in the cursor "cut" buffer may be "pasted" into dialogue areas and text boxes using the middle mouse button.

The primitive "widgets" in the menu interface are used as follows:

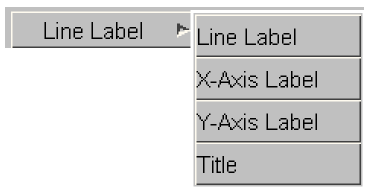
BUTTONS:

Screen buttons are depressed by clicking on them, but action only takes place when the mouse button is released, so it is safe to drag the (depressed) mouse around the screen.



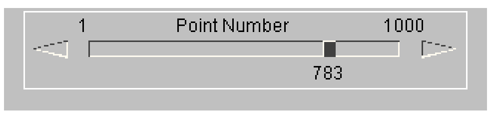
Buttons may also be greyed out to indicate that the option is not currently available. Buttons with " **...** " after them will usually invoke sub-menus.

"Popup" window invocation : Buttons with an " **>** " symbol may be selected normally with the left mouse button, but if the *right* mouse button is depressed over them it will invoke a "popup" window. Holding the right mouse button down move the cursor into this window to make a selection, or move elsewhere and release the button to deactivate the popup.



SLIDERS:

Sliders are moved by clicking on the slider button itself, and then dragging it to a new position. They may also be moved automatically

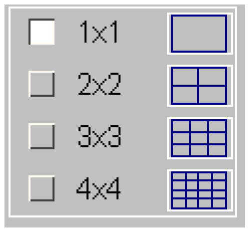


by clicking on, and holding down, one of the arrows at either end.

TEXT BOXES:

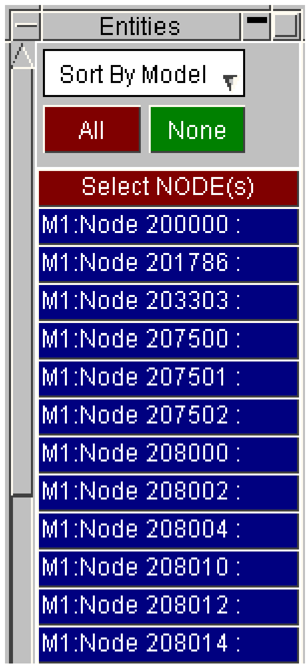
To enter text in a text box: first make it "live" by clicking on it, then type in text, then type **<return>** to enter the string. Clicking on a "live" box for a second time is exactly the same as typing <return>, so clicking twice on a box effectively enters its current contents. You can use the left and right arrow keys for line editing within a box: text entry takes place after the current cursor position.

RADIO BOXES



A "radio" set is provided where only one selection is possible from a range of options. In this example the postscript laser output has been set to a single image per page.

MENU SELECTIONS:



Menus of items are used when you need to make one or more selections from a (potentially) long list. Click on the row you want to select: clicking on a row that is already selected will have the effect of unselecting it. When the list is too long to display in the window you can use the vertical scroll-bars to move up and down it.

A range of items may be selected by either

1) Click on the first item and hold down the mouse key, drag the mouse to the last item in the list. All items between the first and last including the first and last are selected.

or

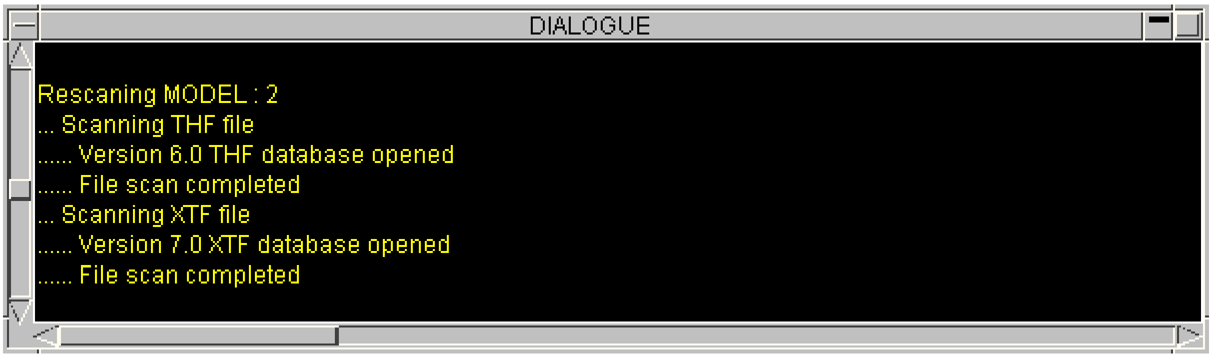
2) Click on the first item, hold down the SHIFT key and click on the last item in the list. All items between the first and last including the first and last are selected.

5.3. Dialogue Input in the Screen Menu Interface

Dialogue input in the screen menu interface

The full command-line capability is preserved when T/HIS is running in screen-menu mode, and you are free to mix command-line and mouse-driven input at will. There are some situations in which command-line input is more efficient: for example when entering lists of explicit entities.

Commands are entered in the dialogue box:



As this example shows the dialogue box is also used for listing messages, warnings and errors to the screen. It can be scrolled back and forth (its buffer is 200 lines long) to review earlier messages. The following colours are used:

Normal messages and prompts	Yellow
Text typed in by you	White
Warning messages	Magenta
Error messages	Red

There is a minor limitation when mixing command-line and screen-menu mode: you cannot perform the same function simultaneously in both modes. If you attempt to do so you will get the message:

`WARNING: recursive access attempted`

And you will not be permitted to continue.

For T/HIS 20.0 onwards, we have now also added the ability to use the Up and Down arrow keys in order to cycle through previously issued commands that you have entered into the Dialogue window. Currently only the last 50 commands are stored in memory. Commands issued after this 50 limit will remove the oldest issued commands first from the list.

5.4. Window Management in the Screen Interface

Window management in the screen interface

Moving, resizing and scrolling of windows is based on the conventions used in the Motif Window Manager.

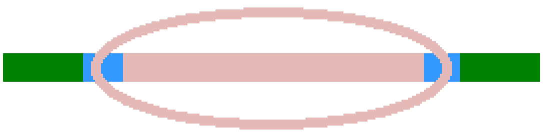
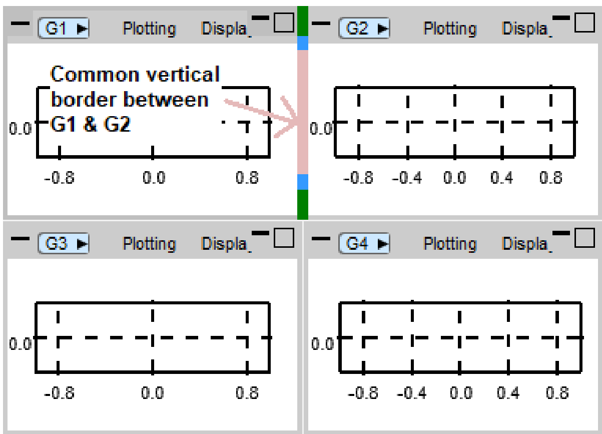
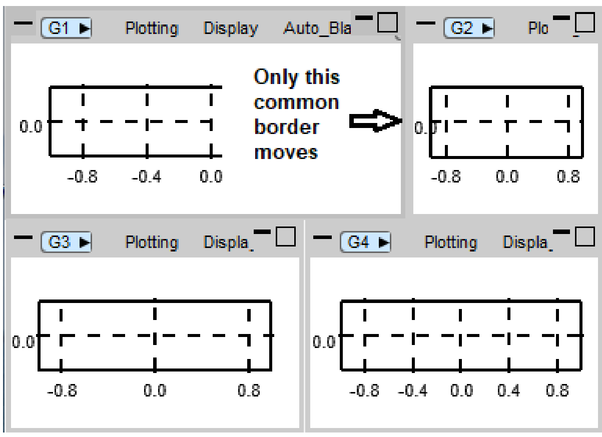
To move a window:	Click down on its title bar, then drag the window to where you want it to be. A "rubber-band" outline moves to show the window's current position.
To resize a window:	Either Click on a border bar to move just that side, or on a corner bar to move both sides attached to that corner. Again, a rubber-band outline shows you the new shape. or Use the **MAXIMISE** button  in the top right hand corner of the window to increase the size of the window to the largest possible size.
To scroll a window:	If a window has got too small for its contents then horizontal and/or vertical scrollbars will appear. Click on a scrollbar slider and move it to the desired position, the window contents will scroll as you do so. Alternatively click on the arrows at either end of the scrollbar for timed motion in that direction.
To minimise a window:	Click on the button  in the top right hand corner of the window. When a window has been iconised it will appear in the **ICON** area at the bottom of the screen.
To restore a window:	Iconised windows may be restored by clicking on the icon in the ICON area.

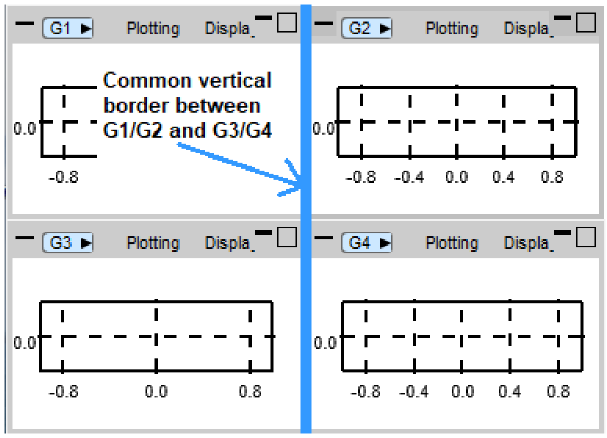
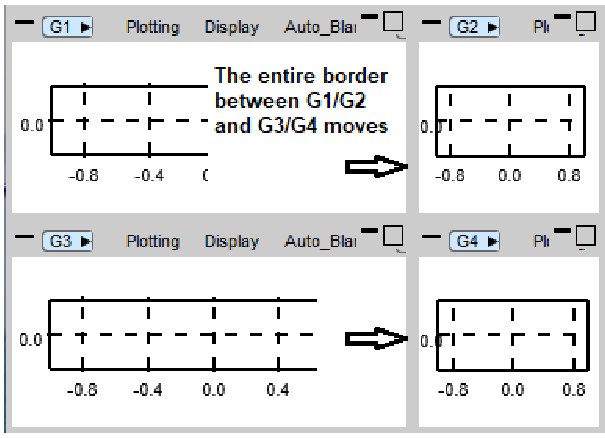
5.5. Common Borders for graphics windows

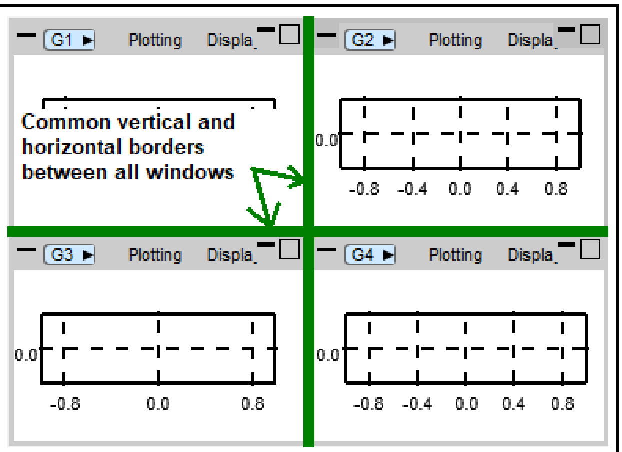
Common Borders for graphics windows

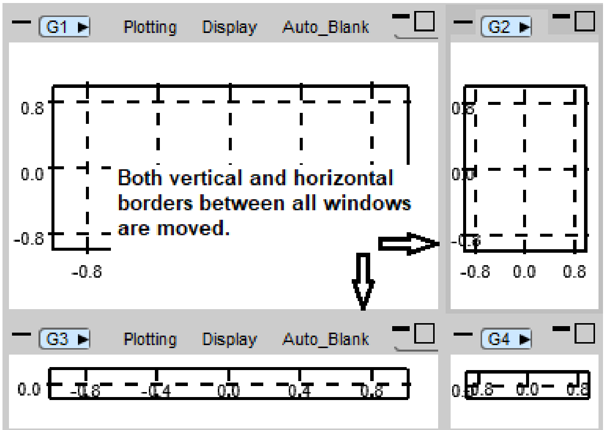
When a page contains more than one graphics window these are laid out in a cellular grid as defined in the Window Layout section. This leads to "common borders" between adjacent windows. From T/HIS 19.0 onwards it is possible to drag this common border with the mouse in a way that resizes windows on both sides of the border as shown in the following images.

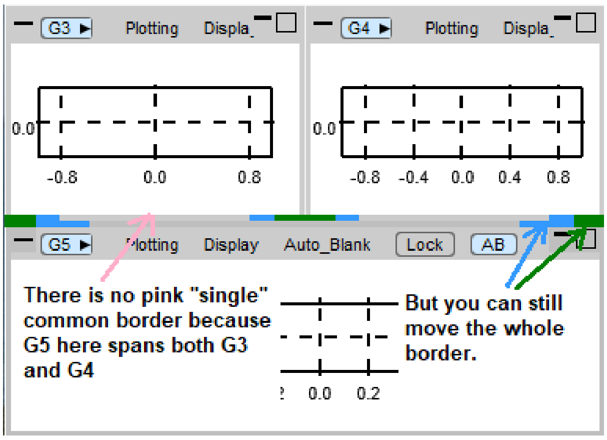
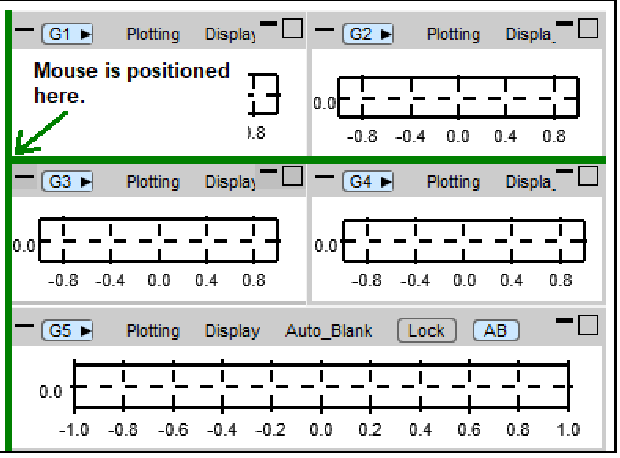
Move the mouse (don't depress a button) over a border region between two windows. This will highlight the drag areas in which a "click and drag" operation will move borders. In order to control which borders are dragged, three zones – coloured pink, blue and green – are shown and these have the following meanings:

Pink zone defines a common border between exactly two adjacent windows.  (Horizontal and vertical borders behave the same way. A horizontal border is shown here. The example on the right shows a vertical border being moved.)	
Dragging in this pink region border moves only that common border between the two windows. So in this example, the vertical border between G1 and G2 is moved, but that between G3 and G4 is unchanged.	

Blue zones define a common border extending the full height or width of the page as appropriate.  (Both blue zones have the same effect, it doesn't matter which end you use.)	
Dragging in the blue zone moves all windows on either side of the border in the appropriate direction. In this example, all four windows are moved.	

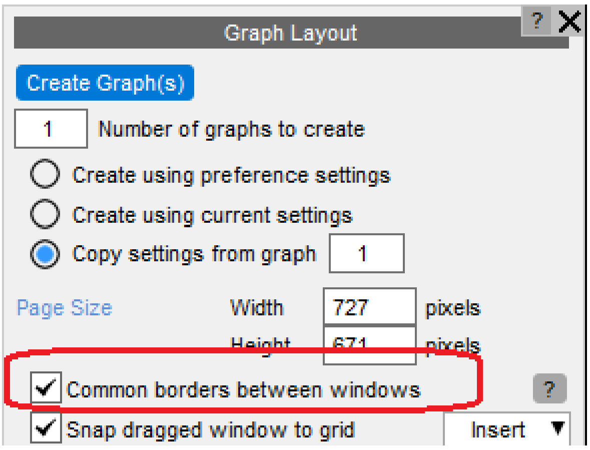
Green zones define two common borders extending both horizontally and vertically to the full width and height of the page 	

(Both green zones have the same effect, it doesn't matter which end you use.)	
Dragging in the green zone moves all windows on either side of the border in the appropriate direction. In this example, all four windows are moved.	

When windows are not the same size: In this example G5 is twice the width of G3 and G4 above it so there is no single common border between G3/G5 or G4/G5. In this situation, there will be no pink zone, only blue and green.		
Positioning the mouse at window edges: When using the green zone to drag both horizontal and vertical axes, the borders that are dragged are those which intersect at the corner where the mouse is located.		

In this example, the mouse is at the bottom left of G1 / top left of G3 and it can be seen that the borders that are highlighted for dragging are those which intersect at this point.	

Switching common border dragging on/off



Common borders are on by default, but they can be controlled from the **[Graph Layout]** panel.

The default behaviour may also be set by the preference:

```
this*common_window_borders:  true  |
 false
```

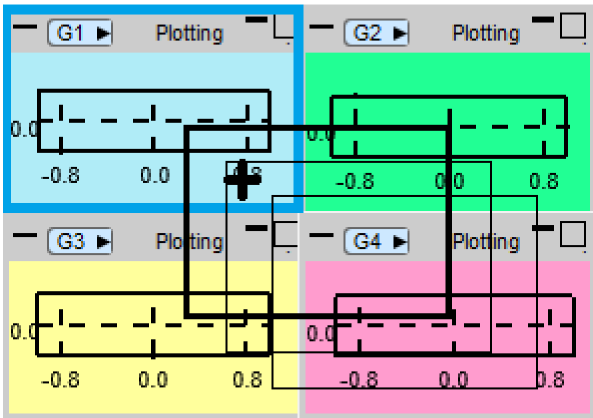
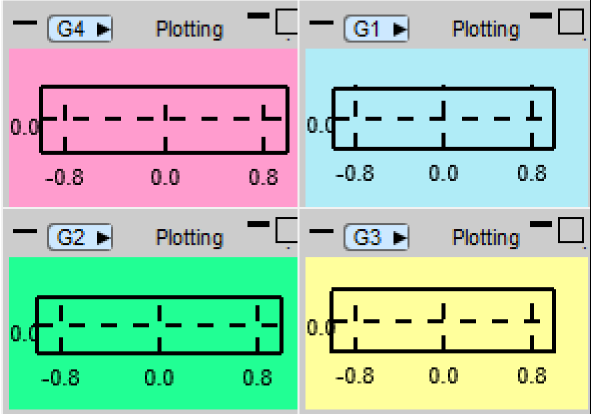
Window "snap to grid" and other options.

When dragging an entire window with the mouse to move its position on the screen (i.e. not resizing it) there are several possible outcomes:

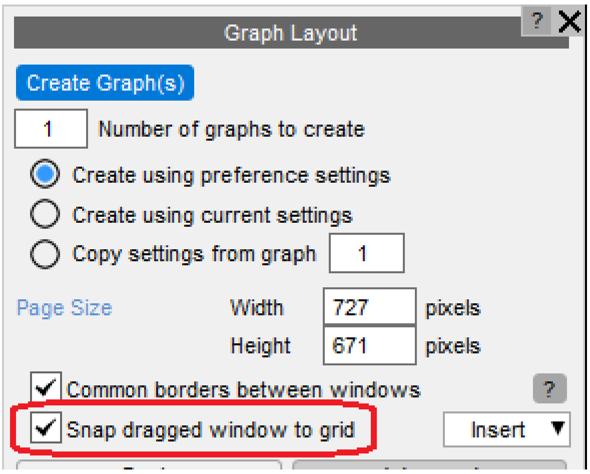
1. **Snap to Grid**: The window is moved from one "cell" in a multi-window page to a different cell, shifting the contents of one or more cells out of the way.
2. **Free positioning #1**: The window is moved from inside the T/HIS master window to a new user-defined position within that window, i.e. positioned where it is "dropped".
3. **Free positioning #2**: The window is moved from inside the T/HIS master window and out onto the desktop.

The behaviour of "Snap to Grid" is illustrated in the following figure:

In this example, G4 is being dragged and the cursor is at the "+" position inside G1. The borders	Releasing the mouse in this example causes G4 to move to the "cell" of G1, and windows G1

<table>
<tr>
<td>of W1 are
highlighted in blue to show it is the destination window.

</td>
<td>to G3
to move down in the grid to make space for it.

</td>
</tr>
<tr>
<td colspan="2">"Snap to grid" behaviour illustrated in "Insert" mode</td>
</tr>
</table>

Switching "Snap to grid" on/off



Snap to grid is on by default, but itcan be controlled from the **Graph Layout** panel.

The default behaviour may also be set by the preference

`this*snap_window_position: true | false`

If turned off, the window positioning within the master T/HIS window reverts to "Free positioning #1" mode with the window positioned where it is dropped with the mouse.

The behaviour of the other windows when a window is moved into a new position depends on whether the mode is **Insert** or **Swap**:

Insert	Other windows circulate either up or down, as in the example above.
Swap	The window being dragged and its destination window swap places

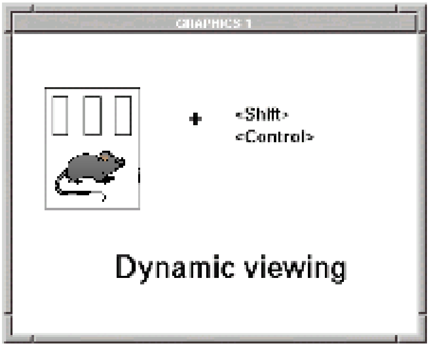
Dragging a window from inside the T/HIS master window onto the desktop, "Free positioning #2", is independent of the "snap to grid" setting: once on the desktop the window does not have any cell membership.

The ordering of windows within cells can also be controlled explicitly within the the **Graph Layout** panel.

5.6. Dynamic Viewing (Using the Mouse to Change Views)

Dynamic Viewing (Using the mouse to change views).

"Dynamic" viewing is the name given to the process in which you perform viewing transformations by moving the mouse around the screen.

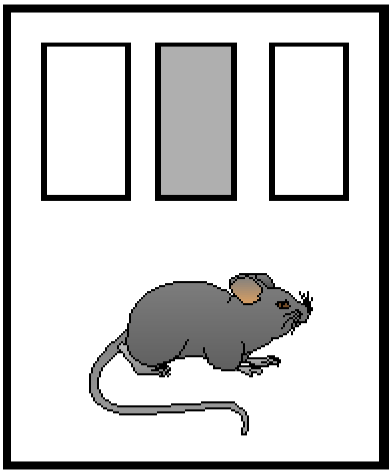


Dynamic Translation.

Dynamic translation uses **`<mid mouse> + <left shift>`**

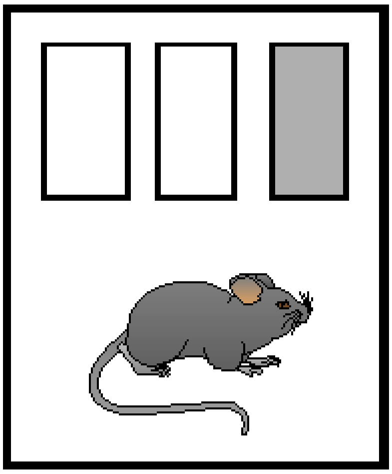
The cursor symbol is yellow, and looks like:

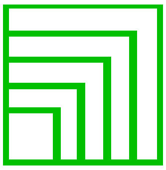
The relationship between mouse and image motion is intuitive: the object tracks the mouse motion in the screen XY plane. The initial position of the mouse is irrelevant.



Dynamic Magnification (Scaling).

Dynamic scaling uses **<right mouse> + <left shift>**



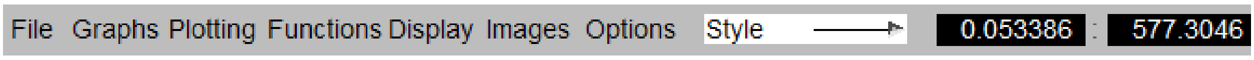
The cursor symbol is green, and looks like: 

Mouse motion to the right and up makes the image larger, left and down smaller. The initial position of the mouse is irrelevant. A horizontal movement will scale just the x-axis while a vertical movement will scale just the y-axis.

5.7. "Tool Bar" Options

"Tool Bar" Options

Across the top of the main graphics window are a number of buttons that can be used to access other T/HIS menus (see Tool Bar) for more details..



If the graphics box is maximised to take up the whole of the main window these buttons can be used to access the rest of the T/HIS menus without having to resize the graphics box between commands. Almost all of the options and functions in these menus may also be accessed from other menu locations, e.g. the Main Menu area.

5.8. Colours

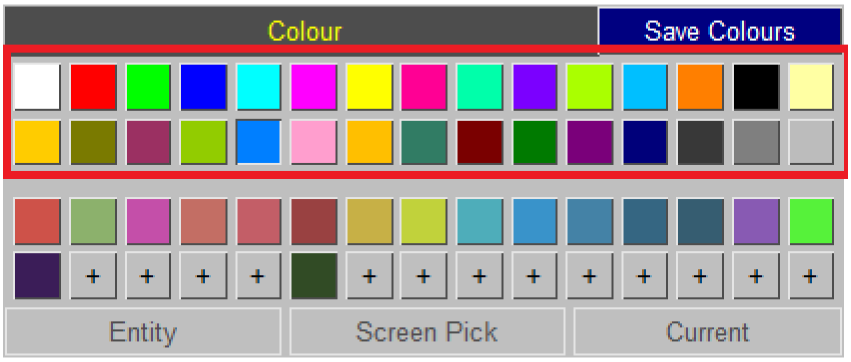
Colours

The colour popup allows users to select a standard colour or set-up and use a user-defined colour.

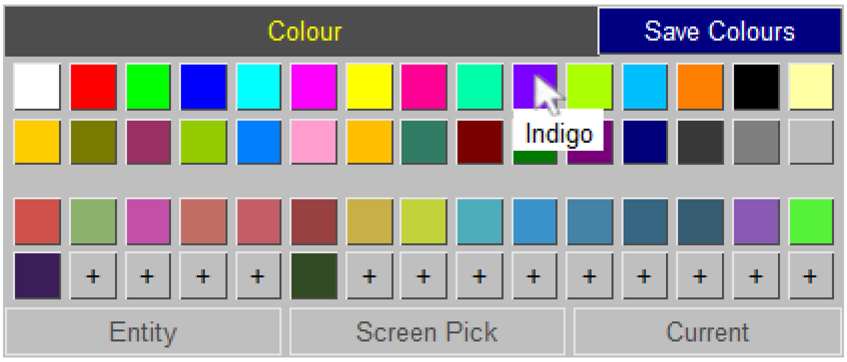
For some menus special context colours are available, for example "Entity", "Default" or "Background". These options are explained in more detail in the sections of the manual about that menu.

Standard Core Colours

The top two rows show the 30 standard core colours.



When you hover over the colour the name is shown.



This name can be used to specify this colour in preferences and dialogue inputs.

In T/HIS this name can be used in JavaScript and FAST-TCF.

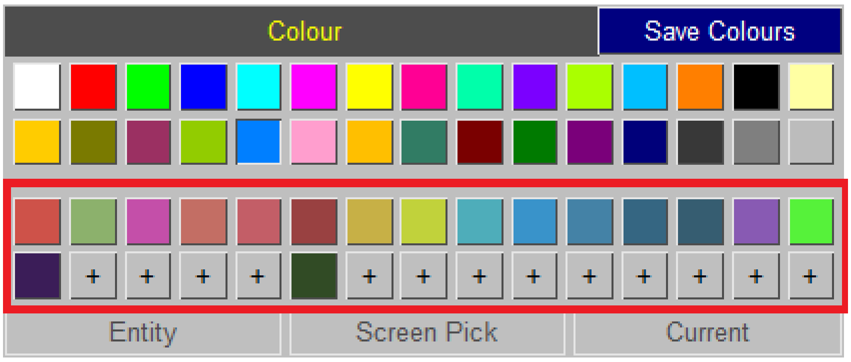
When using the name, "_" is used instead of " ", for example "Hot Pink" becomes "HOT_PINK".

The standard core colours available are very similar in D3PLOT and T/HIS. The following colours are a similar shade but have different names:

D3PLOT	T/HIS
Red/Magenta	Orange
Green/Cyan	Turquoise
Yellow/Green	Lime
Light Blue	Sky
Dark Orange	Pink
Cyan/Blue	Medium Blue
Red/Orange	Light Pink
Grey	Medium Grey

User-Defined Colours

The lower rows show the user-defined colours. There can be up to 150 user-defined colours.



Click on a user-defined colour to apply it, or click on an empty slot to create a new user-defined colour.

User-defined colours can be used in the dialogue input by specifying their name.

In T/HIS user-defined colours can be used in FAST-TCF.

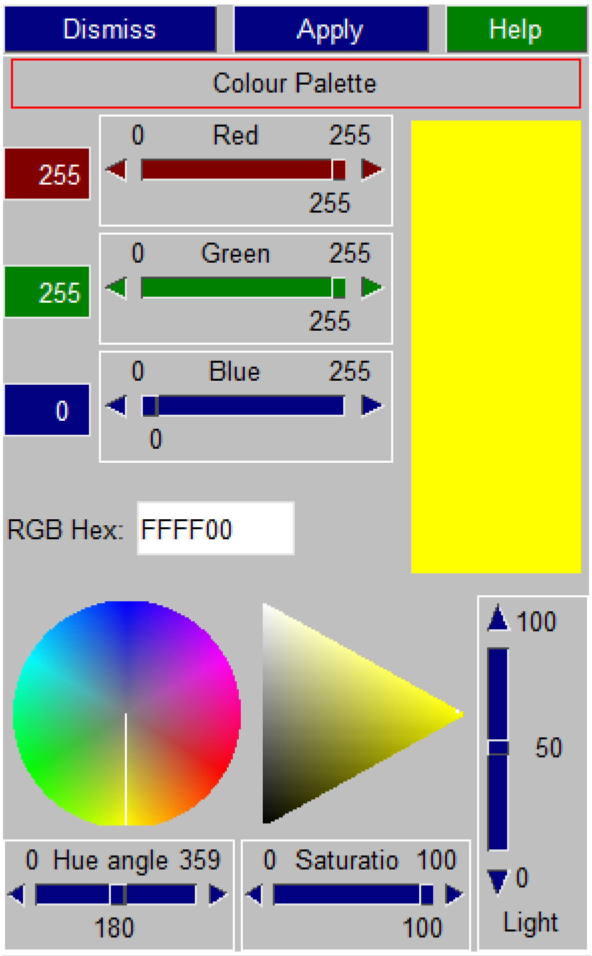
Creating

To create a new user colour click on an empty slot. This maps a colour palette.

The colour can be edited a number of ways:

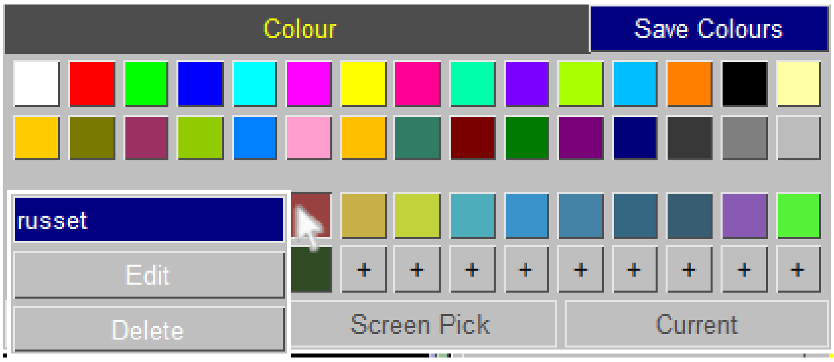
- Using sliders to set the red, green and blue value,
- Inputting a hex colour code,
- Clicking on the colour wheel and cone, or
- Using sliders to set the hue, light and saturation levels.

When you create a colour it is applied.



Editing

Hover over a user colour to edit it. You have the choice to change the name of the colour, **Edit** it, or **Delete** it.



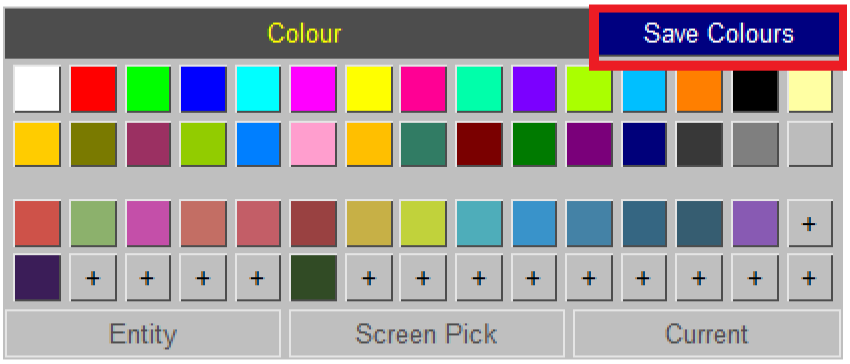
The user-defined colours are given the standard name, for example "user_1". They can be renamed. The name must start with a letter and gets set to all lower case. If the name is not unique, a number will be appended to it, for example "green_1".

Edit maps the colour palette. If you edit a colour it is then applied.

Delete removes a colour. The colour is no longer available when you next open the colour popup.

Saving

The user-defined colours can be saved. The same user-defined colour are then available when you next run D3PLOT or T/HIS.



The user-defined colours are stored in the user_colours.xml file. If the user has permission to modify things in the INSTALL directory, the user is given the option to either save the user colours to the INSTALL directory (which is sometimes visible to multiple users) or their HOME directory.

Alternatively, the preference **`user_colour_file`** can be set to specify an .xml file.

When D3PLOT or T/HIS is next started the user_colours.xml file is read in.

If the same colour, for example "user_1", is defined in the user_colours.xml file in both the INSTALL and HOME directory, the HOME directory user_colours.xml file takes precedence.

If the preference **`user_colour_file`** has been set, any user_colours.xml file in the HOME directory is ignored. If a colour is also defined in the user_colours.xml file in the INSTALL directory, the **`user_colour_file`** .xml file takes precedence.

For T/HIS, if a user colour was previously set-up using a preference, for example **`this*user_colour1`** , and that colour slot is also defined in a user_colours.xml file, the user_colours.xml file takes precedence.

T/HIS Link

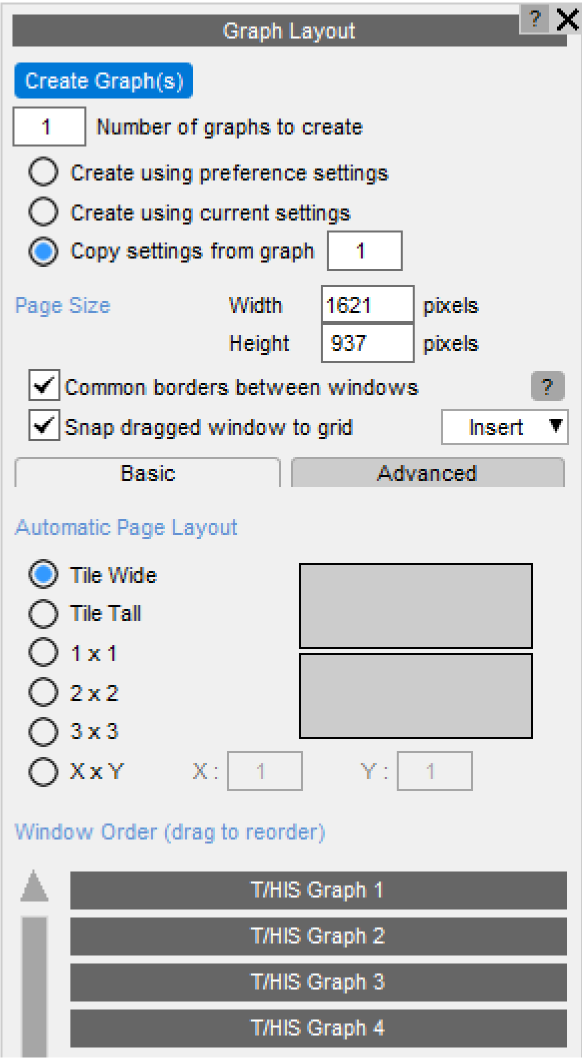
When running the T/HIS link any user colours created in D3PLOT (or in T/HIS) will be available in the other program. When T/HIS is first opened it sets-up the user colours to match the current D3PLOT session, rather than using a saved user_colours.xml file.

6. Graphs and Pages

Graphs and Pages

T/HIS 21.0 can display a maximum of 32 graphs. Each graph can have a different appearance and they can display different curves.

Graphs can be laid out using a number of different formats and they can be organised into Pages.



6.1. Creating Graphs

Creating Graphs

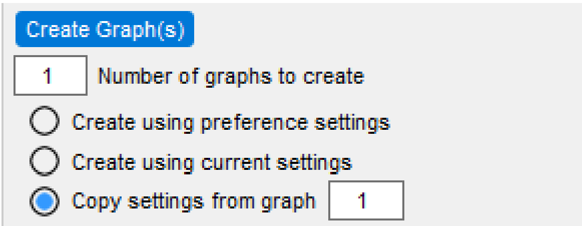
Create Graphs

Create a new graph.

The shortcut key 'G' can also be used to create new graphs.

Number of graphs to create

This option can be used to create multiple graphs.



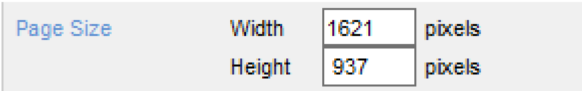
When new graphs are created, the initial settings for each graph can be copied from three different sources:

Create using preference settings	The Display and Axis Settings are copied from the preference file.
Create using current settings	The Display and Axis Settings are copied from the current settings in the Display and Axis menus.
Copy settings from graph n	The Display and Axis Settings are copied from the specified graph.

6.2. Page Size

Page Size

These options can be used to specify the total size of the area (in pixels) used by the graph windows.



6.3. Page Layouts

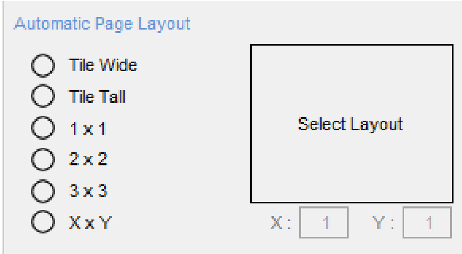
Layout

Graphs can be laid out in a number of different formats and can be organised into Pages.

From D3PLOT and T/HIS 19.0, the Graph Layout menu is split into separate **Basic** and **Advanced** modes.

6.3.1. Basic Mode

Basic Mode



In Basic mode, the menu can be used to select a page layout that is automatically applied to all of the pages.

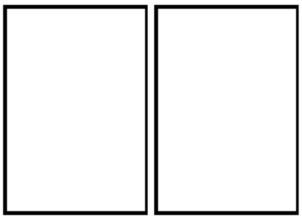
Automatic Page Layout

If an Automatic page layout is used and the layout is set to **Tile Wide** or **Tile Tall** then all Graphs are automatically added to page 1.

In all other layouts, Graphs are automatically added to pages and as many pages as needed are created to hold all the Graphs.

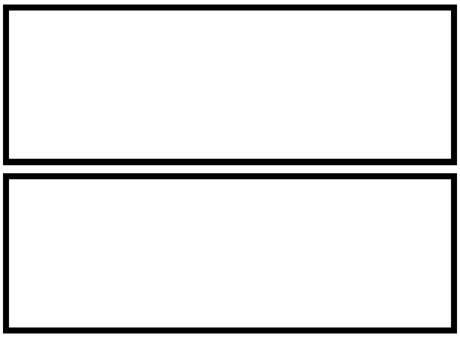
Tile Wide

All of the graphs are positioned on a single page.



Tile Tall

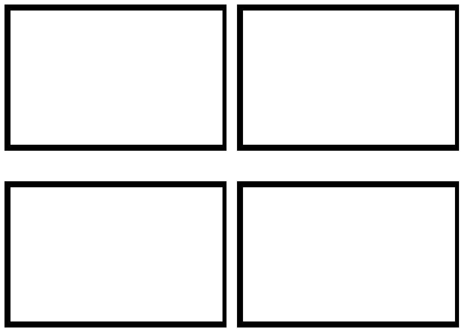
All of the graphs are positioned on a single page.



1 x 1

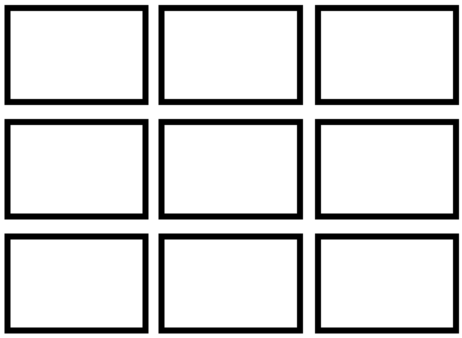
Each graph is positioned on its own page.

2 x 2



Graphs are arranged in a 2 by 2 grid. If there are more than 4 graphs, then graphs 1 to 4 are positioned on page 1, graphs 5 to 8 on page 2, etc.

3 x 3

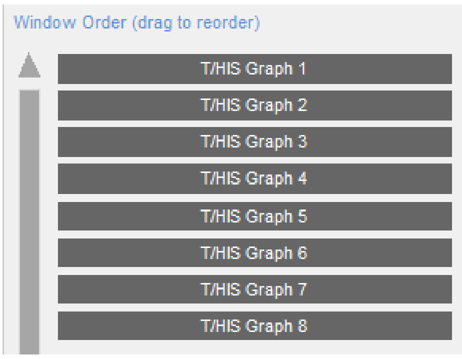


Graphs are arranged in a 3 by 3 grid. If there are more than 9 graphs then graphs 1 to 9 are positioned on page 1, graphs 10 to 18 on page 2, etc.

X x Y

Graphs are arranged in a X by Y grid.

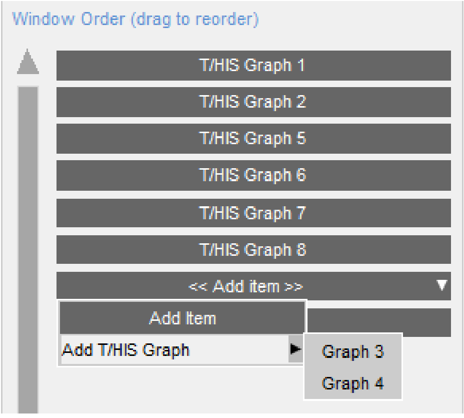
Window Order



By default, Graphs are added to pages in the order they are created.

The order of Graphs can be changed by clicking on a row and dragging it up or down the list to a new position.

Any Graphs that have been dragged out onto the desktop are removed from the list (Graphs 3 and 4 in the example on the right).



If Graphs are on the desktop, the menu will display additional rows that can be used to add the graphs back into the list so that they are displayed on a page again.

Right-clicking on an **<< Add item >>** row will

6.3.2. Advanced Mode

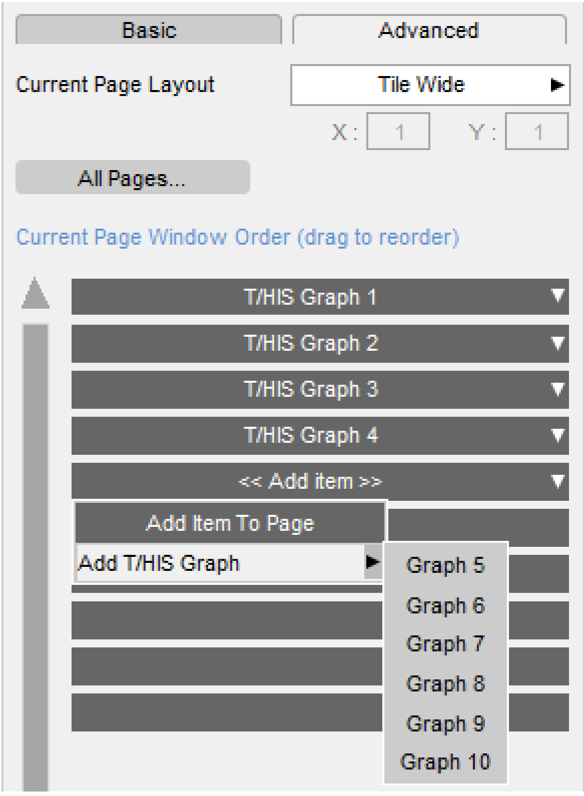
Advanced Mode

Advanced mode can be used to give more control over which graphs appear on which page. Unlike in Basic mode, a graph can appear on more than one page.

Advanced mode works in a similar way to Basic mode except that it controls the settings for the current page only.

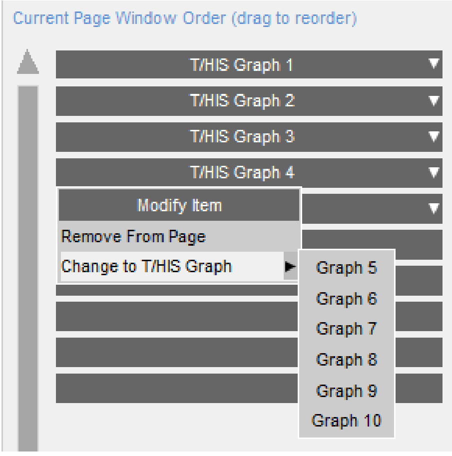
In Advanced mode, the layout and content of each page can be set for that page, and the order of the items displayed on each page can also be controlled by clicking on an item and dragging it up or down to a new position.

Right-clicking on **<< Add item >>** will display a popup menu that can be used to select any Graph that is not currently on the page.



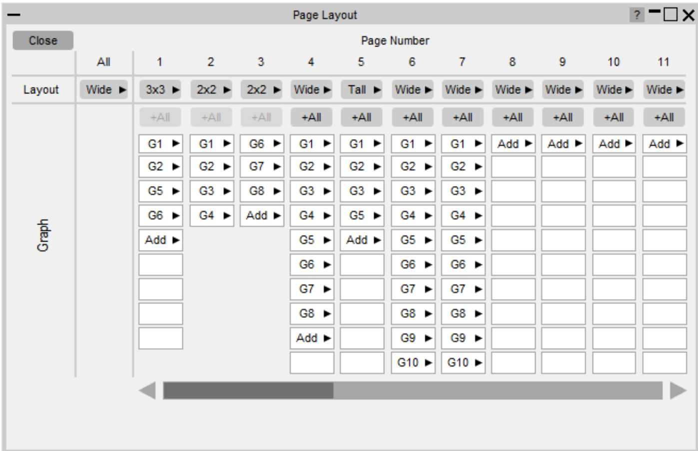
Right-clicking on a row containing a Graph will display a popup menu that can be used to remove the Graph from the current page.

Alternatively, the same popup can also be used to change an item to a different Graph that is not already on the current page.



All Pages menu

In Advanced mode, the **All Pages...** button can be used to display a separate menu that shows the layout and contents of all pages:



This Page Layout menu can be used to select which graphs appear on each page. Each graph can appear on more than one page.

The options to reorder or change the contents of each page are similar to those in the Window Order section of the Layout menu:

- Drag the buttons in each column up and down to reorder graphs on a page
- Use the popup menus to edit page contents

6.4. Pages

Pages

T/HIS can have a maximum of 32 pages, each page can contain multiple graphs. For more information on selecting the currently displayed page Global Commands and Pages . The Image Output options and the FAST-TCF Create option can produce output for either a single page or multiple pages if graphs are located on more than one page.

6.5. Active Graphs

Active Graphs

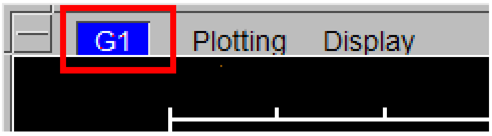
If T/HIS contains more than one graph then each graph can be toggled between being active or inactive.

All the graphs can be activated using the **All** button or deactivated using the **None** button.

There is a popup menu attached to each button that can be used to select that graph **Only** , **Raise** the graph so that it is in front of any others or to **Delete** the graph.

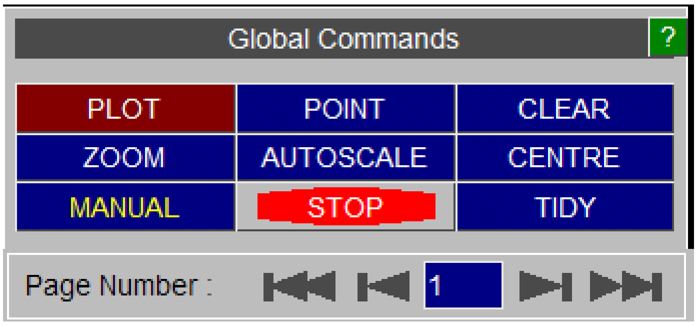
When a graph is deleted any graphs with higher numbers are renumbered downwards to remove any gaps in the graph numbering.

Graphs can also be activated / deactivated using the button located in the top left hand corner or each graph.



7. Global Commands and Pages

Global Commands and Pages

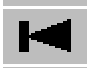
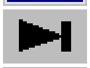


The following commands are to be found as buttons on the **GLOBAL MENU** panel. (The command line codes are given in parentheses.)

All of the commands in the GLOBAL MENU can also be accessed via the **PLOTTING** button at the top of the graphics window.

Page Number

If T/HIS contains more than one graph (see Creating Graphs) then the graphs can be positioned on separate Pages within T/HIS. This menu can be used to select a specific page or it can be used to step through the pages one by one.

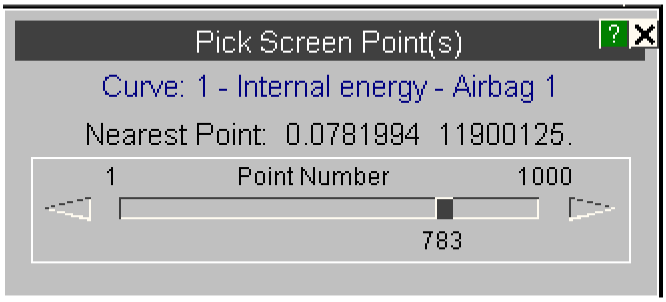
		Shortcut Key
	Goto Page 1	Home
	Go back 1 Page	Page Up
	Goto Page (n)	N/A
	Go back 1 Page	Page Down
	Goto Page 32	End

PLOT (PL)

This option will plot all the curves that are currently UNBLANKED (see Curve Manager).

POINT (PT)

When selected this option waits for the user to pick a point in the main graphics screen.
Once a point has been picked the <x> and <y> values of the point picked are reported along with the ID of the nearest curve and the nearest point on that curve.



After a point has been selected on the screen the slider may be used to move to other points along the same curve.

CLEAR (CL)

Clears the graphics screen.

ZOOM (ZM)

The cursor appears on the screen and may be used to select the required plot area by choosing opposite corners of a box. The graphs are then replotted. Using **ZOOM** implicitly turns autoscaling off.

AUTOSCALE (AU)

Autoscales the plot size for all current unblanked curves in the graphics window and re-displays the plot.

CENTRE (CE)

Pick a point on the screen using the cursor to be the new plot centre. It affects the x/y offsets but not the scales.

MANUAL

Displays the online (HTML) version of the manual

STOP

Some operations, like reading a file containing many curves in to T/HIS, can take a long time. This button can be used to stop some long opertaions without having to exit from T/HIS.

TIDY

This option can be used to reset the menu layout to the default settings.

Additional Commands

A number of additional global commands exist in command line mode. These functions exist in screen menu mode within other menu levels.

(PF) Creates a postscript plot file. Either A4 landscape or A4 portrait formats may be chosen. A title and figure number are also requested. Other plot setting may be made in the command line mode **UTILITIES** menu.

(BL) Blank a currently displayed curve.

(UB) Unblank a curve that has been blanked.

(RM) Remove (delete) a curve. Once a curve has been removed it is lost from the system.

(ER) Erase (delete) all existing curves from T/HIS. (Equivalent to the command **RM *.**)

(GS) Global status: displays the current number of curves, their labels and whether they are blanked.

(CO) Condense: renumbers all curves to fill any gaps in curve numbers.

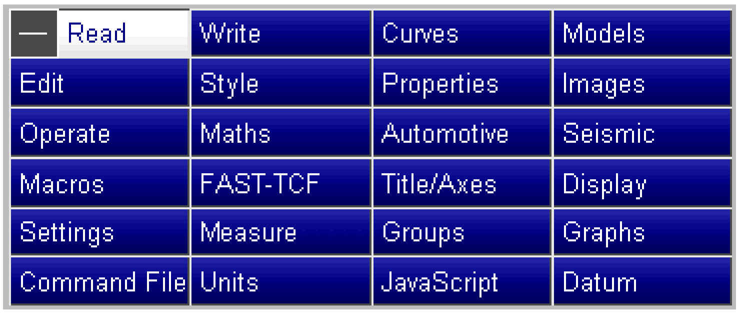
(LM) Gives the current program limits.

(FT) File tracking: lists the 20 files which have been accessed most recently by T/HIS, giving details of the type of file and whether it was read from or written to.

(EX) Exits (leaves) the program.

8. Main Menu

Main Menu



The **MAIN MENU** provides access to a number of separate menus that perform most of the operations available within T/HIS from reading in data to producing postscript laser files.

8.1. Selecting Curves

8.1.1. Input Curves

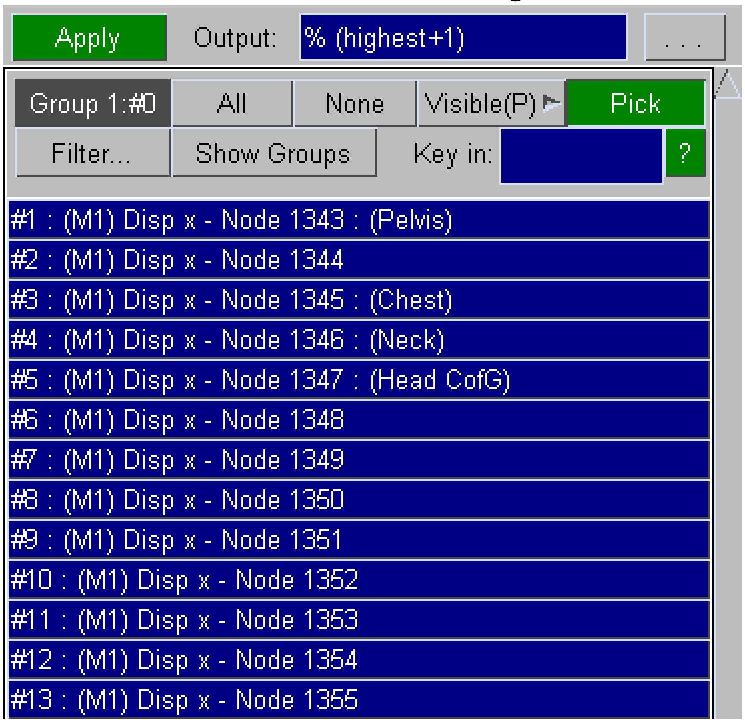
Input Curves

By Curve ID

A number of the menus require a range of curves to be selected. When a range of curves has to be selected a menu containing a list of the available curves will be displayed (see figure, below).

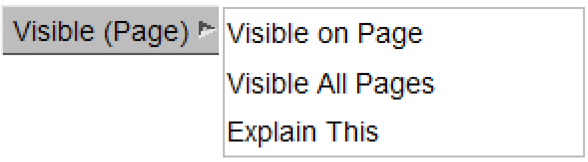
A range or curves may be selected by either

1. Click on the first item and hold down the mouse key, drag the mouse to the last item in the list. All items between the first and last including the first and last are selected.
2. Click on the first item, hold down the SHIFT key and click on the last item in the list. All items between the first and last including the first and last are selected.



VISIBLE (P)age

This option will select all of the curves that are unblanked in any graph on the current page.



VISIBLE (A)ll Pages

This option will select all curves that are unblanked in at least one graph.

PICK

Alternatively curves may be picked from the screen. With this option the left mouse button is used to select curves while the right button deselects curves. As each curve is selected/deselected its name and number will be reported to the user and it will be highlighted on the screen.

A range of curves can be selected interactively by dragging out an area on the screen while holding down the left mouse button.

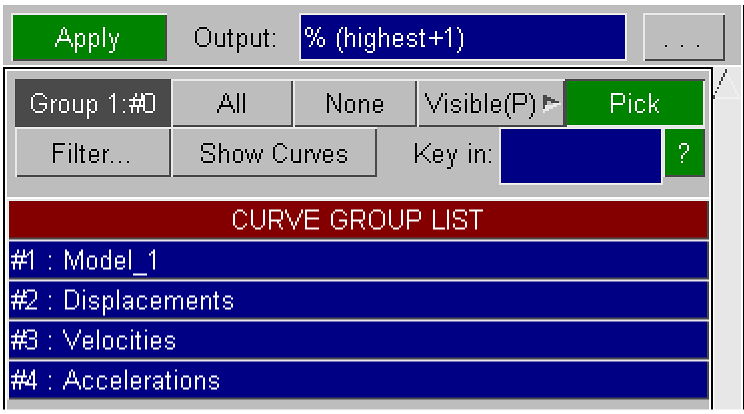
FILTER...

This option can be used to filter the list of curves displayed by model. When this option is selected a list containing all of the current models in T/HIS is displayed and the models can be selected or deselected. Any curves that belong to a deselected model will then be filtered out of the curve list.

SHOW GROUPS

This option will display a list of the currently defined curve groups instead of curves

By Curve Group



In addition to selecting individual curves it is also possible to select curves by Curve Group if they have been defined.

- If a curve is defined in more than one group then it will be selected if at least one of the groups is selected.

- If more than one group containing the same curve is selected then the curve will only be counted once as an input curve.

By Command Line

In command line mode a single curve may be selected by typing in a range. A valid syntax is:

A single curve number	e.g. `#27`
A "from":"to" range	e.g. `#10:#30` (no gaps, " `:` " mandatory)
A compound list in "(..)"	e.g. (`#1 #2 #10:#30 #3 #97`)

In all contexts the order in which a group is defined does NOT influence the order in which it is processed. It is ALWAYS processed in ascending sequential order.

Thus the addition operation

```
/OP ADD (#30 #20 #10) (#1 #2 #3) #40
```

will produce the results

```
#40 = #10 + #1
#41 = #20 + #2
#42 = #30 + #3
```

8.1.2. Output Curves

Output Curves

All operations that generate new curves must have a target curve defined. This must be one of the following:

`#nnn`	a specific curve number `nnn`
`#`	meaning "the lowest free curve"
`%`	meaning "the highest free curve"

In all cases output will start at the relevant curve number, however defined, and will rise sequentially with no gaps. This can cause an existing curve to be overwritten, or the output curve number to exceed the limit of 999. Both conditions are checked for: a warning is given if either will occur should the operation go ahead, and an opportunity given to modify or abort the pending operation.

There is a further output option that is only valid for operations where the input is a curve group:

`.`	meaning "overwrite the input curve(s)"

In this case the input curves are overwritten without warning. For example, this option might be used to integrate a set of curves, overwriting the original results with the integrated values.

Any curve number between 1 and 999 may be used as an input or output curve. It is not necessary to use curves sequentially; gaps are permitted in curve number usage. Therefore curves #1 and #10 can be used, for example, without having to use the intervening curves #2 to #9. Likewise, deleting a curve will no longer cause those above it to be renumbered downwards to fill the gap.

8.1.3. Curve Operations

Curve Operations

The functions available fall into four distinct groups,

1) Separate functions involving two groups of curves, where the result is of the form:

   <Rn> = <G1n> [OP] <G2n>

2) Separate functions involving only one group of curves, where the result is of the form:

   <Rn> = [OP] <G1n>

3) Single output from only one group of curves, where the result is of the form:

   <R> = [OP] <G1(1...n)>

4) Separate functions involving three groups of curves, where the result is of the form:

   <Rn> = <G1n> [OP] <G2n>[OP] <G3n>

   Currently the only function that has 3 curves groups as input is the VEC operation

1) Separate Functions On Two Groups

These functions display a menu in which **two** groups of curves may be selected, (see right).

You must define one or more curves in group #1, and group #2 must be:

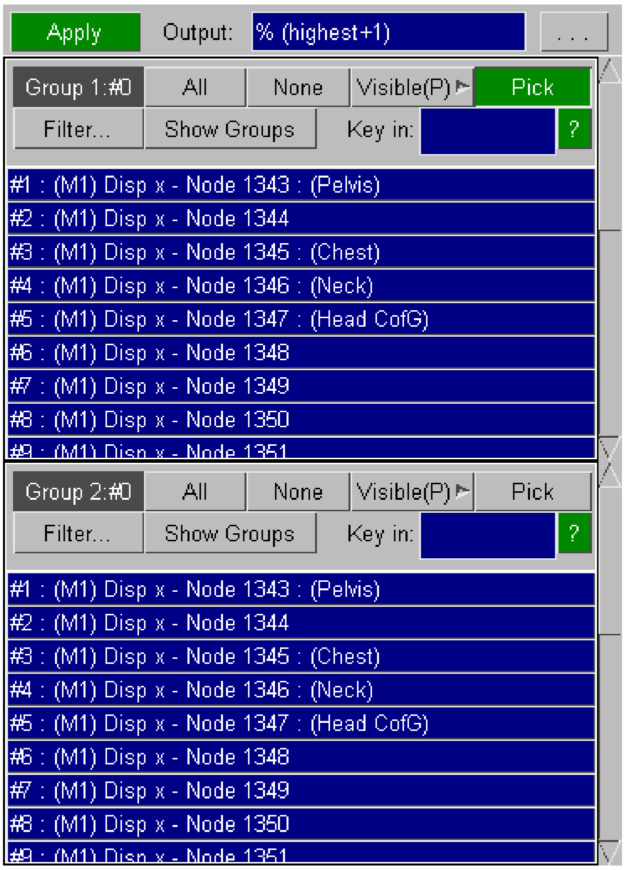
either A group of as many curves as there are in group #1.

or A single curve. Every curve in group #1 is applied to this curve.

or A constant value, entered in the **Key in** : text box.

You can pick curves in either group from their menus, or type a range into the **Key in:** box.

NOTE : the order in which they are processed is ascending sequential, **not the order in which you define them** .

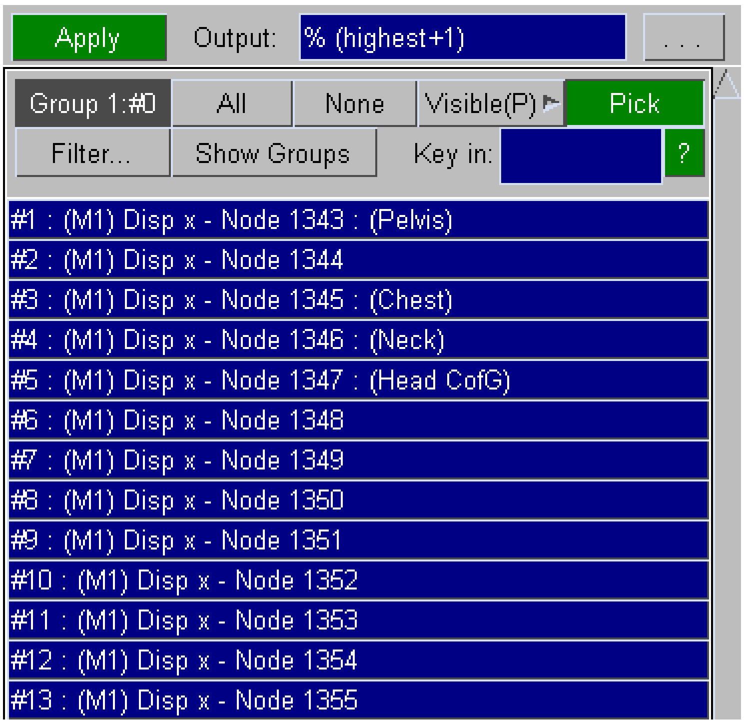


2) Separate Functions On A Single Group

These functions display a menu in which one group of curves may be selected, (see right).

Operations apply separately and uniquely to each selected curve.

As before, the order of processing is ascending sequential, not the order in which you define them.



3) Single Output From A Single Group

These functions require a single group of curves as input like the functions above. The output is a single curve.

8.2. READ Options

READ Options

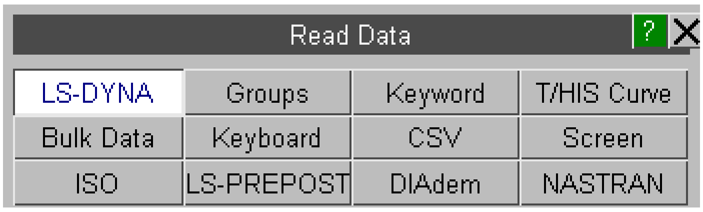
T/HIS can **READ** data from a number of sources including LS-DYNA binary output files, LS-DYNA ASCII files and tabulated x/y data files. In addition this menu allows data for new curves to be entered directly using the keyboard.

8.2.1. LS-DYNA

LS-DYNA

This topic includes:

Users are strongly advised to run each LS-DYNA analysis in a separate directory. Some of the default names for the files generated by LS-DYNA that T/HIS can read are not unique and T/HIS can not tell which files belong to which model. If you do read multiple models from the same directory T/HIS will generate a warning message if you read the same file for more than 1 model.



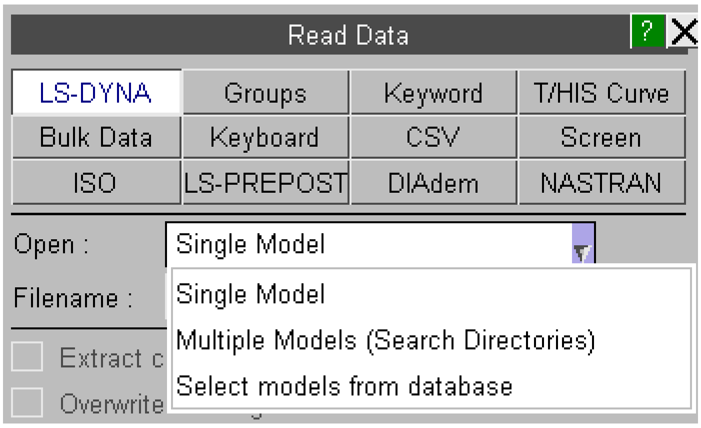
Selecting Models

There are three ways to select the LS-DYNA models that you want to read into T/HIS

(i) Select a single model (see Select Model)

(ii) Search directories for results and open open multiple models (see Search Directories Recursively)

(iii) Open a model database and select the models you want to read (see Select Models From Database)



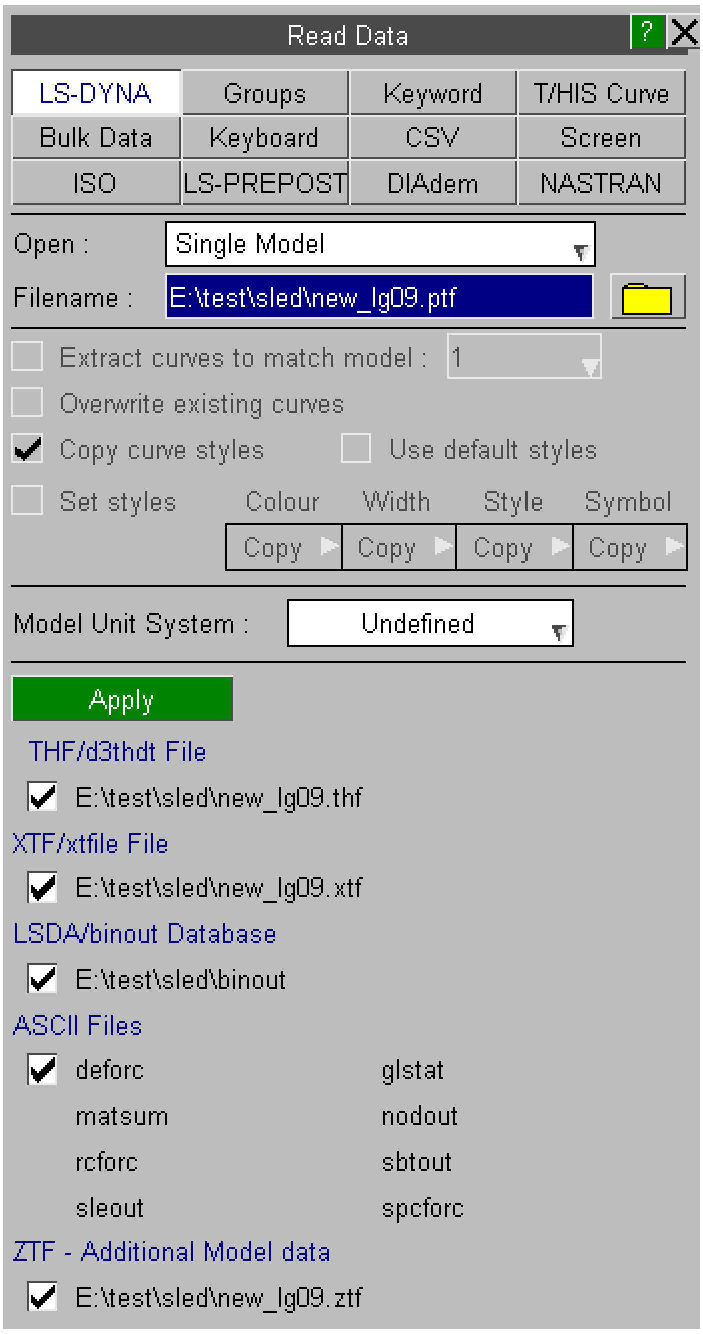
Select Model

Select ANY results file from a model. T/HIS will then search for all the results files in that directory produced by the same analysis as the selected file (as illustrated below) and display a list of all the files found. The user can then select which files to open. The default is to open all the available results files.

If you are using the Oasys Ltd. SHELL to submit jobs then the default filenames will be "jobname.thf", "jobname.xtf", "binout", "abstat" etc. If you use the standard LS-DYNA output file names then the filenames will be "d3thdt", "xtfile", "binout", "abstat".

The T/HIS preference option "this*file_names" can be used to set the default filenames that T/HIS searches for to either the ARUP set or the LST names.

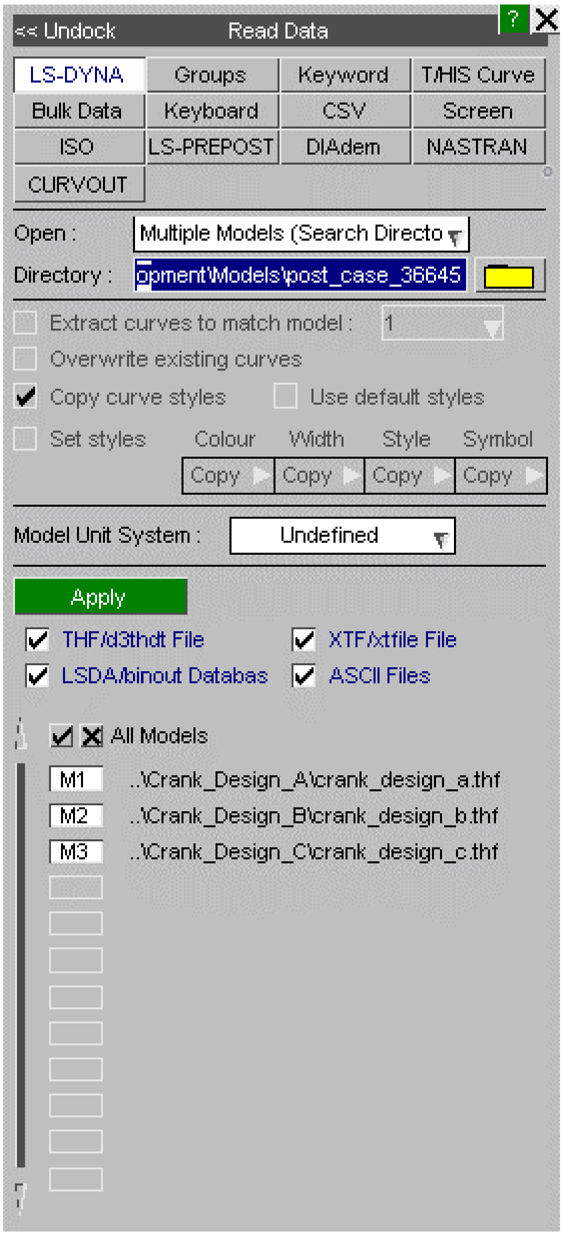
When the user selects **Apply** , the selected files are then opened and the contents scanned. After the files have been scanned the list of available data types will automatically be displayed (see Data Components)



Search Directories Recursively

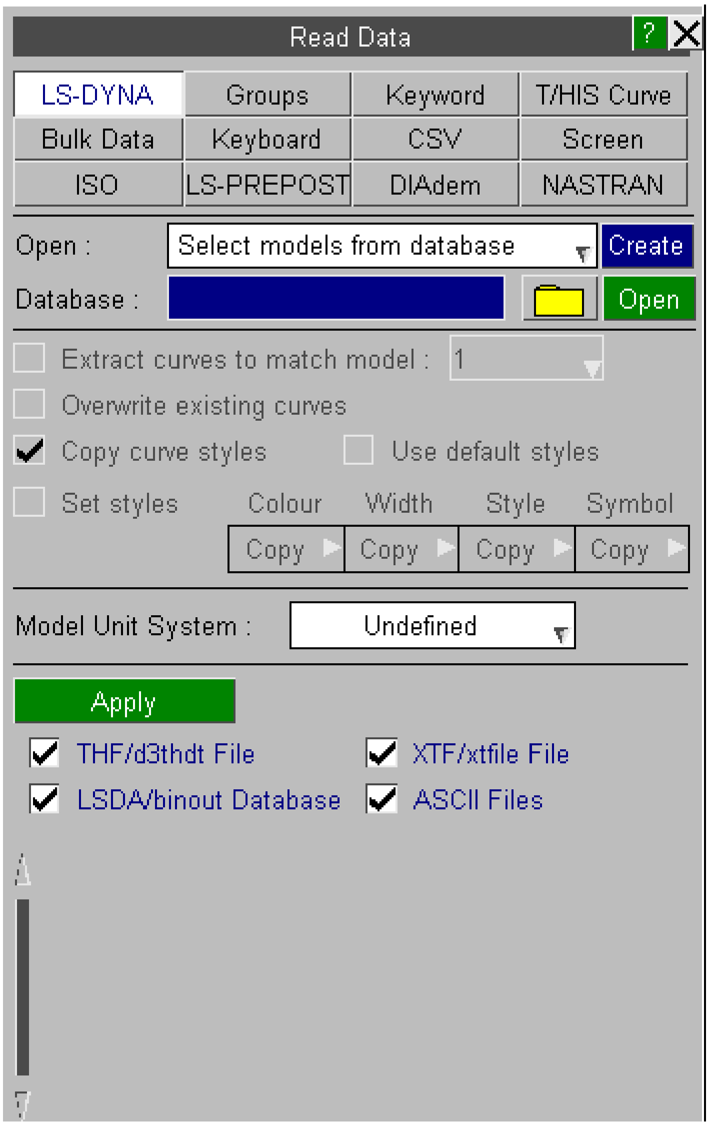
Multiple models can be opened by using the option to search directories recursively.

After a directory has been specified T/HIS will display a list of all the models it can find in the directory structure and each file can be selected. The order in which the models are read in can be specified by selecting the models in the order required. The selection buttons will display the model number that each model will be read into. The model numbering begins from the next free model number and is then sequential.



Select Models From Database

From T/HIS 10.0 onwards users can select models from a model database. The database file is an XML format file that contains information on where models are located along with a brief description of each model, (see below for more details on the file format).



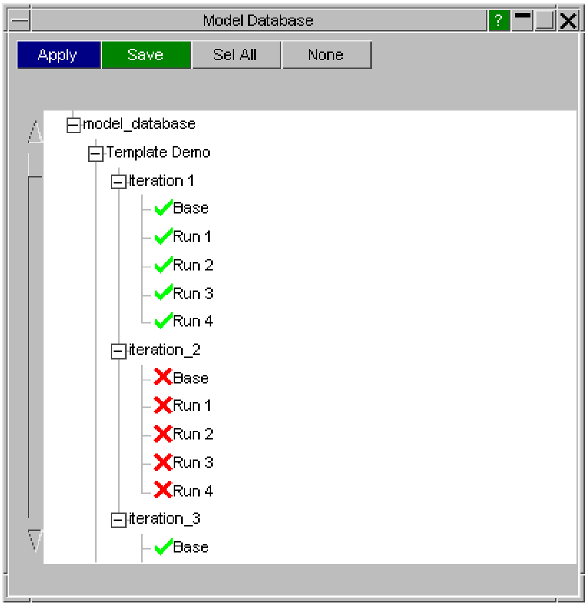
To select a model database either enter it's name in the text box or use the file selector.

The default model database can be specified as a command line argument (see Command Line Options for more details). The default database filename and location can also be specified in the preference file (see Format of the oa_pref File for more details)

```
this*database_dir:
this*database_file:
```

After a database file has been selected it's contents will be read and T/HIS will display a Tree Like menu showing the contents of the database.

As each item is displayed T/HIS will check to see if the files that it refers to exist.



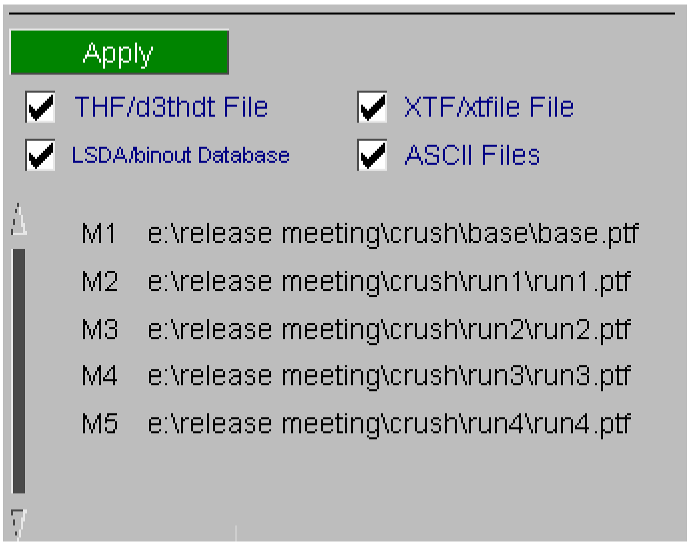
If a file does exist then a green tick will be displayed

If a file does not exist then a red cross will be displayed

The number of levels in the database that are automatically expanded when it is first displayed can be specified in the preference file (see Format of the oa_pref File for more details)

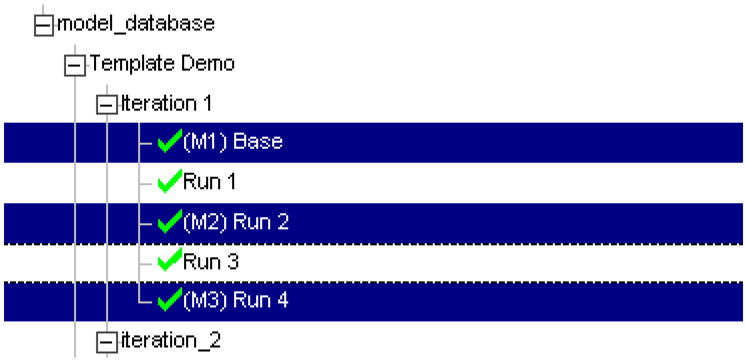
```
this*database_expand:
```

After selecting the required models use **Apply** to close the database window and return to the main menu where the selected models will be displayed along with the model numbers they will be read in as.



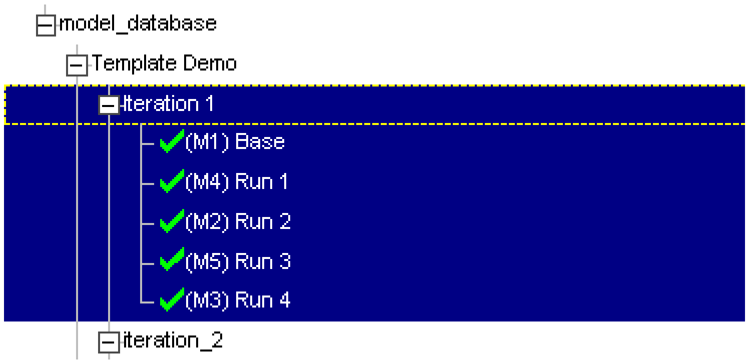
Selecting Models

Models can be selected and deselected by clicking on each row. Multiple models can be selected by clicking on the 1st model and holding down SHIFT while selecting the last model in the range.



As each model is selected, the model number that it will be read in as is automatically displayed alongside the model description.

A complete branch can be selected/deselected by selecting the branch label (Iteration 1).



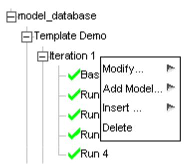
Modifying the Database

Database entries can be added, removed and modified by right clicking on a branch label or a model description

Right clicking on a branch label will display 4 options

Modify ... Modify the branch label.

Add Model ... Add a new model into the selected branch. A menu will be displayed to select a new model and to define the model description that is displayed for the new model.

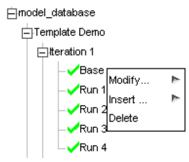


Insert ... Insert a new branch within the selected branch.

Delete	Delete this branch and everything within it.

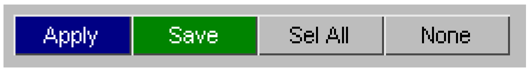
Right clicking on a model description will display 3 options

Modify ...	Modify the model location and description.
Insert ...	Insert a new branch. The selected model will be moved into the new branch.
Delete	Delete the model



Saving the Database

After modifying the database use the **Save** option to save the changes for future sessions.



Creating a new Database

If you do not have a database or if you want to create a new one then T/HIS can create the new database for you. To create a new database click the **CREATE** button and simply enter the name of the new database file in the text box that appears, T/HIS will then check that the file does not already exist and if it doesn't it will create a new empty database.

Alternatively if you type in the name of a file in the main Open Plot File window that does not exist then T/HIS will ask if you want to create a new empty database using that filename.

Once you have done this you can use the Modify options above to add items into the database and then save the file before exiting.

Database Format

The Model Database uses an ASCII XML file format.

All items with the database are either branches or models. Each database entry has an **XML name** and a **LABEL element.** Models also contain a model element that contains the full pathname of one of the files belonging to the model.

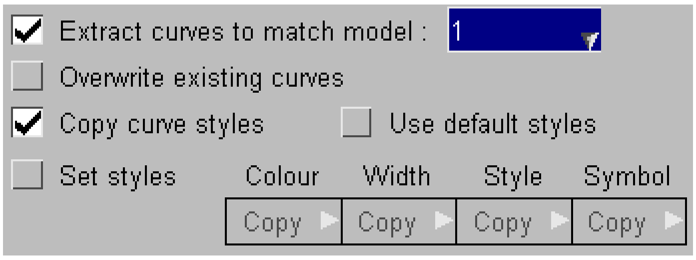
The **XML name** should be unique and should obey the following rules

- Names can contain letters, numbers, and other characters
- Names must not start with a number or punctuation character
- Names must not start with the letters xml (or XML, or Xml, etc)
- Names cannot contain space

The **LABEL** is the string used to display an item within the tree view. Unlike the **XML name** the **LABEL** can contain any ASCII character.

```
<model_database version="10.000000">
    <Template_Demo label="Template Demo">
        <iteration_1 label="Iteration 1">
            <base label="Base"
                model="e:\release\meeting\crush\base\base.ptf"/>
            <run_1 label="Run 1"
                model="e:\release\meeting\crush\run1\run1.ptf"/>
            <run_2 label="Run 2"
                model="e:\release\meeting\crush\run2\run2.ptf"/>
            <run_3 label ="Run 3"
                model="e:\release\meeting\crush\run3\run3.ptf"/>
            <run_4 label="Run 4"
            model="e:\release\meeting\crush\run4\run4.ptf"/>
        </iteration_1>
        <iteration_2 label="Iteration 2">
            <base label="Base"

          model="e:\test\crush2\base\base.ptf"/>
            <run_1 label="Run 1"

          model="e:\test\crush2\run1\run1.ptf"/>
            <run_2 label="Run 2"

          model="e:\test\crush2\run2\run2.ptf"/>
            <run_3 label="Run 3"

          model="e:\test\crush2\run3\run3.ptf"/>
            <run_4 label="Run 4"

          model="e:\test\crush2\run4\run4.ptf"/>
            </iteration_2>
        </Template_Demo>
</model_database>
```

Automatic extraction of model results



When a second or subsequent model is opened in T/HIS this option can be used to automatically generate the same curves as those already read from another model.

This option can also be used if a model is re-read into T/HIS to extract the same curves as those that had already been read from the model.

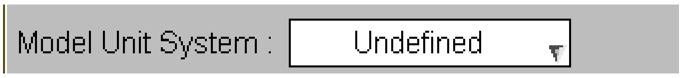
By default this option will attempt to generate curves that match those already read from model 1. If results have already been read from more than one model then the model to match the curves form can be set to any of the existing models.

This option can be used to overwrite the existing curves from a model. If a model has been read into T/HIS and curves have been read from the model while the analysis was still running then this option can be used to automatically update the curves.

When the curves from the 2nd or subsequent model are automatically generated then by default they will be given the same colours, and line styles as the curves in the original model.

Instead of copying the curve styles a new style for all the automatically generated curves can be specified. This make it very easy to set the same style for all of the curves that are read from a model. Alternatively the default T/HIS curve styles can be used.

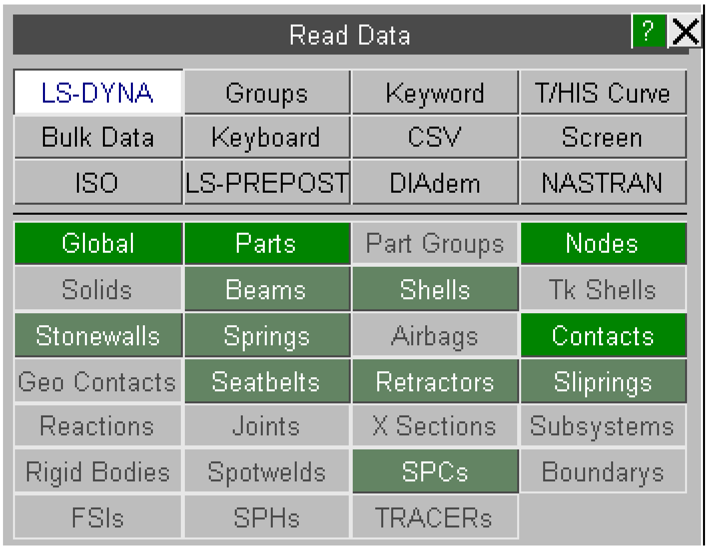
Model Unit System



This option can be used to set the default Unit System that will be applied to the model. For more information on Units see UNITS

Entity Types

Items are shown in bright green if they occur in all the models that have been read into T/HIS and are currently selected. If they occur in at least one model but not all models then they are shown in a duller green (in the case shown in the adjacent picture Beams, Shells, Stonewalls, Springs, Seatbelts, Retractors, Sliprings and SPCs can be found in some but not all of the models).



Data Components

When reading data from any of the LS-DYNA binary files or the LS-DYNA ASCII files multiple components and entities may be selected at the same time.

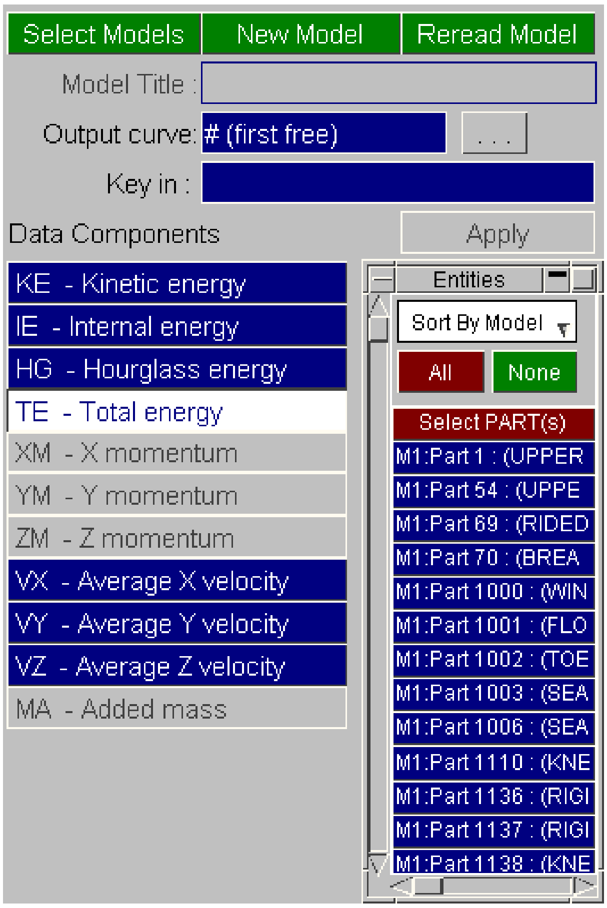
Each data extraction menu consists of a list of available data components and a list of entities.

Data Components

Individual data components can be selected using the mouse. If a component has been selected and a second item is subsequently selected the first item will be deselected.

Multiple components may be selected by

1. Holding down the **CTRL** key when selecting items to add individual items to the list of selected components.
2. Holding down the **SHIFT** key when selecting items to add a range of items to the list of selected components.
3. Clicking on the first item to be selected and then dragging down the list of items without letting go of the mouse button.

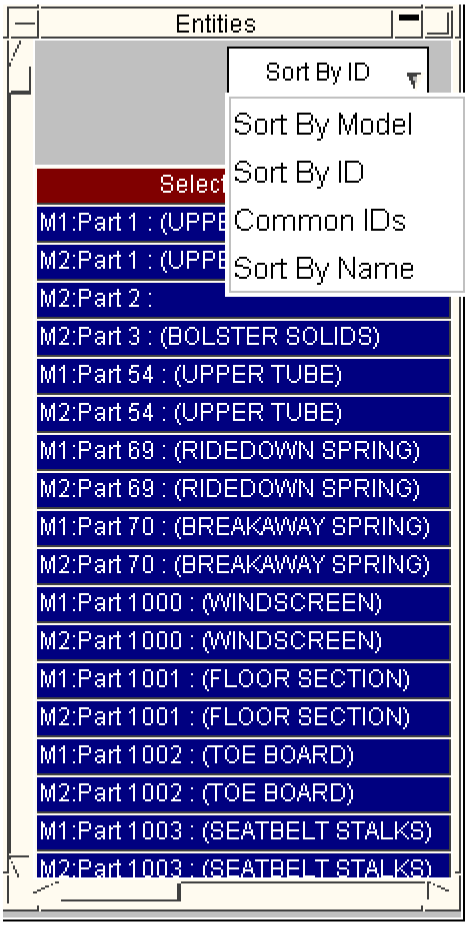


Entities

Individual entities can be selected/deselected using the mouse.

Multiple entities may be selected by

1. Holding down the **CTRL** key when selecting items to add them to the list of selected entities.
2. Holding down the **SHIFT** key when selecting items to add a range of items to the list of selected entities.
3. Clicking on the first item to be selected and then dragging down the list of items without letting go of the mouse button.



Entities can be sorted in four ways:

Sort by model	will list all entities in the lowest number model in order of ascending ID number, then all entities in the next-lowest model, and then move through the rest of the models in ascending order.
Sort by ID	will list all entities in ascending order showing the model ID for each entity
Common IDs	will list only the entities with IDs that are common to all models without showing the model ID's
Sort by Name	arranges the entities in alphabetical order based on their names.

Surfaces/Integration Points

Some BEAM, SHELL, and THICKSHELL data components can be read from multiple integration points.

If a data component is available for multiple integration points then an additional **Select Surface** options is displayed.

Select Surface

This option will display a separate menu listing all of the integration points that are available to read data from.

For Shell and Thick Shell elements the menu will include all of the through thickness integration points plus 3 additional options; TOP, MIDDLE and BOTTOM.

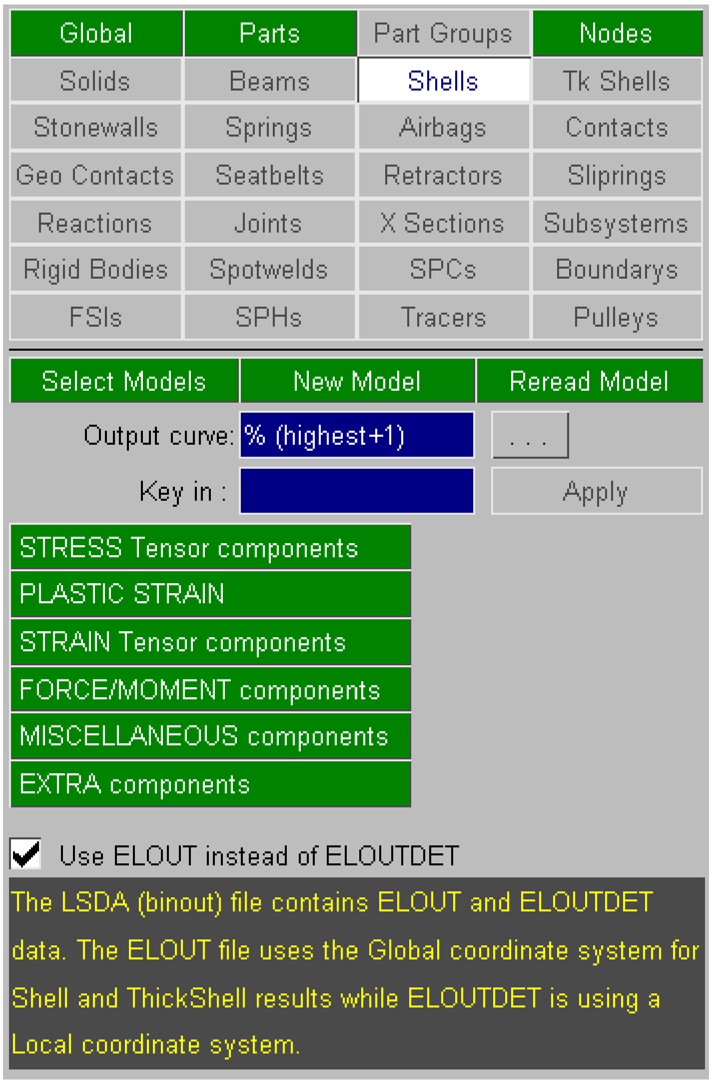
For Beam elements the menu will just display the integration points.

In plane int points

In addition to the through thickness integration points recent versions of LS-DYNA can also output data for multiple in-plane integration points for fully integrated Shell and Thick Shell elements. If T/HIS can identify that the model contains data for multiple in-plane integration points then these options can be used to select the individual in-plane integration points or to average the 4 in-plane points.

For more information on selecting integration points for beams, shells and thick shells see Beam Data Components , Shell Data Components and Thick Shell Data Components.

Shell and ThickShell Data Components



If Shell and ThickShell data is being read from the LDSA (binout) file then the file can contain data components in both the ELOUT and ELOUTDET branches.

By default T/HIS uses the data from ELOUTDET as ELOUT only contains a subset of the data in ELOUDET.

In some versions of LS-DYNA it is possible to change the Shell and ThickShell data components written to the ELOUT so that they are defined using the global coordinate system (see EOCS on *CONTROL_OUTPUT) instead of the default element local coordinate system. If this option is used then only the ELOUT file is modified, the ELOUDET file is still written using the local coordinate system.

Use ELOUT instead of ELOUTDET

If T/HIS detects that the LSDA file contains both ELOUT and ELOUDET and that they are using different coordinate systems then this option can be used to force T/HIS to use the ELOUT file data components using the global coordinate system.

This option can also be set via the preference file (see Appendix H for more details) and via the command line (see Command Line Options)

8.2.2. GROUPS

GROUPS

This option can be used to read a file containing PART group definitions. If a model is read in which contains PART information then the PART groups can be used to read in and sum energies for a group of PARTS in one go.

The 1st time T/HIS finds a group file (groupXXX.asc) in a directory it will automatically read the file and create the PART groups.

After reading the 1st group file T/HIS will by default ignore any other group files it finds in directories when it opens a model.

This option can be changed as follows.

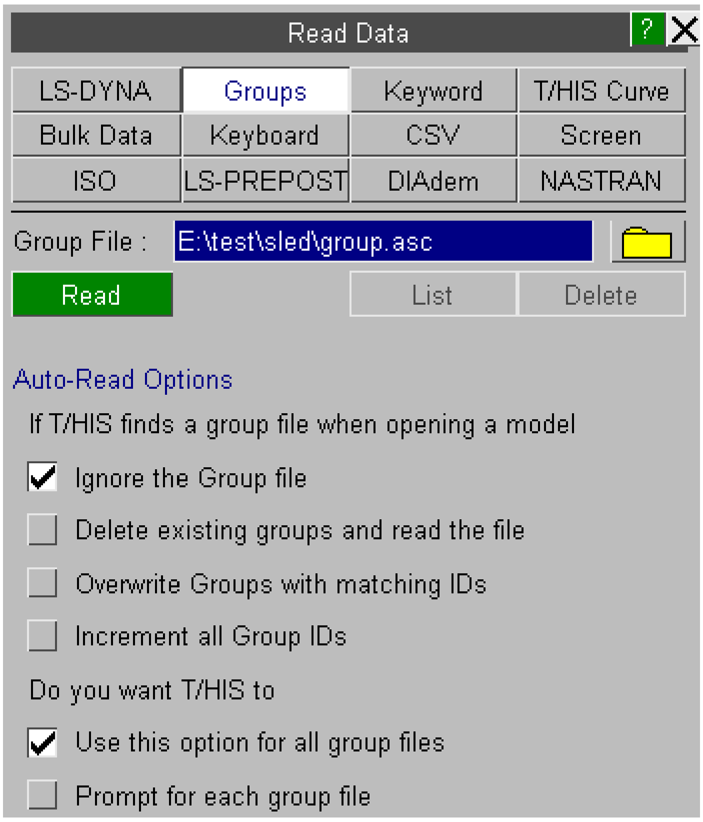
Ignore	This option (the default) will make T/HIS ignore any more group files it finds
Delete	If T/HIS finds a group file when a new model is read in then all existing group definitions will be deleted before the new file is read
Overwrite	If T/HIS finds a group file when a new model is read in then all the new group definitions will be added to the existing ones. If the new file contains a group with the same ID as an existing group then the old definition will be overwritten.
Increment	If T/HIS finds a group file when a new model is read in then all the new group definitions will be added to the existing ones but the group ID's will be incremented to ensure that they do not clash with existing ones.

The default option can be changed using the preference option

```
this*read_group_files:
```

(see Format of the oa_pref File for more details)

If the option to read groups files is set and the directory contains more than one group file then T/HIS will use the newest file.

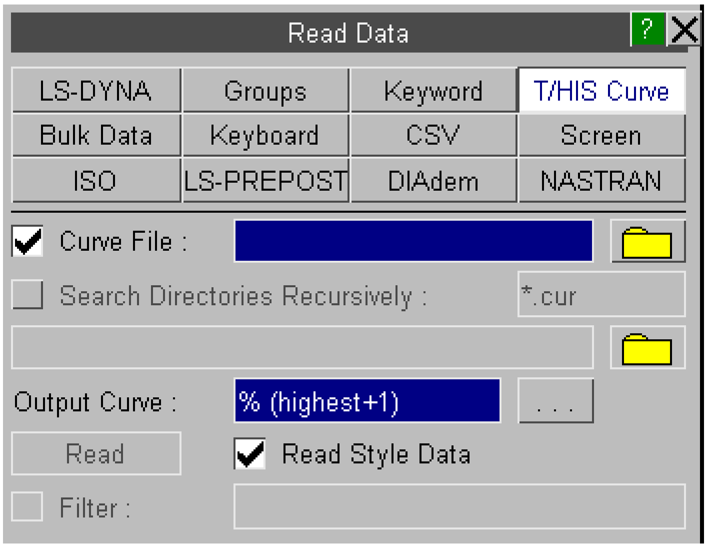

Read Data
?
LS-DYNA
Groups
Keyword
T/HIS Curve
Bulk Data
Keyboard
CSV
Screen
ISO
LS-PREPOST
DIAdem
NASTRAN
Group File :
E:\test\sled\group.asc
Read
List
Delete
Auto-Read Options
If T/HIS finds a group file when opening a model
Ignore the Group file
Delete existing groups and read the file
Overwrite Groups with matching IDs
Increment all Group IDs
Do you want T/HIS to
Use this option for all group files
Prompt for each group file

8.2.3. T/HIS Curve

T/HIS Curve

This option can be used to read in curves stored in T/HIS curve file format (see Appendix B for more details)

By default this option can be used to select a single file. After selecting the file it will automatically be opened and read and all of the curves in the file will be read in.



In addition to reading a single file this option can also be used to search directories recursively for multiple files.

After the search has finished a list showing all of the files that have been found will be displayed so that multiple files can be selected and read in one operation.

By default T/HIS will search for files with the file extension .cur, this can be changed if required.

In addition to changing the default file extension the list of files can also be filtered. The filter string can contain the following wildcards

*	matches multiple characters
?	matches a single character

Note: The filtering ignores case.

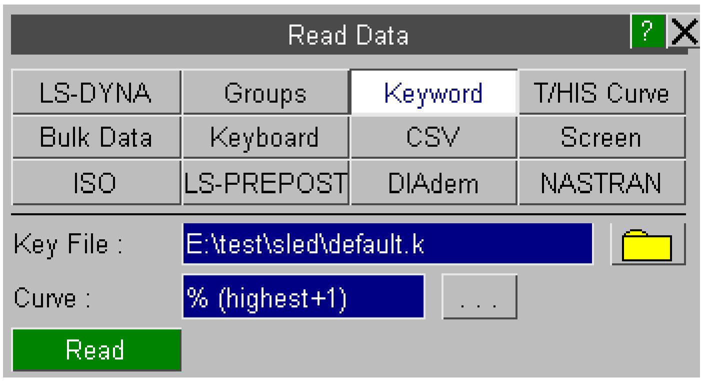
8.2.4. KEYWORD

KEYWORD

Read data into T/HIS from an LS-DYNA KEYWORD input file. All X/Y data defined using **`*DEFINE_CURVE`** will be read in from the specified input file. Any X and Y axis scaling or offsets defined within the **`*DEFINE_CURVE`** definition will be applied to the X,Y as it is read in. If the **`_TITLE`** option has been used the the title will be used as the curve label otherwise the curve ID number will be used.

From version 9.3 onwards this option will also process any files specified using the **`*INCLUDE`** option.

`/re kw "filename"`	read all curves from KEYWORD input file "filename"

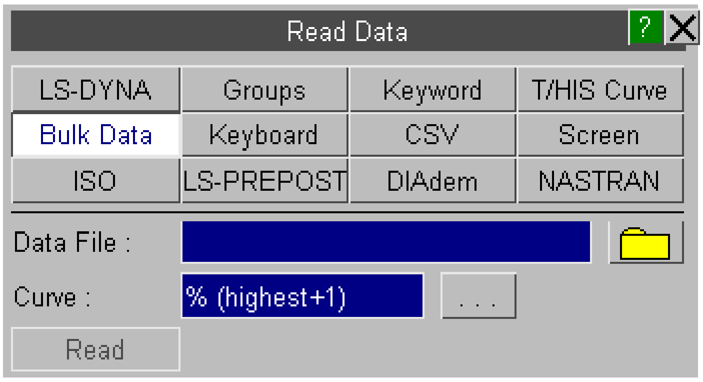


8.2.5. BULK

BULK

Read data into T/HIS from a Bulk Data file. The format of a Bulk Data file is described in Appendix C .

`/re bd "filename"`	read all curves from Bulk Data file "filename"



8.2.6. KEYBOARD

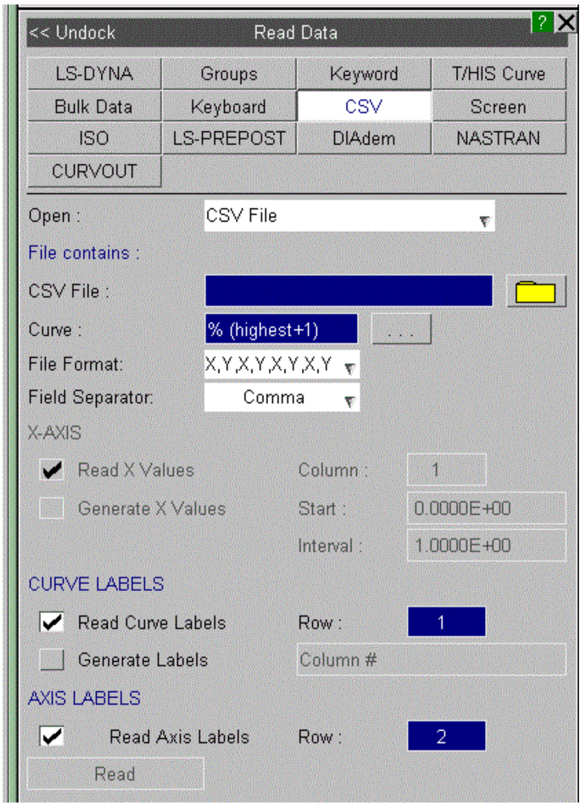
KEYBOARD

Key in curve information directly. A dialogue window is displayed upon requesting this option where the user will be prompted for title, x and y axis labels, a curve identifier and then a series of points. Once all the points required have been entered carriage return should be pressed. The user will then be prompted for the curve or file in which to store this data : # means use the next free curve.

8.2.7. CSV

CSV

The **CSV** menu (see right) can be used to read comma separated variable file(s) into T/HIS. This menu allows to read single CSV file or all the CSV files in a selected directory both recursively and non-recursively.

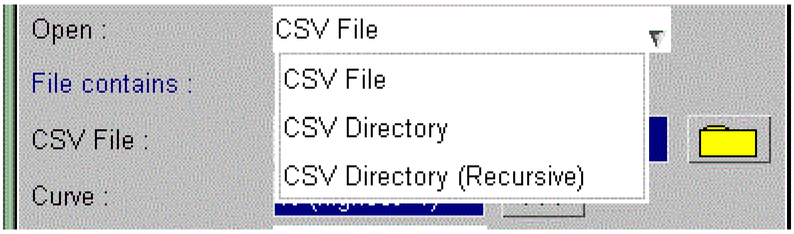


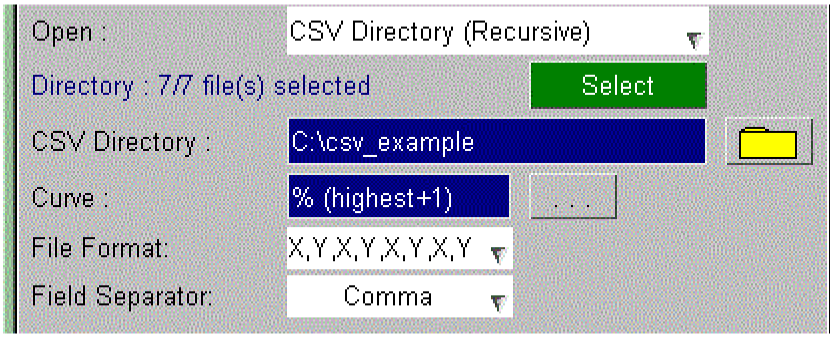
Each file may contain up to 1000 columns of data (separated by commas).

The maximum line length supported by this option is 10240 characters.

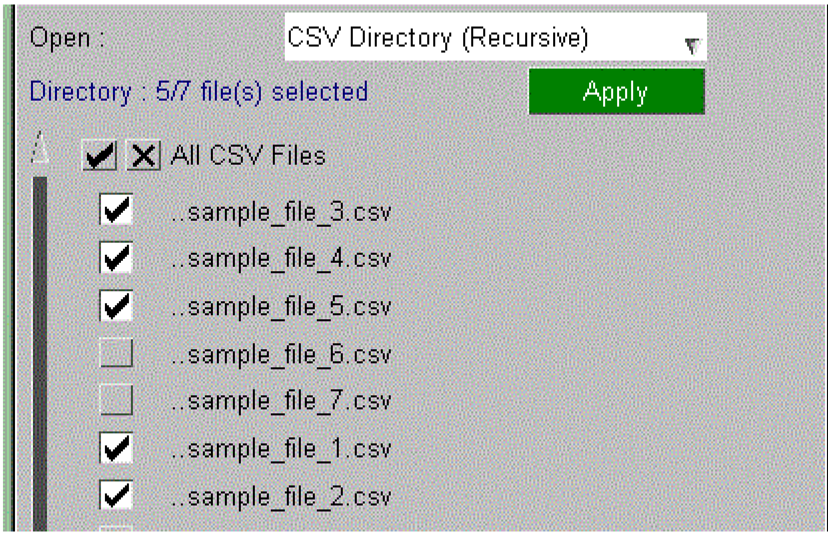
CSV files written from the D3PLOT Write Menu are automatically detected by T/HIS and sets the appropriate read options. The options can be changed, but the data may not read in as expected. Both the **Write**->**Entity** and **Write**->**Scan** formats are supported. The first column of data containing the entity IDs is ignored for both formats. For files written from the **Write**->**Scan** menu the third column is ignored as this also contains entity IDs.

The CSV menu can also read in multiple CSV files in a given directory and also all the sub-directories recursively by changing the **Open** option from **CSV File** to either **CSV Directory** or **CSV Directory (Recursive)**.





For both **CSV Directory** and **CSV Directory (Recursive)** options, CSV menu first scans through the directory and specifies the number of files it has found. By default all the files found will be selected. Users can filter out the files they want to read by clicking on the **Select** button.



On clicking the **Select** button, CSV menu will display the list of all the CSV files found in the specified directory. Users can select which CSV files they would like to read in.

File Format This option can be used to change the CSV file format between the X,Y,X,Y,X,Y format where alternate columns are the X and Y values for each curve and the X,Y,Y,Y format where there is a single column containing the x-axis values for all the curves.

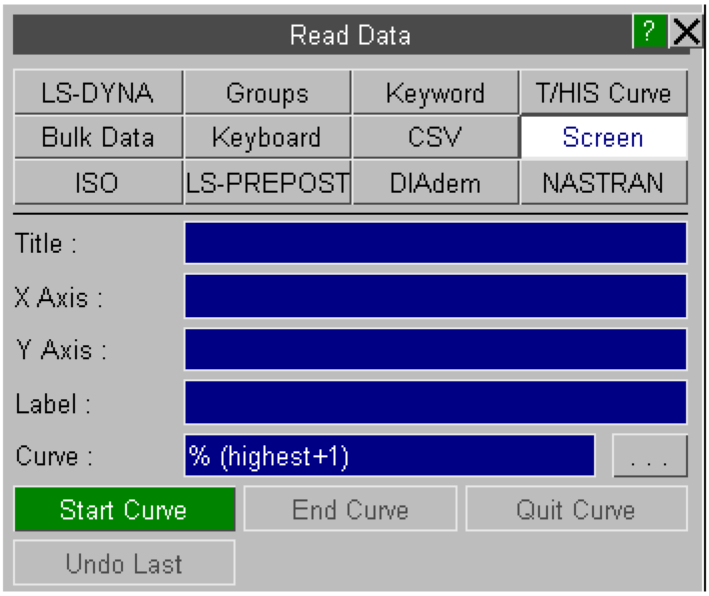
Field Separator By default T/HIS assumes that the columns of data are separated by commas, this option can be used to change the field separator to either a Tab or Spaces. If the 'Space' option is used then multiple spaces are counted as a single field separator. If curve or axis labels are defined in the file and they contain spaces then they need to be enclosed in pairs of " quotes. The default field separator can specified in the

	preference file (see Format of the oa_pref File for more details) `this*csv_separator:`
Read X Values	This option can be used to specify a column within the file that contains the X-axis data values that should be used for all of the other columns of data.
Generate X Values	This option can be used to automatically generate the X-axis values if none of the columns within the file contain the data.
Read Labels	This option can be used to specify a row within the file that contains labels for each of the columns of data that can be used as the curve labels within T/HIS.
Generate Labels	This option can be used to automatically generate labels for each set of data. A single string can be specified which will then have the column number appended to it to generate unique labels.
Read Axis Labels	This option can be used to specify a row within the file that contains the axis labels.

8.2.8. SCREEN

SCREEN

The **SCREEN** menu (see right) can be used to interactively create a curve T/HIS by selecting points using the mouse.



Start Curve	This option will start point selection process. Once you have started creating a curve all the other T/HIS menus will be disabled until you end the point selection using either the **End Curve** or **Quit Curve** options. ***Dynamic viewing will still be available.***
End Curve	This option will end the current curve creation and save the curve.
Quit Curve	This option will end the current curve creation without saving the curve.
Undo Last	This option can be used delete the last point created (the middle mouse button will also delete the last point).

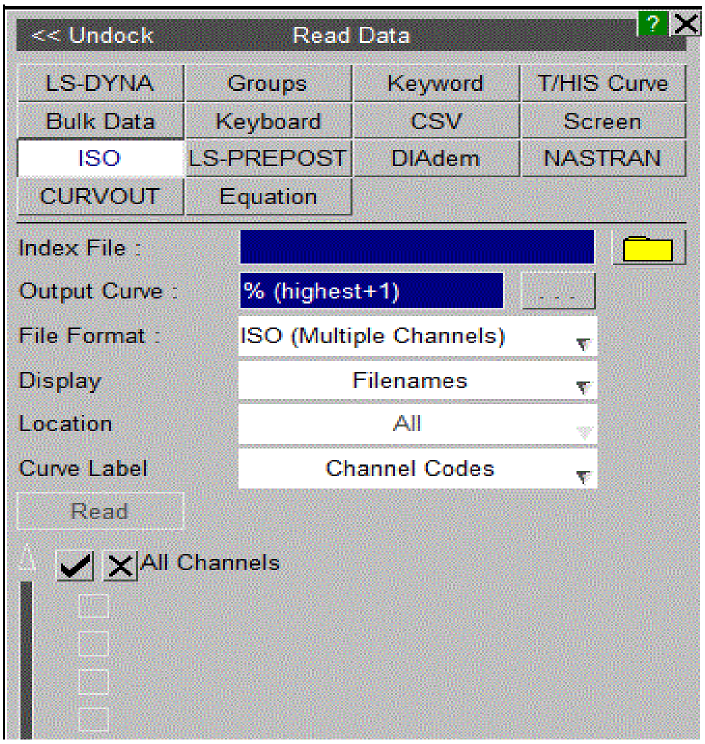
8.2.9. ISO

ISO

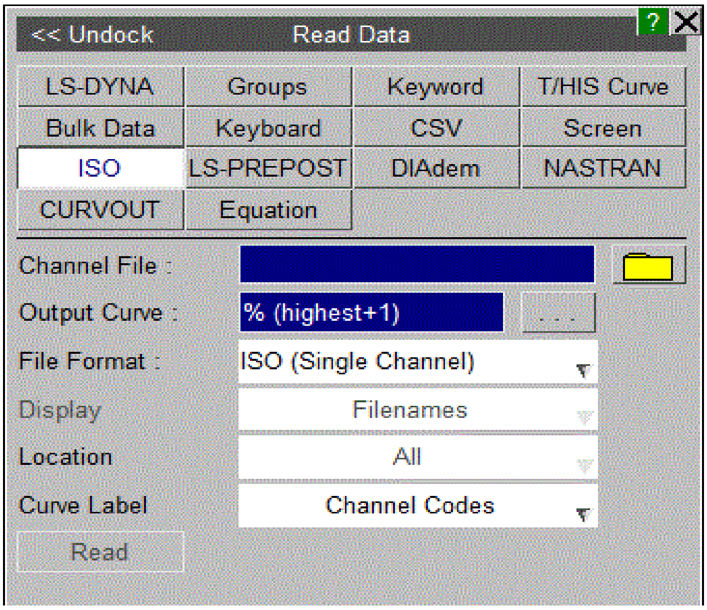
This option can be used to read in curves from files written using the ISO/TS 13499:2003 file format. Two versions of the format are supported; v1.6 and v2.0.

The default option in T/HIS is to read in an Index file containing information on multiple channels. After the file has been opened and read a list of all the available channels will be displayed so the required channels can be selected.

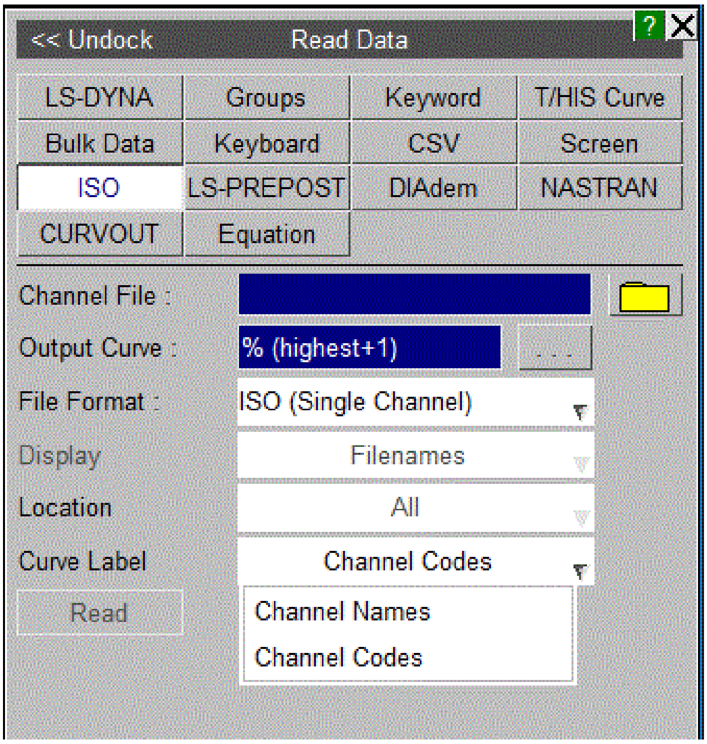
When listing the channels the default is to display the filenames for each of the channel files. Alternatively the channel names (read from the Index) file can be displayed.



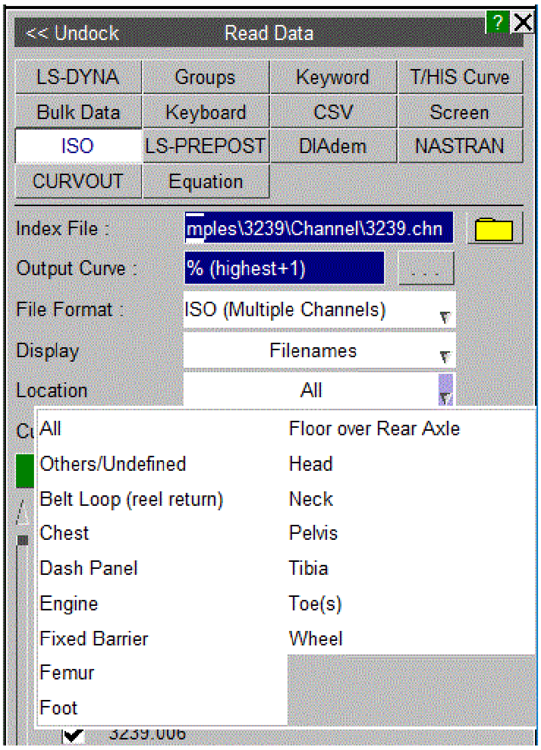
Instead of reading an Index file and then selecting which channels to read individual channel file can be read in directly.



For Curve labels the default is set to Channel Codes. Alternatively it can be changed to Channel Names.



A dynamic location pop up has been added. The options displayed in this popup will be according to the options available in the channel list.



<< Undock
Read Data
LS-DYNA
Groups
Keyword
T/HIS Curve
Bulk Data
Keyboard
CSV
Screen
ISO
LS-PREPOST
DIAdem
NASTRAN
CURVOUT
Equation
Index File :
mples\3239\Channel\3239.chn
Output Curve :
% (highest+1)
File Format :
ISO (Multiple Channels)
Display
Filenames
Location
All
All
Others/Undefined
Belt Loop (reel return)
Chest
Dash Panel
Engine
Fixed Barrier
Femur
Foot
Floor over Rear Axle
Head
Neck
Pelvis
Tibia
Toe(s)
Wheel

8.2.10. LS-PREPOST

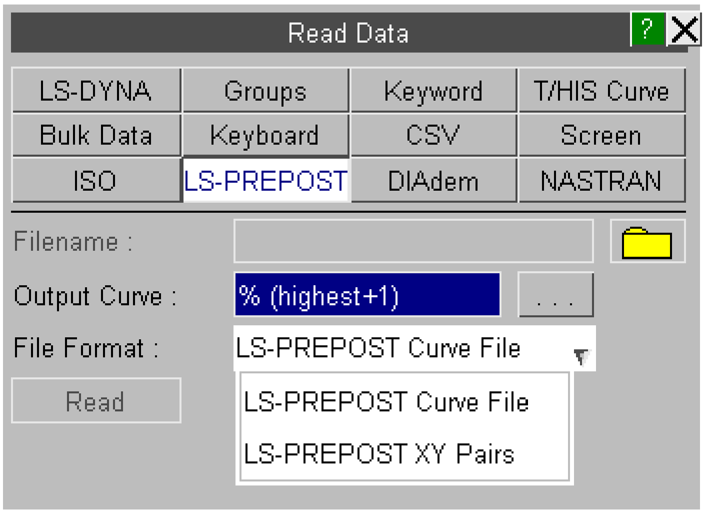
LS-PREPOST

This option can be used to read in curves from files written out from LS-PREPOST.

Two different file formats are supported

LS-PREPOST Curve Files

LS-PREPOST XY Pairs



8.2.11. DIAdem

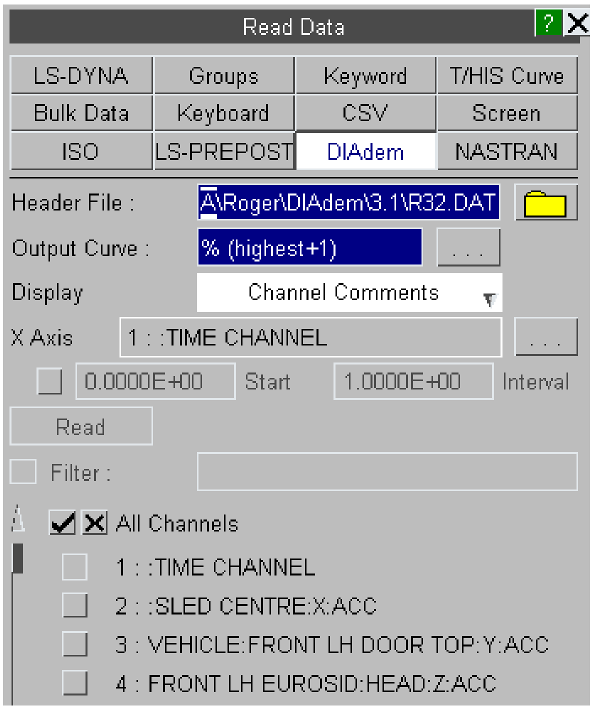
DIAdem

This option can be used to read in data from DIAdem format data files. After selecting a DIAdem header file a list of all the available channels will be displayed so the required channels can be selected.

Version 11.0 of T/HIS supports the following DIAdem data file formats

REAL32
REAL48
REAL64
INT16
INT32
WORD8
WORD32
ASCII

The MSREAL32, TWOC12 and TWOC16 are not supported.



By default T/HIS will display the channel comments (header block 201) for each channel. This can be changed to the channel names (header block 200) using the popup menu if required.

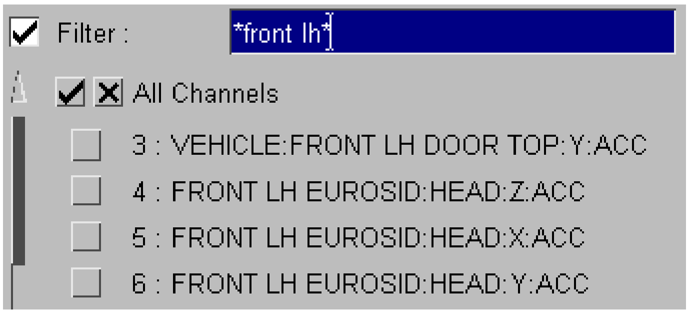
When channels are read in this option is also used to create the labels for each curve.

As well as displaying either the channel comments or the channels names the list of channels can also be filtered if required .

The filter string can contain the following wildcards

*   matches multiple characters
?   matches a single character

Note: The filtering ignores case.



Normally one of the DIAdem data channels contains the x-axis (time) values. By default T/HIS assumes this is channel 1 but this can be changed using the button labelled **...**

If none of the channels contain the x-axis values then a start value and an increment can be specified to generate curves with evenly spaced x-axis values.

Supported DIAdem header file blocks

The following DIAdem header file data blocks are supported. All other data blocks are ignored.

GLOBAL HEADER

111	Value for NoValues in the data file
112	Interchange high- and low-bytes

CHANNEL HEADER

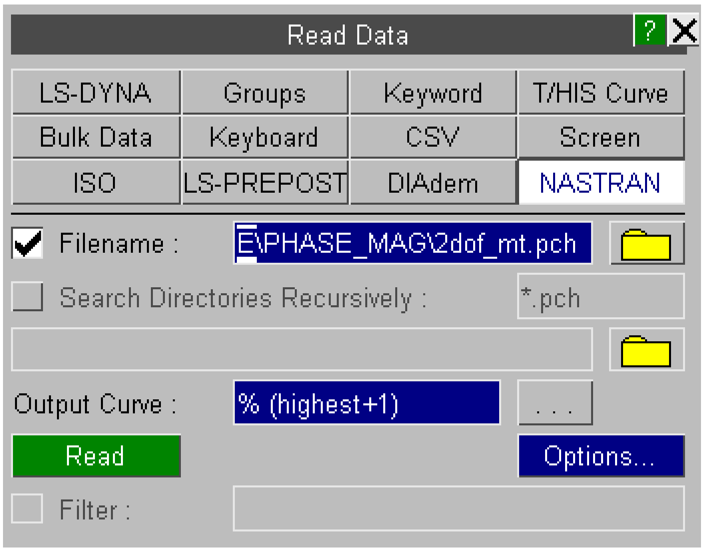
200	Channel name
201	Channel comment

210	Channel type
211	File from which channel data is read
213	Method of storing the data
214	Data type
220	No. of values in the channel
221	Pointer to the 1st value in the channel
222	Offset for ASCII block files Offset for binary block files with header
223	Local ASCII-pointer in the case of ASCII block files
230	Separator character for ASCII-block files
231	Decimal character in ASCII-files
232	Exponential character in ASCII-files
240	Exponential character in ASCII-files
241	Step width / Factor
252	Keyword for NoValues in the channel
254	Value for NoValues in the channel

8.2.12. NASTRAN

NASTRAN

This option can be used to read in data from from NASTRAN PCH files.



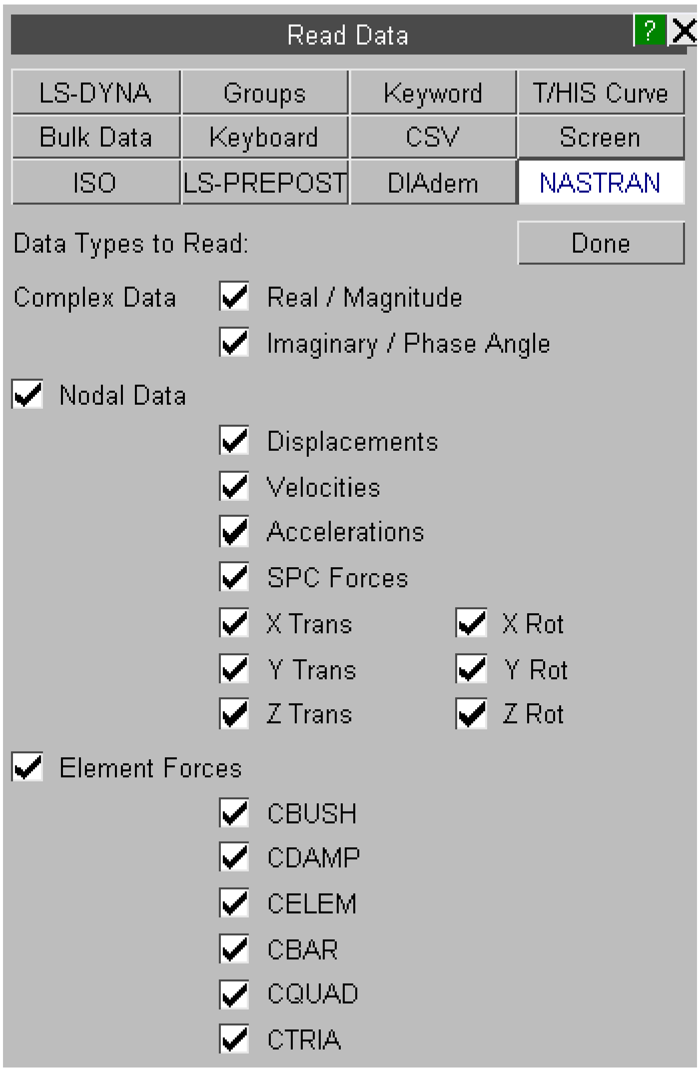
Currently the following types and data components are supported along with the SORT1, SORT2 and XYPUNCH file formats

Nodal	Displacements
Nodal	Velocities
Nodal	Accelerations
Nodal	SPC Forces
CBUSH	Element Forces
CDAMP	Element Forces
CELEM	Element Forces
CBAR	Element Forces
CQUAD	Element Forces
CTRI	Element Forces

By default T/HIS will read in every curve that it finds in the file so if you read in a file containing nodal displacements from a SORT2 format file you will end up with 12 curves being produced in T/HIS for each node.

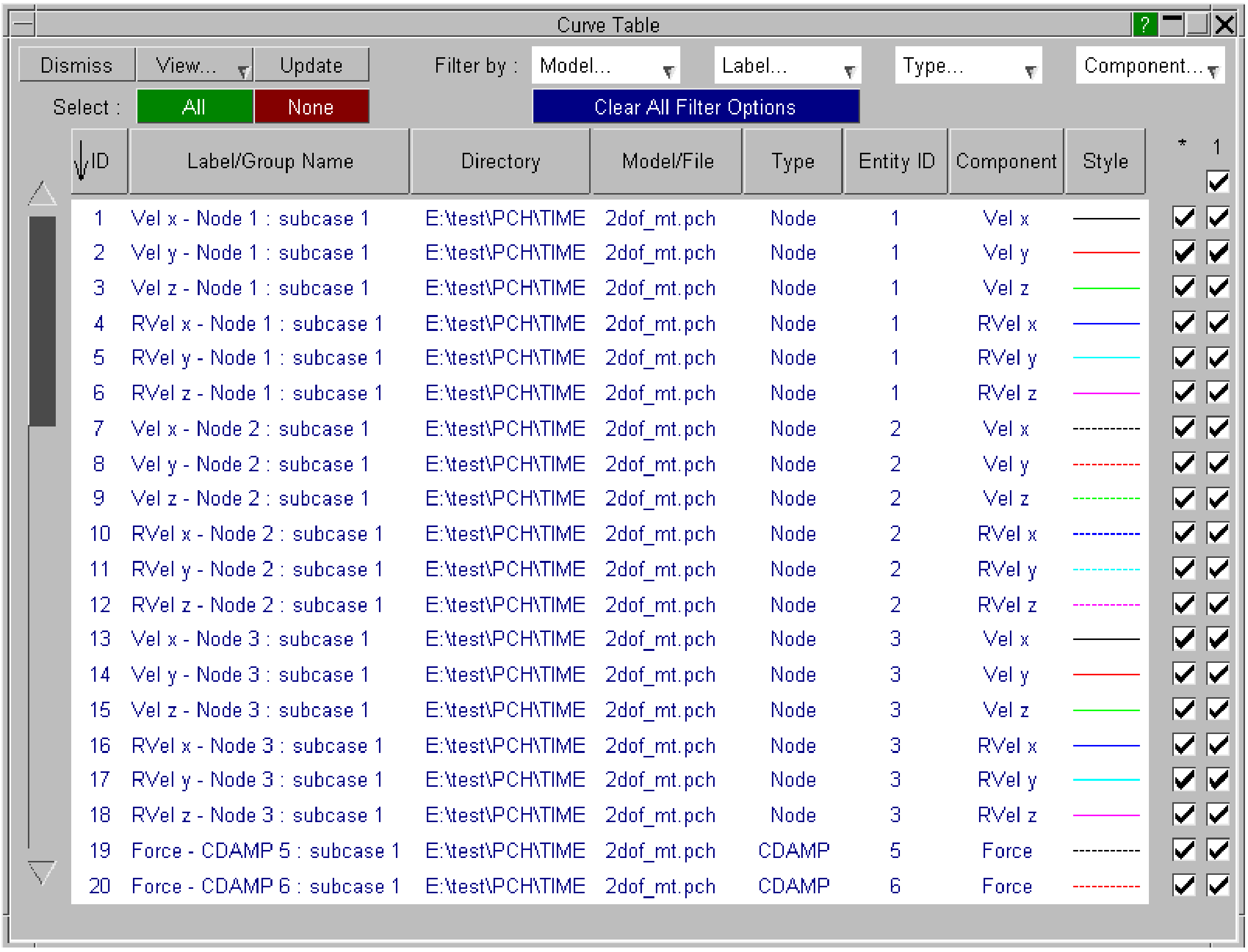
X,Y,Z translation (Real) / (Magnitude)
X,Y,Z translation (Imaginary) / (Phase angle)
X,Y,Z rotational (Real) / (Magnitude)
X,Y,Z rotational (Imaginary) / (Phase angle)

The **Options...** button will display the following menu that will allow some components to be deselected before reading the file.



Complex Data	For complex data components written out as a pair of real and imaginary numbers or as a magnitude and phase angle either of the components can be deselected.
Nodal Data	For nodal data any of the 4 data types can be deselected along with any of the 6 translational/rotational directions.
Element Forces	For element forces each individual element type can be deselected.

T/HIS will automatically create curve labels for each curve generated from the PCH file. The entity types, ID's and components will also be stored with the curves to allow the curves to be sorted using the curve table (see Table)

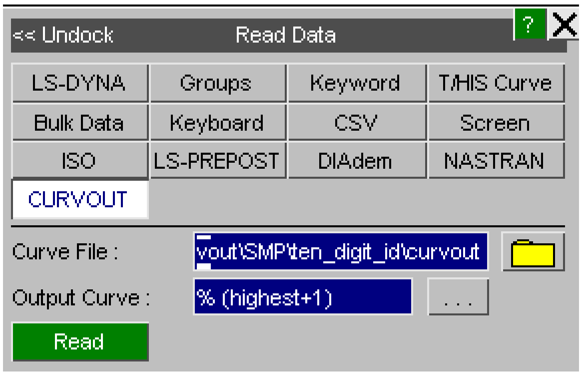


ID	Label/Group Name	Directory	Model/File	Type	Entity ID	Component
1	Vel x - Node 1 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	1	Vel x
2	Vel y - Node 1 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	1	Vel y
3	Vel z - Node 1 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	1	Vel z
4	RVel x - Node 1 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	1	RVel x
5	RVel y - Node 1 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	1	RVel y
6	RVel z - Node 1 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	1	RVel z
7	Vel x - Node 2 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	2	Vel x
8	Vel y - Node 2 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	2	Vel y
9	Vel z - Node 2 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	2	Vel z
10	RVel x - Node 2 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	2	RVel x
11	RVel y - Node 2 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	2	RVel y
12	RVel z - Node 2 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	2	RVel z
13	Vel x - Node 3 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	3	Vel x
14	Vel y - Node 3 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	3	Vel y
15	Vel z - Node 3 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	3	Vel z
16	RVel x - Node 3 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	3	RVel x
17	RVel y - Node 3 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	3	RVel y
18	RVel z - Node 3 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	Node	3	RVel z
19	Force - CDAMP 5 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	CDAMP	5	Force
20	Force - CDAMP 6 : subcase 1	E:\test\PCH\TIME	2dof_mt.pch	CDAMP	6	Force

8.2.13. CURVOUT

CURVOUT

This option can be used to read in data from from a CURVOUT ASCII file (curves defined by *DEFINE_CURVE_FUNCTION).



All the curves defined in the file are read. However, CURVOUT data (from both ASCII and binout files) can now be read in via the LS-DYNA option in the read panel. The curves in the file will then be treated as entities, allowing them to be selected individually in the entities list.

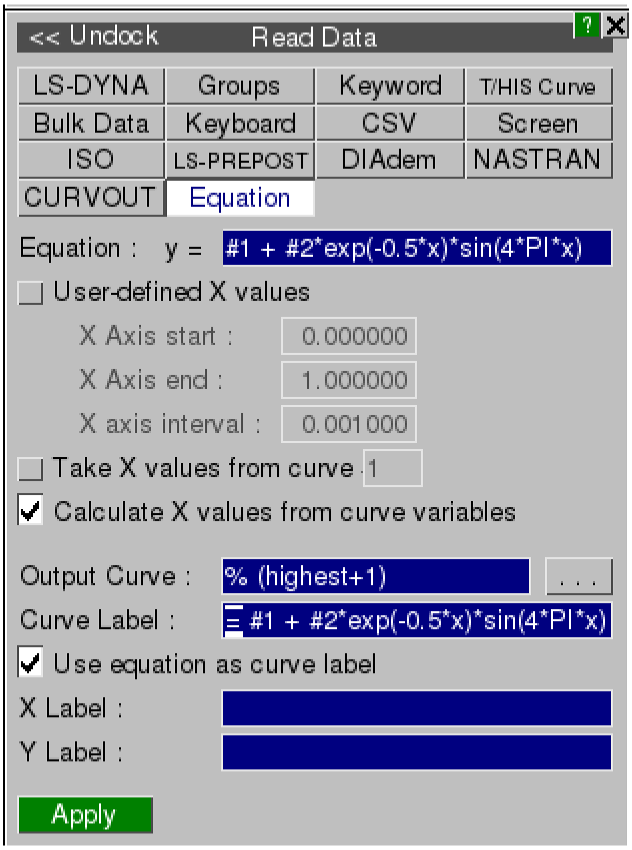
8.2.14. Equation

Equation

This option can be used to create a curve by definining an equation of the form '**y=f(x)**'. Here 'x' can be replaced by any of x, X, t, T, time or TIME.

The usual operators + - * / ^ % can all be used. The following standard mathematical functions can be used: SIN, COS, TAN, SEC, CSC, COT, ASIN, ACOS, ATAN, ATAN2, SINH, COSH, TANH, ASINH, ACOSH, TANH, ASINH, ACOSH, ATANH, EXP, CEIL, FLOOR, LOG, LOG10, SQRT, MOD, MAX, MIN, SIGN, ABS, INT, AINT, NINT, FLOAT.

Additonally, some of the functions specified in the LS-DYNA manual under *DEFINE_CURVE_FUNCTION are also available. These are: IF, STEP, POLY, CHEBY, FORSIN, FORCOS, SHF.



This allows PRIMER to send *DEFINE_CURVE_FUNCTION definitions to T/HIS, as long as they only depend on TIME and no other values that change during the LS-DYNA run. In the *DEFINE_CURVE_FUNCTION edit panel, if the expression is suitable for evaluation, then the **T/HIS** button will be active and the equation can be sent across. The curve will be plotted from TIME = 0 until the termination time specified on the *DATABASE_CONTROL_TERMINATION card. The value of any parameters appearing in the expression will be maintained. The curve can be edited via right-clicking and selecting **Edit equation...**. It can then be sent back to PRIMER by right-clicking and selecting **Update curve in PRIMER**.

Curves can be referenced in equations using variables of the form 'c1', 'C1' or '#1' to refer to curve #1. For example, equations such as '**y = 2*#1 + 3*#3**' are valid. This allows multiple curve operations to be replaced by a single equation.

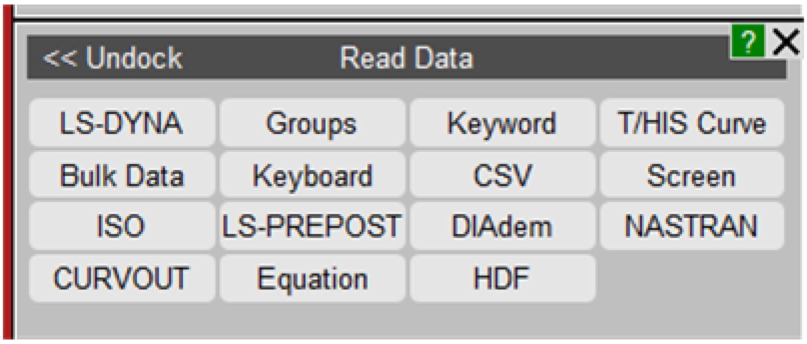
There are multiple options for defining the x-values used to plot the equation curve. There is an option to specify directly the start value, end value and interval between points. Alternatively, the X values can be copied from a specified curve. The final option is only relevant if the equation contains curve variables. The x-values from all of the curves that appear in the equation will be combined to give one potentially larger set of x-values, which will then be used to plot the equation curve.

Equation curves can also be created using the JavaScript API, see 'Read.Equation' in the JavaScript API reference manual.

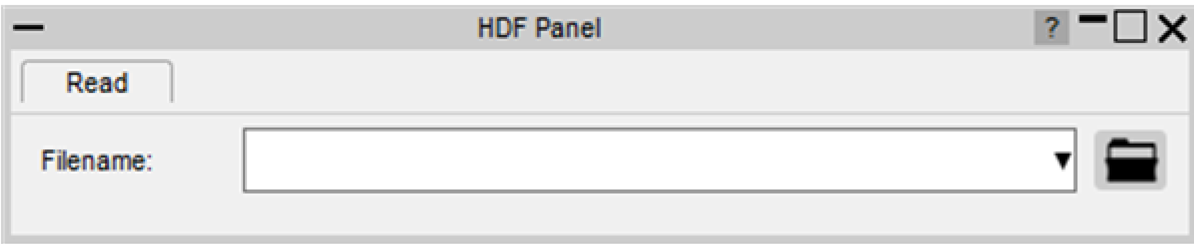
8.2.15. HDF

HDF

T/HIS can read HDF files. The version of HDF file supported in T/HIS is HDF5. HDF4 files require conversion to HDF5 before they can be read (see Converting HDF4 to HDF5). Currently, T/HIS supports Float data types within Atomic datasets and Float data types within Compound datasets. Reading HDF files is supported in FAST-TCF. We plan to improve our support for HDF in future releases of T/HIS, so please send us any feedback you have.



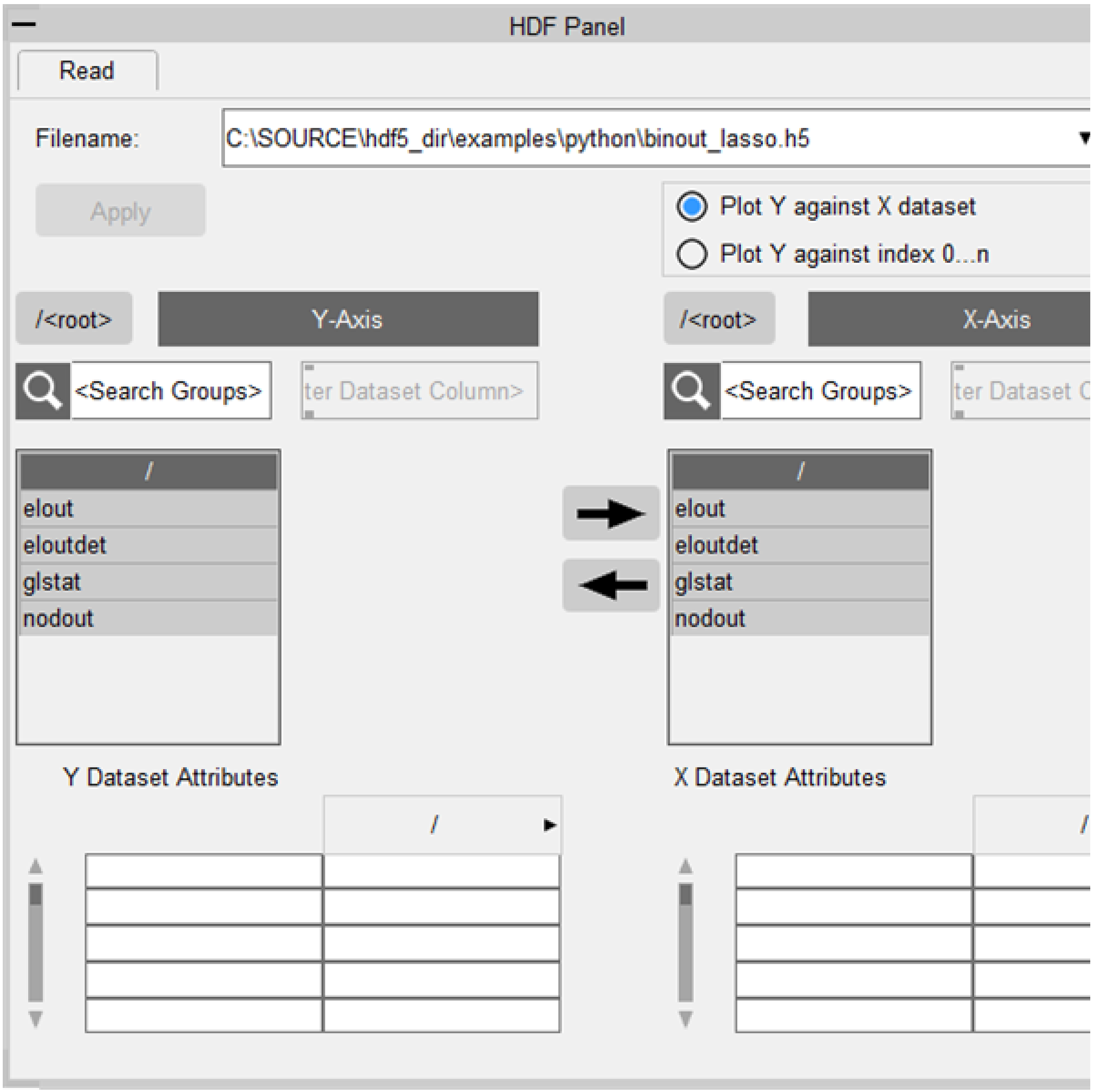
An HDF5 file can be read by entering the path in the textbox or using the file selector:



Plot Y against X dataset

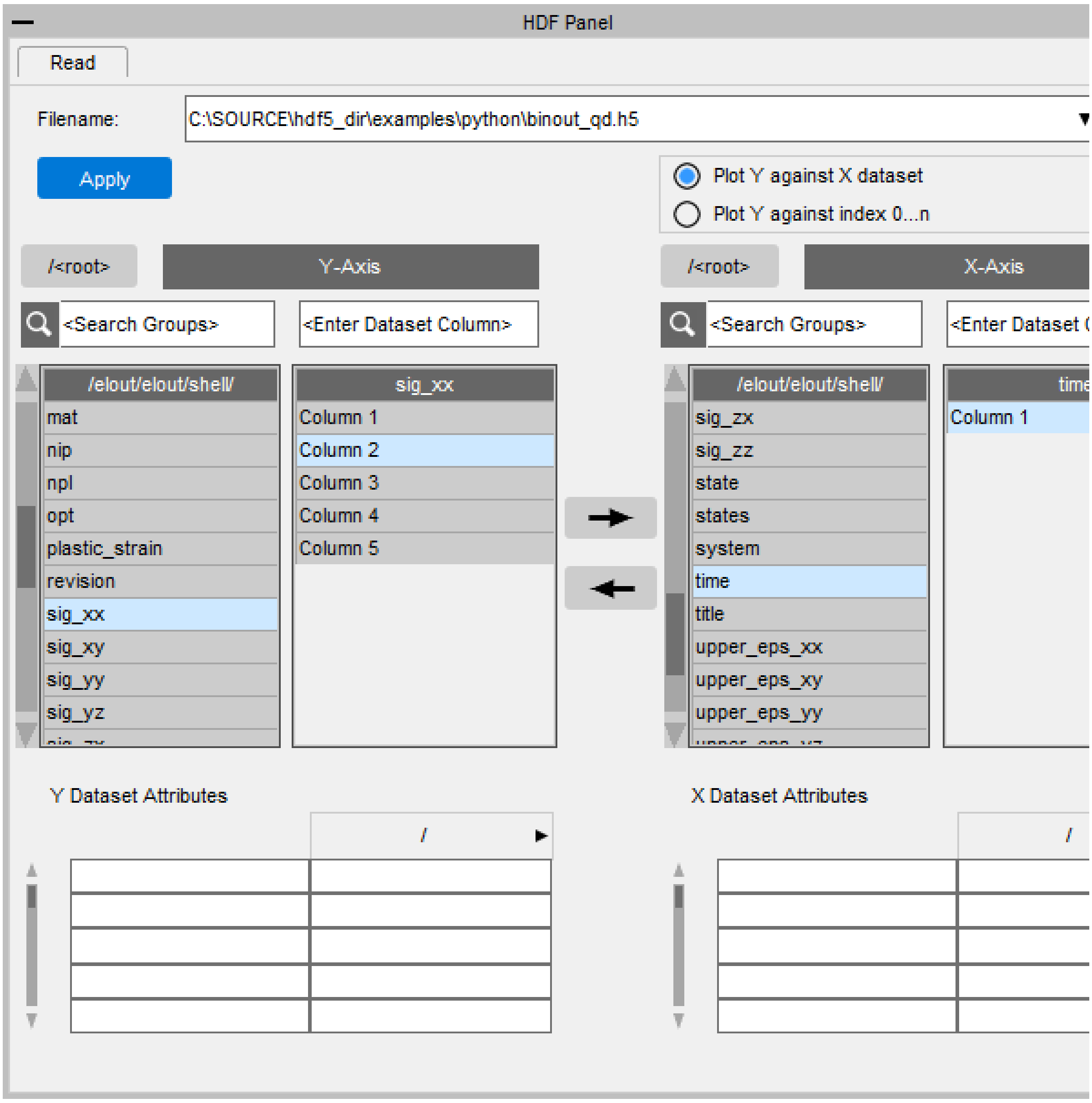
Once a valid HDF file is selected, the HDF panel will open with two navigation lists mapped in "Plot Y against X dataset" mode – one related to Y-Axis and the other for X-Axis – showing the contents in the root group (/). Using these navigation lists, the entire HDF5 file can be accessed. If the selected datasets have any associated attributes, they will be listed in the attributes lists below.

An example of a curve reading in from a HDF5 file has been shown below. The file that has been read in is a binout file converted to HDF5 format:

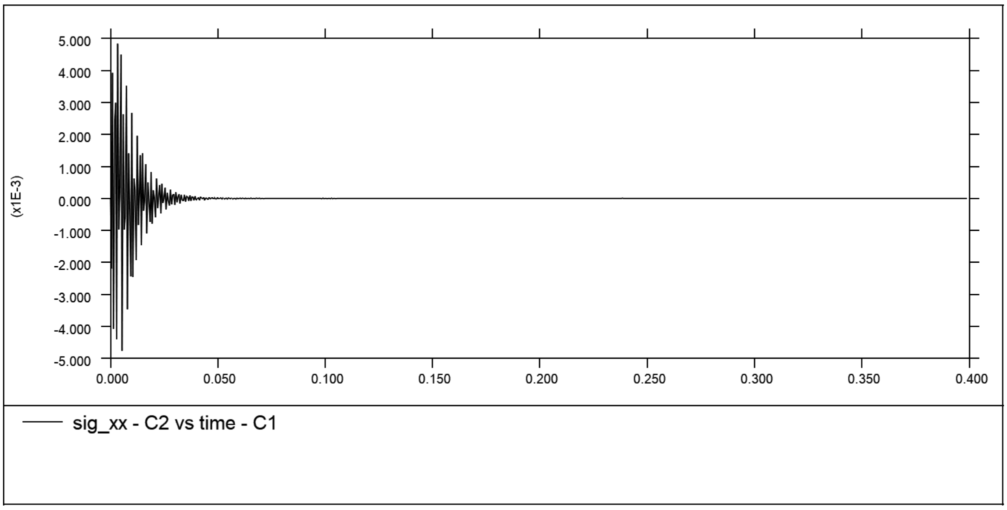


Clicking on a dataset will map an adjecent list of the dataset contents. An empty dataset box or disabled rows just means that data is not readable into T/HIS or we do not support it yet.

In this example, for Y-Axis we have selected **elout → elout → shell** followed by **sig_xx** (which is a 2D dataset), and selected Column 2 in the dataset list. For X-Axis we have selected **elout → elout → shell** followed by **time** (which is a 1D dataset) and selected Column 1:

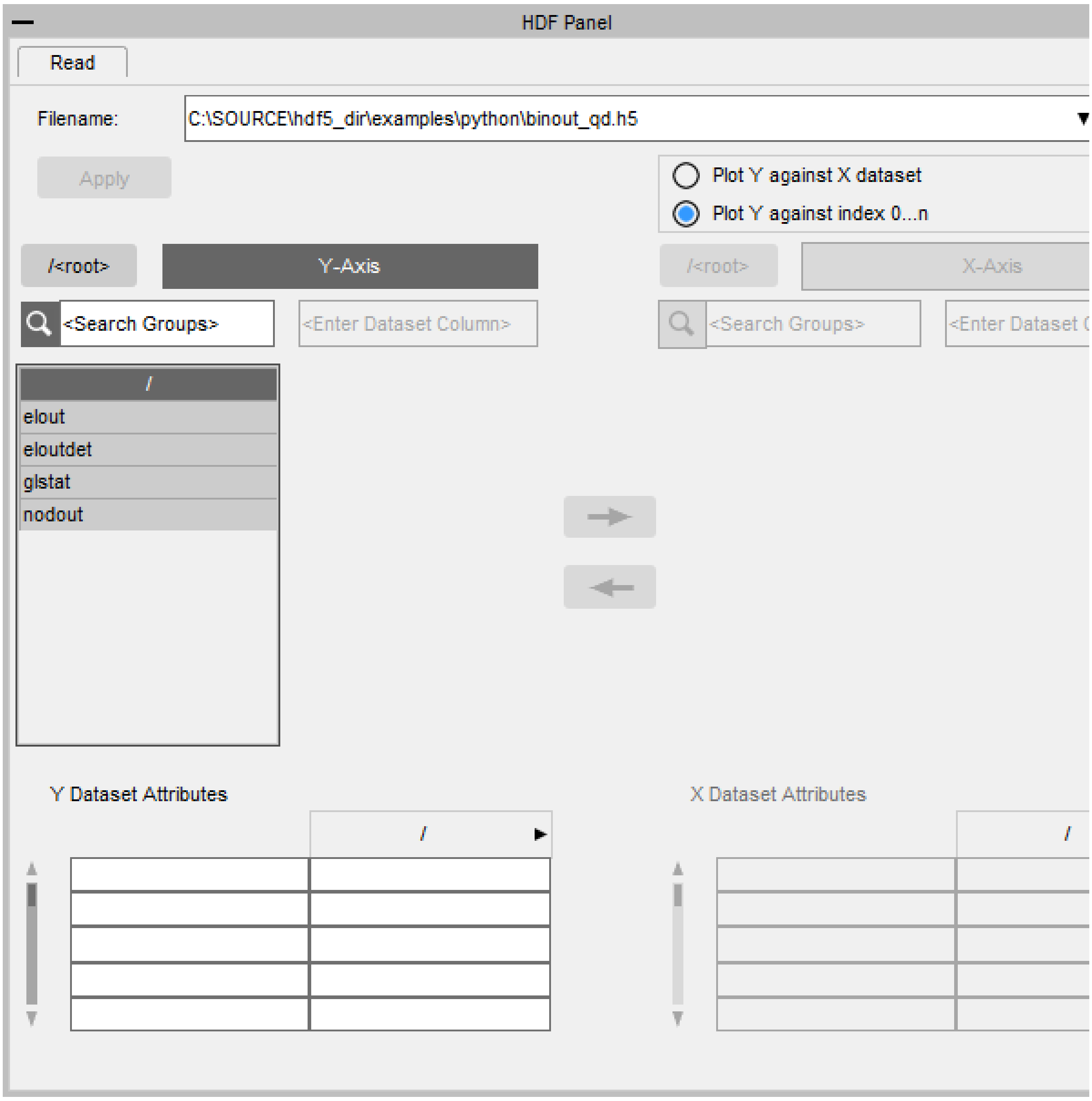


When you click **Apply**, a curve will be mapped:



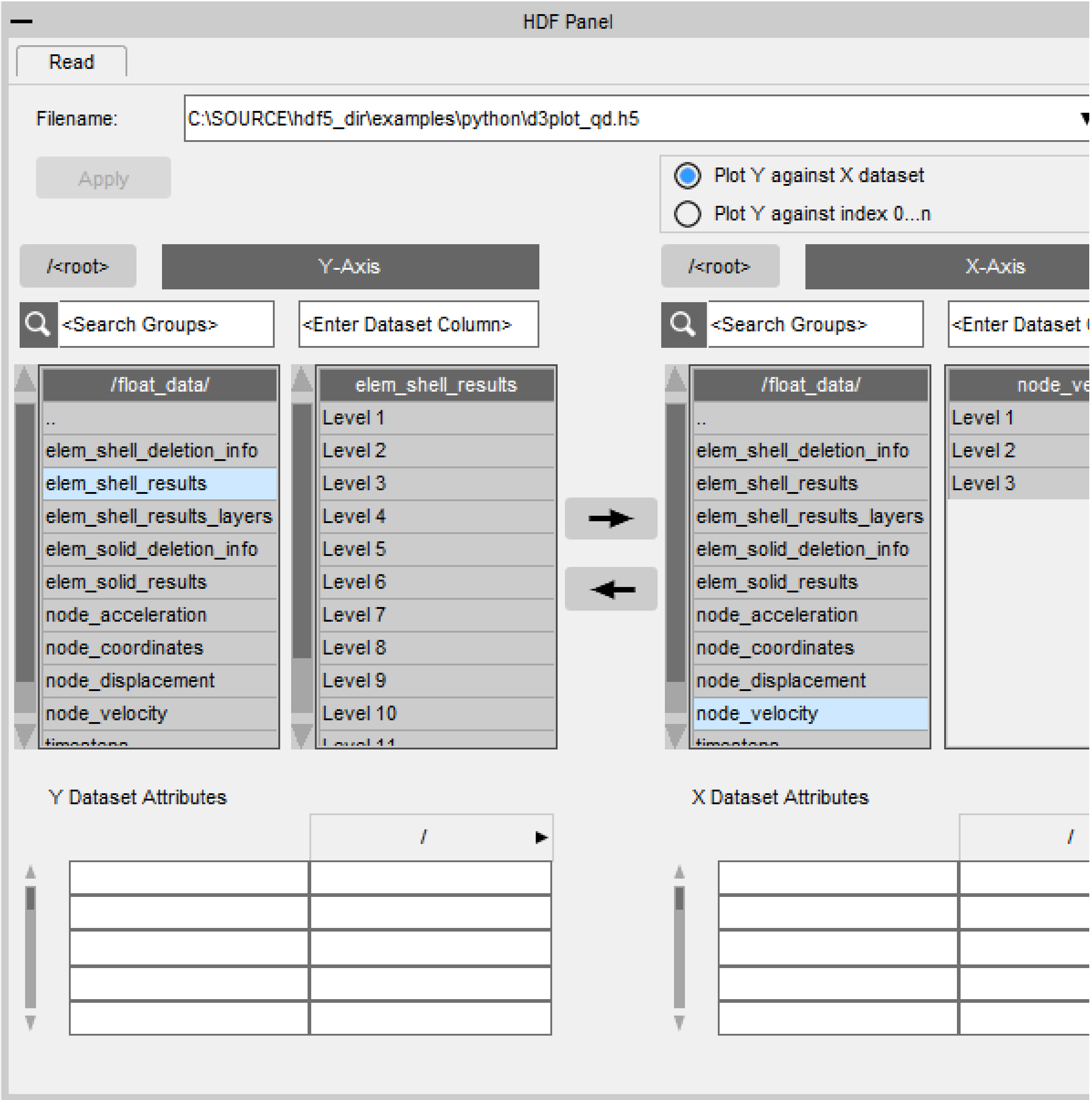
Plot Y against Index 0...n

The second mode available is "Plot Y against Index 0...n", where only the Y-Axis points need to be selected. Y-values will then be plotted against their index i.e. X-values of 0, 1, 2, 3, etc.

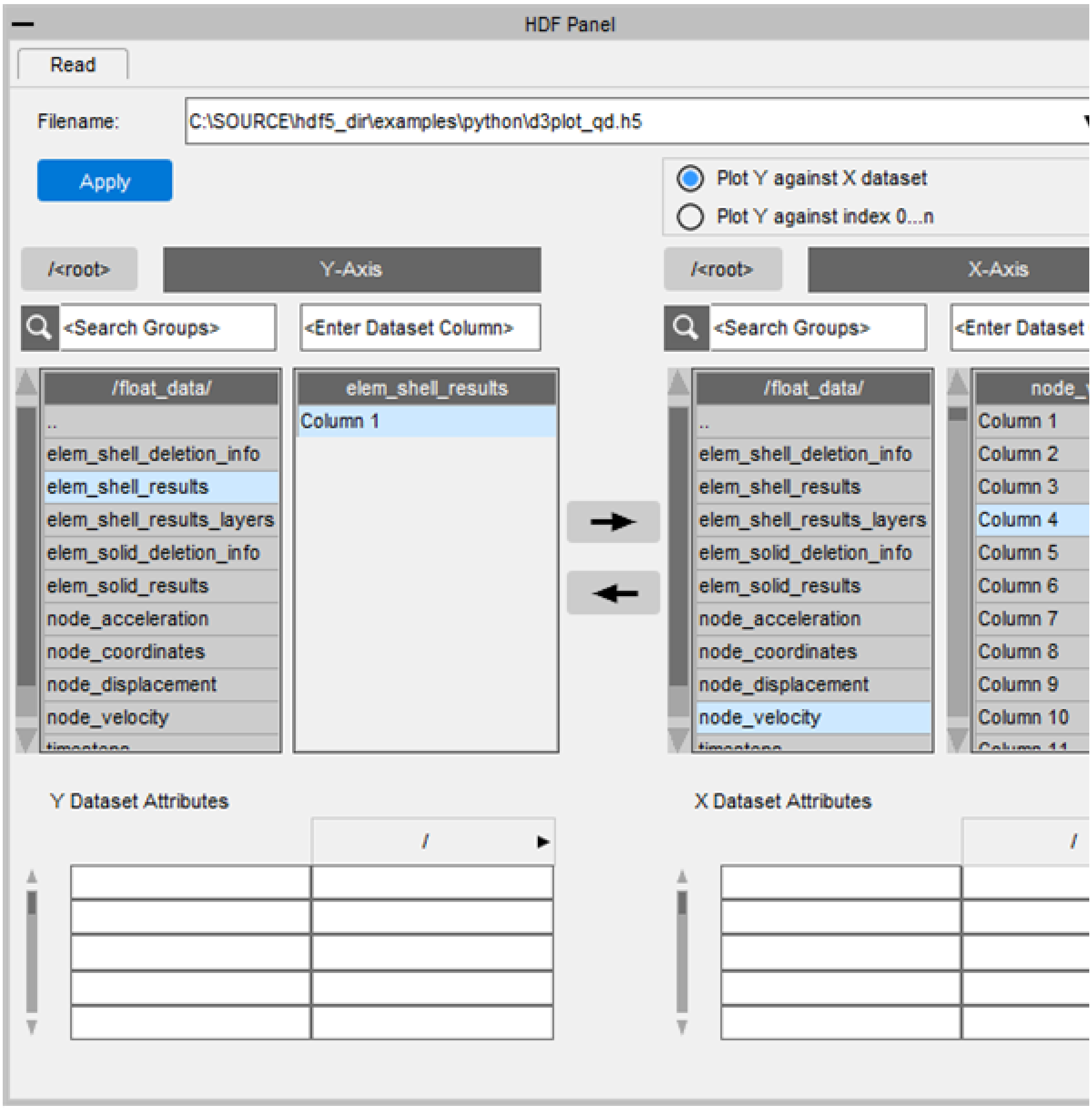


3D Datasets

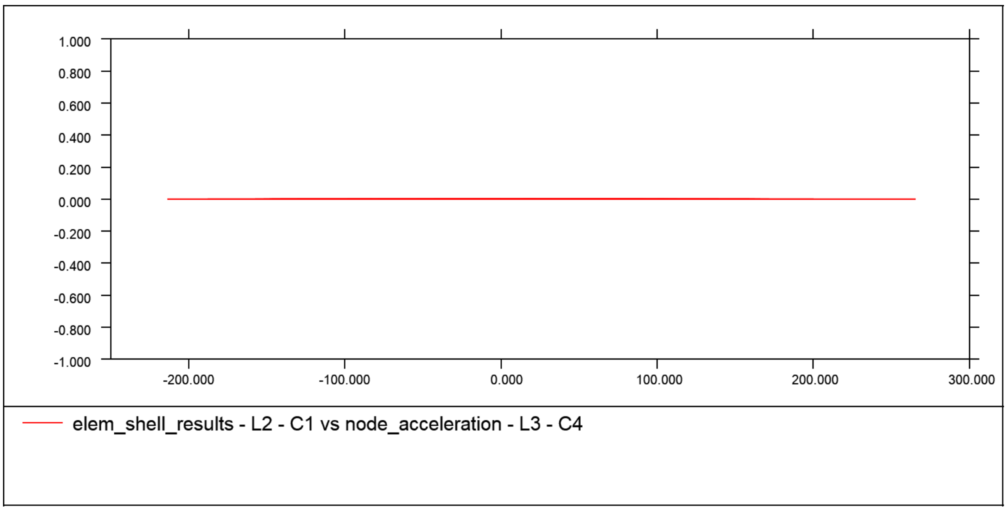
The contents of the dataset list change depending on the data. For a 1D or 2D dataset, a list of columns is shown (see previous examples) but for a 3D dataset, first the Level has to be selected:



After selecting the Level, the dataset box will show the list of Columns for selection:

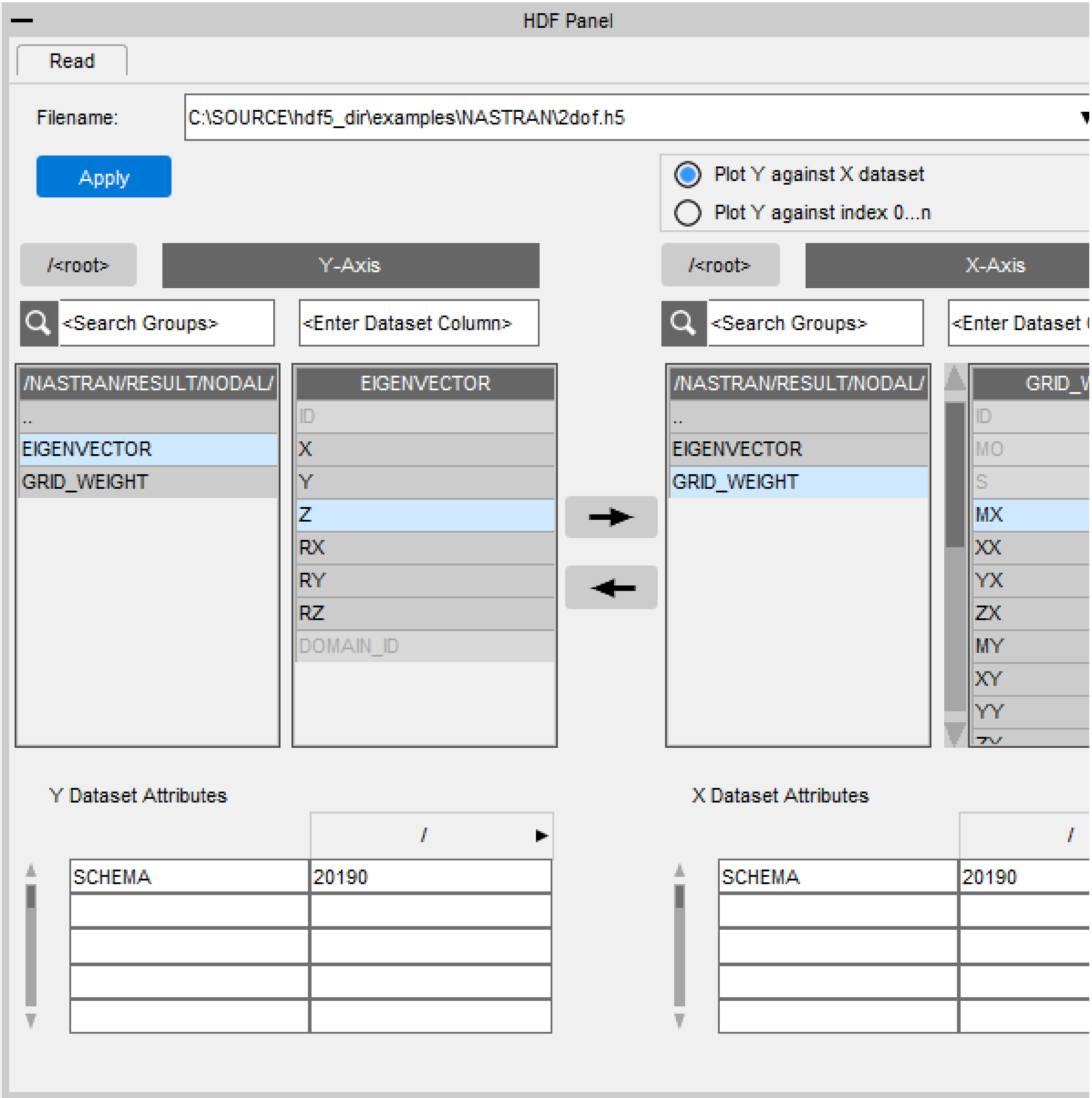


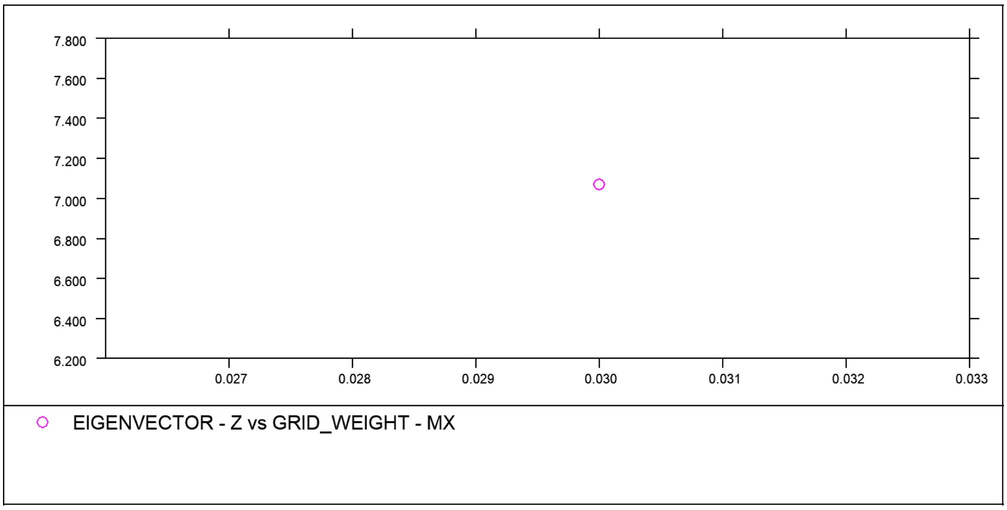
After clicking **Apply**, the curve label will show the Level and the Column selected:



Compound Datasets

For a compound dataset, the curve label shows the list label:



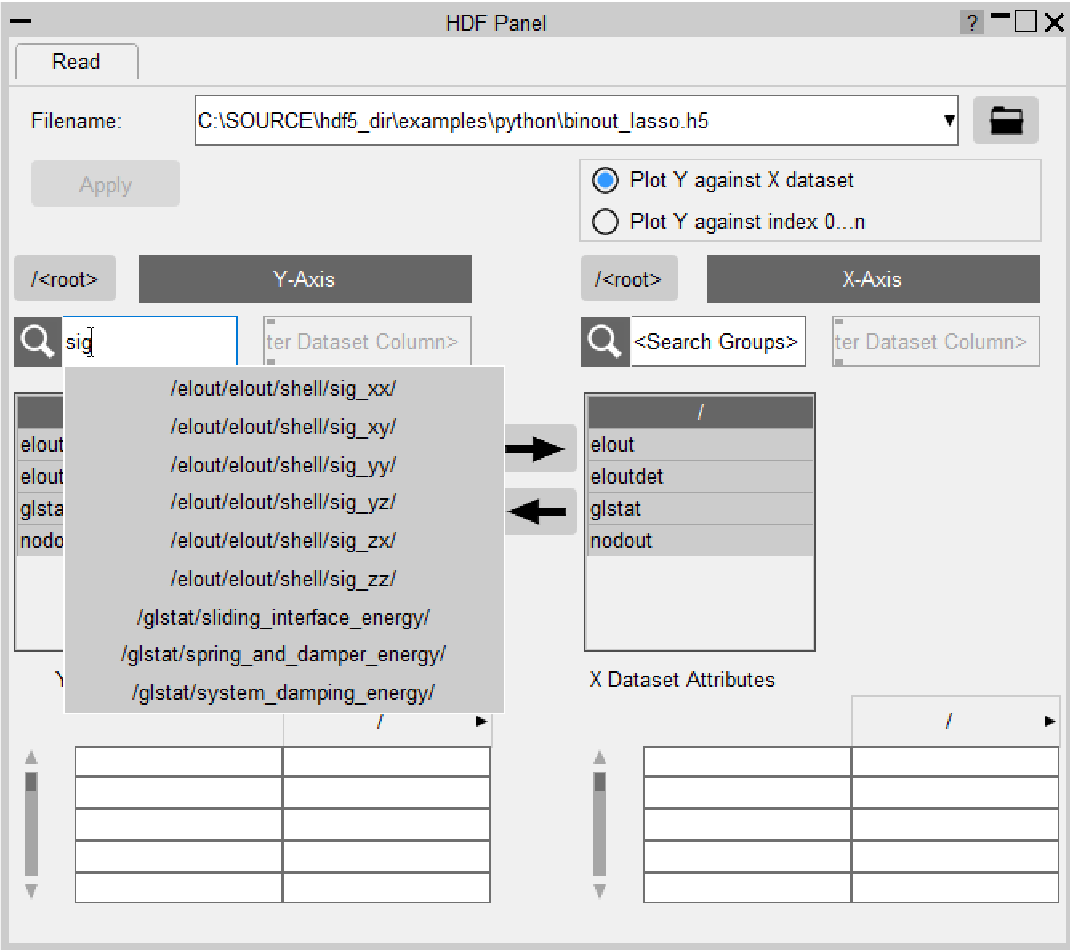


Navigation aids

Navigation aids have been introduced in order to traverse through the file easily and with minimum number of clicks.

Search

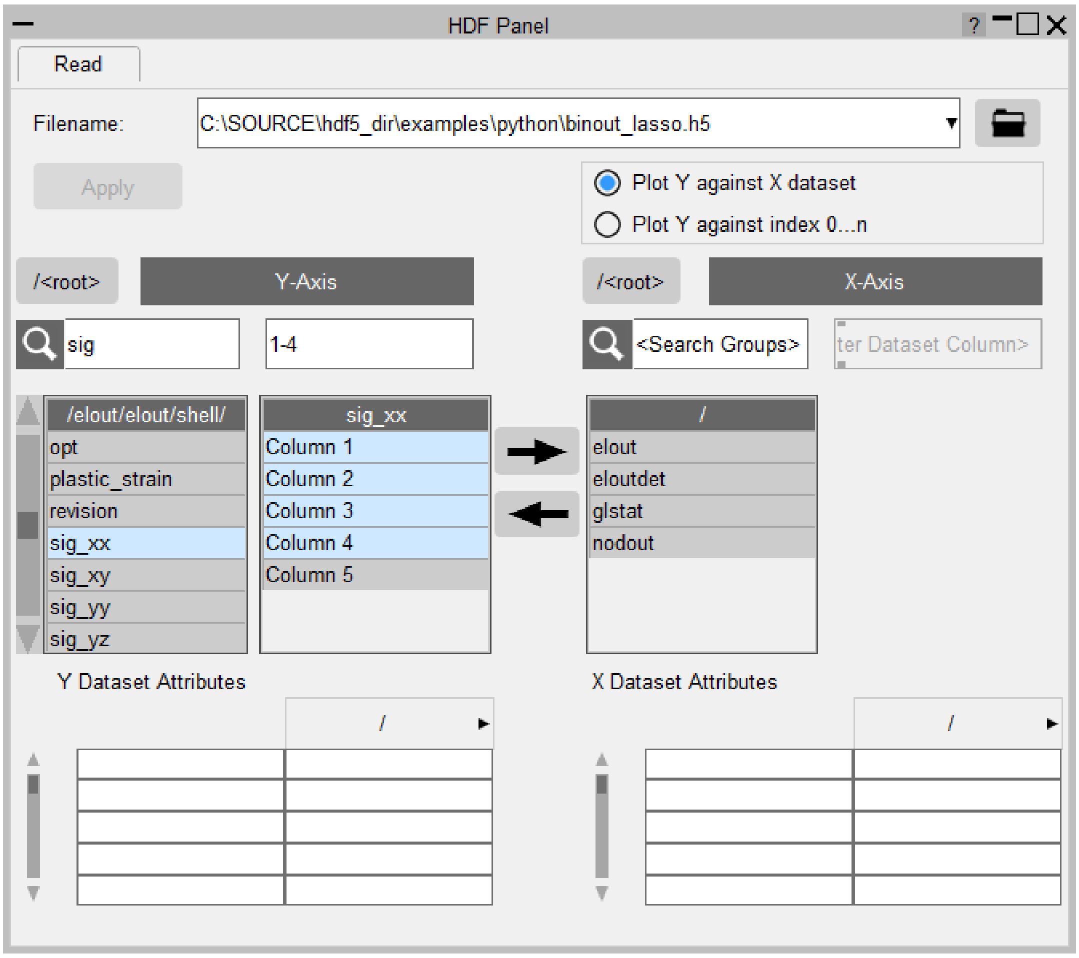
Let us consider the "Plot Y against X dataset" example above. We had to follow the sequence **elout** → **elout** → **shell** → **sig_xx**. Instead, if you already know that you want to retrieve **sig_xx** you can search for it in the **<Search Groups>** textbox directly, and select from the matching paths listed:



Column selection

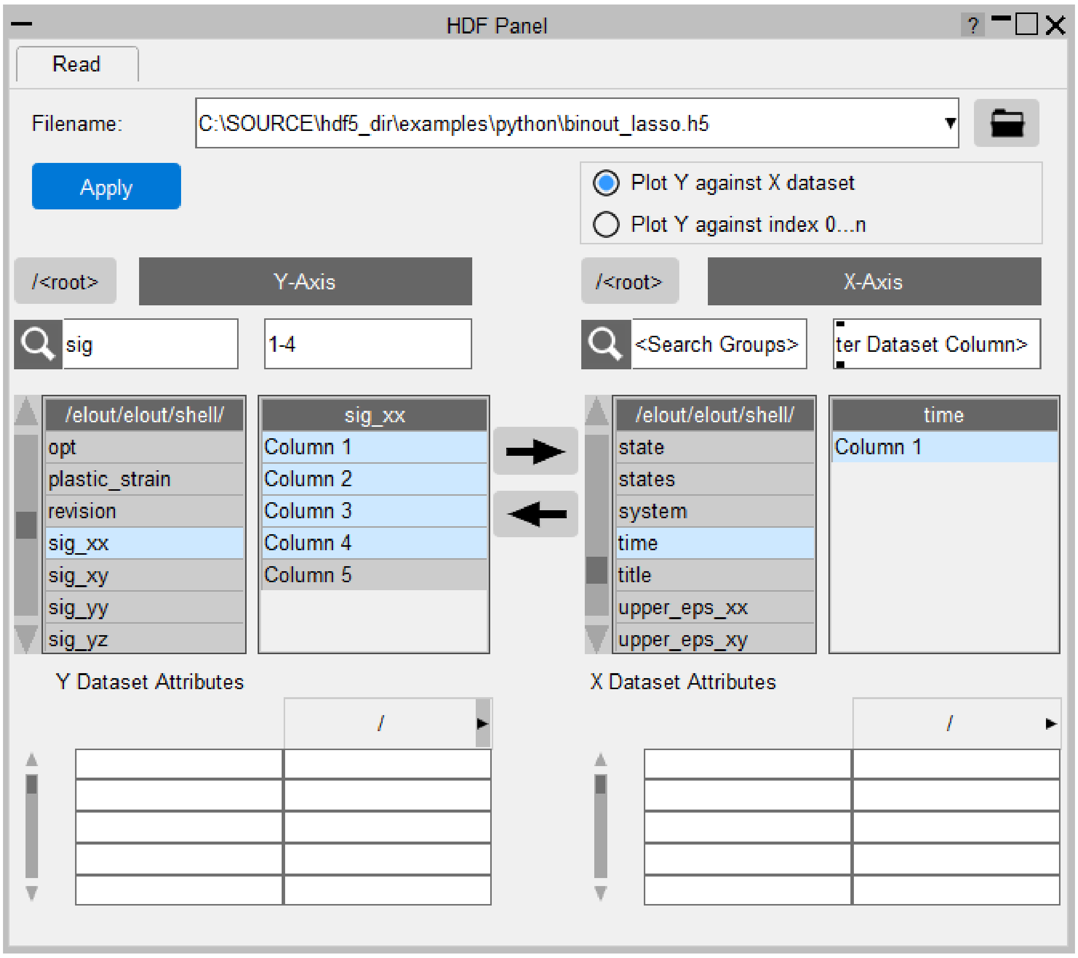
The **<Enter Dataset Column>** textbox allows you to select columns quickly. The following are all valid searches:

- “1” or “Column 1”
- “1-4” or “Column 1-4”
- “1, 2, 3, 5“ or “1 2 3 5” or “Column 1, 2, 3, 5” or “Column 1 2 3 5”



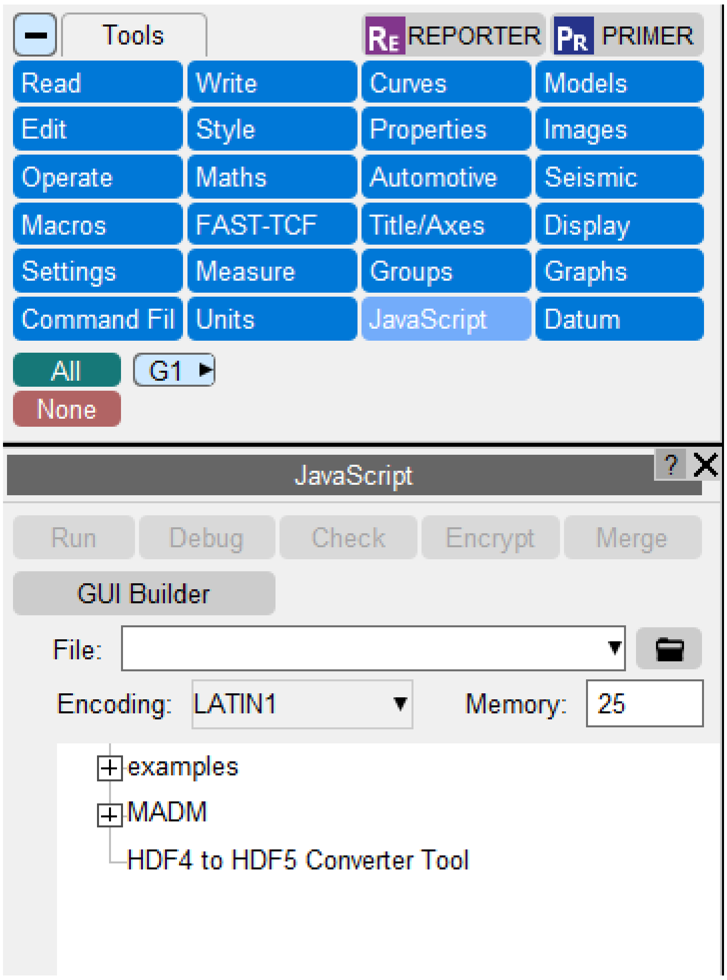
Copy path

The left and right arrows simply copy the Y-Axis path to X-Axis path and vice versa. This can make it quicker to plot *Y* versus *X* data. Supposing you wish to plot *x-stress* versus *time*. Once you have selected the **sig_xx** dataset for the Y-axis, you can copy the path to the X-Axis and quickly switch the X-Axis selection to **time** if it is in the same location in the file:



Converting HDF4 to HDF5

To convert an HDF4 file to the HDF5 format, you need to install the h4h5tools application, which can be found on the HDF Group website (we tested *h4h5tools-1.10.6-2.2.5-win10_64-vs15.zip*). Once installed, you can use the **HDF4 to HDF5 Converter Tool** script included with T/HIS to aid file conversion. The script converts selected HDF4 files into HDF5 files, writing the new files in the same directory as the originals. The script can be found in the T/HIS **JavaScript** menu:

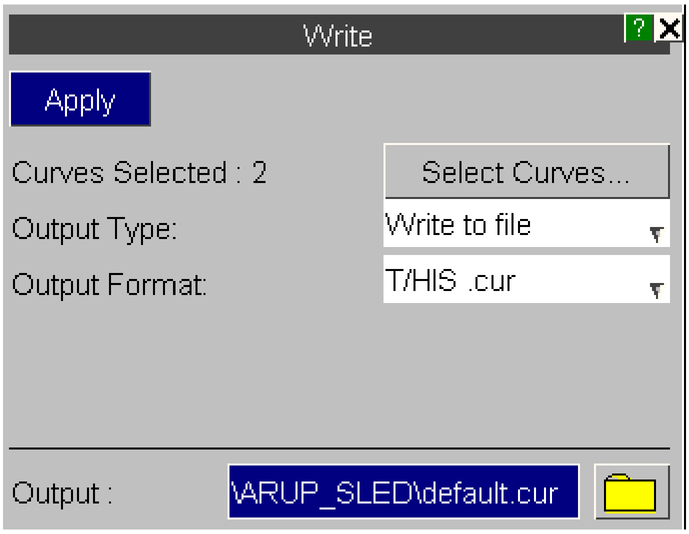


We plan to improve our support for HDF in future releases of T/HIS, so please send us any feedback you have.

8.3. WRITE Options

WRITE Options

Writes a group of curves out to a file for later use or to the screen.



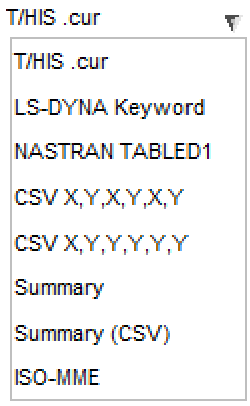
8.3.1. WRITE TO FILE

WRITE TO FILE

Writes a group of curves out to a file for later use if required. The user is prompted for the list of curves to write out after a filename has been specified.

FILE FORMAT

Writes a group of curves out to a file for later use if required. The user is prompted for the list of curves to write out after a filename has been specified.



T/HIS .cur format

This option will write out curves using the default T/HIS curve format. One curve file will be written containing all the selected curves along with their Titles, Axis Labels, Line Labels and styles. From version 9.4 onwards the curve file can also contain information on the UNIT system and the X and Y axis units for each curve (see Appendix B for more details on the curve file format)

LS-DYNA Keyword

One file will be written containing all the selected curves using the LS-DYNA *DEFINE_CURVE format so that the file is suitable for inclusion in a LS-DYNA keyword file.

NASTRAN D1

This option will write out curves using the NASTRAN TABLE D1 format. Curves are listed sequentially in the file.

CSV X,Y,X,Y,X,Y

This option will write out curves using as a CSV (comma separated variable) file that can be read into other programs like Microsoft EXCEL. The columns written are x-values for the 1st selected curve, y-values for the 1st selected curve, x-values for the 2nd selected curve, y-values for the 2nd selected curve ...

CSV X,Y,Y,Y,Y,Y

This option also writes out a CSV file. All the curves are output using a single consistent set of X values that can either be taken from one of the curves or they can be generated automatically.

Summary

Gives a summary of the curve. This includes the type of data being plotted and the maximum and minimum values in the curve.

Summary (CSV)

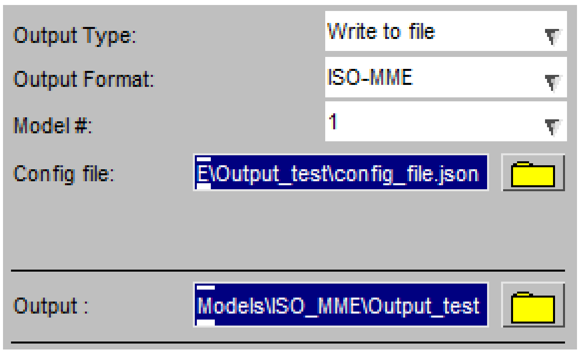
CSV (comma separated variable) version of the summary file.

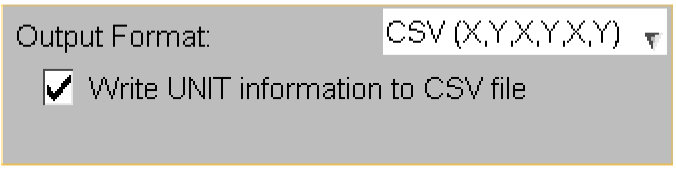
ISO-MME

The ISO-MME format (ISO/TS 13499) is a data exchange format for crash analyses comprising a number of folders and files. This option works slightly differently to the others, in that you need to select a model for T/HIS to extract the data from and provide a configuration file to specify what data should be written out, rather than selecting curves.

This is needed because the naming conventions of the output files, set out in the ISO standard, follow specific rules which require extra data that isn't present in the curves alone and they also contain lines at the top of the files which describe the data in more detail. The contents of the configuration file are described in ISO-MME Configuration File .

In addition, an output directory is specified rather than a output file since multiple files are written out.





From version 9.4 onwards the CSV files generated by T/HIS can also contain information on the UNIT system and the X and Y axis units for each curve. If you don't want to output this information then you can turn it off.

The default setting for this option can be set via the preference option

```
this*write_csv_units:
```

This option can also be turned on and off in FAST-TCF scripts (see FAST-TCF)

ISO-MME Configuration File

The ISO-MME configuration file is a JSON format file which is used to tell T/HIS which curves to generate: the data to extract, from which entities to extract it, their locations in the model and whether any filtering or operations are required. This configuration file is required so T/HIS can write various ISO-MME files with the correct names and any additional data required at the top of the files.

To set the scene, the structure of the directory where various files get written out and the files it contains is:

<table>
<tr><th>ISO-MME Version 1.6</th><th>ISO-MME Version 2.0</th></tr>
<tr><td>Output Dir
|
|-- <Virtual Testing reference ID>.mme <= Test Data
|
|-- Channel
|
|-- <Virtual Testing reference ID>.chn <= Channel index file
|-- <Virtual Testing reference ID>.001 <= Channel data files</td><td rowspan="2">Output Dir
|
|-- <testname>.mme <= Test information file
|
|-- Channel
| |
| |-- <testname>_Channel.mmi <= Channel index file
| |-- <testname>_<channel_code>.mmd <= Channel data files
| |-- <testname>_<channel_code>.mmd .
| |-- <testname>_<channel_code>.mmd .
|</td></tr>
<tr><td></td></tr>
</table>

|-- <Virtual Testing reference ID>.002 . |-- <Virtual Testing reference ID>.003 .	|-- Object | |-- <testname>_<object_code>.mmi <= Object files |-- <testname>_<object_code>.mmi . |-- <testname>_<object_code>.mmi .

Test information file (.mme file)

The test information file describes the test and the objects (vehicles, dummies, barriers) in it. As an example:

ISO-MME Version 1.6	ISO-MME Version 2.0
Data format edition number :1.6 Customer name :ARUP Customer test ref. number :NOVALUE Customer project ref. number:9999 Title :NOVALUE Timestamp :2022-09-29 13:30:00 Type of the test :NOVALUE Subtype of the test :NOVALUE Virtual Testing reference ID:NOVALUE Regulation :NOVALUE Date of the test :2022-10-02 Name of test object 1 :NOVALUE Ref. number of test object 1:NOVALUE Velocity test object 1 lon. :NOVALUE Velocity test object 1 lat. :NOVALUE	Data format edition number :2.0 Timestamp :2020-11-10 Laboratory name :Arup Laboratory contact name :NOVALUE Laboratory contact phone :NOVALUE Laboratory contact fax :NOVALUE Laboratory contact email :NOVALUE Laboratory test ref number :NOVALUE Type of the test :NOVALUE Subtype of the test :NOVALUE Regulation :NOVALUE Date of the test :NOVALUE Number of test objects :2 #Begin of testobject Type :D

Mass test object 1 :1230 Driver position object 1 :3 Impact side test object 1 :LE Name of test object 2 :NOVALUE Ref. number of test object 2:NOVALUE Velocity test object 2 lon. :NOVALUE Velocity test object 2 lat. :NOVALUE Mass test object 2 :1230 Driver position object 2 :2 Impact side test object 2 :LE Type of data source :Simulation	Filename :my_test_D0.mmi #End of testobject #Begin of testobject Type :1 Filename :my_test_1.mmi #End of testobject

Object files (.mmi file)

These files are only present in ISO-MME Version 2.0. The object files describe each object in the test, for example:

```
Name                            :H350
Velocity                        :NOVALUE
Mass                            :NOVALUE
Impact side                     :00
#Begin of biomechanical
Gender                          :male
Age                             :21
#End of biomechanical
```

Channel index file (.chn or .mmi file)

The channel index file lists the channel data files:

ISO-MME Version 1.6 (.chn file)	ISO-MME Version 2.0 (.mmi file)
Instrumentation standard : Not applicable	Number of channels :3 Data origin :S

Number of channels :7 Name of channel 001 :13HEAD0000WSACX0 Name of channel 002 :13HEAD0000WSACY0 Name of channel 003 :13HEAD0000WSACZ0 Name of channel 004 :13HEAD0000WSDCX0	Data source : simulation #Begin of channel Extended channel code :D0HEADMI0000000B #End of channel #Begin of channel Extended channel code :D0CHSTMI0000000C #End of channel #Begin of channel Extended channel code :D0PELVMI0000000C #End of channel

Channel data files (.001 or .mmd file)

The channel data file that gives the details about the curve:

ISO-MME Version 1.6 (.001 file)	ISO-MME Version 2.0 (.mmd file)
Test object number :1 Name of the channel : Laboratory channel code :NOVALUE Customer channel code :NOVALUE Channel code :13HEAD0000WSACX0 Unit :m / (s * s) Reference system :NOVALUE Transducer type :NOVALUE Pre-filter type :NOVALUE Cut off frequency :NOVALUE Channel amplitude class :NOVALUE Sampling interval :0.0001 Bit resolution :NOVALUE Time of first sample :NOVALUE	Data structure :Channel Instrumentation standard :NOVALUE Name of the channel :Accel x - Node 52503304 : (PelvisAccel_INJURY) (Reg 0.100E-03) (C 180) Data source :simulation Data status :ok Cut off frequency :NOVALUE Channel amplitude class :NOVALUE Sampling interval :0.0001 Bit resolution :NOVALUE Time of first sample :0 Number of samples :1500 Reference channel :implicit #Start of data -6.09125e-05 -1785.28 -3315.55

Number of samples :1999 -0.42144 -0.00030 -0.00028	. . #End of data

Configuration structure

Below is an illustration of the configuration file structure.

-
  - **Test data:** At the top of the file is a list of test data properties, followed by a list of descriptors for the test.
-
  - **Objects array:** Next is a list of objects (e.g. vehicles, dummies, barriers) and their properties and descriptors.

Channels and Operations arrays are used to specify the Curve properties to output required curves:

-
  - **Channels array:** Each channel in the array is defined as an object with properties such as entityType, id, etc. to create a single curve with its ISO-MME channel code defined in properties like mainLocation, fineLocation etc.
-
  - **Operations array:** Each operation object defines an operation to be performed on any previously defined or multiple curves. Operations can be chained together. The final output should be an ISO-MME channel code.

Configuration File Structure

```
{
   |
   | -- Test Data Property 1
   | -- Test Data Property 2
   | -- Test Data Property 3
   | -- Descriptors
   |       | -- Descriptor 1
   |       | -- Descriptor 2
   |
   | -- Object Type 1
   |       | -- Object 11
   |       |       | -- Property 1
```

```
    |       |       | -- Property 2
    |       |       | -- Descriptors
    |       |               | -- Descriptor 1
    |       |               | -- Descriptor 2
    |       |
    |       | -- Object 12
    |       |       | -- Property 1
    |       |       | -- Property 2
    |
    | -- Object Type 2
    |       | -- Object 21
    |       |       | -- Property 1
    |       |       | -- Property 2
    |       |
    |       | -- Object 22
    |       |       | -- Property 1
    |       |       | -- Property 2
    |
    |
    | -- Channels
    |       | -- Channel 1
    |       |        | -- Channel Property 1
    |       |        | -- Channel Property 2
    |       |
    |       | -- Channel 2
    |       |        | -- Channel Property 1
    |       |        | -- Channel Property 2
    |       |
    |       | -- Channel 3
    |       |        | -- Channel Property 1
    |       |        | -- Channel Property 2
    |       |        | -- Descriptors
    |       |               | -- Descriptor 1
    |       |               | -- Descriptor 2
    |
    | -- Operations
    |       | -- Operation 1
    |       |        | -- Operation Property 1
    |       |        | -- Operation Property 2
    |       |
    |       | -- Operation 2
    |       |        | -- Operation Property 1
    |       |        | -- Operation Property 2
    |       |        | -- Descriptors
    |       |               | -- Descriptor 1
    |       |               | -- Descriptor 2
}
```

Configuration file example

Below is an example configuration file, showing all the available options that can be set. Not all of them are required – the list below shows which ones are optional and what they can be set to.

```
{
    "testName": "Far side",
    "timestamp": "12/18/2023, 4:24:25 PM",
    "formatVersion": "1.6",
    "laboratoryName": "Oasys LS-DYNA Environment",
    "customerName": "Euro NCAP",
    "customerTestRefNumber": "001",
    "customerProjectRefNumber": "1234",
    "virtualTestingReferenceId": "FS_Pole_75_x-ref_z-ref_50M_Sim_1",
    "typeOfTest": "SideImpact",
    "subtypeOfTest": "Far Side + VTC",
    "regulation": "EuroNCAP",
    "testDate": "12/18/2023",
    "unitSystemModel": "U2",
    "unitSystemDisplay": "U1",
    "xAxisInterval": 0.0001,
    "descriptors": [
        {
            "description": "Type of data source",
            "value": "Simulation"
        },
        {
            "description": "Dummy Simulation Model Specification",
            "value": "WSID 50 M v3.4.1"
        }
    ],
    "dummies": [
        {
            "testObject": "1",
            "position": "1",
            "vehicleTestObject": "1",
            "name": "PDB WSID 50M 4.0",
            "gender": "male"
        }
    ],
    "vehicles": [
        {
            "testObject": "1",
            "position": "0",
            "driverPosition": "1",
```

```
            "name": "Lighting McQueen",
            "refNumber": "NO VALUE",
            "mass": "NO VALUE",
            "impactSide": "RI",
            "descriptors": [
                {
                    "description": "Vehicle Model",
                    "value": "95"
                }
            ]
        }
    ],
    "channels": [
        {
            "testObject": "1",
            "position": "1",
            "mainLocation": "ABRI",
            "fineLocation1": "RI",
            "fineLocation2": "01",
            "fineLocation3": "WS",
            "physicalDimension": "AN",
            "direction": "#",
            "entityType": "spring",
            "component": "rotation",
    "xAxisInterval": 0.0001,
            "id": 10324,
            "write": false
        },
        {
            "testObject": "1",
            "position": "1",
            "mainLocation": "ABRI",
            "fineLocation1": "RI",
            "fineLocation2": "02",
            "fineLocation3": "WS",
            "physicalDimension": "AN",
            "direction": "#",
            "entityType": "spring",
            "component": "rotation",
            "id": 10325,
            "write": false
        },
        {
            "testObject": "0",
            "position": "0",
            "mainLocation": "EKIN",
            "fineLocation1": "SU",
            "fineLocation2": "00",
```

```
            "fineLocation3": "00",
            "direction": "0",
            "physicalDimension": "EN",
            "entityType": "whole",
            "filter": "C",
            "component": "kinetic",
            "write": true
        },
        {
            "testObject": "0",
            "position": "0",
            "mainLocation": "EHOU",
            "fineLocation1": "SU",
            "fineLocation2": "00",
            "fineLocation3": "00",
            "direction": "0",
            "physicalDimension": "EN",
            "entityType": "whole",
            "filter": "A",
            "component": "hourglass",
            "write": true,
            "descriptors": [
                {
                    "description": "Energy Type",
                    "value": "Hourglass Energy"
                }
            ]
        }
    ],
    "operations": [
        {
            "operation": "sub",
            "input": [
                "11ABRIRI02WSAN#0",
                1.570796
            ],
            "output": "11ABRIRI02WSANZ0",
            "write": false
        },
        {
            "operation": "c180",
            "input": [
                "11ABRIRI02WSANZ0"
            ],
            "output": "11ABRIRI02WSANZC",
            "write": true,
            "descriptors": [
                {
```

```
                    "description": "Abdomen Compression valid Value",
                    "value": "46"
                },
                {
                    "description": "Abdomen Compression Invalid Value",
                    "value": "65"
                }
            ]
        }
    ]
}
```

Configuration file properties

The following tables list all the required and optional properties and what they can be set to. If they aren't set, the default value is used.
Some values should follow the guidance in the ISO Related Electronic Documents A and B (RED A and RED B) and they are indicated below. They can be found on the ISO website https://www.iso-mme.org/forum/.

Test Data properties

The first Table specifies the Test Data properties which provide details about the test. It is necessary to include testName and formatVersion.

Property	Description	Valid values	Default	Type	Necessity
testName	Name of test. This is used for the test directory name and in the filenames.	Any value	-	String	Required
formatVersion	Format Version specifies the ISO-MME output	1.6 or 2.0	2.0	String	Required

	version required.				
thisVersion	T/HIS Build Version	Any Numerical value. For Eg:- “21.0”	-	String	Required
thisBuildNumber	T/HIS Build Number	Any numerical value. For Eg:- “6100”	-	String	Required
timestamp	A timestamp dates. It is written in the header of the test information *.mme file.	Any value, but the expected format is "yyyy-mm-dd".	"NOVALUE"	String	Optional
laboratoryName	The laboratory name. It is written in the header of the test information *.mme file.	Any value.	"NOVALUE"	String	Optional
laboratoryContactName	The laboratory contact name. It is written in the header of the test information *.mme file.	Any value.	"NOVALUE"	String	Optional
laboratoryContactPhone	The laboratory contact phone number. It is written in the header of the test information *.mme file.	Any value.	"NOVALUE"	String	Optional

laboratoryContactFax	The laboratory contact fax number. It is written in the header of the test information *.mme file.	Any value.	"NOVALUE"	String	Optional
laboratoryContactEmail	The laboratory contact email. It is written in the header of the test information *.mme file.	Any value.	"NOVALUE"	String	Optional
laboratoryTestRef	The laboratory test reference. It is written in the header of the test information *.mme file.	Any value.	"NOVALUE"	String	Optional
typeOfTest	The test type. It is written in the header of the test information *.mme file.	See the ISO Related Electronic Document A for valid values, e.g. "Frontal Impact".	"NOVALUE"	String	Optional
subtypeOfTest	The test subtype. It is written in the header of the test	See the ISO Related Electronic Document A for valid values, e.g. "0 Degree Active".	"NOVALUE"	String	Optional

	information *.mme file.				
virtualTestingReferenceId	The virtual testing ref id is used to write the filename for channel index file and channel data file.	Any value.	"NOVALUE"	string	Optional
typeOfDataSource	Type of data source. It is written in the header of the test information *.mme file.	Any value.	"NOVALUE"	String	Optional
regulation	The test regulation. It is written in the header of the test information *.mme file.	See the ISO Related Electronic Document A for valid values, e.g. "EuroNCAP".	"NOVALUE"	String	Optional
testDate	The date of the test. It is written in the header of the test information *.mme file.	Any value, but the expected format is "yyyy-mm-dd".	"NOVALUE"	String	Optional
unitSystemModel	Used to set up units for model. If these are not	"U1","U2","U3","U4","U5","U6".	blank	String	Optional

	provided the units would be as they were prior to write ISO-MME or undefined if units were not set.				
unitSystemDisplay	Used to set up units for display. Will be set to SI if model units specified and display units not specified.	"U1","U2","U3","U4","U5","U6".	blank	String	Optional
xAxisInterval	Used to specify X Axis Interval for all curves to regularised.	0 to 1e-7	blank	Number	Optional

Descriptor properties

Along with the predefined properties listed in the tables detailing about Test, Objects and Channels, you can define any additional descriptor you like by providing a description and value using the syntax in the table below.

The descriptor JSON object allows you to define any number of descriptors that can appear in the ISO MME Files.

1.
    - Test data file(.mme) if defined in Test data block
2.
    - Object files(.mmi) if defined in Object Blocks
3.
    - Data Files (.001 or. mmd) if defined in channel or operation block.

To provide any valid description/value please see the see the details below.

Property	Description	Valid values	Default	Type	Necessity
description	The Name of the Property for which you are creating the description.	This is supposed to mentioned as String as required by protocols. For eg. "Time step setting", "Number of CPUs"	NOVALUE	String	Optional
value	The Property which is defined in the 'description' is specified a value in this section.	This can be either string or integer for example -0.2 or " Value"	NOVALUE	Number or String	Optional

Object arrays

The table lists the 'Standard' object types that are supported by T/HIS. Currently T/HIS supports the following objects:-

Objects	Description	Properties	Definition String	Default TestObject
vehicles	The vehicles array contains vehicle objects.	name, velocity, mass, impactSide	"vehicles"	Numerical starting from 1
dummies	The dummies array contains dummy objects.	name, gender, age, position	"dummies"	'D'
barriers	The barriers array contains barrier objects.	name, barrierWidth, barrierHeight, yawAngle	"barriers"	'B'
mobileBarriers	The mobileBarriers array contains mobile barrier objects.	name, barrierWidth, barrierHeight, yawAngle	"mobileBarriers"	'M'

Object properties

This table lists the properties that are supported by T/HIS for different objects. The properties provide details about the object.

Property	Description	Valid values	Default	Type	Necessity
testObject	testObject classification. It is the first character in the ISO-MME channel code. This property is used to name Object files (.mmi)	See the 'Test Object' section in the ISO Related Electronic Document B for valid values, e.g. "1", "2", “D” etc.	“D” for Dummies “B” for Barriers “M” for mobileBarriers Numerical Value for Vehicle	String	Required
position	The dummy’s position in the vehicle. It is the second character in the ISO-MME channel code. This property is used to name Object files(.mmi)	See the ‘Position’ section in the ISO Related Electronic Document B for valid values, e.g. “1”, “2”.	“0”	String	Required
name	The test object name. It is written in the header of the object *.mmi file for Version 2.0 and mentioned in *.mme in Version 1.6.	Any value	blank	String	Optional
velocity	The object’s velocity. It is written in the header of the object *.mmi file.	Any value	"NOVALUE"	Number	Optional
mass	The object’s mass. It is written in the header of the object *.mmi file.	Any value	"NOVALUE"	Number	Optional
impactSide	It is written in the header of the object *.mmi file.	See the 'Fine Location 1' section in the ISO Related Electronic Document B for valid values, e.g., "LE", "LO".	"00"	String	Optional
gender	The dummy gender. It is written in the object *.mmi file.	Any value	"NOVALUE"	String	Optional

age	The dummy age. It is written in the object *.mmi file.	Any value	"NOVALUE"	Number	Optional
barrierWidth	The barrier width. It is written in the object *.mmi file.	Any value	0.0	Number	Optional
barrierHeight	The barrier height. It is written in the object *.mmi file.	Any value	0.0	Number	Optional
yawAngle	The barrier yaw angle. It is written in the object *.mmi file.	Any value	0.0	Number	Optional

Channel Data properties

Channel Properties are used to define details about the Channel.

Property	**Description**	**Valid values**	**Default**	**Type**	**Necessity**
testObject	testObject classification. It is the first character in the ISO-MME channel code.	See the 'Test Object' section in the ISO Related Electronic Document B for valid values, e.g. "1", "2", “D” etc.	“D” for Dummies “B” for Barriers “M” for mobileBarriers Numerical Value for Vehicle	String	Required
position	The dummy’s position in the vehicle. It is the second character in the ISO-MME channel code.	See the 'Position' section in the ISO Related Electronic Document B for valid values, e.g. "1", "2".	"0"	String	Required
mainLocation	Main location on the object.	See the ISO Related Electronic Document B for valid values, e.g. "HEAD", "CHST".	-	String	Required

	This is required for the channel code.				
entityType	The LS-DYNA entity types to extract data for.	A FAST-TCF data extraction keyword (See Section 7.4.5), e.g. "node", "beam"	-	String	Required
id	The LS-DYNA entity ID to extract data for.	This can be the numerical ID or a *DATABASE_HISTORY_ID name or an array of IDs or names. If an array is defined, the output generated is the sum of the individual curves. In either case, the IDs/names should be specified as a string or number, e.g. "100",100, "my_node_id", [100, 200, 101], ["my_node_id_1","my_node_id_2"]. The <id> property is not required for global data (e.g. global kinetic energy).	-	Number, String, Array of String or Numbers	Optional
component	The component to read.	FAST-TCF data extraction component words (See Section 7.4.5), e.g. "displacement x", "energy"	-	String	Required
source	The LS-DYNA file to read data from.	A FAST-TCF file keyword (See Section 7.4.2.1), e.g. "lsda", "thf"	Blank (will extract data from the default file for the specified entity type).	String	Optional
fineLocation1	Fine location on the object. This is	See the 'Fine Location 1' section in the ISO Related Electronic Document B for valid values, e.g. "IN", "OU".	"00"	String	Optional

	used for the channel code.				
fineLocation2	Fine location on the object. This is used for the channel code.	See the 'Fine Location 2' section in the ISO Related Electronic Document B for valid values, e.g. "IN", "OU".	"00"	String	Optional
fineLocation3	Fine location on the object. This is used for the channel code.	See the 'Fine Location 3' section in the ISO Related Electronic Document B for valid values, e.g. "IN", "OU".	"00"	String	Optional
physicalDimension	Physical dimension data is determined automatically from the required "component" property but can be overwritten with this optional property. This is used for the channel code.	See the 'Physical Dimension' section in the ISO Related Electronic Document B for valid values, e.g. "AN", "DS".	"00"	String	Optional
direction	The data direction is determined	See the 'Direction' section in the ISO Related Electronic Document B for valid values, e.g. "R", "X".	"0"	String	Optional

<table>
<tr><td></td><td>automatically from the required "component" property but can be overwritten with this optional property. This is used for the channel code.</td><td></td><td></td><td></td><td></td></tr>
<tr><td>xAxis Interval</td><td>Used to specify x Axis Interval for the curve to be regularised. This will overridethe x Axis Interval property specified in test info block</td><td>0 to 1e-7</td><td>blank</td><td>Number</td><td>Optional</td></tr>
<tr><td>filter</td><td>A filter to use on the extracted data.</td><td>The Filter Class Code are positioned 16th in the ISO-MME Channel Code. The following Filter Class Code are supported by T/HIS: -
<table>
<tr><td>Filter Class Code</td><td>Description</td></tr>
<tr><td>“0”</td><td>Unfiltered</td></tr>
<tr><td>“1”</td><td>FIR100</td></tr>
<tr><td>“2”</td><td>Combined A and B</td></tr>
<tr><td>“A”</td><td>CFC1000</td></tr>
<tr><td>“B”</td><td>CFC600</td></tr>
<tr><td>“C”</td><td>CFC180</td></tr>
<tr><td>“D”</td><td>CFC60</td></tr>
<tr><td>“E”</td><td>CFC1000</td></tr>
<tr><td>“F”</td><td>CFC600</td></tr>
</table></td><td>blank (UnFiltered)</td><td>String</td><td>Optional</td></tr>
</table>

		"G"	CFC180			
		"H"	CFC60			
write	An option to select whether the curve should be written to ISO-MME format.	Acceptable values are 'true' and 'false'. The default case is taken as true. If set to false, at the end of the ISO-MME writing process in FAST-TCF, the curve is deleted.		true	Boolean	Optional

Operation Data properties

Some ISO-MME channels can be written directly from the LS-DYNA results data. However, many require further operations, for example to apply filtering or convert a rotation sensor to a displacement. Each operation takes one or more inputs, uses FAST-TCF to perform the underlying operation, and produces an output. You can "chain" several operations together to produce the desired final output. Intermediate outputs can be given any name; the final output names must match the desired ISO-MME channel code. The following table lists the properties required to define each operation.

Property	Description	Valid values	Default	Type	Necessity
operation	The type of operation to be performed on the input curves.	The accepted values for the operation are FAST-TCF Operation values as mentioned in Section 7.8 operation commands (e.g. "muly", "add").	-	String	Required
input	To process operations by FAST-TCF File, T/HIS requires arguments based on the type of operation.	Accepted inputs are ISO-MME channel codes for already created curves in the channel block or output curves created in Operation Block, or Numerical Values. The elements in the Input array are required to be in same order as mentioned in FAST-TCF Manual 7.8.	-	Array of strings or Numbers	Required

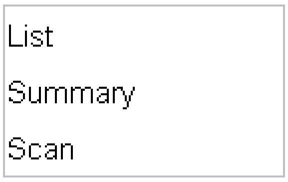
		Eg:- ["11ABRIRI02WSAN#0", 1.570796]			
output	The output property is used to specify the FAST-TCF curve tag.	For intermediate operation outputs, this can be any string. For the final output, this should be a valid ISO-MME channel code. Eg:- "intermediate curve" or ”11ABRIRI02WSANZC”	-	String	Required
write	An option to select whether the curve should be outputted.	Acceptable values are 'true' and 'false'. The default case is taken as true. If set to false, at the end of the ISO-MME writing process in FAST-TCF, the curve is deleted.	true	Boolean	Optional

8.3.2. WRITE TO SCREEN

WRITE TO SCREEN

Writes data to a text window on the screen.

OUTPUT FORMAT



LIST	This option will write out all the points in the selected curves.
Summary	Gives a summary of the curve. This includes the type of data being plotted and the maximum and minimum values in the curve.
SCAN	Scans a group of curves and reports the maxima and minima values for each individual curve along with the overall maxima and minima

8.4. Curve Manager

Curve Manager

In screen menu mode curves are managed using the **CURVE MANAGER** window, shown in the figure (below).

By default the **CURVE MANAGER** menu only displays 1000 curves. An unlimited amount of curves can be used and these are displayed in the menu in blocks of 1000. If an attempt is made to use a curve higher than 1000 then the Range options are used to select which group of 1000 curves you wish to display.

Against each curve that currently contains information is a curve number button. The colour of this button indicates the current blanking status of a curve

The curve is unblanked in all active graphs (Active Graphs)

The curve is blanked in all active graphs

The curve is unblanked in some active graphs

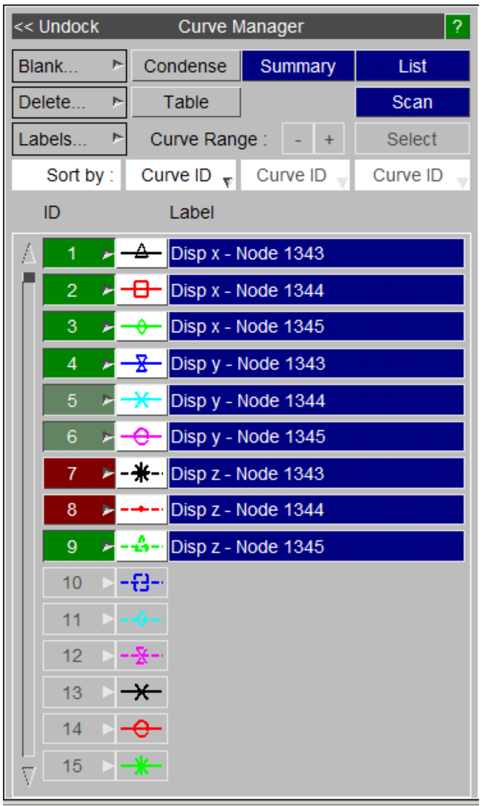
The blanking status of each curve can be changed by clicking on this button. The Curve Table can also be used to change the blanking status of a curve.

A range of curves may either be blanked or unblanked by selecting the first button in the range and then holding down the **SHIFT** key while selecting the last button in the range. All buttons that lie between the first and last buttons selected will have their status changed to match that of the first button that was selected.

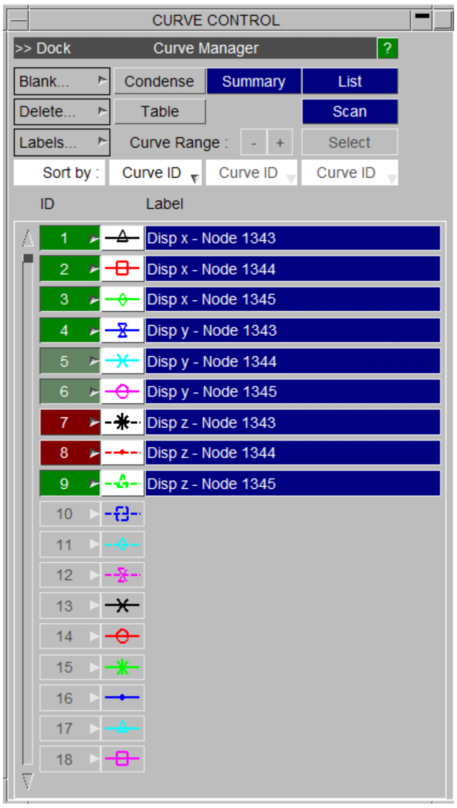
The line label for each curve may be changed by over-typing the label currently displayed in the line label box.

The button located between the curve number button and the curve label shows the current colour, line style and symbol that will be used to plot the curve. These properties can be modified by clicking on this button to display the line style menu, see LINE STYLES

The **CURVE CONTROL** window can also be accessed via the **File....Curves** option at the top of the graphics window or from the **Curves** button in the main menu.



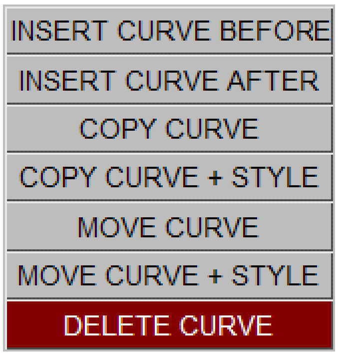
If the curve labels are too long to be seen in the standard Curve Manager menu then the menu can be turned into a floating menu by selecting the **<<<Undock** option in the menu header. After undocking the menu it can be re-docked by selecting **>>>Dock.**



8.4.1. Reordering Curves

Reordering Curves

Attached to each of the curve number buttons is a popup menu that can be used to reorder curves by copying and moving them. This menu is accessed by clicking the right mouse button over the curve number buttons.



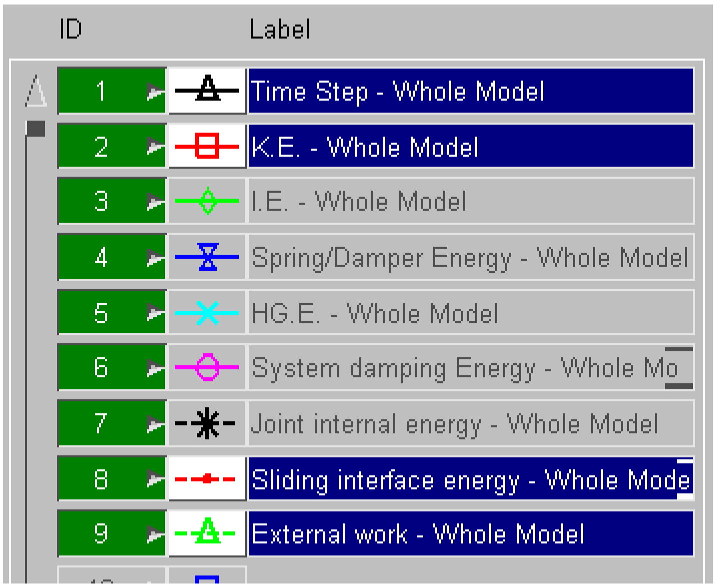
INSERT CURVE BEFORE	Inserts the last curve copied to a scratch definition before the selected curve.
INSERT CURVE AFTER	Inserts the last curve copied to a scratch definition after the selected curve.
COPY CURVE	Copies the curve to a scratch definition.
COPY CURVE + STYLE	Copies the curve along with its line style settings to a scratch definition.
MOVE CURVE	Copies the curve to a scratch definition and then deletes the original curve
MOVE CURVE + STYLE	Copies the curve along with its line style settings to a scratch definition and then deletes the original curve
DELETE CURVE	Deletes the selected curve

Block Moving/Copying Curves

Since version 16, T/HIS has had the option to move or copy a selection of curves and insert them before/after a given curve.

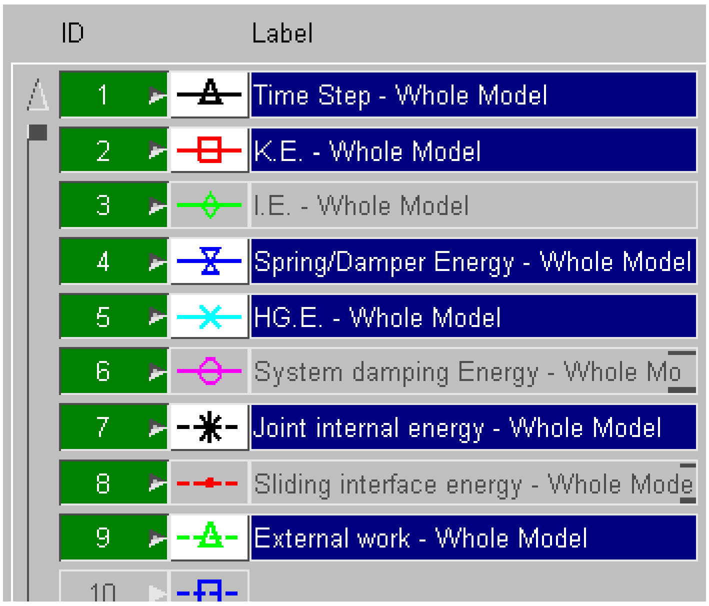
This can be done via selecting the first curve as per usual and then either:

- **SHIFT + "Move/Copy Curve"** - To select a block of curves
- **CTRL + "Move/Copy Curve"** - To add an additional curve to your previous selection



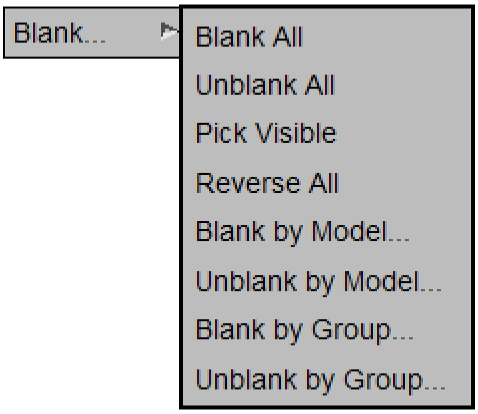
At any time, the operation can be cancelled by right clicking a curve which is in its "pending" status (greyed out), and selecting either **Cancel Move/Copy** to cancel the curve selected or via **Cancel Move/Copy - All** which cancels all "pending" curves.

The order in which the curves are inserted is the same order in which they are currently in the list. For example, the image below has curves 3,6 and 8 as pending. When these are inserted they will be condensed so that they are next to one another. So, if this selection was inserted after curve 9, then these three curves will occupy slots 10,11 and 12 respectively.



8.4.2. Blank...

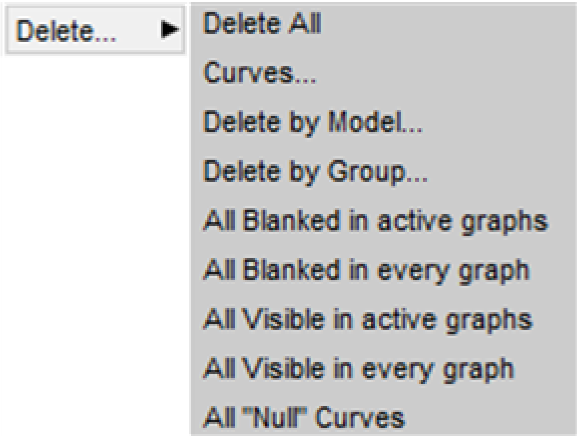
Blank...



Blank All	Blank all curves
Unblank All	Unblank all curves
Pick Visible	Pick curves from the screen to be blanked.
Reverse All	Reverse the blanking status of all curves
Blank by Model...	Blank curves belonging to a Model
Unblank by Model...	Unblank curves belonging to a Model
Blank by Group...	Blank curves by Curve Group
Unblank by Group...	Unblank curves by Curve Group

8.4.3. Delete...

Delete...

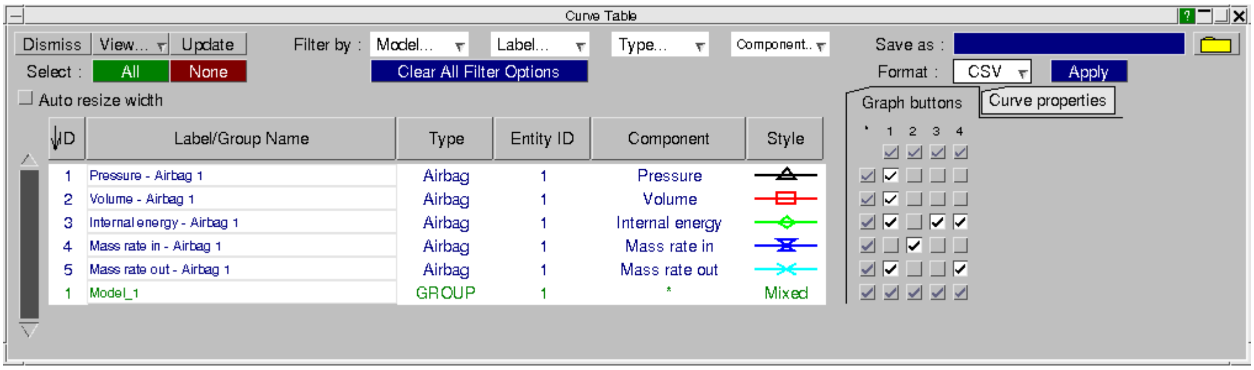


Delete All	Deletes all current curves. You are prompted for confirmation first!
Curves...	Select groups of curves for deletion
Delete by Model...	Delete curves belonging to a Model
Delete by Group...	Delete curves by Curve Group
All Blanked in active graphs	Delete all the curves that are currently blanked within all currently active graphs
All Blanked in every graph	Delete all the curves that are blanked over every graph, irrespective of active graph status (equivalent to if every graph is active)
All Visible in active graphs	Delete all the curves that are currently unblanked within all currently active graphs
All Visible in every graph	Delete all the curves that are unblanked over every graph, irrespective of active graph status (equivalent to if every graph is active)
All "Null" Curves	Delete any curve which is "NULL"

8.4.4. Table

Table

The Table option can be used to give more control over which curves are blanked and unblanked in all of the currently defined graphs, as well as display curve properties and injury values in a tabulated format. By default the Curve Table displays a scrolling list of all of the currently defined curves and curve groups along with a set of tick boxes that display the status of the curve in the current graphs. Curves are displayed in BLUE text while curve groups are displayed in GREEN.



For each curve the following information is displayed by default.

ID	Curve ID or Group ID for curve groups
Label	Curve Label or Group Name
Directory	If the curve has been read in from a model then this will be the directory that all all the model files are in, if the curve had been read in from a file (.cur. .csv) then this will be the file location. No information is displayed for curve groups.
Model/File	If the curve has been read in from a model then by default this will be the ID of the model. If the curve had been read in from a file then this will be the filename. No information is displayed for curve groups.
Type	The entity type that the curve was generated from. If the curve was read in from a file then this will display "FILE".
Entity ID	ID of the item that the data was read from. If the curve was read from a file then this will be the index within the file for each curve. If the row represents more than one curve (e.g. curve groups) and the curves have different components then it will display '*'
Component	Data component name.

	If the row represents more than one curve (e.g. curve groups) and the curves have different components then it will display '*'
Style	This will show the line colour, style and width used to display the curve.

The column widths of any of the above columns can be adjusted by clicking on the bars between the header columns and the order of the columns can be changed by dragging the column headers.

The contents of the table can also be sorted by any column by clicking on the relevant header button. Clicking on the same header a 2nd time reverses the sort order for the column.

Adding / Removing Curves from graphs and Locking / Freezing

To add an individual curve (or curve group) to a graph the tick boxes on the right hand side of the curve table can be used.

The first column of tick boxes (under the *) can be used to add/remove a curve from all the currently defined graphs, while the top row of tick boxes can be used to add/remove all the currently defined curves from a graph.

- If all of the curves are unblanked in a graph then the tick box will display a black tick in a white box.
- If some of the curves are unblanked in a graph then the tick box will display a dark grey tick in a grey box.
- If none of the curves in a group are unblanked in a graph then the tick box will be empty.

Multiple tick boxes can be set/unset by clicking on the first one and then holding SHIFT while clicking on the last one.

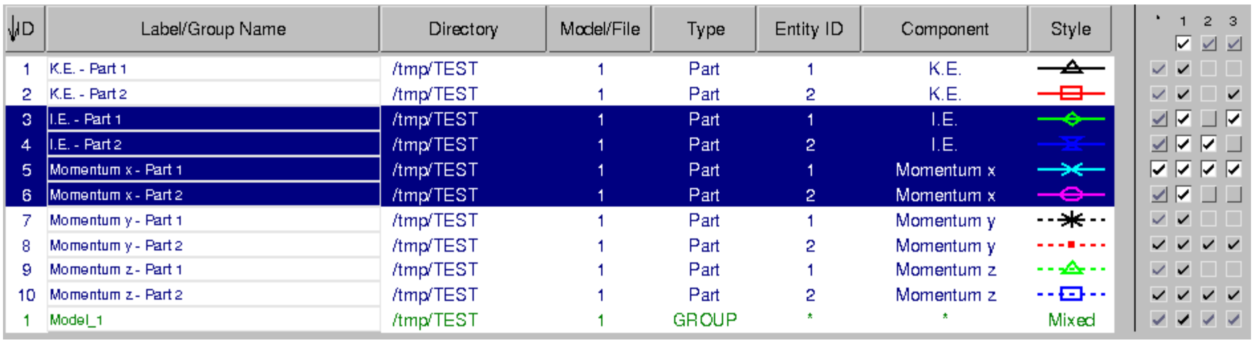
These tickboxes can also be used to lock or freeze curves. If the 'Locking/Freezing' button, or the 'Locking/Freezing' option in the 'Graph buttons' popup is selected, then the tickboxes are re-purposed. Locking a curve means fixing it as blanked in a graph so it cannot be made visible until it is unlocked. Freezing a curve is the equivalent for visible curves. The curve will be visible in that graph until it is unfrozen. These curves will no longer be affected by shortcut keys such as 'u', 'r' and 'b'.

Instead of changing the ticks, locking and freezing will change the background colour of the tickbox. When a curve is locked, the background will be red. When it it is frozen, the background will be green.

The buttons relating to multiple graphs or multiple curves behave in the same way as for blanking, as does multiple-selection using CTRL or SHIFT.

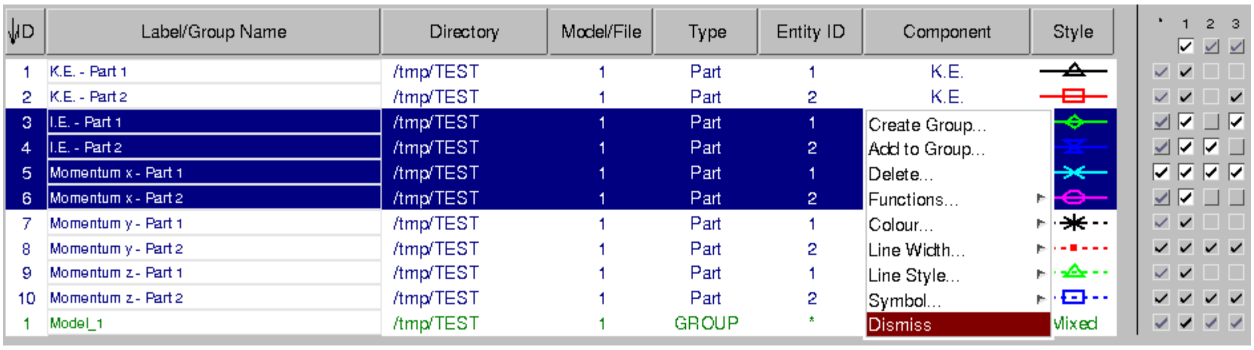
Individual curves can also be selected by clicking on them in the main part of the curve table. Multiple curves can be selected using either CTRL to select a single curve or SHIFT to select a range of curves. As curves are selected they are highlighted in blue and the tick boxes for any unselected curves are greyed out.

When multiple curves have been selected then clicking on a tick box sets the status for all the selected curves.



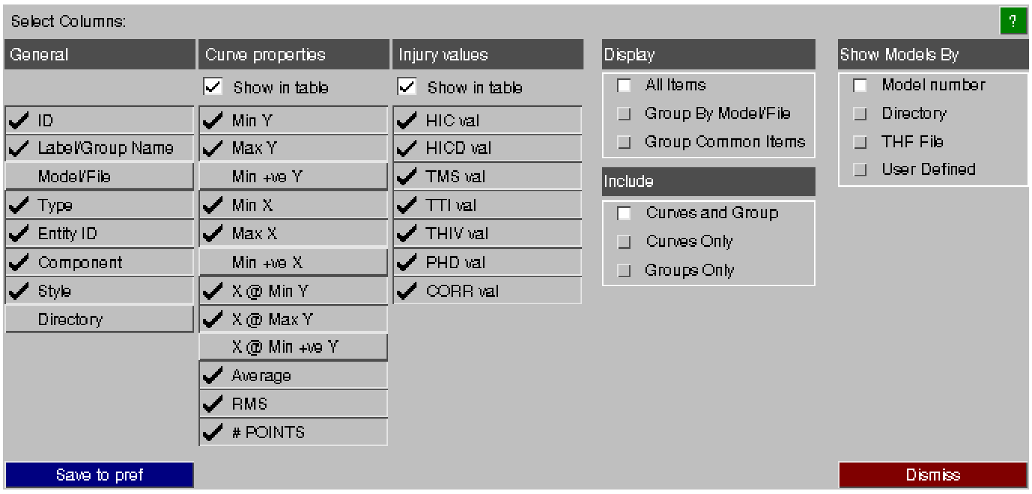
ID	Label/Group Name	Directory	Model/File	Type	Entity ID	Component	Style
1	K.E. - Part 1	/tmp/TEST	1	Part	1	K.E.	
2	K.E. - Part 2	/tmp/TEST	1	Part	2	K.E.	
3	I.E. - Part 1	/tmp/TEST	1	Part	1	I.E.	
4	I.E. - Part 2	/tmp/TEST	1	Part	2	I.E.	
5	Momentum x - Part 1	/tmp/TEST	1	Part	1	Momentum x	
6	Momentum x - Part 2	/tmp/TEST	1	Part	2	Momentum x	
7	Momentum y - Part 1	/tmp/TEST	1	Part	1	Momentum y	
8	Momentum y - Part 2	/tmp/TEST	1	Part	2	Momentum y	
9	Momentum z - Part 1	/tmp/TEST	1	Part	1	Momentum z	
10	Momentum z - Part 2	/tmp/TEST	1	Part	2	Momentum z	
1	Model_1	/tmp/TEST	1	GROUP	*	*	Mixed

As well as blanking and unblanking curves in graphs a number of other options can be applied to selected curves by right clicking on them, such as applying operations and changing the line style.



View Options

The viewing options popup, found in the top left of the curve table window, can be used to control which columns of data are displayed and what items are displayed in the curve table.

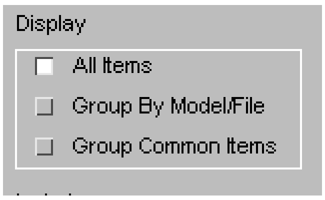


By default all 8 columns of general information about each curve will be displayed, each column can be turned on and off but T/HIS will ensure that at least one column is always displayed. Curve properties and injury values can also be displayed, but this will be discussed further below.

The columns that initially displayed can be specified in the preference file (see Format of the oa_pref File for more details). If the columns are changed then **Save to Pref** can be used to update the preference file.

Display

This option can be used to control how items are displayed in the curve table.



All Items

By default the curve table will contain one row for each curve and one row for each curve group.

1	K.E. - Whole Model	E:\BASE	1	Model	1	K.E.	
2	I.E. - Whole Model	E:\BASE	1	Model	1	I.E.	
3	K.E. - Whole Model	E:\RUN1	2	Model	1	K.E.	
4	I.E. - Whole Model	E:\RUN1	2	Model	1	I.E.	
11	Disp mag - Node 100	E:\BASE	1	Node	10000	Disp ma	
12	Disp mag - Node 100	E:\RUN1	2	Node	10000	Disp ma	
1	Model_1	N/A	N/A	GROUP	*	*	Mixed
2	Model_2	N/A	N/A	GROUP	*	*	Mixed

Group By Model/File

*	*	E:\BASE	1	*	*	*	Mixed
*	*	E:\RUN1	2	*	*	*	Mixed
1	Model_1	N/A	N/A	GROUP	*	*	Mixed
2	Model_2	N/A	N/A	GROUP	*	*	Mixed

This option will display a single row for all the curves that were read from the same model or file.

When this option is selected the columns for curve ID, Label, Type, Entity ID and component display a '*' as they represent multiple values.

This option can be used to quickly assign all of the curves from a single model or file to the same graph.

Group By Common Items

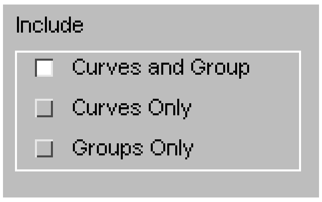
*	*	*	*	Model	1	K.E.	Mixed
*	*	*	*	Model	1	I.E.	Mixed
*	*	*	*	Node	10000	Disp ma	Mixed
1	Model_1	N/A	N/A	GROUP	*	*	Mixed
2	Model_2	N/A	N/A	GROUP	*	*	Mixed

This option will display a single row for all the curves that were created using the same entity type, ID and component.

In the example opposite the 1st row represents all of the curves that contain a model Kinetic Energy while the 3rd row represents all the curves that contain a displacement magnitude for Node 10000.

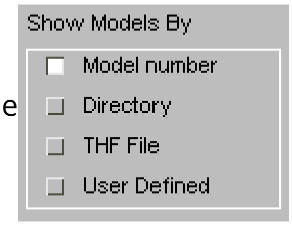
This option can be used to quickly assign all of the curves for the same entity and component to the same graph when comparing results from multiple models.

Include



By default the curve table contains both curves and curve groups. This option can be used to display either just the curves only or just the curve groups.

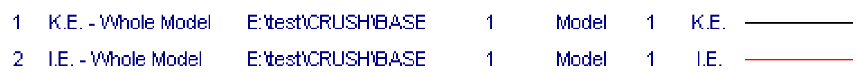
Show Models By



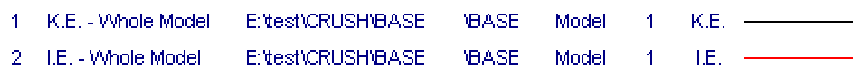
If the column displaying the model ID is displayed in the curve table then by default it will display the model number.

This option can be used to display either.

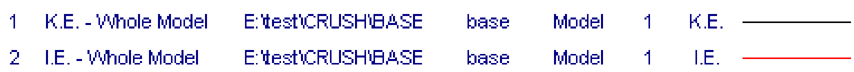
The model ID



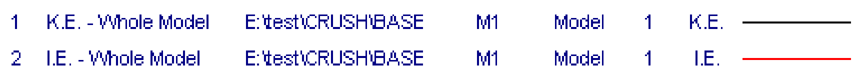
The model directory



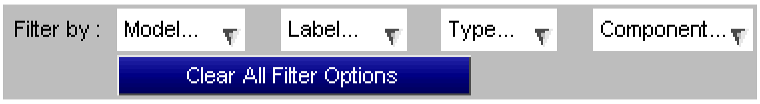
The name of the THF file



A user defined model description



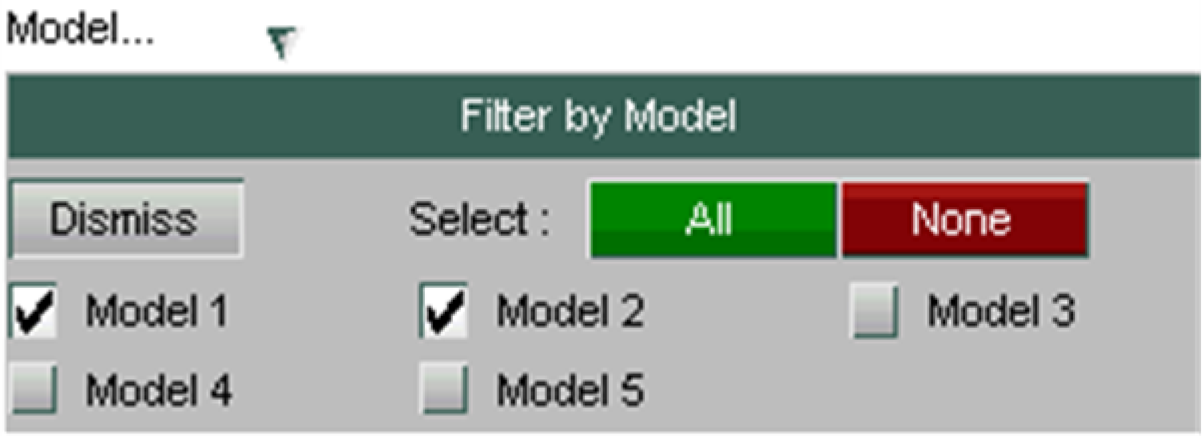
Filter Options



The filter options can be used to filter the list of curves displayed in the curve table.

Multiple filters can be active at the same time

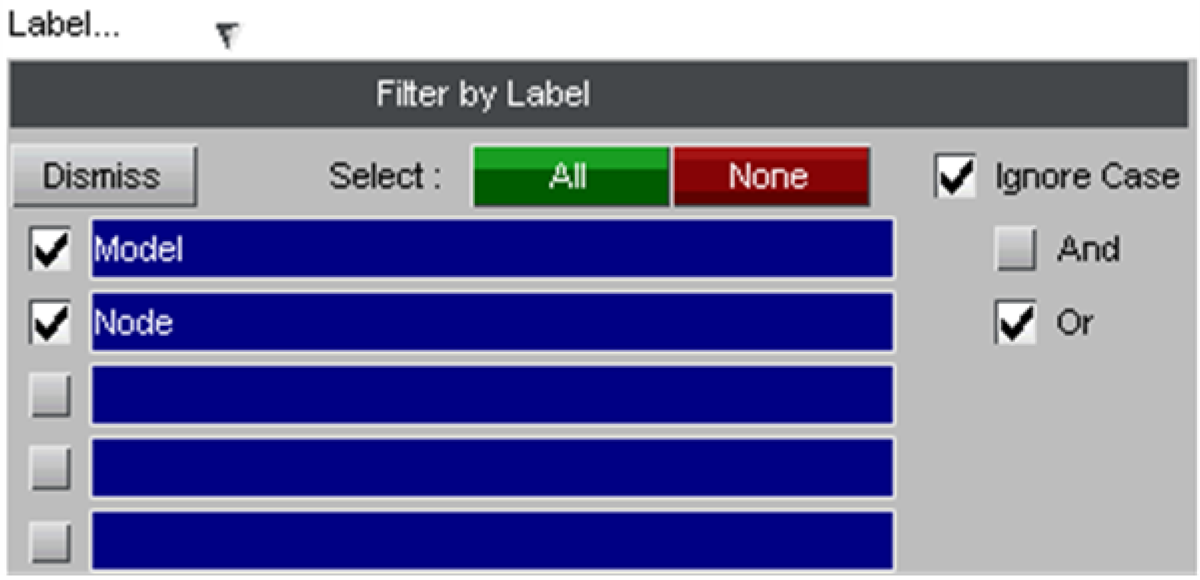
Filter By Model



This option can be used to filter the list of curves by model number. If curves have been read in from a file then an "Other" option will be shown.

In the example opposite only curves that are either from model 1 or model 2 will be displayed.

Filter By Label

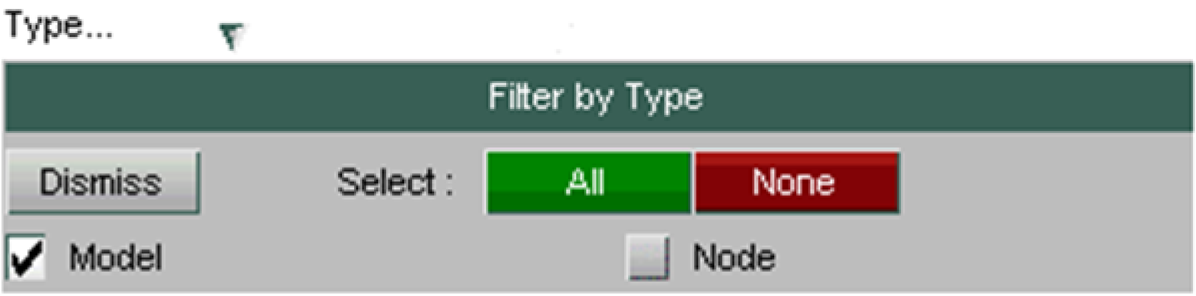


This option can be used to filter the list of curves by label. Up to 5 different strings can be entered and the list of curves displayed will be filtered using those strings. If multiple strings are entered then the strings can either be combined using either "AND" or "OR".

A separate option can be used to ignore the case so that "model" will match both "Model" and "model"

In the example opposite only curves that contain either the word "Model" OR the word "Node" in their labels will be displayed.

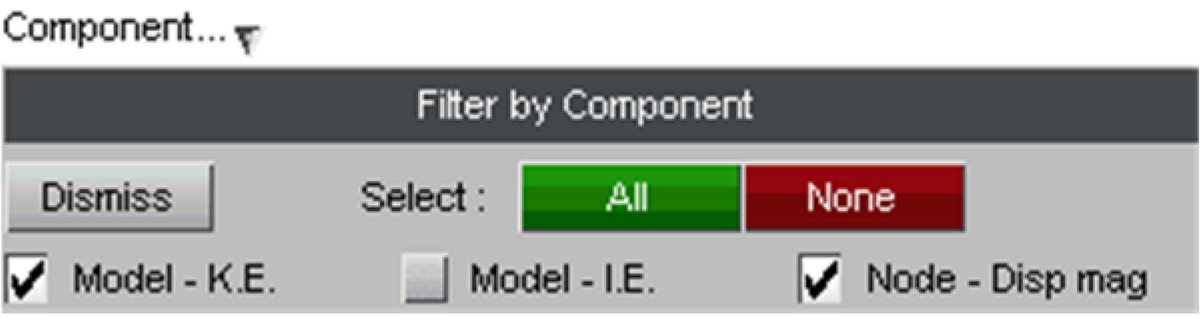
Filter By Type



This option can be used to filter the list of curves by entity type. The list of entity types displayed will automatically update to show the entity types for all the curves that are currently stored in T/HIS.

In the example opposite only curves that contain "Model" data are displayed.

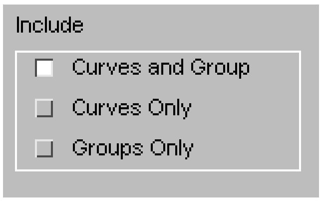
Filter By Component



This option can be used to filter the list of curves by component type. The list of components displayed will automatically update to show the components for all the curves that are currently stored in T/HIS.

In the example opposite only curves that are either Model Kinetic Energy or Nodal Displacement Magnitudes are displayed.

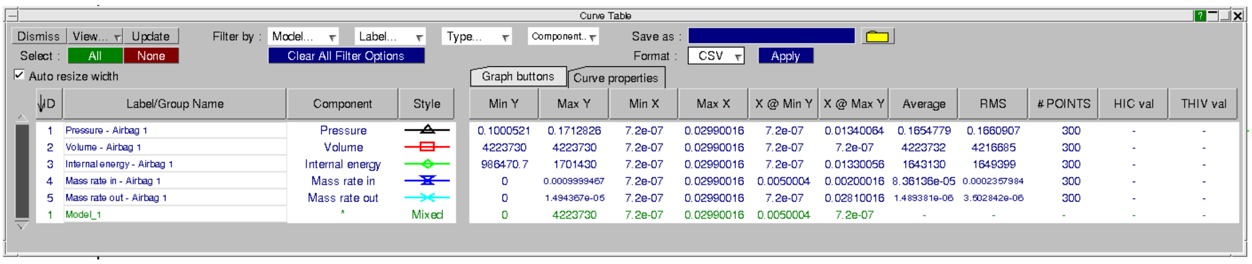
Include



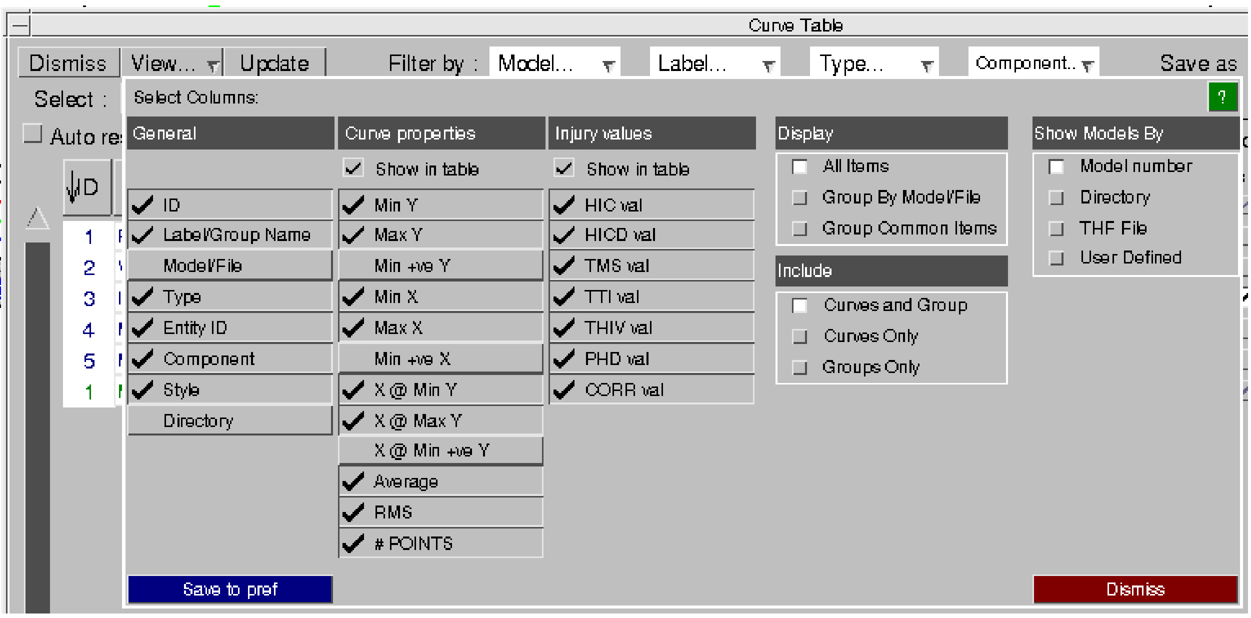
By default the curve table contains both curves and curve groups. This option can be used to display either just the curves only or just the curve groups.

Curve Properties

The properties of each curve and any calculated injury values can also be displayed in the Curve Table. These are displayed by selecting the **Curve Properties** tab above the graph tickboxes. The Curve Table, including the values in all the displayed columns (except the Style column), can be written out to either a .csv or .xlsx file.

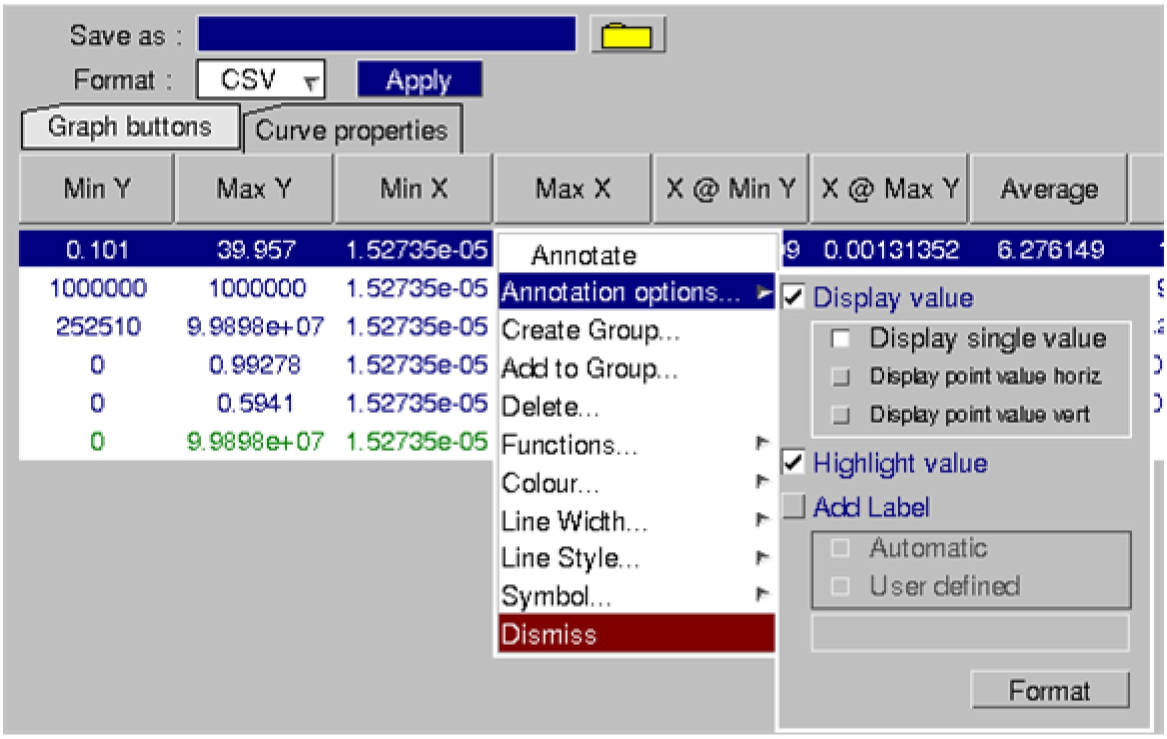


The Curve Properties and Injury Values columns that are displayed can be customized in the **View...** popup, both individually by clicking their name in the popup and as a group using the **Show in table** tickboxes. The choice of displayed columns can be saved to preferences.



Annotating Curves

Curves can be annotated with most of the properties and injury values by right-clicking the values in the table and selecting **Annotate**. Options for customising these annotations can be found in the **Annotation options** popup. The options include the format of the displayed value, i.e. whether it should appear as a single value (usually either an X or Y value depending on the property), or as a point (X,Y). The choice to highlight the value on the curve with a cross is also given, as well as the ability to add either an automatic or user-defined label to the annotation.



8.4.5. Summary

Summary

Displays a window from which a group of curves may be chosen. The maximum and minimum values of the selected curves are then displayed.

8.4.6. List

List

Displays a **LIST CURVES** window, from which a number of curves may be selected. The data point values for the selected curves are then listed in a listing box.

8.4.7. Scan

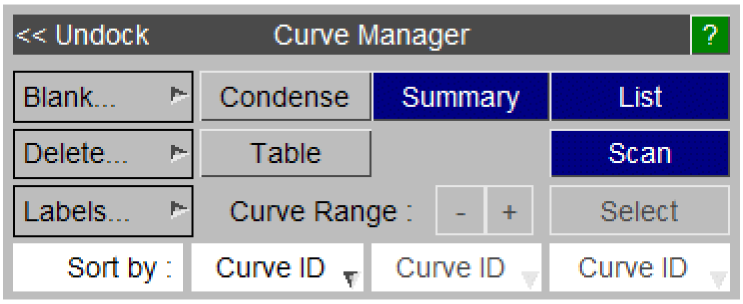
Scan

Displays a window from which a group of curves may be chosen. The maximum and minimum values of the selected curves are then displayed.

8.4.8. CURVE RANGE SELECTION

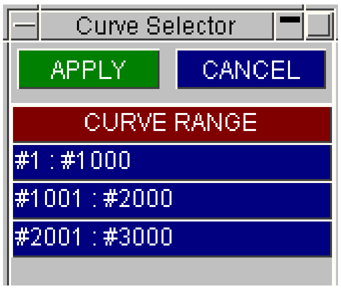
CURVE RANGE SELECTION

The range buttons in the Curve Control menu can be used to when you are working with more than 1000 curves to move between groups of 1000 curves. Pressing the green **+** button will display the next group of 1000 curves in the menu, whilst pressing the red **-** button will display the previous group of 1000 curves.



Alternatively pressing the **Select** button will bring up the following new window.

Select the appropriate group of 1000 curves and press **Apply** to display those 1000 curves in the Curve Manager.



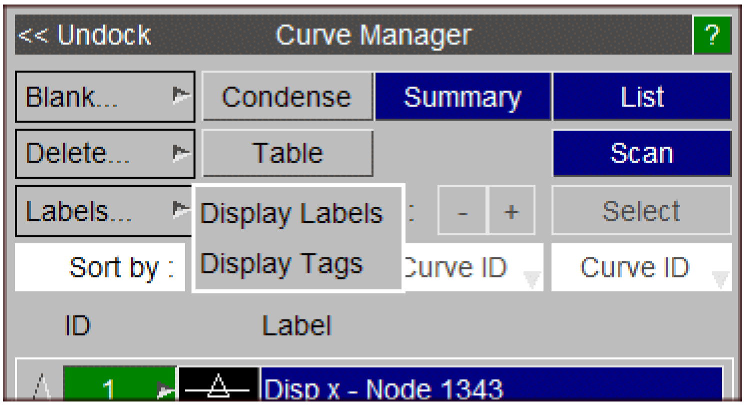
8.4.9. CURVE TAGS

CURVE TAGS

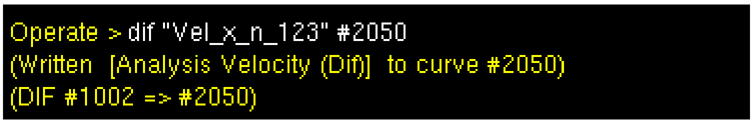
Curves can be given tags to act as internal identifiers within T/HIS which can be used to reference curves in order to perform operations on them.

In order to display the curve tags, toggle on the Show Labels arrow and select Show Tags. The tag names can be defined in the input boxes.

When a curve file is written, T/HIS will save the tags of all the tagged curves in the file.



When performing operations in the dialogue box, curves can be referenced by their tags. The tag must be placed in double quotes.

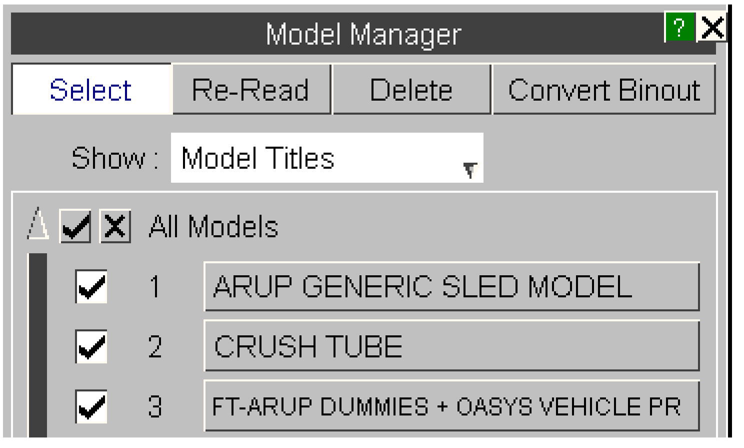


8.5. Model Manager

8.5.1. Select

Select

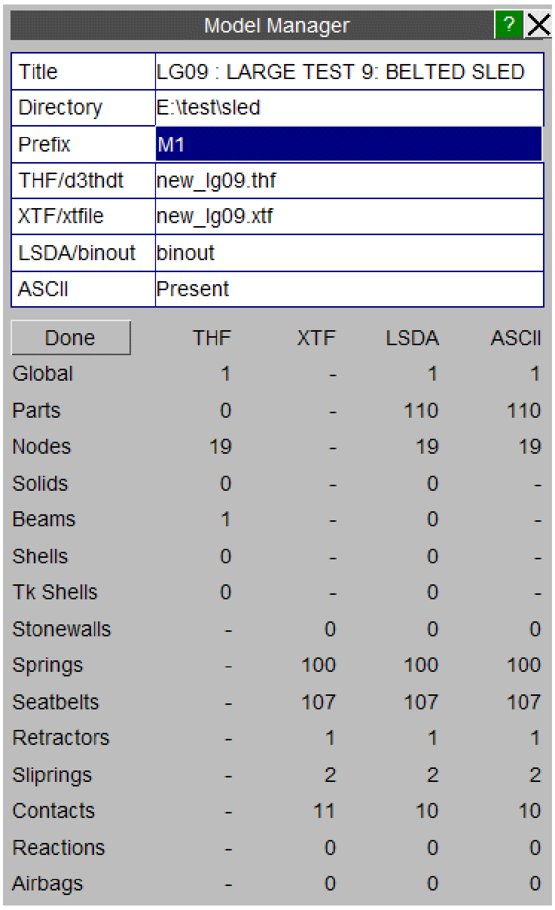
This allows the user to turn models on/off. Deselecting a model will result in removal of its entities as options when reading data. Models can be displayed according to their titles or alternatively by the directories they were read in from.



Clicking on the button displaying a model title will produce a menu similar to that illustrated below. The number of each type of item in the model and the sources T/HIS found for that item type's data will be shown. The user can select which file type is preferred for the data for each type of item (see Settings).

Prefix

This menu can also be used to define a user defined model prefix. This prefix can be added automatically to the start of curve labels to help identify which model they belong to.



Model Manager

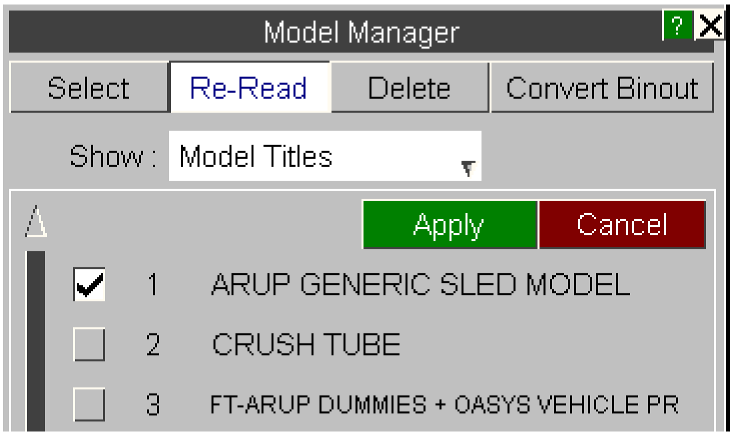
Title	LG09 : LARGE TEST 9: BELTED SLED
Directory	E:\test\sled
Prefix	M1
THF/d3thdt	new_lg09.thf
XTF/xtfile	new_lg09.xtf
LSDA/binout	binout
ASCII	Present

Done	THF	XTF	LSDA	ASCII
Global	1	-	1	1
Parts	0	-	110	110
Nodes	19	-	19	19
Solids	0	-	0	-
Beams	1	-	0	-
Shells	0	-	0	-
Tk Shells	0	-	0	-
Stonewalls	-	0	0	0
Springs	-	100	100	100
Seatbelts	-	107	107	107
Retractors	-	1	1	1
Sliprings	-	2	2	2
Contacts	-	11	10	10
Reactions	-	0	0	0
Airbags	-	0	0	0

8.5.2. Re-Read

Re-Read

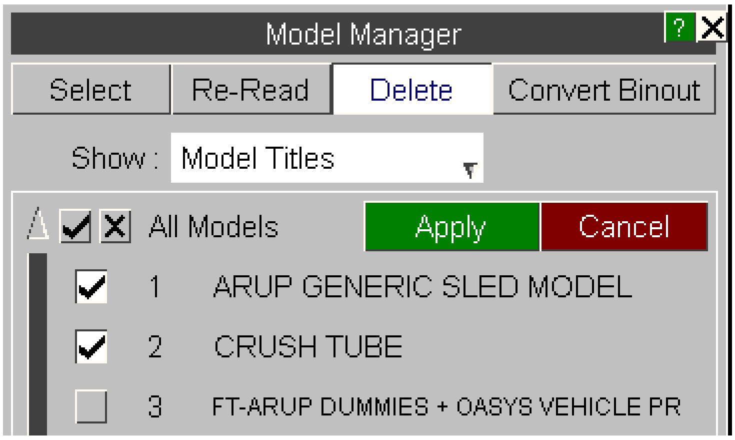
The re-read option can be used to rescan and update the model. This will find any new data written to disk since the file was last read.



8.5.3. Delete

Delete

This option allows the user to select and delete models from T/HIS. Any curves that have been read in from a model that is deleted are NOT deleted with the model. Any number of models to be deleted from T/HIS.



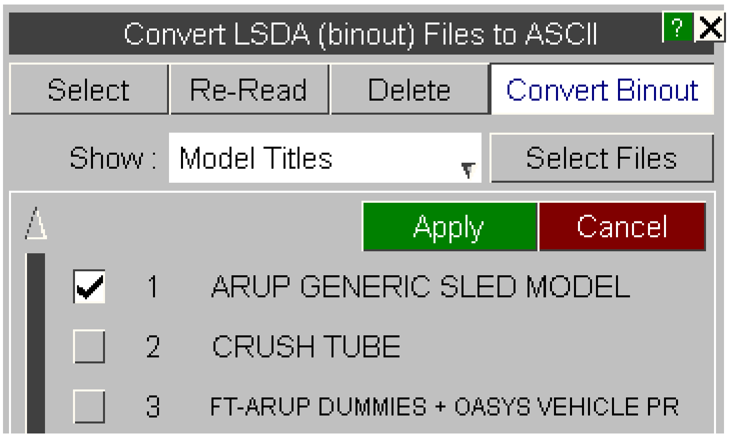
8.5.4. Convert Binout

Convert Binout

This option can be used to convert LSDA binout files into the older ASCII files. The menu allows a number of models to be selected.

The **Select Files** button allows the user to specify which ASCII files are to be created.

All of the ASCII files are written into the directory containing the LSDA file.

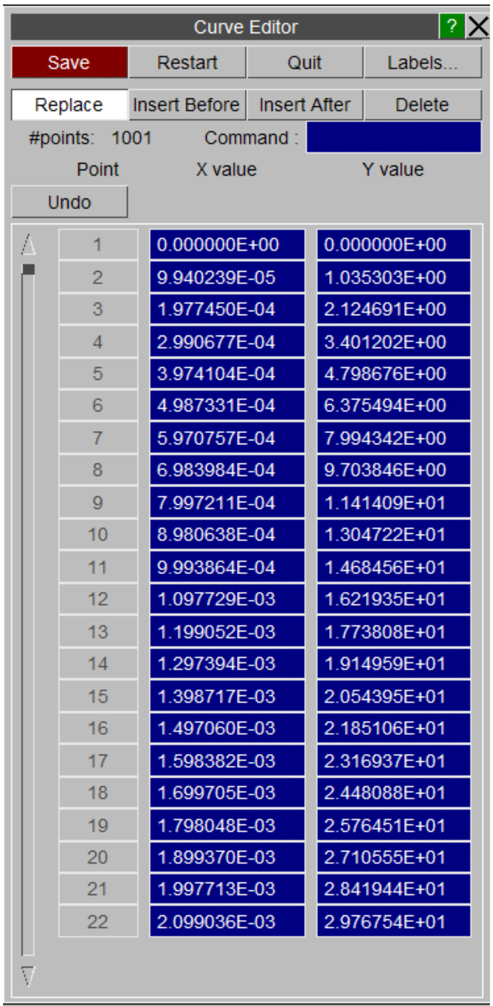


8.6. EDIT Options

EDIT Options

This menu allows you to examine and make modifications to the curve data points. You are always working on a "scratch" copy of the curve. The permanent curve is only updated when you **Save** it explicitly.

Moving around the curve data is done through the use of scroll bars on the data panel.



Save

Saves the edited curve as either a new curve or overwrites the original.

Restart

Resets the curve being edited to the values at the start of the edit session.

Quit

Quits the Curve Editor without making any changes to the curve.

Labels...

Allows the title, axis and line label to be changed (see Curve Labels for more details).

Replace

Allows curve values to be changed by overtyping the x and y values.

Insert Before

Inserts a new point in the curve before the selected point.

Insert After

Inserts a new point in the curve after the selected point.

Delete

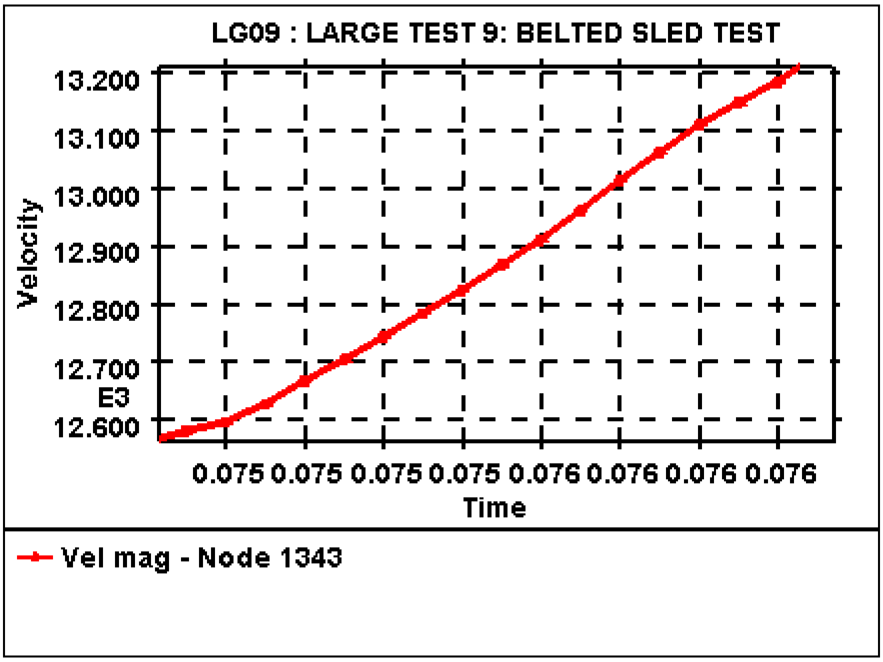
Deletes the selected point.

The **Command** text-box allows control by command line (see Command line mode for more details).

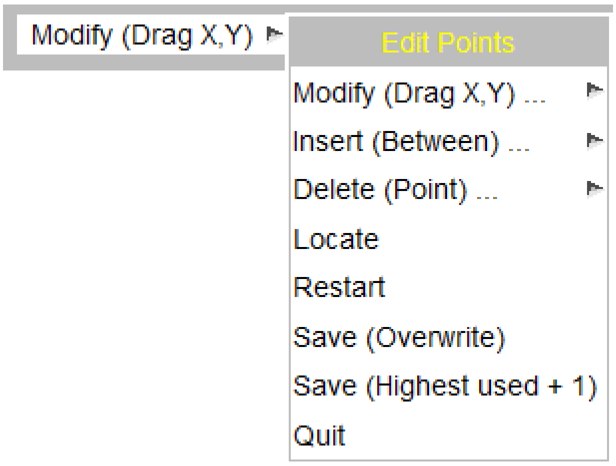
8.6.1. Interactive Curve Editing

Interactive Curve Editing

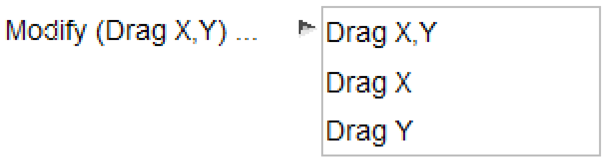
After a curve has been selected it is displayed using a thicker line to highlight it in any graphs that it is visible in.



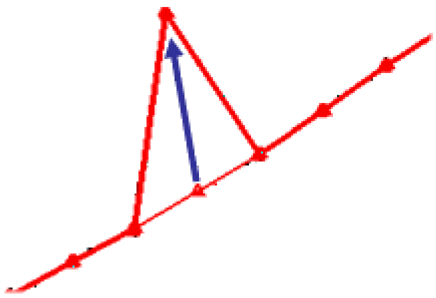
As well as being highlighted the curve points can be edited interactively and the Quick Pick menu in the main Tool Bar (see Tool Bar for more details) is replaced with the EDIT menu.



Modify



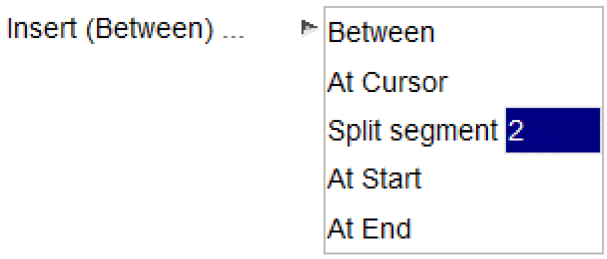
Drag X,Y Modify the point nearest to the screen pick by dragging it's position in both the X and Y axis directions.



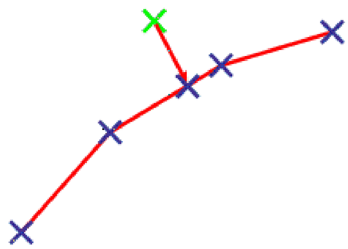
Drag X Drag a point in the X axis direction only.

Drag Y Drag a point in the Y axis direction only.

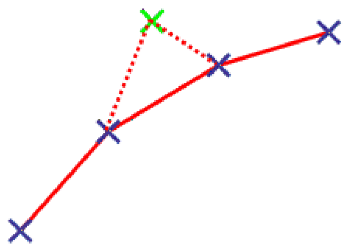
Insert



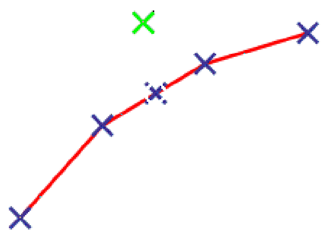
Between Finds the nearest segment to the point selected on the screen and then projects the point onto the segment.



At Cursor Finds the nearest segment to the point selected on the screen and then inserts the a point at the screen location between the 2 ends of the segment.



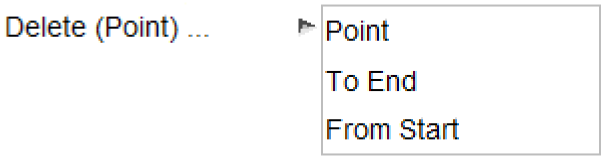
Split Segment Finds the nearest segment to the point selected on the screen and then splits the segment in 2 or more parts.



At Start Inserts a new point at the screen location before the first point in the curve.

At End Inserts a new point at the screen location after the last point in the curve.

Delete



Point	Finds the nearest point to the screen pick and deletes it.
To End	Finds the nearest point to the screen pick and deletes all points in the curve from that point onwards.
From Start	Finds the nearest point to the screen pick and deletes all points in the curve up to that point.

Locate

Finds the nearest point to the screen pick and updates the list of points in the main edit panel so that the points either side of the picked point are displayed.

Restart

Resets the curve being edited to the values at the start of the edit session.

Save (Overwrite)

Overwrite the original curve with the edited one.

Save (Highest used + 1)

Save the edited curve as a new curve without overwriting the original curve.

Quit

Quits the Curve Editor without making any changes to the curve.

8.6.2. Command Line Mode

Command line mode

In command line mode editing of curves is done in a similar fashion using the following commands.

`Moving around the curve:`	`F`	**Forward**	Move forward 16 lines
	`B`	**Back**	Move back 16 lines
	`T`	**Top**	Move to the top of the curve
	`E`	**End**	Move to the end of the curve
	`N`	**Number**	Move to given line number
`Modifying the curve:`	`Cn`	**Change**	Change line n
	`In`	**Insert**	Insert points before line n
	`An`	**Append**	Append points after line n
	`D n1 n2`	**Delete**	Delete lines n1 to n2
	`L`	**Label**	Change the line label
	`R`	**Reset**	Reset the curve back to the original curve
`Saving and Plotting the curve:`	`W`	**Write**	Write the curve
	`S`	**Save**	As write
	`PE`	**Plot Edited**	Plot the edited curve
	`PA`	**Plot All**	Plot the edited and original curve
	`PL`	**Plot**	Plot the current T/HIS curves

`Q`	**Quit**	Quit the editor

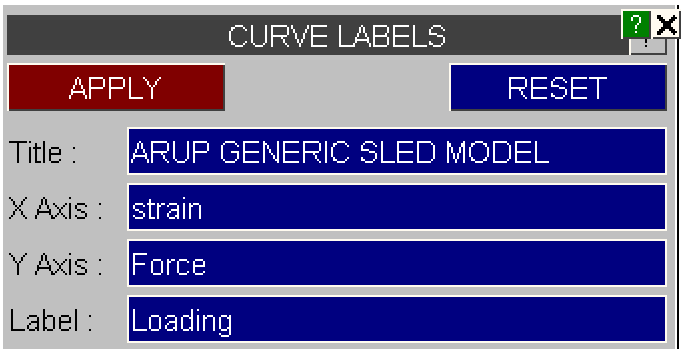
In command line mode the EDIT menu is reached by typing **`/ED`**

8.6.3. Curve Labels

Curve Labels

Each curve has four labels associated with it:

Title	The title string at the top of the plot
X label	The label for the X axis of the plot
Y label	The label for the Y axis of the plot
Label	The label applied to the line itself



The first three are only used on a plot if this curve is the first (or only) curve to be plotted, and the relevant labels are in "automatic" mode (see TITLE and AXIS).

You can change any of these by simply overtyping whatever is currently there. When you are happy with the result use the **APPLY** button to dismiss this box, saving the new values. The labels here are scratch values, current only in this editor, the permanent curve labels are only overwritten with them if you **SAVE** this edited curve.

RESET will restore the scratch labels to the original values of the permanent curve being edited.

The title, axis and line labels can also be modified using the dialogue box.

8.7. LINE STYLES

LINE STYLES

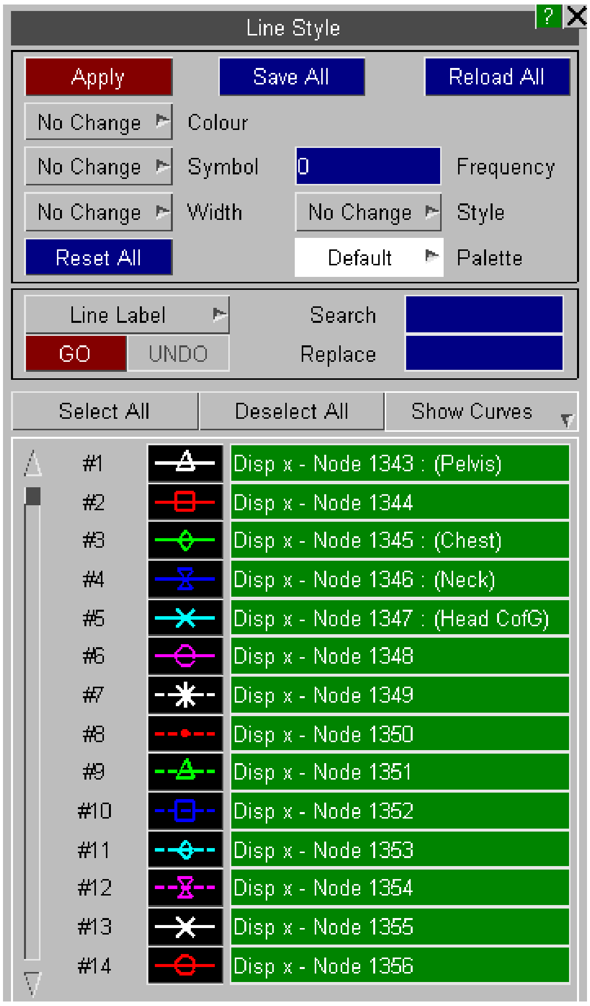
The **Line Style** menu is shown in the figure (right). This menu can be used to change the colour, width, style and symbol for any of the curves that are currently being used.

When a curve file is written, T/HIS will save the line style for each curve in the file.

The lower half of this panel contains a list of all the curves that are currently being used. By default the curve that was clicked on in the **Curve Manager** will be highlighted and the Colour and Symbol buttons in the top section of the menu will show the setting for that curve.

The **Save All** button can be used to save the current set of line styles to a file while the **Reload All** button can be used to reload a set from a previously saved file. The **Reset All.** button will reset all the curve styles to the original T/HIS settings.

If you wish to modify the colour/style of more than one curve at a time additional curves may be selected by pressing the



8.7.1. APPLY

APPLY

This button will **APPLY** the current line colour, symbol, width and style selection to all the curves that have been selected.

8.7.2. COLOUR

COLOUR

Pressing the right mouse button while over the colour button will invoke a colour popup menu.

T/HIS has a built in palette of 30 predefined colours and 6 user defined colours. Colours are defined using 6 digit Hexadecimal values using the format RRGGBB.

RR Red Component (0-255)
GG Green Component (0-255)
BB Blue Component (0-255)

Colour ID	Name	Alternative Name	Value
1	COL_1	WHITE	FFFFFF
2	COL_2	RED	FF0000
3	COL_3	GREEN	00FF00
4	COL_4	BLUE	0000FF
5	COL_5	CYAN	00FFFF
6	COL_6	MAGENTA	FF00FF
7	COL_7	YELLOW	FFFF00
8	COL_8	ORANGE	FF9C00
9	COL_9	TURQUOISE	21FF94
10	COL_10	INDIGO	7B00FF
11	COL_11	LIME	BDFF39
12	COL_12	SKY	39BDFF
13	COL_13	PINK	FF7B7B
14	COL_14	BLACK	000000
15	COL_15	PALE_YELLOW	FFFF9C
16	COL_16	GOLD	FFCE00
17	COL_17	OLIVE	7B7B00

18	COL_18	DARK_MAGENTA	9C3163
19	COL_19	MEDIUM_GREEN	9CCE00
20	COL_20	MEDIUM_BLUE	7B7BFF
21	COL_21	HOT_PINK	FF9CCE
22	COL_22	LIGHT_PINK	FFCE9C
23	COL_23	SEA_GREEN	317B63
24	COL_24	MAROON	7B0000
25	COL_25	DARK_GREEN	007B00
26	COL_26	PURPLE	7B007B
27	COL_27	NAVY	00007B
28	COL_28	DARK_GREY	393939
29	COL_29	MEDIUM_GREY	7B7B7B
30	COL_30	LIGHT_GREY	BDBDBD
31	COL_31	USER_1	-
32	COL_32	USER_2	-
33	COL_33	USER_3	-
34	COL_34	USER_4	-
35	COL_35	USER_5	-
36	COL_36	USER_6	-

As well as the 36 colour options **Foreground** and **Background** can be selected to change the colour to the **Foreground** and **Background** colours defined in the Display menu.

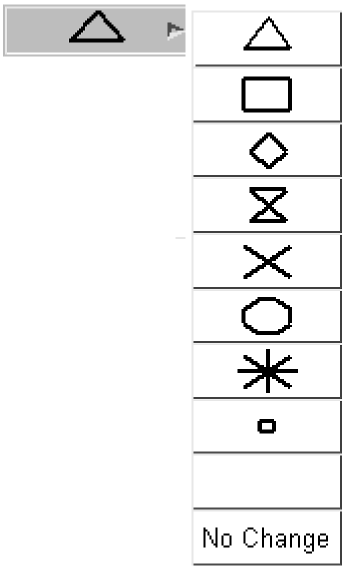
If **No Change** is selected then the **Apply** button will have no effect on the colour of the currently selected curves..

8.7.3. SYMBOL

SYMBOL

Pressing the right mouse button while over the **Symbol** button will invoke a symbol popup menu that allow any of the 9 T/HIS symbols to be selected (the 9 th is a blank symbol that can be selected so that a curve can be plotted without a symbol). As well as the 9 symbols the menu also contains a **No Change** option.

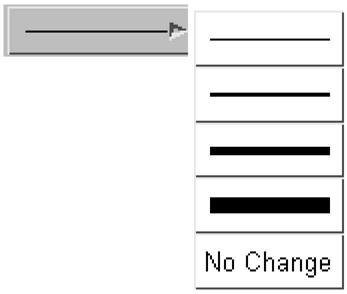
The **Symbols Frequency** controls how often a symbol is drawn on a curve. By default, symbols are not drawn; they can be switched on using the Display menu.



8.7.4. WIDTH

WIDTH

Pressing the right mouse button while over the width button will invoke a popup menu that allows 10 different line widths to be selected or **No Change**.

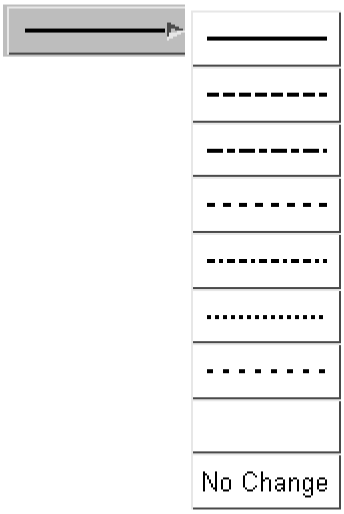


8.7.5. STYLE

STYLE

Pressing the right mouse button while over the style button will invoke a popup menu that allows 8 different line styles to be selected (the 8th is actually a blank line that can be selected so that a curve can be plotted without a line).

As well as the 8 line styles the menu also contains a **No Change** option.

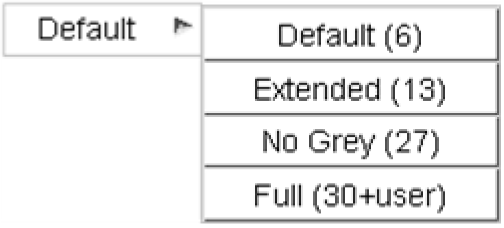


8.7.6. CURVE PALETTE

CURVE PALETTE

By default T/HIS uses 6 colours (White, Red, Green, Blue, Cyan and Magenta) for any curves that have not had a colour explicitly defined for them. Curves 1,7,13... will be White, 2,8,14... will be Red.

This option can be used to change the default number of colours T/HIS uses.



Default	Use the default 6 colours
Extended	Use the first 13 colours
No Grey	Use all 30 predefined colours except the 3 grey ones
Full	Use all 30 predefined colours plus any user defined ones.

The default value for the curve palette can also be specified in the "preferences" file (see Appendix H for more details).

8.7.7. MODIFYING LABELS

MODIFYING LABELS

Multiple curve labels may be edited using the Search and Replace option to enter the string to search for and the string to replace it with **^** can be used to insert text at the beginning of a label while **$** can be used to append to the end of a label. The table below shows the effect of 4 search and replace examples.

	Example 1	Example 2	Example 3	
Original Label	Displacement N1034	Time	iso18571-a1.1-cae1	iso18571-a1.1-cae1
Search String	N1	$	*	*
Replace String	Node 1	(s)	ISO_A1.1	
Modified Label	Displacement Node 1034	Time(s)	ISO_A1.1	

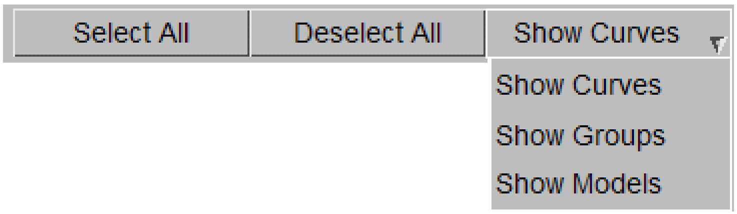
The **GO** button will initiate the search and replace on all the curves that are currently selected (highlighted in the bottom half of the menu), while the **UNDO** button can be used to reset the labels to what they were before the search and replace.

Pressing the right mouse button while over the **Line Label** button will invoke a popup menu that allows the label that is being modified to be swapped between the **Line Label** , **X-Axis Label** and the **Y-Axis Label** .

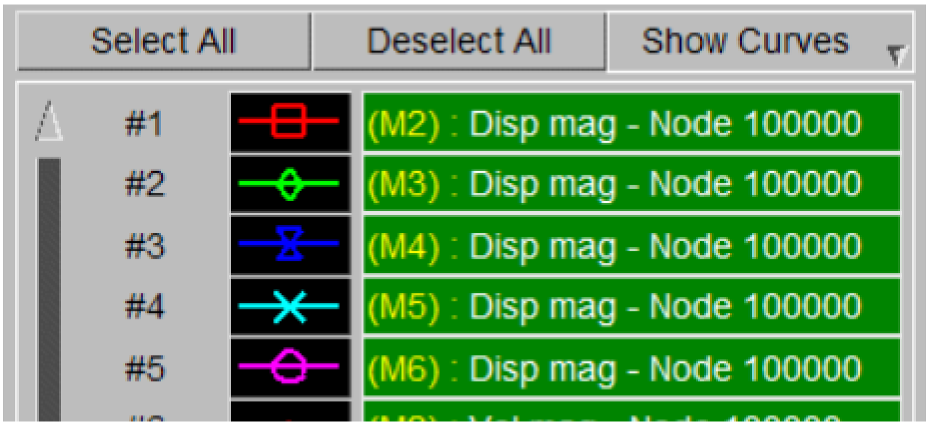
Line labels can also be modified by using the dialogue box.

8.7.8. SELECTING CURVES

SELECTING CURVES

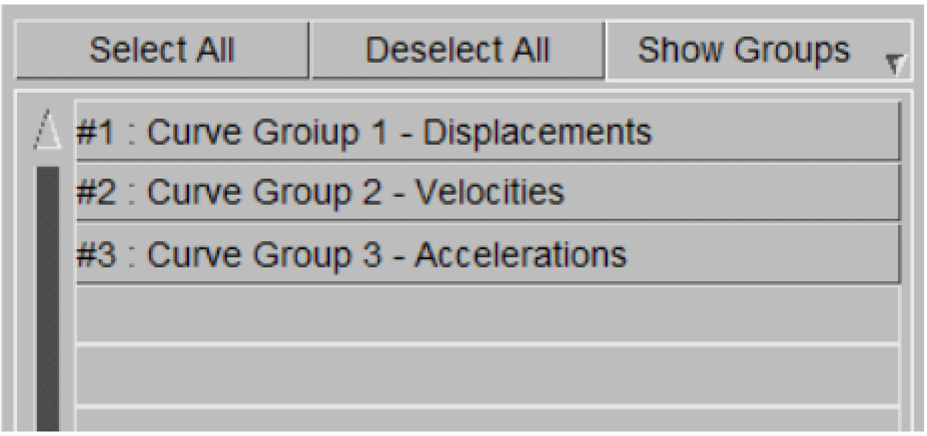


By default the Style menu will display a list of all the current defined curves so that the style for individual curves can be modified



Instead of displaying individual curves the style menu can be changed to display a list of any currently defined curve groups.

If curve groups are selected then the style will be applied to all of the curves in the curve group.



The style menu can also display a list of all the models currently loaded in T/HIS.

If models are selected then the style will be applied to any curve that was created using data from the model.

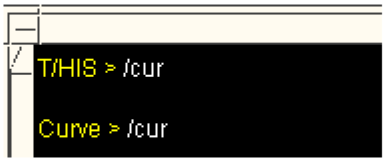

Select All
Deselect All
Show Models
#2 : BASE T = 1.50
#3 : RUN 1 T = 1.30
#4 : RUN 2 T = 1.40
#5 : RUN 3 T = 1.60
#6 : RUN 4 T = 1.70

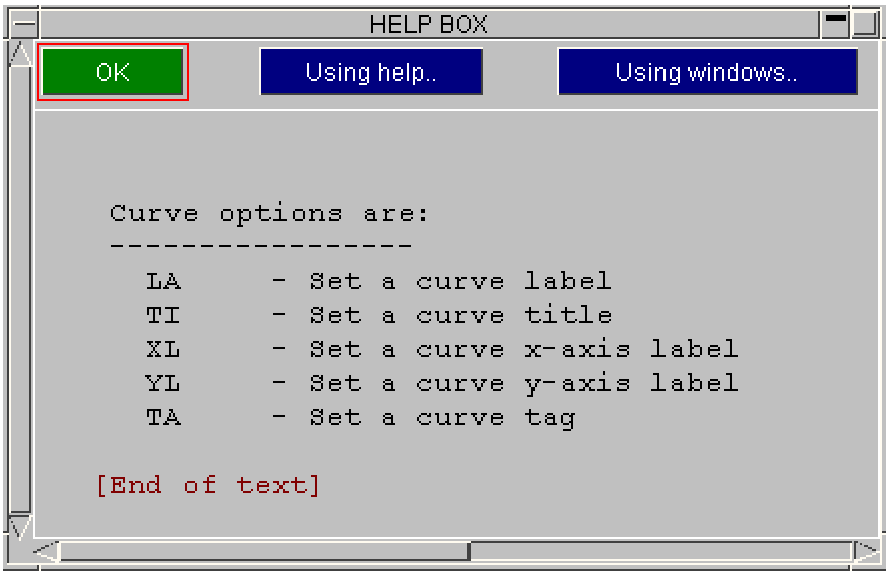
8.7.9. LINE STYLE EDITING IN THE DIALOGUE BOX

LINE STYLE EDITING IN THE DIALOGUE BOX

The dialogue box can be used to edit curve styles.

To access this feature, enter the command `/style` at the Command Prompt.



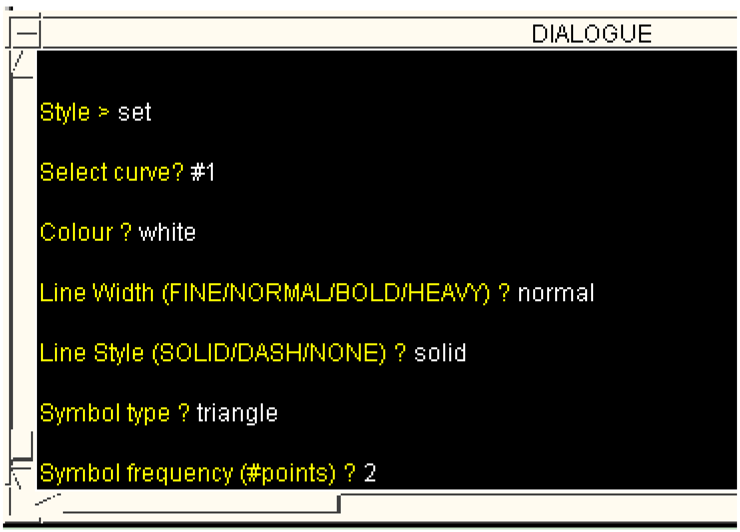


Enter `M` at the **STYLE >** command prompt for a list of all available dialogue box commands.

The following commands are available:

SET
READ
WRITE
DEFAULT
FIX
GM

SET



This option allows the user to set the style properties for individual curves.

Enter the curve number (e.g `#1` for curve 1) at the **`Select Curve?`** command prompt.

T/HIS will prompt the user to input the desired style properties in the order:

`Colour`; Enter the colour for the line
`Line Width`; Enter the desired line width for the line
`Line Style`; Enter the desired line style (e.g. dashed) for the line
`Symbol Type`; Enter the desired Symbol Type
`Symbol Frequency`; Enter the desired frequency of the symbols in the format

READ

This option allows the user to read a style file containing style information and apply that style to a particular curve.

Enter the name of the style file at the **`Style File?`** command prompt.

WRITE

This option allows the user to write a style file containing style information.

DEFAULT

This option allows the user to reset all the curve styles to the default settings.

FIX

This is an **ON** / **OFF** switch which resets the curve styles when they are plotted on the screen so that the curves cycle through the default T/HIS colours and styles as they are plotted. This will result in the first curve being plotted always being white, the second red, the third green, etc regardless of their curve numbers. The default is **OFF** .

GM

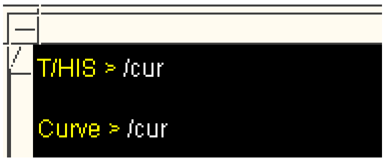
This option will display the Global Menu in a separate window.

8.7.10. LABEL AND TITLE EDITING IN THE DIALOGUE BOX

LABEL AND TITLE EDITING IN THE DIALOGUE BOX

The dialogue box can be used to edit curve labels, x-axis and y-axis labels and curve titles.

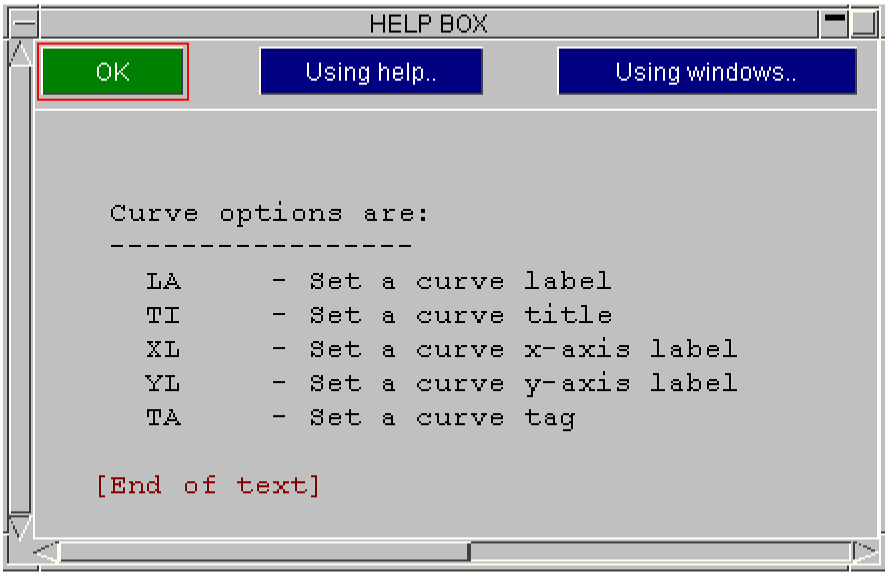
To access this feature, enter the command `/cur` at the Command Prompt.



Enter `M` at the **CURVE >** command prompt for a list of all available dialogue box commands.

The following commands are available:

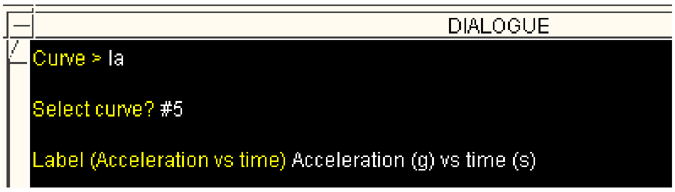
LA
TI
XL
YL
TA



LA

This option allows the user to edit the label for individual curves.

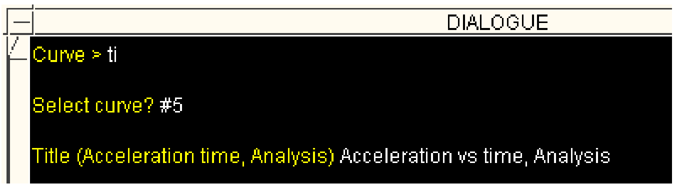
Enter the curve number at the **Select curve?** prompt.



Enter the desired new label at the Label prompt, the current Label will be displayed in brackets.

TI

This option allows the user to edit the title for individual curves.



Enter the curve number at the **`Select curve?`** prompt.

Enter the desired new title at the Title prompt, the current title will be displayed in brackets.

XL

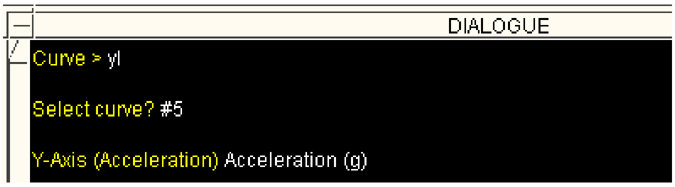
This option allows the user to edit the x-axis label for individual curves.

Enter the curve number at the **`Select curve?`** prompt.

Enter the desired new title at the X- Axis prompt, the current x-axis label will be displayed in brackets.

YL

This option allows the user to edit the y-axis label for individual curves.



Enter the curve number at the **`Select curve?`** prompt.

Enter the desired new title at the Y- Axis prompt, the current y-axis label will be displayed in brackets.

TA

This option allows the user to edit the tag for individual curves.

Enter the curve number at the **`Select curve?`** prompt.

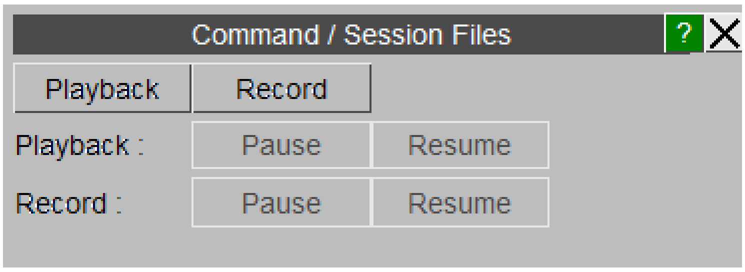
Enter the desired new Tag at the Tag prompt, the current tag will be displayed in brackets.

8.8. Command/Session Files

Command / Session Files

Command and session files are used to drive or record a T/HIS session. Both session (save) and command (playback) files have been set up to act like tape recorders; and the concept of "recording" and "playing back" files will be used below.

These files ("button click" command files) are not easy to edit by hand and they are not always backwards compatible between versions of T/HIS. For these reasons most users prefer the newer FAST-TCF format, which can also be recorded and played back from within T/HIS.



In screen menu mode a command has a meaning beyond the simple command word. For example, **HELP** appears in many different places, with a distinct meaning (or relevance) in each place. Therefore, context information is stored when saving screen menu session files.

In practice the following information is saved:

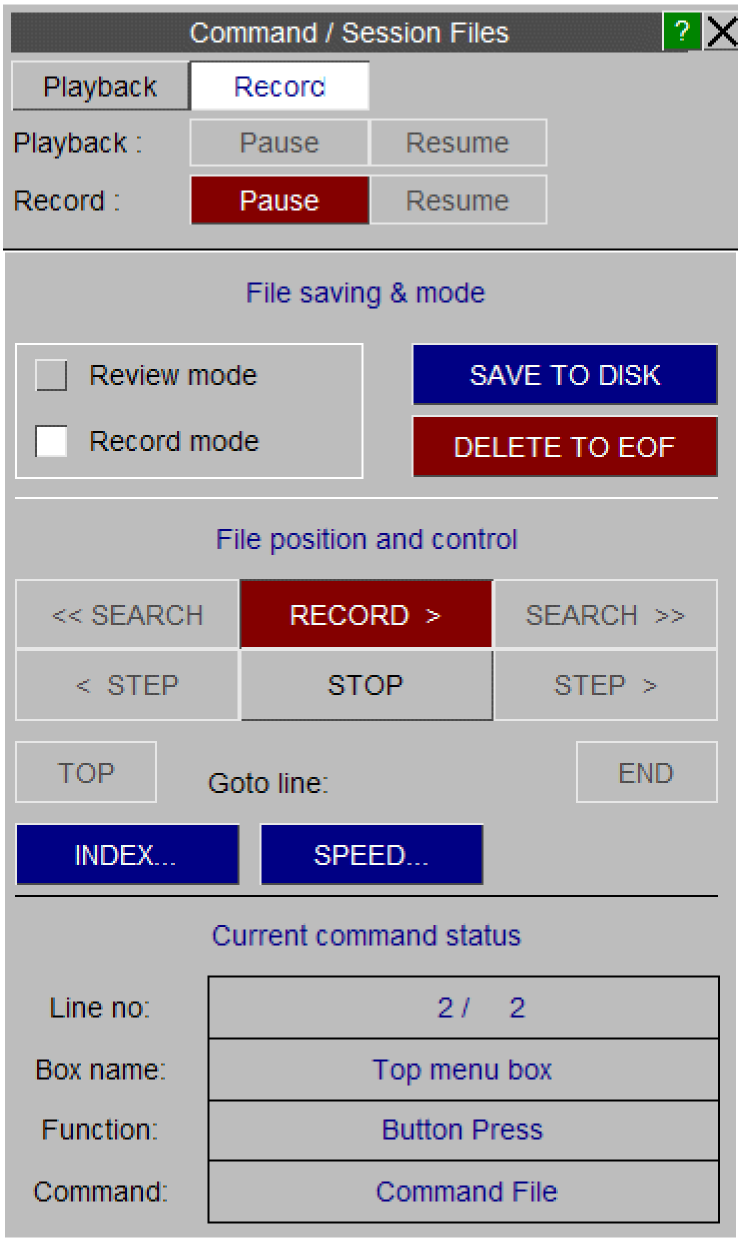
- the command itself - whether typed of inferred from a button
- the button identification (if any)
- the parent window identification
- the menu item (if relevant)
- the action type (screen pick, button press, etc)
- any x/y coordinates that may be relevant.

A choice of either writing ("recording") session files or executing ("playing back") command files is given. By default commands are not saved. If they are to be saved the session file record switch must be turned on.

8.8.1. Writing ("Recording") Session Files

Writing ("Recording") Session Files

To write a session file the **Record** tab must be pressed, displaying the **RECORD COMMAND FILES** menu shown below.



Pressing the **RECORD >** button will start the session file. Thereafter, all commands (except those in the session/playback windows) are saved in an internal scratch file. In order to save these commands to disk they must be written explicitly using the **SAVE TO DISK** button. They can then be read back in and replayed

A variety of features are available to help move around the file. These are shown in the **FILE POSITION AND CONTROL** area of the panel. The file can be indexed at particular user defined points using the **INDEX MARKS** menu is accessed by pressing the **INDEX...** button. These may be used as targets of a search and also to control recording.

The scratch file is random access, and can be moved back and forth and reviewed at will. To help with this it is possible to switch between **RECORD** and **REVIEW** modes in the session file control box:

RECORD	records all your commands when running
REVIEW	plays back your recorded commands

A command file can be stepped through or run backwards or forwards. It may also be searched for a particular command. As with a real tape recorder, if the pointer is moved backwards and recording continued the commands that were previously stored will be overwritten from that point.

The session file recording and command file playback operations are totally separate: they can be thought of as two separate tape recorders. As a consequence it is possible to record commands that are being played back: in effect it is possible to edit and combine files.

8.8.2. Executing ("Playing Back") Command Files

Executing ("Playing Back") Command Files

Similar to recording Session Files, the **PLAYBACK COMMAND FILES** menu, shown below, must be invoked from the **COMMAND/SESSION FILES** window.

This is done by pressing the **Playback** tab. An existing file must then be read. This is analogous to loading a tape into the tape recorder: it is then converted into an internal scratch format (random access, as above) and can be played back or previewed at will.

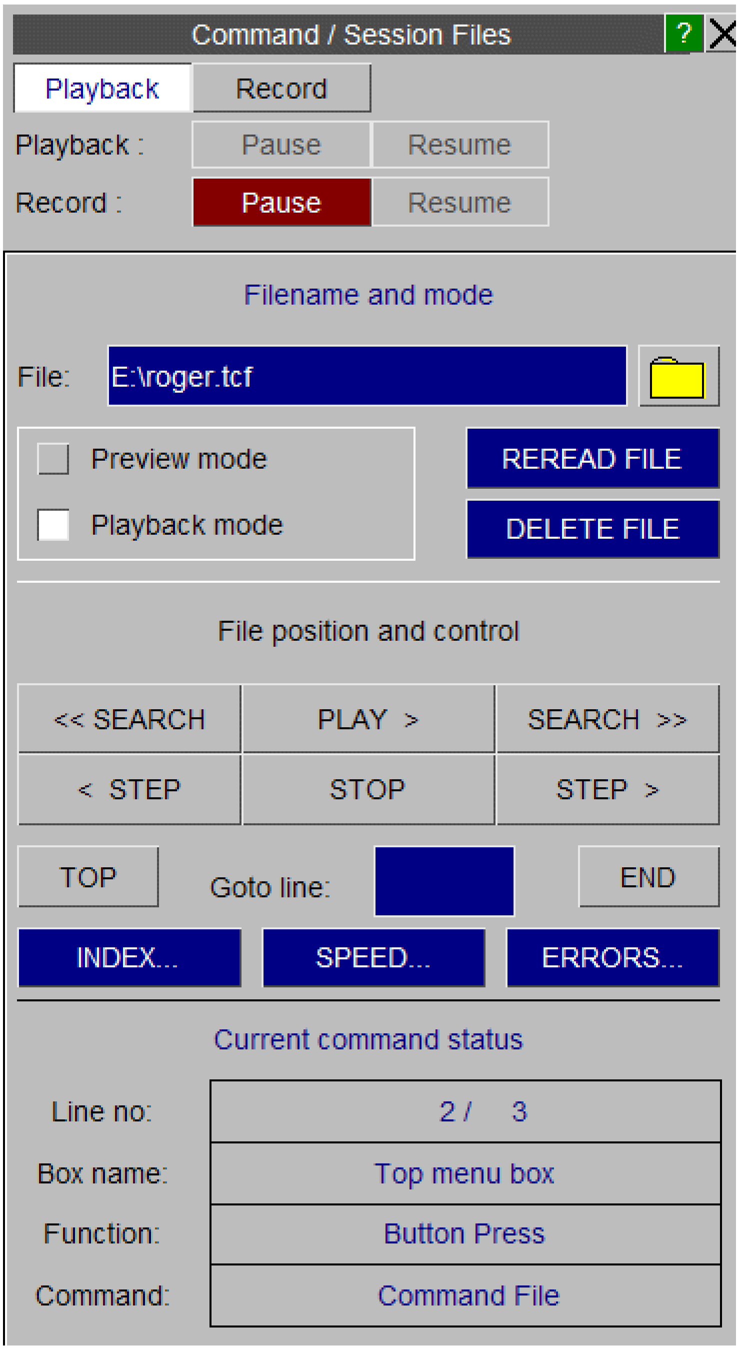
Once a file is read in either **PLAYBACK** or **PREVIEW** mode may be selected:

PLAYBACK actually executes the commands,

PREVIEW simply lists them without executing them.

The file may be stepped through backwards or forwards at will, and searches made for commands. Playback commences at the current line when **PLAY** is pressed, so it is possible to skip unwanted commands or repeat a sequence.

As with **RECORD** above, index marks can be inserted, which may be used as targets of a search and also to stop playback.

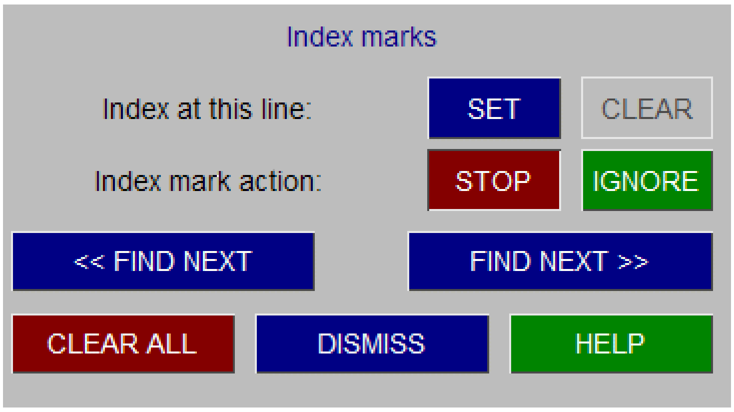

Command / Session Files
?
Playback
Record
Playback :
Pause
Resume
Record :
Pause
Resume
Filename and mode
File:
E:\roger.tcf
Preview mode
Playback mode
REREAD FILE
DELETE FILE
File position and control
<< SEARCH
PLAY >
SEARCH >>
< STEP
STOP
STEP >
TOP
Goto line:
END
INDEX...
SPEED...
ERRORS...
Current command status
Line no:
2 / 3
Box name:
Top menu box
Function:
Button Press
Command:
Command File

8.8.3. INDEX MARKS

INDEX MARKS

"Index marks" are optional flags that you can set at any line in a file. They are not interpreted as commands but rather treated as markers which are used as targets of SEARCH operation. Index mark functions are:

SET	Set an index mark on this line;
CLEAR	Clear an index mark set on this line
STOP	Stop in PLAY/REVIEW mode when index found
IGNORE	Ignore index marks during PLAY/REVIEW
FIND INDEX	Finds the next index mark: "<<" searching backwards, ">>" searching forwards
CLEAR ALL	Clear all index marks in the file

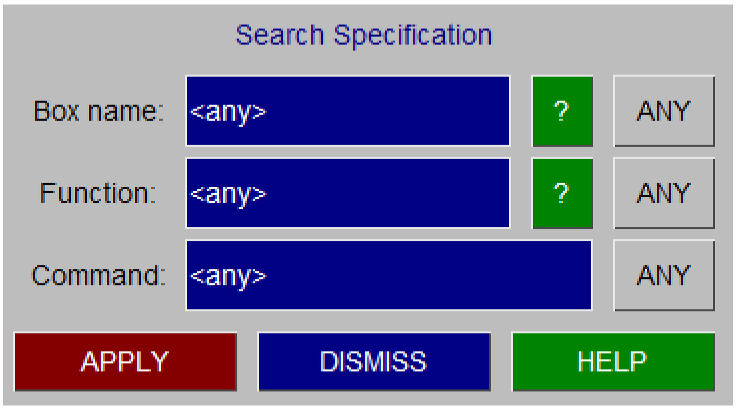


8.8.4. SEARCH

SEARCH

The **SEARCH** buttons can be used to find a specific command when in **REVIEW** mode. You can search through the command file for a match to any permutation of the following:

Box name	The name of a screen menu box inside which an event occurred
Function	The screen menu function type. This is "button press", "dialogue", etc;
Command	The command word(s) to look for.



The default for all of these is "<any>", i.e. a wildcard search, but you can specify a value by typing into the appropriate text box. When you have filled in all the fields you need, press **APPLY** to start the search. **Box name** and **Function** fields are unlikely to be of use to most users, you can list all valid events using the **?** button to provide a menu to pick from. The **ANY** button may be used for any field to restore it to its default (wildcard) status.

8.8.5. Command Line Mode Session/Command File Control

Command Line Mode Session / Command File Control

The available features in command line mode for command and session file control are very basic. A session file can recorded at any point by typing `SF` (in the **GLOBAL MENU**) followed by the desired filename. This is equivalent to the **RECORD** button in screen menu mode. The session file can be closed by typing `CS` and is automatically written to disk. This is equivalent to pressing the **STOP** and **SAVE TO DISK** buttons in screen menu mode.

To execute an existing command file in T/HIS simply type `CF` , followed by the filename.

No previewing/reviewing or editing of command/session files is possible in command line mode.

8.8.6. Command Files From Earlier Versions of T/HIS

Command Files From Earlier Versions Of T/HIS

Command files recorded in Version 9.0 or earlier will not work in T/HIS 21.0 .

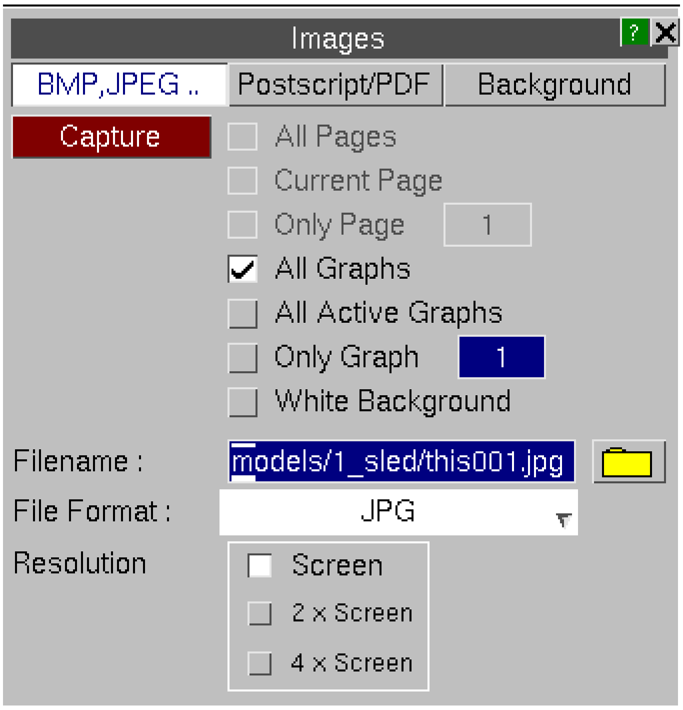
8.9. IMAGE Options

IMAGE Options

8.9.1. BMP, JPEG...

BMP,JPEG ...

This menu can be used to save an image containing one or more graphs in a number of different formats.

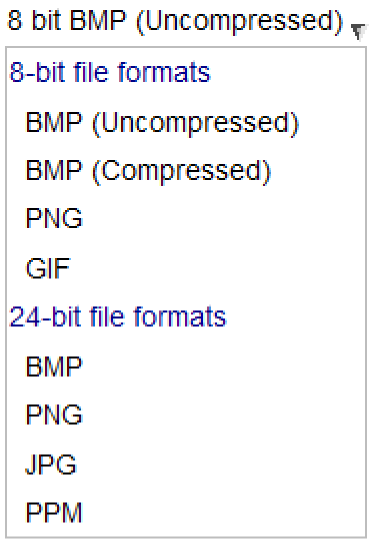


All Pages	Each page will be saved as a single image to multiple files. The filenames used will be based on the filename selected by the user. *This option will only be available if T/HIS contains multiple graphs on more than one page (see Page Layouts).*
Current Page	A single image containing currently displayed page will be generated.
Only Page (n)	A single image containing the selected page will be generated. *This option will only be available if T/HIS contains multiple graphs on more than one page (see Page Layouts).*
All Graphs	A single image will be generated containing all of the graphs. *This option will only be available if T/HIS only contains a single page (see Page Layouts).*
All Active Graphs	A single image will be generated containing all of the currently active graphs. *This option will only be available if T/HIS only contains a single page (see Pae Layouts).*
Only Graph (n)	A single image containing the selected graph will be generated.

White Background	Captures the image with a white background and black foreground. Once the image is captured the colours are reset to their original values.

File Format

8-bit file formats

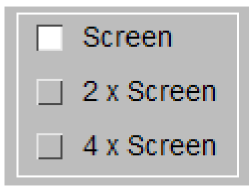


BMP Uncompressed	Uncompressed 8 bit Microsoft Windows bitmap. The approximate size of the file (in bytes) is file size= image width * image height
BMP Compressed	8 bit RLE Microsoft Windows bitmap.
PNG	8 bit Portable Network Graphics
GIF	Graphics Interchange Format

24-bit file formats

BMP	Uncompressed 24 bit Microsoft Windows bitmap. The approximate size of the file (in bytes) is file size = 3 * image width *image height
PNG	24 bit Portable Network Graphics
JPG	JPEG (Joint Photographic Experts Group) file
PPM	Uncompressed **P** ortable **P** ix **M** ap. The approximate size of the file (in bytes) is file size = 3 * image width *image height

Resolution



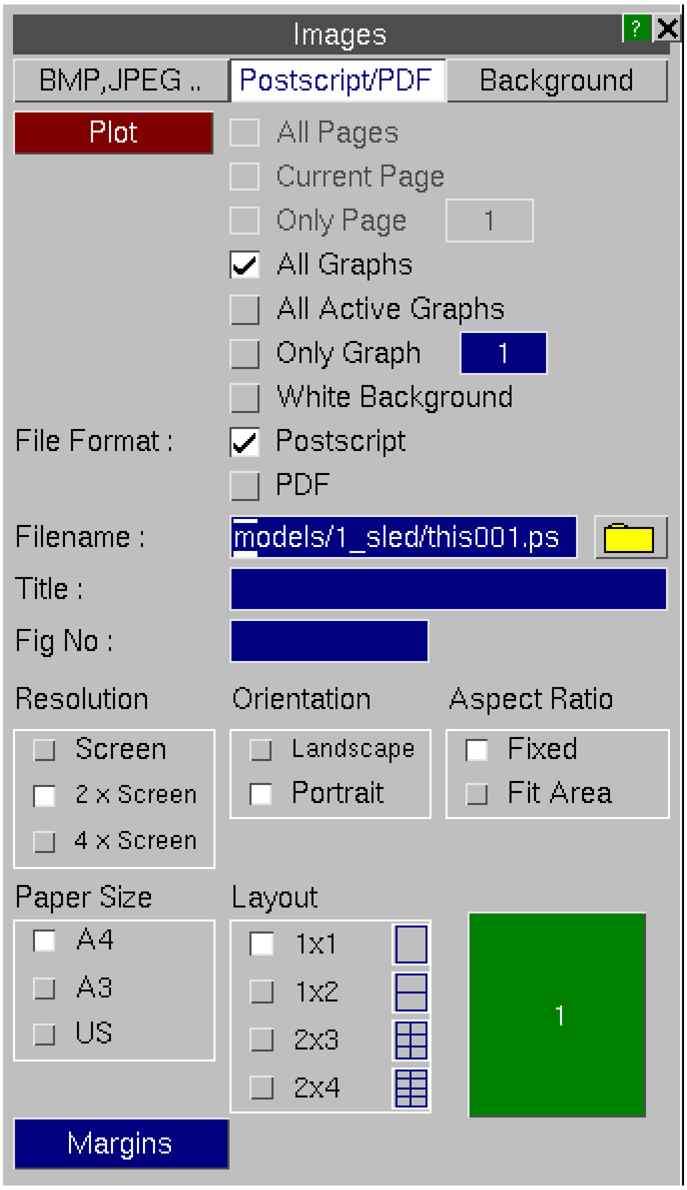
All images can be output at either the screen resolution or at a resolution of either 2 or 4 times the screen resolution.

8.9.2. Postscript

Postscript

This menu can be used to save an image containing one or more graphs to either a PDF or Postscript file.

All PDF and Postscript files are generated using raster images so that the contents of the screen is exactly reproduced.



All Pages All T/HIS pages containing 1 or more graphs will be saved to a single file. *This option will only be available if T/HIS contains multiple graphs on more than one page (see Page Layouts).*

Current Page The current T/HIS page will be saved. *This option will only be available if T/HIS contains multiple graphs on more than one page (see Page Layouts).*

Only Page (n) A single image containing the selected page will be generated. *This option will only be available if T/HIS contains multiple graphs on more than one page (see Page Layouts).*

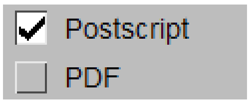
All Graphs A single image will be generated containing all of the graphs. *This option will only be available if T/HIS only contains a single page (see Page Layouts).*

All Active Graphs A single image will be generated containing all of the currently active graphs. *This option will only be available if T/HIS only contains a single page (see Page Layouts).*

Only Graph (n) A single image containing the selected graph will be generated.

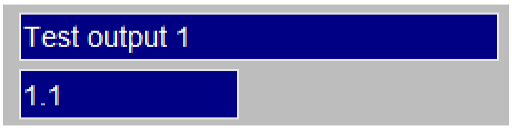
White Background Captures the image with a white background and black foreground. Once the image is captured the colours are reset to their original values.

File Format



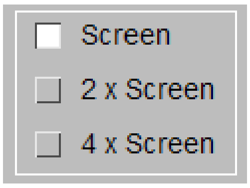
All images can be output in either Postscript or PDF file format.

Title



By default PDF and Postscript files are not labeled and have no figure number, but you may add either or both of these. They are always put at the bottom of each page, along the short edge, regardless of the orientation used for plots.

Resolution

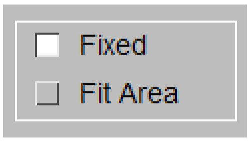


All images can be output at either the screen resolution or at a resolution of either 2 or 4 times the screen resolution.

Orientation

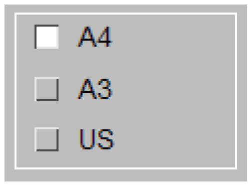
All images can be output in either landscape or portrait format.

Aspect Ratio



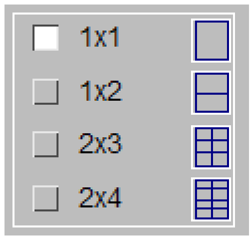
By default all images are output using a fixed aspect ratio. This option can be used to stretch each image to fit the available space on the page. Different scaling factors will be applied to the horizontal and vertical directions and the image will be distorted.

Paper Size



The paper size can be set to be either A4 (210 x 296mm), A3 (296 x 420mm) or US (letter - 216 x 279mm). The default size is A4.

Layout

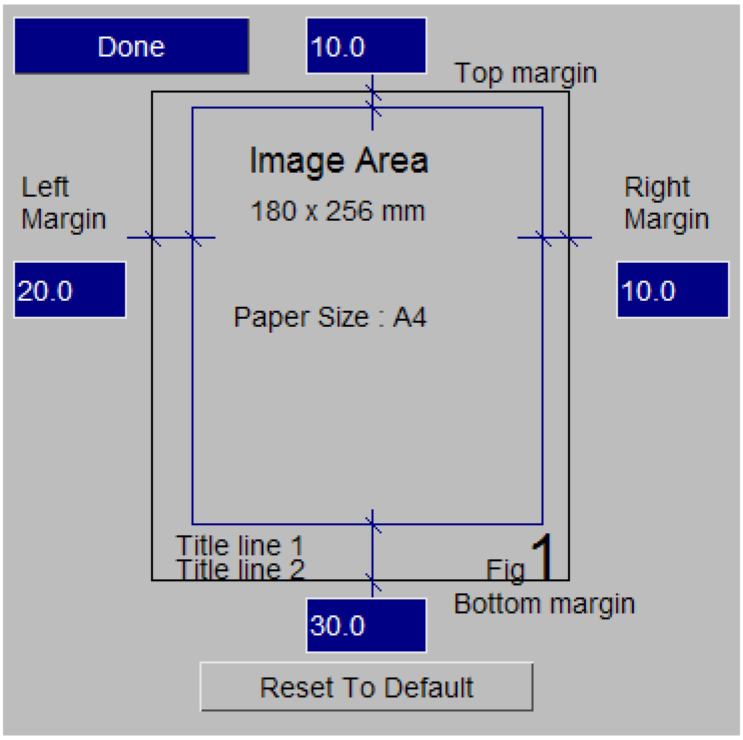


Multiple plots on a page are also available. In landscape format there is a choice of 1, 2x2, 3x3 and 4x4 plots to a page. In portrait format there is a choice of 1, 1x2, 2x3 and 2x4 plots on a page. By default there is a single plot on a page.

When multiple plots are requested the order in which they are performed can be defined.

Margins

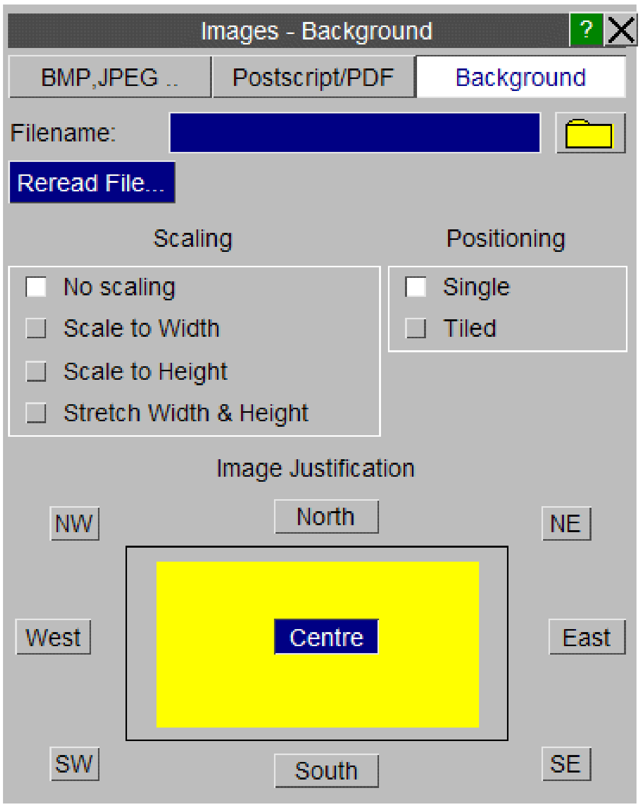
The Margins can be used to change the top, bottom, left and right margins for each page.



8.9.3. Background

Background

This option can be used to add a background image to each graph (see section Background for more details).

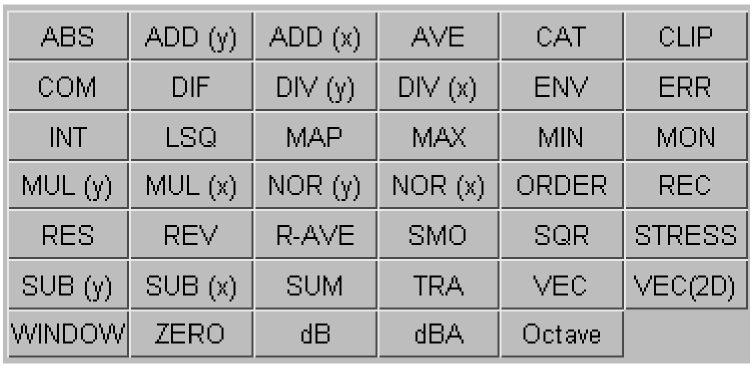


8.10. OPERATE Options

OPERATE

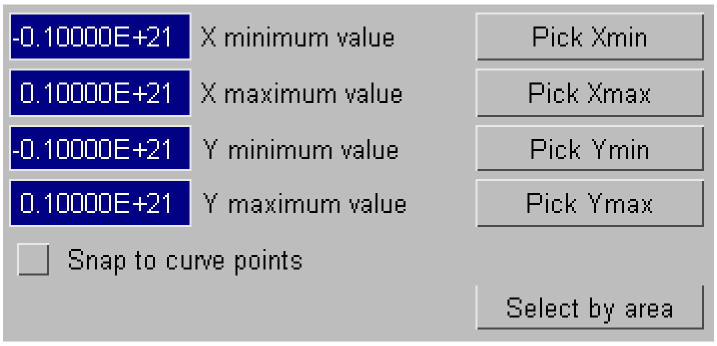
The **OPERATE** menu shown in the figure (below). If the mouse is left hovering over an option a short description of the function will appear.

For these functions, the user selects a range of curves to be operated on. A range may be one or more curves, making it possible to operate on multiple curves, for example add 20 curves to 20 curves.

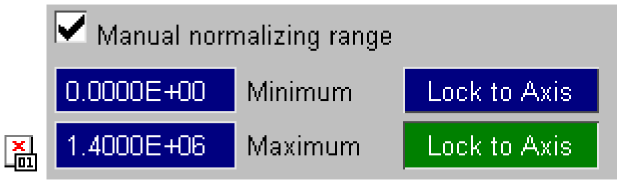


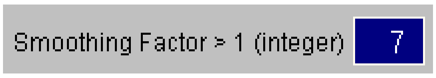
The options with the **OPERATE** menu are split into 3 groups. The first group require 2 sets of curves as input. The second group require a single set of curves as input. The third group also require a single set of curves as input but the output from these functions is a single curve. (See Curve Operations for more information on curve groups).

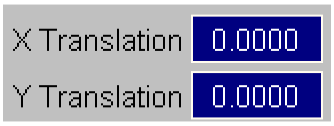
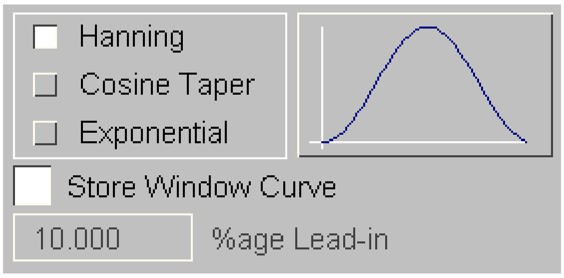
ABS	Produces the absolute y-values of a curve.
ADD (y)	Add the y axis values together for two curves or add a constant value to all the y-values. If two curves are being added together they must have identical x-axis values. If not, the resultant curve is generated by considering every x-coordinate on both curves and by interpolating the other curve as needed. Any duplicate points as well as points outside the range where the input curves overlap are culled as needed.
ADD (x)	Add the x axis values together for two curves or add a constant value to all the x-values. If two curves are being added together they must have identical y-axis values. If not, the nth x-value from the second curve is simply added to the corresponding x-value on the first curve and the y-value from the first curve gets used by the resultant as is.
AVE	Produces a single curve that is the average of the input curves.

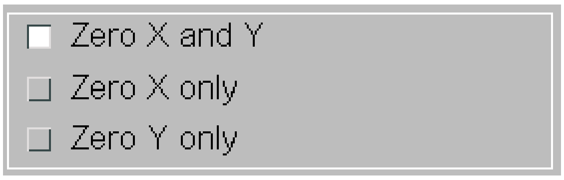
CAT	Concatenate the second curve to the end of the first.
CLIP	Clip a curve to remove any points that exceed a set of specified minimum and maximum x & y axis value. The user is prompted for minimum and maximum values after the curves have been selected..  Instead of typing in values for the limits individual x and y axis minimum and maximum values can be selected by picking screen points. In addition to picking individual points an area can be dragged out interactively to set all 4 limits. When picking screen points the default is to allow any point to be selected. Snap to curve points can be used to select the point on the nearest curve instead of the screen coordinates.
COM	Two curves are combined to give a new curve. For example if a displacement/time curve is combined with a velocity/time curve a velocity/displacement curve will result. If the 2 curves do not contain points at the same x values then the curve with the larger x-axis intervals is automatically mapped on to the x-axis values of the other curve. If the curves do not start and finish at the same x-axis values then only the points for which the two curve x-axes overlap are mapped onto each other.
DIF	A curve is differentiated with respect to the x-axis variable.
DIV (y)	Divide the y axis values of the first curve by the y axis values of the second curve (or a constant). If two curves are being used they must have identical x-axis values. If not, the resultant curve is generated by considering every x-coordinate on both curves and by interpolating the other curve as needed. Any duplicate points as well as points

	outside the range where the input curves overlap are culled as needed.
DIV (x)	Divide the x axis values of the first curve by the x axis values of the second curve (or a constant). If two curves are being used they must have identical y-axis values. If not, the nth x-value on the first curve is simply divided by the corresponding x-value on the second curve and the y-value from the first curve gets used by the resultant as is.
ENV	Produces a single curve that bounds the maximum and minimum values of the group of input curves.
ERR	This option reports the degree of correlation between 2 input curves. The first curve selected is used as a reference curve and the following parameters are then reported : Maximum difference : Value & Time Value as a %age of reference curve Value as a %age of reference curve peak value. Average difference - Value %age of reference curve peak value Area Weighted Difference Correlation Parameter - 0 to 1 For more details on this function please see Appendix G.
INT	A curve is numerically integrated with respect to the x-axis variable using Simpson's rule.
LSQ	Fits a straight line through the points using the least squares method.
MAP	The second curve is mapped onto the first curve, the resulting curve has identical x-axis values to the reference (first) curve with y-axis values obtained from the mapped (second) curve.
MAX	Produces a single curve that bounds the maximum values of the group of input curves.
MIN	Produces a single curve that bounds the minimum values of the group of input curves.
MON	Sorts a curve into monotonically increasing x-axis values.
MUL (y)	Multiply the y axis values together for two curves or multiply all the y-values by a constant. If two curves are being multiplied together they must have identical x-axis values. If not, the resultant curve is generated by considering every x-coordinate on both curves and by interpolating the other curve as needed. Any duplicate points as well

	as points outside the range where the input curves overlap are culled as needed.
MUL (x)	Multiply the x axis values together for two curves or multiply all the x-values by a constant. If two curves are being multiplied together they must have identical y-axis values. If not, the nth x-values on the two curves are simply multiplied together and the y-value from the first curve gets used by the resultant as is.
NOR (y)	Normalise a curve so that the y axis values lie in the range [-1, +1].  If the manual normalising range is checked then a custom value can be chosen to normalise with. The custom value chosen for normalising is calculated by taking the absolute maximum of the user-defined textbox values. In addition to this is the option to Lock to the specified Min and Max axis values which can be used for normalising the curve to the axis value. $y_{val} = \dfrac{y_{val}}{\max(\lvert\mathrm{MinV}\rvert, \lvert\mathrm{MaxV}\rvert)}$ Where, $\mathrm{MinV} = \mathrm{Minimum\ Value}$ $\mathrm{MaxV} = \mathrm{Maximum\ Value}$
NOR (x)	Normalise a curve so that the x axis values lie in the range [-1, +1].  If the manual normalising range is checked then a custom value can be chosen to normalise with. The custom value chosen for normalising is calculated by taking the absolute maximum of the user-defined textbox values.

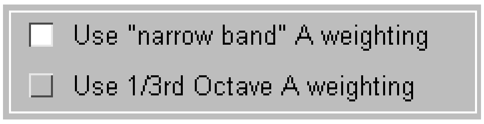
	In addition to this is the option to Lock to the specified Min and Max axis values which can be used for normalising the curve to the axis value. $x_{val} = \dfrac{x_{val}}{\max(\lvert\mathrm{MinV}\rvert,\lvert\mathrm{MaxV}\rvert)}$ Where, $\mathrm{MinV} = \mathrm{Minimum\ Value}$ $\mathrm{MaxV} = \mathrm{Maximum\ Value}$
ORDER	Reverse the order of all the points in the curve.
REC	Produces the reciprocal of the y-values of a curve.
RES	Calculate the vector magnitude from a group of input curves.
REV	Reverses the x and y axes of a curve. For example if you start with a curve with displacement (y axis) against time (x axis) you end up with a curve of time (y axis) against displacement (x axis).
R-AVE	Produces a single curve of the running average on the input curve.  If the time window is set to 0 then the y values for the output curve are the average value of all the point up to that point. If the time window is non-zero (T) then the y values at each point are calculated by averaging the values between -T/2 and +T/2.
SMO	A moving average technique is used to smooth (filter) a curve. The user will be prompted for a smoothing factor.  The integer refers to the number of points included in the averaging of each point. The value you want will depend on the number of points in the curve and the amount of smoothing required. A certain amount of trial and error is necessary to get the required result.
SQR	Take the square root of the y-values of a curve.

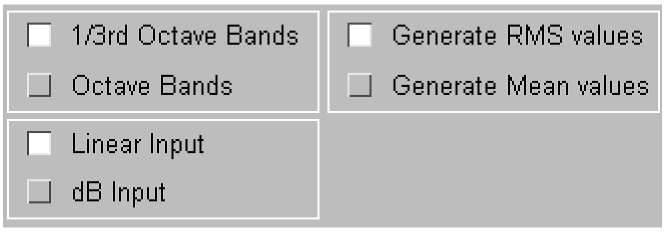
STRESS	Converts a stress / strain curve between True and Engineering Stress /Strain.
SUB (y)	Subtract the y axis value (or constant) of the second curve from the first curve. If two curves are being subtracted they must have identical x-axis values. If not, the resultant curve is generated by considering every x-coordinate on both curves and by interpolating the other curve the other curve as needed. Any duplicate points as well as points outside the range where the input curves overlap are culled as needed.
SUB (x)	Subtract the x axis value (or constant) of the second curve from the first curve. If two curves are being subtracted they must have identical y-axis values. If not, the nth x-value from the second curve is simply subtracted from the corresponding x-value on the first curve and the y-value from the first curve gets used by the resultant as is.
SUM	Calculates the sum of a group of curves. This "sums" up the y-axis values of a group of curves, and maps the result onto the x-axis of the first curve.
TRA	Translate a curve with respect to the x and y axes. The user is prompted for the x and y values. 
VEC	Calculate the vector magnitude from three input curves.
VEC(2D)	Calculate the vector magnitude from two input curves.
WINDOW	This function is typically used to modify a curve before carrying out an FFT on it.  The y-axis values for each of the input curves is multiplied by a factor between 0 and 1. Three different window shapes are available. The Store Window Curve option can be used to output the multiplying factors to a separate curve if required.

<table>
<tr><td>ZERO</td><td>Translate a curve so that the first data point is moved to (0,0).


By default this option will translate the curve in both X and Y, alternatively the curve can be translated in X only or Y only.</td></tr>
<tr><td>dB</td><td>Converts a curve to dB.

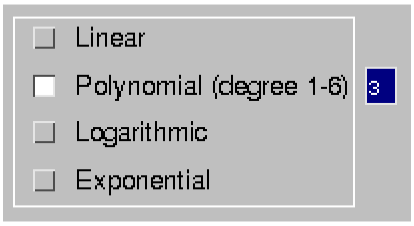
f(x) = 20log(y/ref)

</td></tr>
<tr><td>dBA</td><td>Converts a curve from dB to dBA by applying "A" weighting factors to the curve values.


• Narrow band A weighting values are calculated using a formula.
• 1/3 Octave A weighting values are calculated from a lookup table.</td></tr>
<tr><td>Octave</td><td>Converts a curve from narrow band to either Octave bands or 1/3 rd Octave bands.


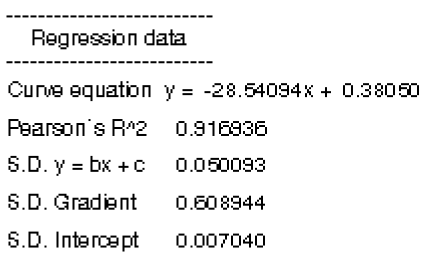
The input curve can either be a curve that has already been converted to dB or it can be an unconverted "linear" curve.

The output curve can also be generated using either Mean values or RMS values.</td></tr>
<tr><td>Regres</td><td>Fits data with either a linear, polynomial (degree 1-4), logarithmic or exponential regression curve, using least squares.</td></tr>
</table>



The equation of the regression curve, as well as the value of Pearson's correlation coefficient, R^2, can be found by right-clicking the output curve and selecting properties. The value of R^2 gives a measure of the goodness of fit of the regression curve, with a value close to 1 corresponding to a good fit.

In the case of linear regression, the standard deviation of the gradient, intercept and y values are also provided in the properties pop-up accessed through right-clicking the curve. Additionally, 95% confidence and prediction bands can be displayed around the linear regression curve by selecting **Properties** in the top-right Tools panel, then selecting **All Curves** and ticking **Show CBands** next to the output regression curve. The confidence band, which is the inner of the two bands, gives a 95% confidence interval at each x value for the best value of y. The prediction band gives a 95% confidence interval at each x value for predicting a new value of y.



8.11. MATHS Options

MATHS Options

The **MATHS** menu is shown in the figure (below). This menu allows mathematical operations to be applied to curves. These options are self explanatory and work on the y-value of the curve (except where explicitly stated).

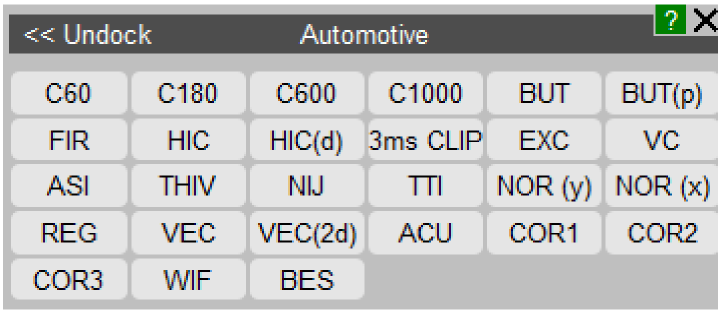
Note: Trigonometrical functions expect the user to work in radians.

SQRT	The square root of a curve.
LOG	Natural log (to base e)
EXP	e to power of
LOG10	Log to base 10
****n**	Raise to power n
LOG(x)	Natural log (x-axis values)
LOG10(x)	Log to base 10 (x-axis values)
SIN	Sine (radians assumed)
ASIN	Arc sine
COS	Cosine
ACOS	Arc cosine
TAN	Tangent
ATAN	Arc tangent
ATAN2	Arc tangent using two curves

8.12. AUTOMOTIVE Options

AUTOMOTIVE Options

The **AUTOMOTIVE** menu is shown in the figure (below). The automotive options are a number of operations that can be performed on curves, typically finding their use in the Automotive industry. They consist of filters and injury criteria calculations, along with a number of other useful functions.



All the options in the **AUTOMOTIVE** menu require a single set of curves as input except the **VEC** and **VEC(2D)** options which require groups of 3 or 2 curves respectively as input but only output a single curve. (See Curve Operations for more information on curve groups).

Notes on using the various filters

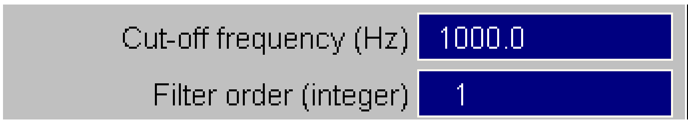
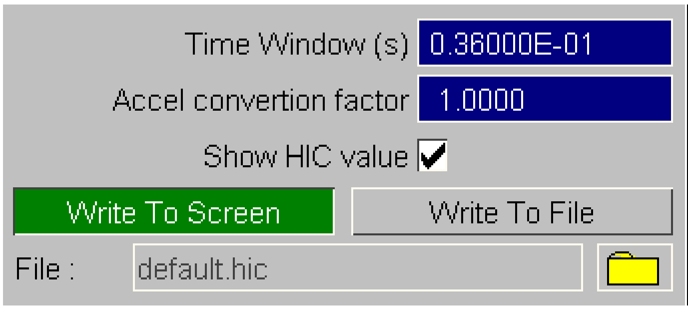
When filtering curves the sampling rate of the data should be considered: it should be at least twenty times the filter cutoff frequency if good results are to be obtained.

T/HIS will reject attempts to filter curves for which the sampling rate is too low, if this happens the **REG** option can be used to increase the number of points. This will allow the filter to function although it is not a good substitute for obtaining data at a higher sampling rate.

For more information on the filters and injury criteria calculations see Appendices D & E.

All of the filters expect the input curve to have a consistent time interval. When using one of the filter options the user can specify a time interval for the curve to be automatically regularised to (**REG**) before filtering if the time interval is not consistent. The user can set a default time interval for regularising the input curves in the PREFERENCE menu. The PREFERENCE menu can also be used to automatically convert the x axis values from milliseconds to seconds before filtering and to convert the curve back to milliseconds afterwards.

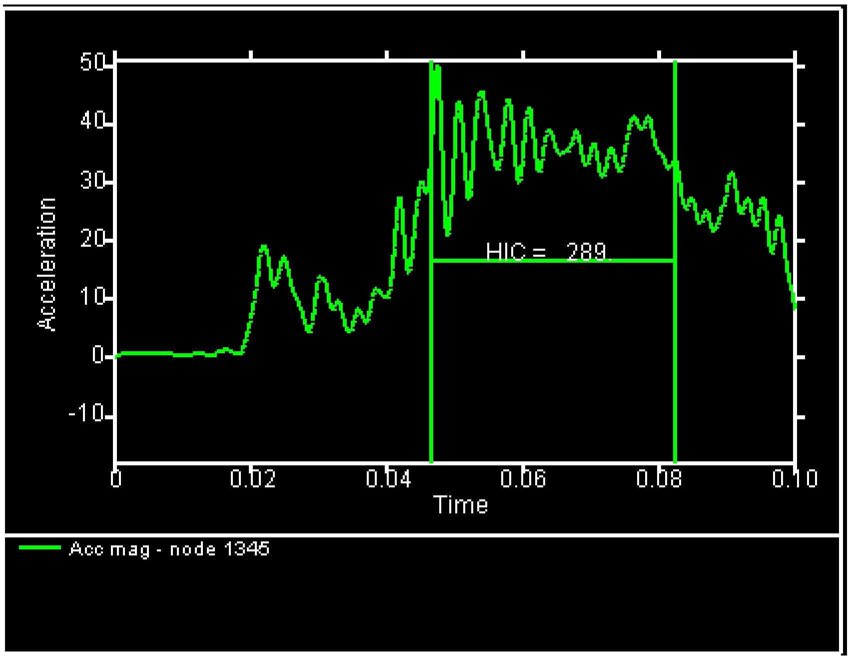
C60	Filter a curve using a standard SAE Class 60 filter.

C180	Filter a curve using a standard SAE Class 180 filter.
C600	Filter a curve using a standard SAE Class 600 filter.
C1000	Filter a curve using a standard SAE Class 1000 filter.
BUT	The curve is passed through a Butterworth filter. The user is prompted for the cutoff frequency and the order of the filter. 
BUT(p)	This passes a curve through a Pure Butterworth filter. This is the same as the BUT function above, but the two refinements, described in Appendix D, to minimise end-effects and phase change errors are not included.
FIR	Special filter for US "SID" dummy.
HIC	Calculates the Head Impact Criteria from an acceleration time history. The user is prompted for the time window and the acceleration conversion factor.  Normally this option writes the HIC value to the screen. If required the values may also be written out to a file using the **WRITE TO FILE** option. The time unit for the input curve should be seconds. T/HIS looks at the range of the x-axis values and if the range is >1 then T/HIS will assume the x-axis values are in ms and it will automatically divide the x-axis values by 1000. If the y-axis values are not in (G) then an optional factor can be specified that T/HIS will DIVIDE the y-axis values by to convert them to (G). Example factors for different units are :

Unit	Factor
m/s^2	9.81
mm/s^2	9810
mm/ms^2	0.00981

In addition to calculating and reporting the HIC value the time window and value can be displayed on the graph using the **Show HIC Value** option.

See Appendix E for more details on the Head Impact Criteria calculation.



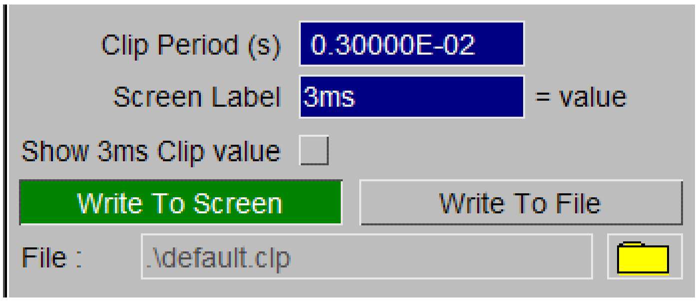
HIC(d)

HIC(d) is used to calculate the Head Injury Criteria for the Free Motion Headform used within the FMVSS201 legislation. The equivalent dummy HIC(d) is calculated as follows

$$HIC(d) = 0.75446 * (\textrm{free motion headform HIC}) + 166$$

3ms CLIP

Calculates the **3ms clip** value from an acceleration time history. Normally this option writes the value to the screen, and produces a curve of the clip region.



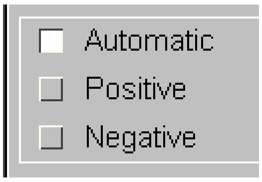
By default the screen value will be labeled as " `3ms = value` ". This label can be modified by specifying a different **Screen Label.**

If required the values may also be written out to a file using the **WRITE TO FILE** option. In addition to calculating and reporting the 3ms clip value the time window and value can be displayed on the graph using the **Show 3ms Clip Value** option.

See Appendix E for more details on the 3ms clip calculation.

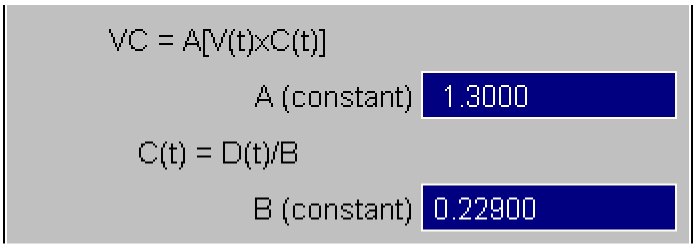
EXC

Calculate and displays an **EXC**eedence plot. This is a plot of force (y-axis) versus cumulative time (x-axis) for which the force level has been exceeded. By default the **Automatic** option will create an exceedence plot using either the +ve OR the -ve values depending on which the input curve contains most of. The **Positive** option will calculate the exceedence plot using only the points with +ve y values.
The **Negative** option will calculate the exceedence plot using only the points with -ve y values.



VC

Calculates the **V**iscous **C**riteria from an acceleration time history. The user is prompted for the constants A and B. See Appendix E for more details.



<table>
<tr>
<td>

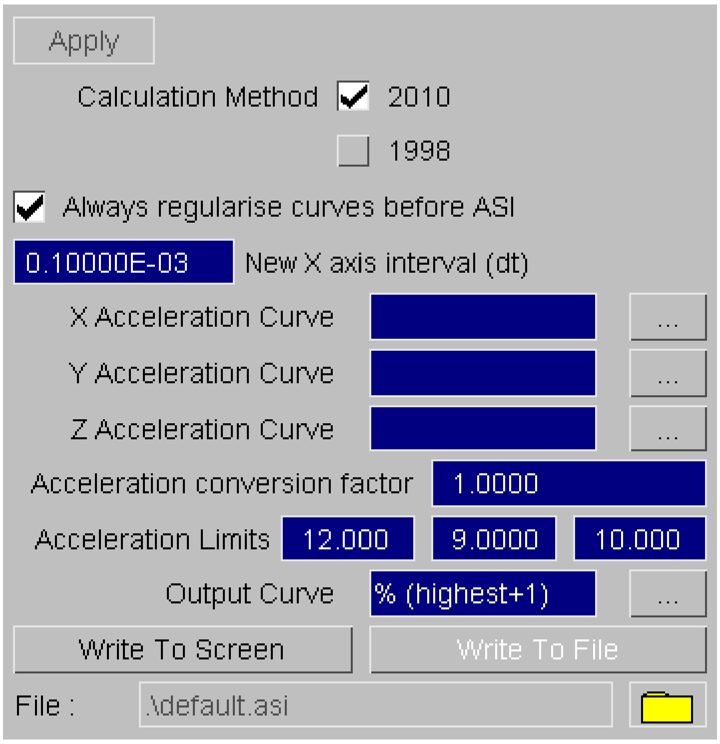
ASI

</td>
<td>

Acceleration **S**everity **I**ndex. This value is used to assess the performance of road side crash barriers.

This option requires 3 acceleration input curves. The user is prompted for the acceleration limits in the 3 directions.

The calculation method can be set to 2010 (BS EN 1317-1:2010) or 1998 (BS EN 1317-1:1998). See Appendix E for more details on this calculation.



</td>
</tr>
<tr>
<td>

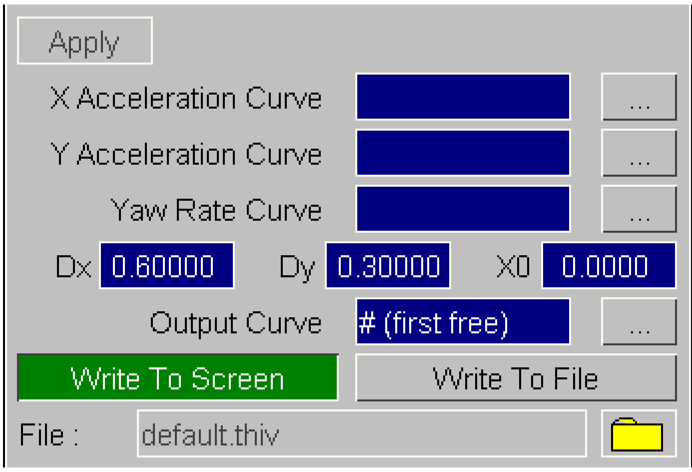
THIV

</td>
<td>

Theoretical **H**ead **I**mpact **V**elocity and
the **P**ost **I**mpact **H**ead **D**eceleration. These values are used to assess the performance of road side crash barriers.

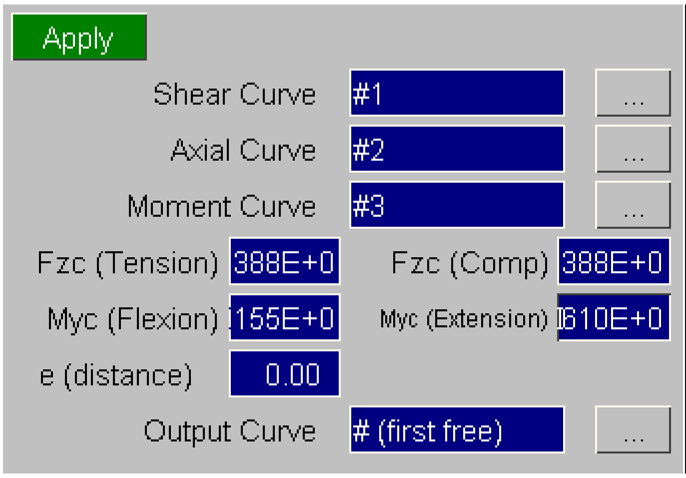
This option requires 3 input curves, a longitudinal and lateral acceleration and a rotation rate. The user is prompted for the constants **Dx**, **Dy** and **X0**. See Appendix E for more details on these calculations.

</td>
</tr>
</table>



NIJ

Biomechanical neck injury predictor. Used as a measure of injury due to the load transferred through the occipital condyles.



This option requires 3 input curves. 1 to represent Shear force, 1 to represent Axial force and a third to represent bending moment in the dummy's upper neck loadcell. Enter these curves in the corresponding input boxes.

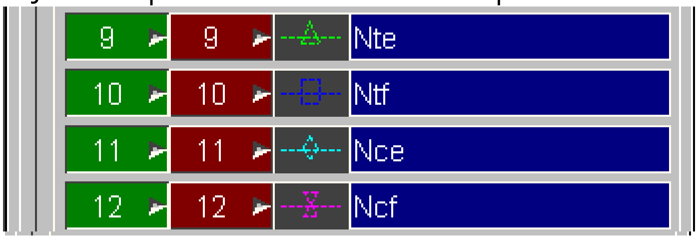
The 4 critical constants used to calculate NIJ; **Fzc (Tension)**, **Fzc (Comp)**, **Myc (Flexion)** and **Myc (Extension)** default to the values specified by the test creators. These can be changed by entering different values into the respective boxes.

Enter the e distance into the **e (distance)** box.

Select which curves you wish to output to in the **Output** box.

For more information on the calculation of NIJ, refer to Biomechanical neck injury predictor (NIJ).

NIJ will output 4 curves due to the 4 possible loading conditions for Nij;



Nte is the tension-extension condition

Ntf is the tension-flexion condition

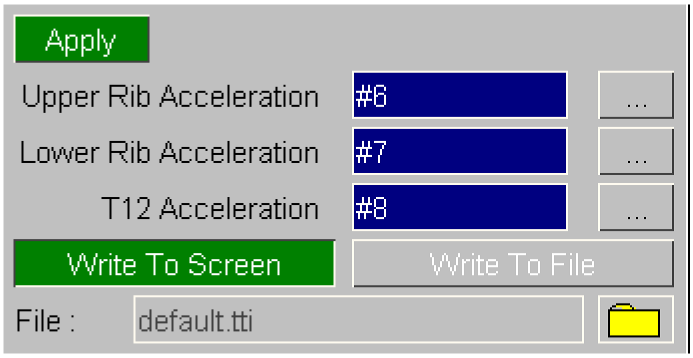
Nce is the compression-extension condition

Ncf is the compression-flexion condition

TTI

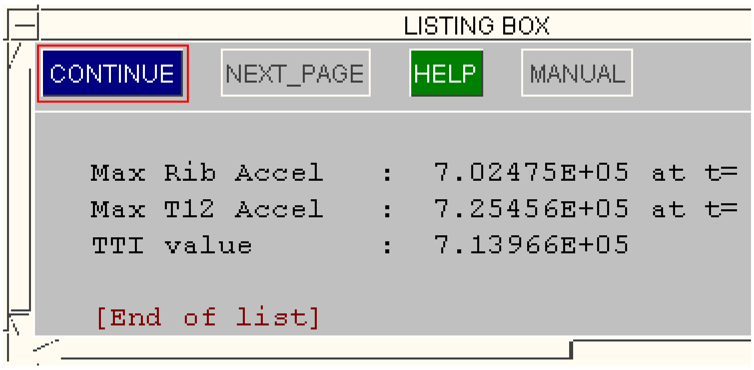
Thorax Trauma Index:

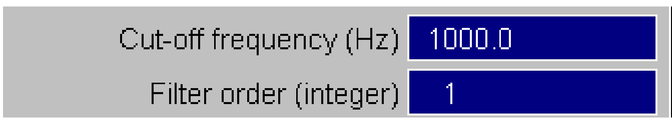
This option requires 3 input curves. 1 to represent the Upper Rib Acceleration, 1 to represent the Lower Rib Acceleration and a third to represent the Lower Spine (T12) Acceleration. Enter these curves in the corresponding input boxes.



The output can either be written to the screen, appearing in a listing box, or written to a file specified in the **File:** input box, or both.

If the **Write To Screen** option is toggled on, the following window will appear:

	 For more information on the calculation of TTI, refer to The Thoracic Trauma Index (TTI).
NOR(y)	Normalise the curve so that the Y values are within the range [-1, +1].
NOR(x)	Normalise the curve so that the X values are within the range [-1, +1].
REG	Make a curve have a constant time step. It is necessary for a curve to have a constant time step between points for it to be filtered. This option takes an existing curve and prompts the user for a new time step. The points of the output curve are calculated by linear interpolation. Regularising a curve may alter its peak values, and could change filtered output slightly.
VEC	Calculate the vector magnitude of three input curves.
VEC(2D)	Calculate the vector magnitude of two input curves.
ACU	Airbag Control Unit This option evaluates the following equation: \(ACU(T) = \int_{T-n}^{T} (a(t) - m)dt \\ \begin{align*} \textrm{Where } m &= \textrm{user-defined offset} \\ n &= \textrm{time to integrate over} \end{align*} \)
COR1	Curve correlation function. The Correlation function provides a measure of the degree to which two curves match. When comparing curves by eye, the quality of correlation may be judged on the basis of how well matched are the patterns of

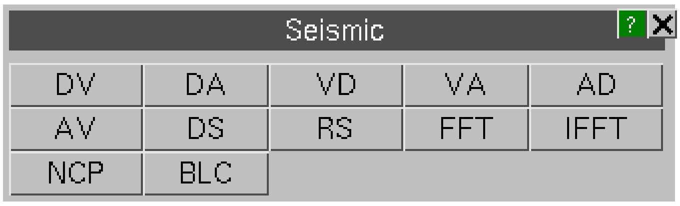
	peaks, the overall shapes of the curves, etc, and can allow for differences of timing as well as magnitude. Thus a simple function based on the difference of Y-values (such as T/HIS ERR function) does not measure correlation in the same way as the human eye. The T/HIS correlation function attempts to include and quantify the more subtle ways in which the correlation of two curves may be judged. The input parameters for the COR1 function have been chosen so as to produce a strict judgement of the correlation (see Appendix F for more details).
COR2	The COR2 function is the same as COR1 except the input parameters have been chosen so as to produce a less strict judgement of the correlation (see Appendix F for more details).
COR3	Another curve correlation function. This function first normalises the curves using two factors either specified by the user or defaults calculated by the program (the maximum absolute X and Y values of both graphs). For each point on the first normalised curve, the shortest distance to the second normalised curve is calculated. The root mean square value of all these distances is subtracted from 1 and then multiplied by 100 to get an index between 0 and 100. The process is repeated along the second curve and the two indices are averaged to get a final index. The higher the index the closer the correlation between the two curves. Note that the choice of normalising factors is important. Incorrect factors may lead to a correlation index outside the range of 0 to 100 (see Appendix F for more details).
WIF	Weighted Integrated Factor (WIFAC) curve correlation function. Compares curves using the Weighted Integrated Factor method (WIFAC). A value between 0 and 100 is calculated, the higher the index the closer the correlation between the two curves. See Appendix F for more details.
BES	The curve is passed through a Bessel filter. The user is prompted for the cutoff frequency 

	and the order of the filter.
CORA	T/HIS includes CORA (**COR**relation and **A**nalysis), an implementation of the methodology used by the Partnership for Dummy Technology and Biomechanics (PDB) software CORA. For more details, see Appendix F – CORA implementation.
MADM	The minimum area discrepancy method (MADM) is ideal for correlation between LS-DYNA simulations and physical tests when force versus deflection is the relationship of interest, and offers benefits over other correlation methods that focus on parameters versus time. For more details, see Appendix F – MADM Correlation tool.

8.13. SEISMIC Options

SEISMIC Options

The **SEISMIC** menu is shown in the figure (right). T/HIS can be used to handle response spectra information. In particular, displacement, velocity or acceleration spectra can be read and converted to another format.



DV Displacement spectrum is converted to a velocity spectrum

DA Displacement spectrum is converted to an acceleration spectrum

VD Velocity spectrum is converted to a displacement spectrum

VA Velocity spectrum is converted to an acceleration spectrum

AD Acceleration spectrum is converted to a displacement spectrum

AV Acceleration spectrum is converted to a velocity spectrum.

DS

Produce a design spectrum from a response spectrum through the specification of a broadening factor..

RS

Produce a response spectrum from input accelerations. This gives the response of a damped single degree of freedom system, given its damping factor and period, to the input acceleration time-history.

FFT

Perform a fast Fourier transform. Convert an input signal from the time to the frequency domain.

There are three options for output;
- magnitude only
- magnitude and phase
- real and imaginary components of the time signal.

The frequency is calculated in Hz NOT radians/s if the time axis is in seconds.

T/HIS automatically adds points with zero y-value to the end of the curve to pad the curve out so that the number of points is increased to the next power of 2.

There are two options for scaling the curves output:

- Scaling Option 1 - Consistent with other signal processing software giving a magnitude independent of any padding. This is the default and recommended for most purposes. Performing an inverse FFT on the resulting curves will NOT get back exactly to the original curve if it did not have a number of points equal to a power of 2.
- Scaling Option 2 - With this option, applying an inverse FFT to the resulting curves will generate a curve the same as the original even if the original curve did not have a number of points equal to a power of 2. This is useful if users wish to create their own filters, where the filter characteristic is defined in the frequency domain.

An option to regularise the curve before performing the function is on by default. The spacing between points on the frequency axis of the resulting curve is determined by the time duration of the padded input curve; dx = 1.0/(time).

The highest frequency in the output curve is determined by the time interval of the input curve; $F_{(max).}$ = (#points)/dt

IFFT

Performs an inverse fast Fourier transform. Converts two input signals from the frequency to the time domain. The two input signals can be the magnitude and phase or real and imaginary components of the time signal.

NOTE: If an FFT using scaling option 1 is performed on a curve that does not have a number of points equal to a power of 2 and then an IFFT is performed on the resulting curves you will NOT get back exactly to the original curve. This is because the FFT and IFFT both scale their output curves by the number of points in the curve, which in this case will be different. For the FFT the number of points used to scale the curves is the original number of points before padding. For the IFFT the number of points used is the original number of points plus the points needed to make it a power of 2.

If the number of points in the original curve is a power of 2 and no padding is required, the IFFT of the resulting curves will get back to the original curve.

NCP

By default beam element plastic rotations are always written out by LS-DYNA as being increasing +ve (i.e. cumulative). This option allows a non-cumulative plastic rotation to be calculated by taking two input curves: the moment/time and the cumulative rotation/time histories for the beam in question.

BLC

Baseline correction.

8.14. MACRO Options

MACRO Options

The **MACRO** menu can be used to play FAST-TCF based macro files on existing T/HIS curves.

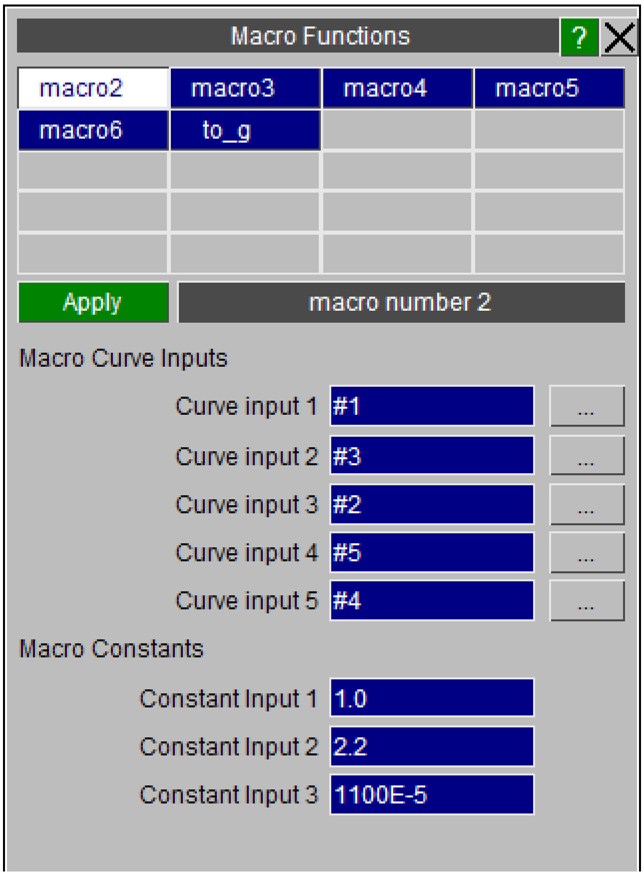
T/HIS macro files can be stored in any directory. Each user can define up to three macro areas using the oa_pref option:

`this*macro_directory : C:\blah\macros`

T/HIS will read any **`.thm`** files within the macro directories and generate the macro menu (shown right) using keywords within the macro scripts. T/HIS will search the macro directories in the following order.

`this*macro_directory` from SYSTEM oa_pref file
`this*macro_directory` from users oa_pref file
`this*macro_directory` from local oa_pref file

The scripting for a macro in T/HIS is based upon FAST-TCF using variables within the script (see [FAST-TCF section](#) for more details). There are some extra command options to make T/HIS aware of input curves and constants within the macro. These macro keywords are as follows:



`macro acr <name>`	acronym for the macro button in T/HIS
`macro title <description>`	a more descriptive title for the macro
`macro curve <curve variable> <curve description>`	FAST-TCF variable name for input curve followed by curve description

`macro const <const variable> <curve description>`	FAST-TCF variable name for input constant followed by constant description

If one or more macro files are found with duplicate acronyms then only the last file read will be displayed so users can override SYSTEM macros with their own definitions if they want to.

When the user selects one of the Macro functions the macro file associated with the function is read and T/HIS creates a selection menu for the user to define the relevant curve numbers and constant values to input into the macro script. These inputs will **replace** the variables used within the macro.

For example: If the user selects **`#1`** for the macro curve variable **`macro_input`** , then any occurrence of **`$macro_input`** in the macro script will be replaced by **`#1`** .

An example macro script follows. This macro asks the user for a filter option (e.g. c60, c600, c1000) and also an input curve number. The macro then filters the input curve and divides by 9810.

```
# Macro to convert a file to g after filtering
#
macro acr to_g
macro title filter and convert curve to g
macro curve macro_input input curve
macro const macro_filter filter to use
#
model none
model 1
oper $macro_filter $macro_input tag filtered
oper div filtered 9810.0
```

8.15. FAST-TCF Options

FAST-TCF Options

The **FAST-TCF** menu can be used to capture and playback FAST-TCF scripts. FAST-TCF is a simple and intuitive scripting language for T/HIS. See FAST-TCF for more details and commands.

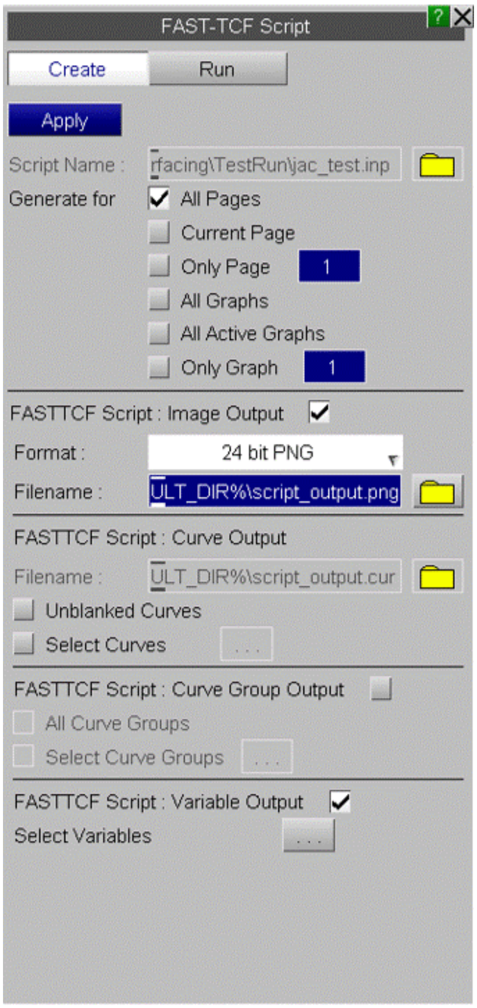
8.15.1. Create

Create

T/HIS 9.2 onwards has the ability to automatically generate FAST-TCF scripts using the CREATE menu.

The FAST-TCF script will contain all of the commands required to

1. Create and position multiple graphs on pages.
2. Extract the data from models or other files
3. Carry out any curve operations required to reproduce the chosen curves
4. Set any curve styles and labels
5. Set plot attributes such as titles, axis labels, colours, fonts and scaling
6. Generate the output image and/or curve file
7. Generate curve groups
8. Generate variable and tabular output requests



Before generating the FAST-TCF script the following options can be set:

Generate For

All Pages	The FAST-TCF script will contain all of the commands required to regenerate all of the pages that contain 1 or more graphs. If the option to generate images is selected then the FAST-TCF script will contain the commands to generate multiple images with the page number appended to the filename specified.
Current Page	The FAST-TCF script will contain all of the commands required to regenerate the currently displayed page.
Only Page (n)	The FAST-TCF script will contain all of the commands required to regenerate the selected page.
All Graphs	The FAST-TCF script will contain all of the commands required to regenerate all the currently defined graphs. All of the graphs will be positioned on page 1 using the currently defined layout. *This option will only be available if T/HIS only contains a single page (see Page Layouts).*
All Active Graphs	The FAST-TCF script will contain all of the commands required to regenerate all of the active graphs. All of the graphs will be positioned on page 1 using the currently defined layout. *This option will only be available if T/HIS only contains a single page (see Page Layouts).*
Only Graph (n)	The FAST-TCF script will contain all of the commands required to regenerate the selected graph. The graph will be positioned on page 1.

FAST-TCF Script : Image Output

If this option is selected then the FAST-TCF script will contain the commands required to generate an image of each of the pages/graphs selected for output. The **Image Format** can be set to any of the supported image types (see IMAGE Options).

If the FAST-TCF script generates multiple pages then the **Filename** specified will be used for the first image. Subsequent images will use the specified filename with "_2", "_3" ... appended.

FAST-TCF Script : Curve Output

By default the FAST-TCF script will only contain the command needed to reproduce the curves that are unblanked in 1 or more of the graphs selected for output. This option can be used to select additional curves for which the commands required to generate them are also added to the FAST-TCF script. If a curve is selected that is also unblanked in one of the graphs the command to regenerate it are only added to the FAST-TCF script once.

In addition to selecting additional curves this option can also be used to add commands to the FAST-TCF script to write the additional curves out to a T/HIS .cur curve file.

FAST-TCF Script : Curve Group Output

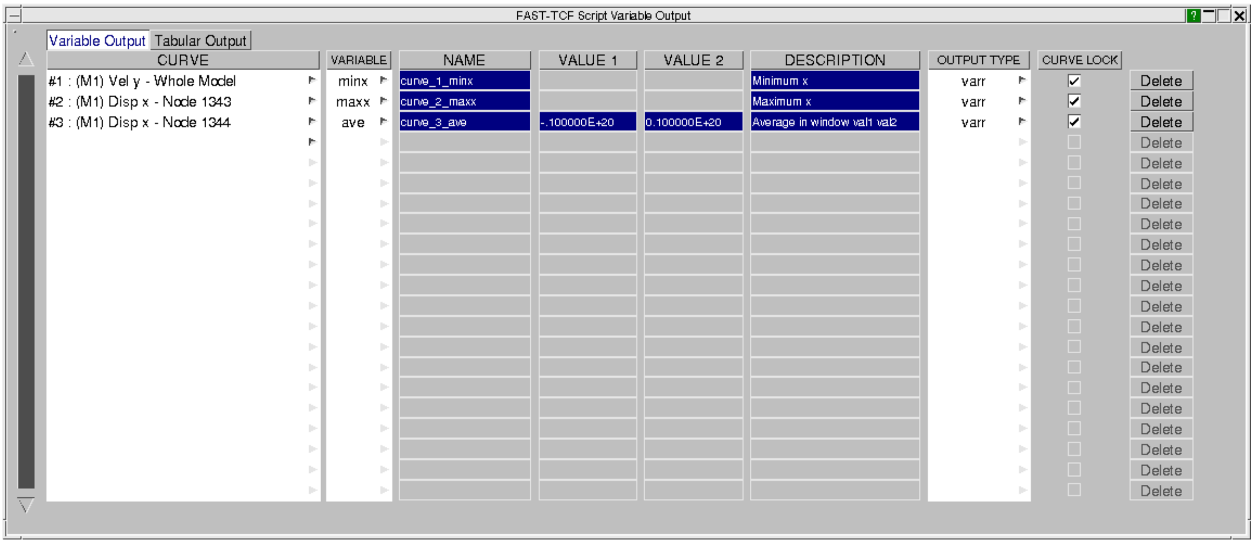
This option can be used to select additional curves for output to the FAST-TCF script by curve group. If a curve is selected that is also unblanked in one of the graphs the command to regenerate it are only added to the FAST-TCF script once. This option will also add the commands to regenerate the selected curve groups to the FAST-TCF script.

FAST-TCF Script : Variable Output

This option can be used to define variable and tabular output requests for output to the FAST-TCF script via the menu launched by pressing the button. Variable and tabular output requests defined in a FAST-TCF script that is read in will appear in the menu.

Variable Output or **Tabular Output** are selected at the top of the menu. Each output request is defined on a row of the table. The curve and variable type are selected using the drop down menu. A default variable name is generated and can be manually edited or a name can be selected from variable names that are present in . Additional value fields are populated with default values if required and these can be edited. The output description is also populated with default text that can be edited. The output type is selected using the drop down menu. Curve Lock prevents the curve that the variable refers to from being deleted. If a curve is not locked and is deleted, then any variables associated with that curve will also be deleted.

Pressing the heading buttons will sort the output request definitions.


FAST-TCF Script Variable Output
Variable Output
Tabular Output
CURVE
VARIABLE
NAME
VALUE 1
VALUE 2
DESCRIPTION
OUTPUT TYPE
CURVE LOCK
#1 : (M1) Vel y - Whole Model
minx
curve_1_minx
Minimum x
varr
Delete
#2 : (M1) Disp x - Node 1343
maxx
curve_2_maxx
Maximum x
varr
Delete
#3 : (M1) Disp x - Node 1344
ave
curve_3_ave
-.100000E+20
0.100000E+20
Average in window val1 val2
varr
Delete

8.15.2. Run

Run

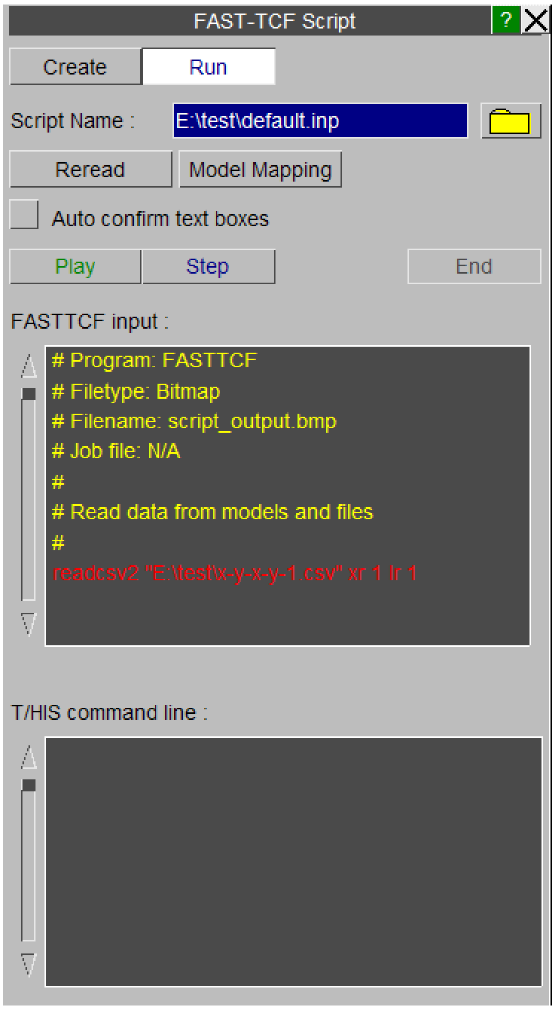
This menu allows the user to run a FAST-TCF file from within T/HIS. After the user has selected the FAST-TCF file T/HIS scans the file for data requests and model requests to see what input the FAST-TCF file requires. Note that there must be a model read into T/HIS before a FAST-TCF file that contains data extraction can be run.

The next FAST-TCF command line is displayed in red in the upper text area, at this point the user can select to **Play** the FAST-TCF file or **Step** through it line by line. After every line of FAST-TCF the resulting command in T/HIS is shown in the lower text area. Select **End** during stepping through the lines to go to the end of the file. **Reread** will re read the file and start back at the beginning.

The **Model Mapping** option allows the user to define which model in T/HIS should be used for the equivalent model number in the FAST-TCF script. The model number **zero** is equivalent to the default model in FAST-TCF if no models are defined. The default model mapping will use the same model numbers as in the FAST-TCF script.

Auto confirm text boxes will force T/HIS to confirm any text boxes that should appear in the interactive playback of a FAST-TCF script (such as HIC results and so on).

The FAST-TCF script will ignore any existing T/HIS curves and their tags. This guarantees that the user can run a single FAST-TCF file many times and it will only use the new curves created by FAST-TCF.


FAST-TCF Script
Create
Run
Script Name :
E:\test\default.inp
Reread
Model Mapping
Auto confirm text boxes
Play
Step
End
FASTTCF input :
Program: FASTTCF
Filetype: Bitmap
Filename: script_output.bmp
Job file: N/A
#
Read data from models and files
#
readcsv2 "E:\test\x-y-x-y-1.csv" xr 1 lr 1
T/HIS command line :

8.16. TITLE/AXES/LEGEND Options

TITLE/AXES/LEGEND Options

The **TITLE/AXES** menu is shown in the figure (below).

This menu controls the contents of the title and axes labels and the axis scaling.

The individual axis, title and legend menus can also be accessed by clicking over the appropriately highlighted area on the graph.

Changes to the TITLE/AXES/LEGEND options are only applied to active graphs (see Active Graphs).

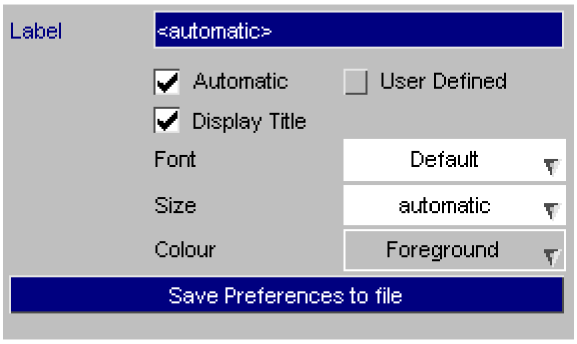
8.16.1. TITLE

TITLE

The plot title may be set automatically or manually. When the **Automatic** option is selected the text box will display **`<automatic>`** and the plot title will be set to the title of the first curve that is currently being plotted. The plot title may be turned on and off by toggling the **Display Title** button.

Save Preferences to File

Launches a popup to quickly save preferences to the oa_pref file. See Save Preferences Popup.



8.16.2. X-AXIS

X-AXIS

AXIS LABELS

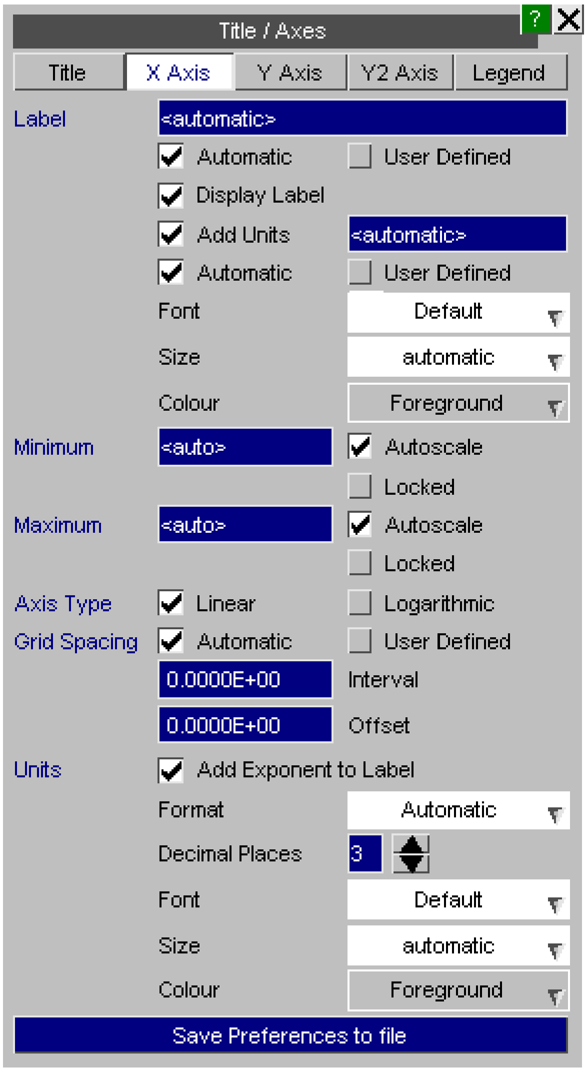
The x-axis label may be set automatically or a user defined label can be specified.. When the **Automatic** option is selected the text box will display `<automatic>` and the axis label will be set to the x axis label of the first curve that is currently being plotted. The axis label may be turned on and off by toggling the **Display Label** button.

In addition to displaying the axis labels an optional unit label can also be appended to the axis label. If the option to add a unit label is set to Automatic then the unit label displayed will depend on the current curves that are visible and the current unit system being used to display results (see UNITS for more information on Unit Systems). If the curves being displayed do not have the same axis unit then no unit label will be displayed. The unit label may be turned on and off by toggling the **Add Units** button.

AXIS LIMITS

The minimum and maximum x axis values can be controlled using a combination of the text box and the popup menu opposite.

Autoscale The axis values will be set to the maximum and minimum values of all the curves that are currently being plotted.
Locked The axis limit is set to the user defined value specified in the text box. If the curves are translated of scaled dynamically the limit will be reset.



Note : The global command **AUTOSCALE** (see Autoscale) will reset the minimum and maximum values to **AUTO** .

AXIS TYPE

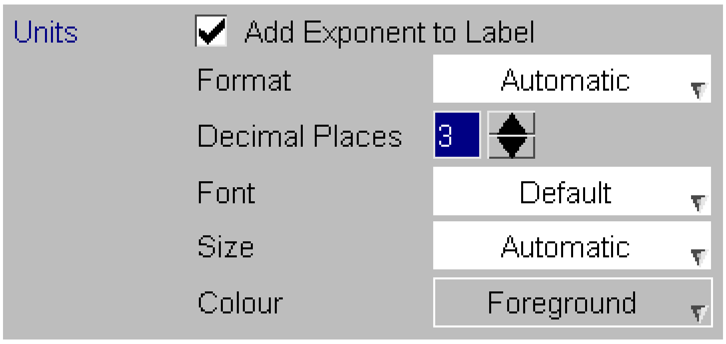
The x-axis can be switched between a **Linear** or **Logarithmic** scale. If a logarithmic scale is selected, a warning will be generated if an attempt is made to plot points that have -ve or zero X values and the points will be skipped.

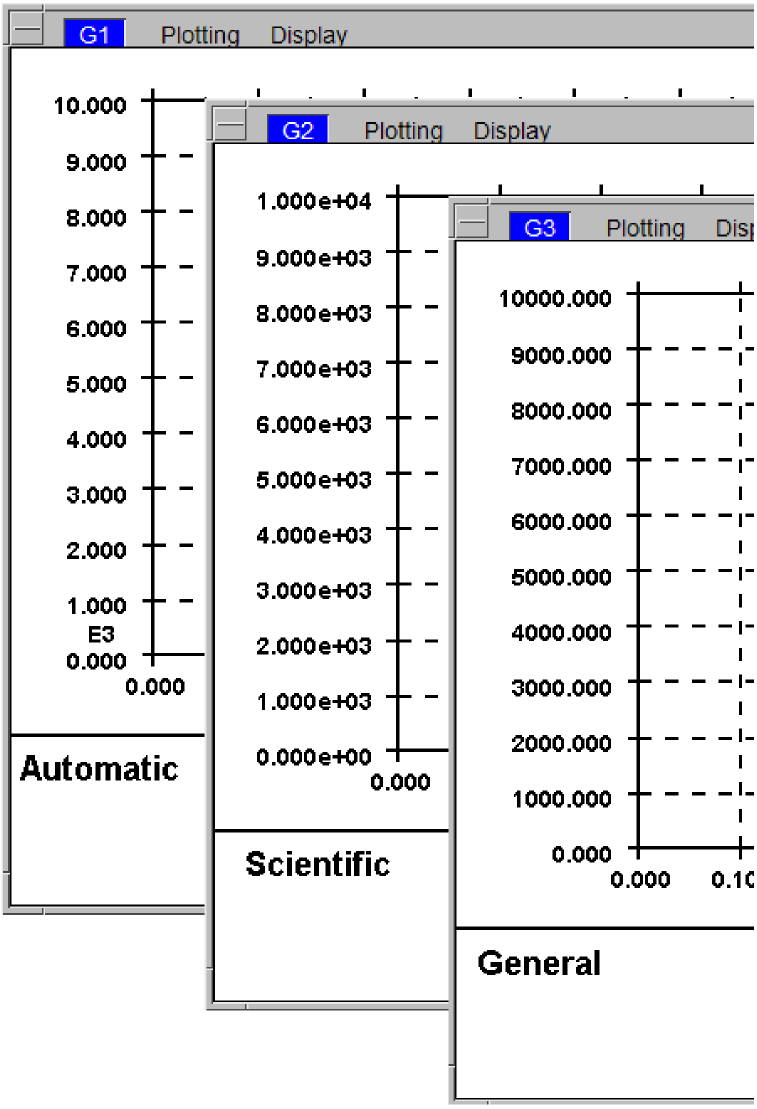
GRID SPACING

By default T/HIS will automatically set the grid line intervals for the x-axis when the grid is tuned on (see GRID). If the GRID option is changed from **Automatic** to **Manual** a grid **Interval** and **Offset** my be specified. If the **Interval** is set to 0.1 and the **Offset** to 0.02 then grid lines will be produced at 0.02, 0.12, 0.22 ....

UNITS

Axis values can be displayed using 3 different formats:





Automatic

Values are displayed using exponential format, all values are displayed as values of E0, E3, E6 etc.

e.g 11.234E+03

Scientific

Values are displayed using exponential format.

e.g 1.123E+04

General

Values are displayed as real numbers.

e.g 11234.000

In addition to specifying the format, the number of decimal places can also be set between 0 and 9 and the colour and font used to display the values can be set.

Save Preferences to File

Launches a popup to quickly save preferences to the oa_pref file. See Save Preferences Popup.

8.16.3. Y-AXIS

Y-AXIS

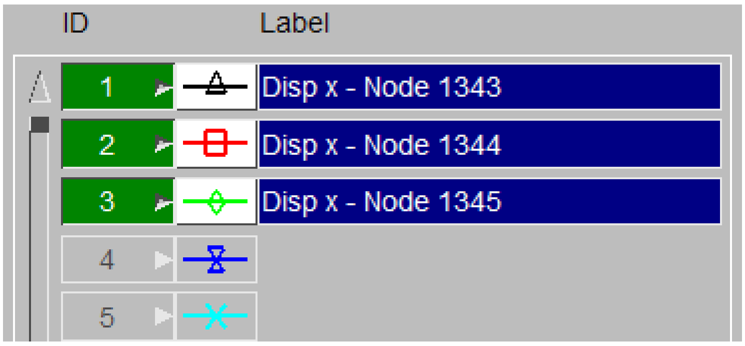
The same options for LABELS, LIMITS, SCALE, GRID LINES and UNITS apply to the Y-AXIS as those available for the X_AXIS.

8.16.4. Second Y-AXIS

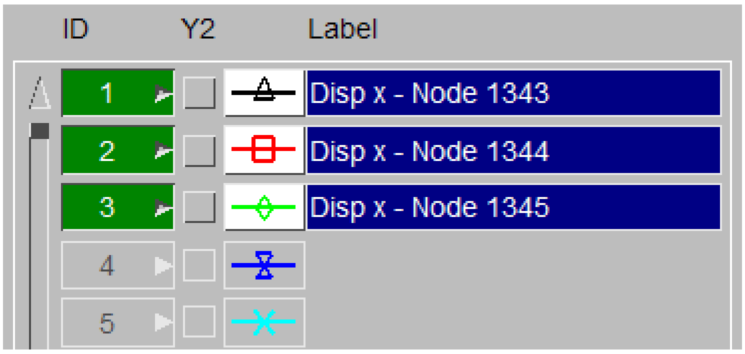
Second Y-AXIS

Curves can be plotted in T/HIS using 2 different y-axis scales. When **Add Second Y Axis** is selected using the tickbox in the Y2 Axis menu the Curve Manager changes

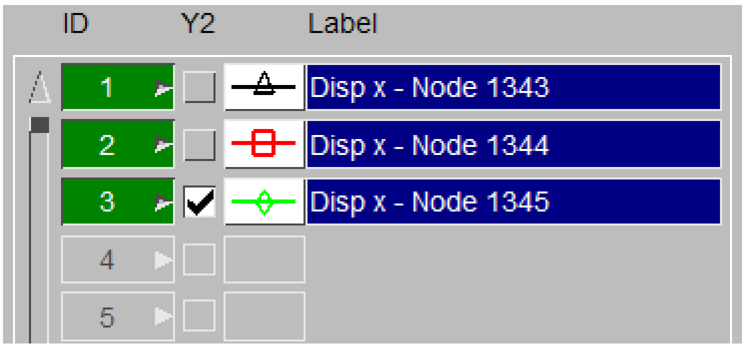
from



to

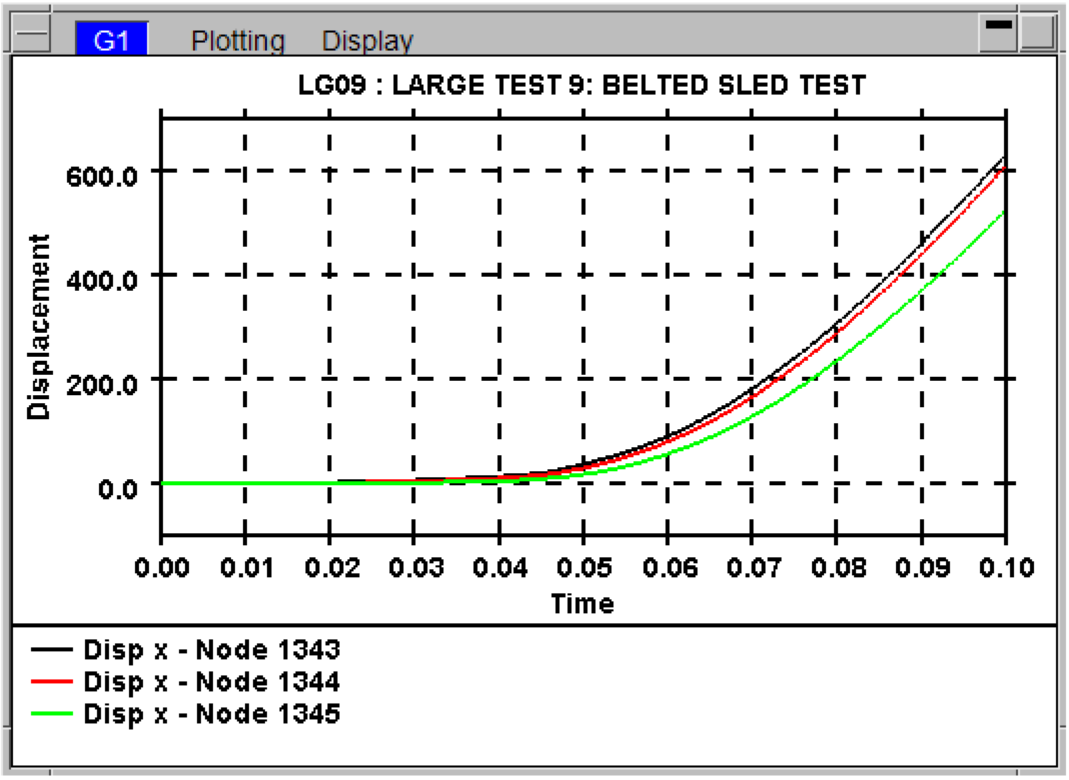


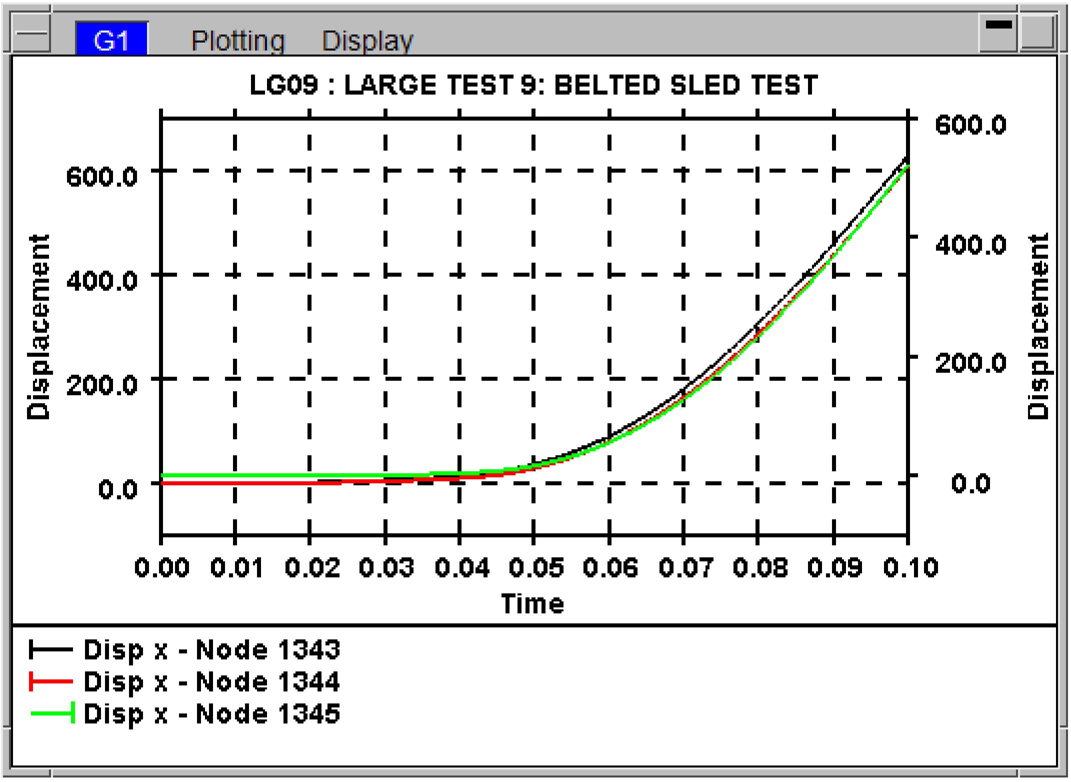
with an additional tick box for each curve that controls which curves are plotted against the second (right hand) y-axis.



If only one y-axis scale is used it is not possible to meaningfully plot curves with different units or very different values. A second scale allows more information to be displayed at once, as demonstrated below.

To identify which axis a curve is being plotted against the line labels on the plot are automatically modified.	Second Y axis disabled	— Disp x - Node 1343
	Left hand Y axis	⊢ Disp x - Node 1343
	Right hand y axis	⊣ Disp x - Node 1343





All of the options that are available to control the label, scale and type of the y-axis are also available for the second y-axis except for the Grid option.

NOTE : When the DOUBLE AXIS option is used with GRID lines a grid is only plotted for the left hand y-axis.

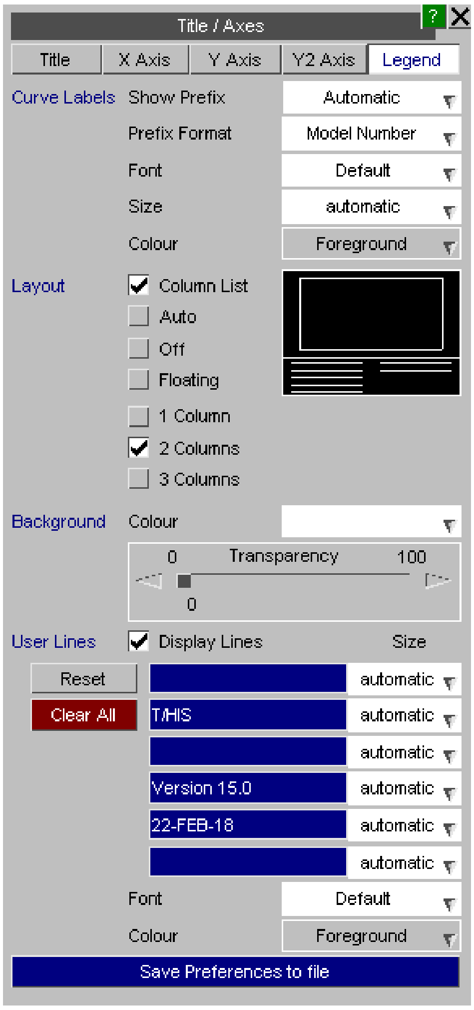
8.16.5. Legend

Legend

Curve Labels

Show Prefix

This option can be used to automatically add a prefix to each of the curve legends when a curve is plotted.



This option has 3 settings:

Automatic If there is more that 1 model loaded in T/HIS then a prefix will automatically be added to any curves that have been read in from a model. Curves read in from other files will not be prefixed.

On A prefix will automatically be added to any curves that have been read in from a model regardless of the number of models currently loaded in T/HIS. Curves read in from other files will not be prefixed.
Off No prefixes will be added

Prefix Format

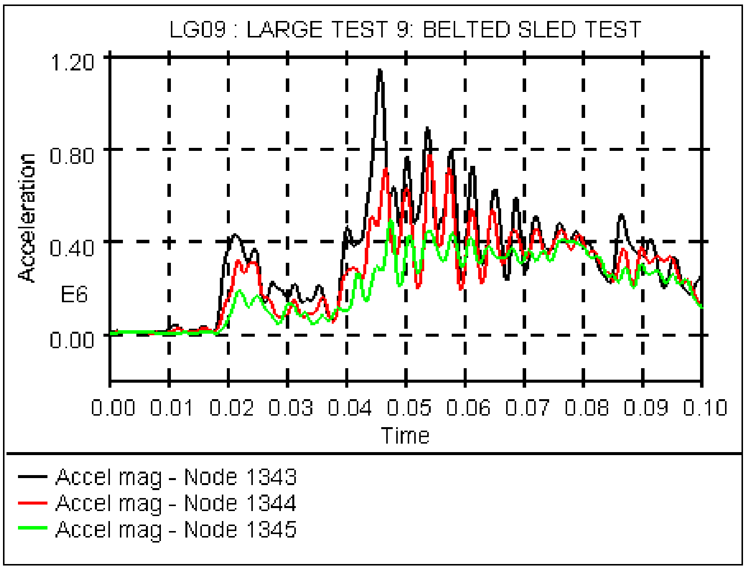
This option can be used to set the format used for the curve prefix. This option has 4 settings

Model Number	The model number will be used as the prefix. e.g **(M1)**
Directory	The directory name the model was read from will be used at the prefix. e.g. **(/run1)**
THF File	The root name of the THF file will be used as the prefix. e.g **(sled_test)**
User Defined	A user defined prefix will be used. The prefix can be defined on a model by model case using the Model Menu.

The font, size and colour of the text used to display the legends can also be specified.

Layout

T/HIS has 4 different plotting formats as described below. The number of columns used to display the curve legends can also be set between 1 & 3. When multiple columns are used curve labels will automatically be truncated to fit the available space.



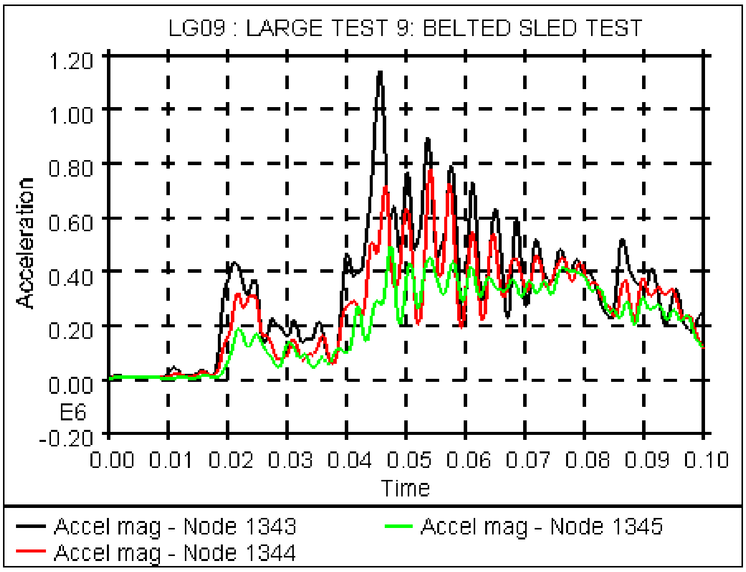
Column List (default)

This format fixes the size of the plotting area. The maximum number of curve legends that can be displayed will depend on the font family and size selected by the user and the number of columns.

If any USER LINES have been defined then the area used to display the legend will be reduced so that the text does not cover the user lines.

Automatic

This format automatically adjusts the plot size to maximise the plotting area while still showing a maximum of 18 line labels. Any text entered using the USER LINES option will be ignored in this plotting mode.



Off

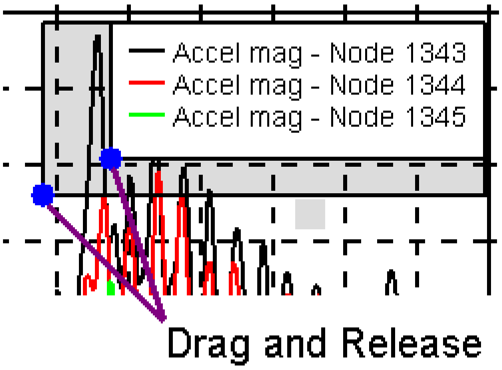
This format turns **OFF** the display of the graph legend and maximises the plotting area by not showing any line labels. Any text entered using the USER LINES option will be ignored in this plotting mode.

Floating

This format maximises the plotting area and positions the legend on top of the graph area.

The size of the legend can be modified by clicking with the left mouse button on the legend border/corner and dragging.

The legend can also be moved by clicking with the left mouse button inside the legend and dragging.



BACKGROUND

This option can be used to alter the default background colour of the floating legend. By default the colour will be the same as the background colour of the graph. As well as setting a different background colour for the floating legend a %age transparency can also be specified if the legend obscures any curves.

USER LINES

This option can be used to alter the default text that appears on the bottom right-hand corner of each plot. Text can be typed into any of the panels or they can be left blank. The **Size** of the text may be altered. If no text is specified the area used by the curve legends will be increased.

The default values are read from the preferences file (see Appendix H for more details).

Save Preferences to File

Launches a popup to quickly save preferences to the oa_pref file. See Preferences.

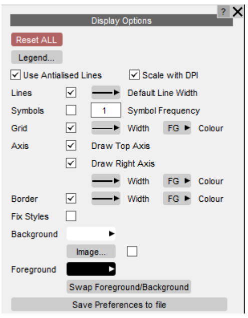
8.17. DISPLAY Options

DISPLAY

The **DISPLAY** menu is shown in the below figure. This menu controls the overall appearance of plots.

As well as controlling basic things like the background colour and whether a grid is drawn, this menu also controls a number of default settings that are applied to all curves. These default settings may be overwritten for individual curves using the Curve Manager.

Changes to the Display options are only applied to Active Graphs. (With the exception that line anti-aliasing and scaling with DPI are both programme-wide.)



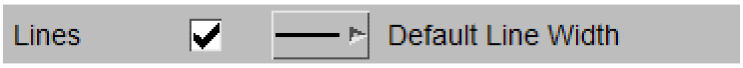
8.17.1. LEGEND...

LEGEND...

This option will map the Legend settings panel (see Legend).

8.17.2. LINES

LINES



This is an **ON** / **OFF** switch for the lines between points to be drawn for all curves. The default is **ON** . The **Default Line Width** is used for all curves that have not had their widths explicitly set in the **Line Style** menu.

The default line width can be specified in the preferences file (see Appendix H for more details).

8.17.3. SYMBOLS

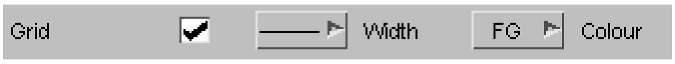
SYMBOLS

This is an **ON** / **OFF** switch which controls whether symbols are plotted on top of the curves to help identify them. This option affects all the curves that are currently being used. If you wish to turn the symbols on for only some of the curves then this switch should be set to **ON** and the **Line Style** menu should be used to turn the symbols off on the curves for which you do not want symbols drawn on. The default is **OFF**.

The **Symbols Frequency** is used for all curves that have not had a frequency explicitly set in the **Line Style** menu. This value controls how often a symbol is drawn on a curve.

8.17.4. GRID

GRID



This is an **ON** / **OFF** switch which determines whether or not grid lines are shown on the plot. The default is **OFF**. The **Grid Width** can be used to change the width of the grid and axis lines. The **COLOUR** button can be used to change the colour of the grid lines (see COLOUR for details on the available colours).

`/de grid on`	turns grid lines on
`/de grid off`	turns grid lines off
`/de grid th 2`	sets the grid thickness to 2 pixels

The default grid width and visibility can be specified in the preferences file (see Appendix H for more details).

8.17.5. AXIS

AXIS

The **Axis Width** can be used to change the width of the axis lines. The **COLOUR** button can be used to change the colour of the axis lines (see Section COLOUR for details on the available colours).

The default axis width can be specified in the preferences file (see Appendix H for more details).

Draw Top Axis	This option can be used to turn on and off the display of the graph's top axis
Draw Right Axis	This option can be used to turn on and off the display of the graph's right hand axis

The default settings for these 2 options can also be specified in the preferences file (see Appendix H for more details).

8.17.6. BORDER

BORDER

This is an **ON** / **OFF** switch which determines whether or not a border is drawn round the plot. The default is **ON** . The **Border Width** can be used to change the width of the border. The **COLOUR** button can be used to change the colour of the border (see Section COLOUR for details on the available colours).

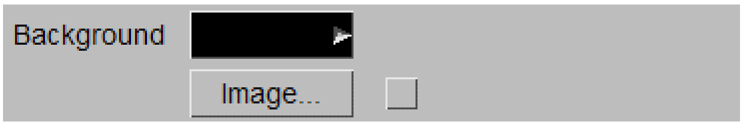
8.17.7. FIX LINE STYLES

FIX LINE STYLES

This is an **ON** / **OFF** switch which resets the curve styles when they are plotted on the screen so that the curves cycle through the default T/HIS colours and styles as they are plotted. This will result in the first curve being plotted always being white, the second red, the third green .etc regardless of their curve numbers. The default is **OFF** .

8.17.8. Background

Background

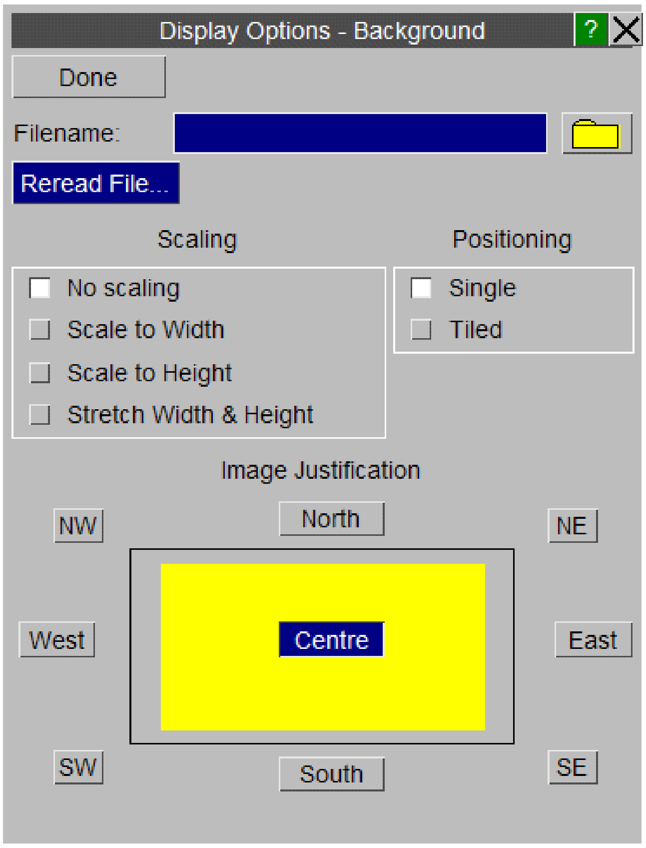


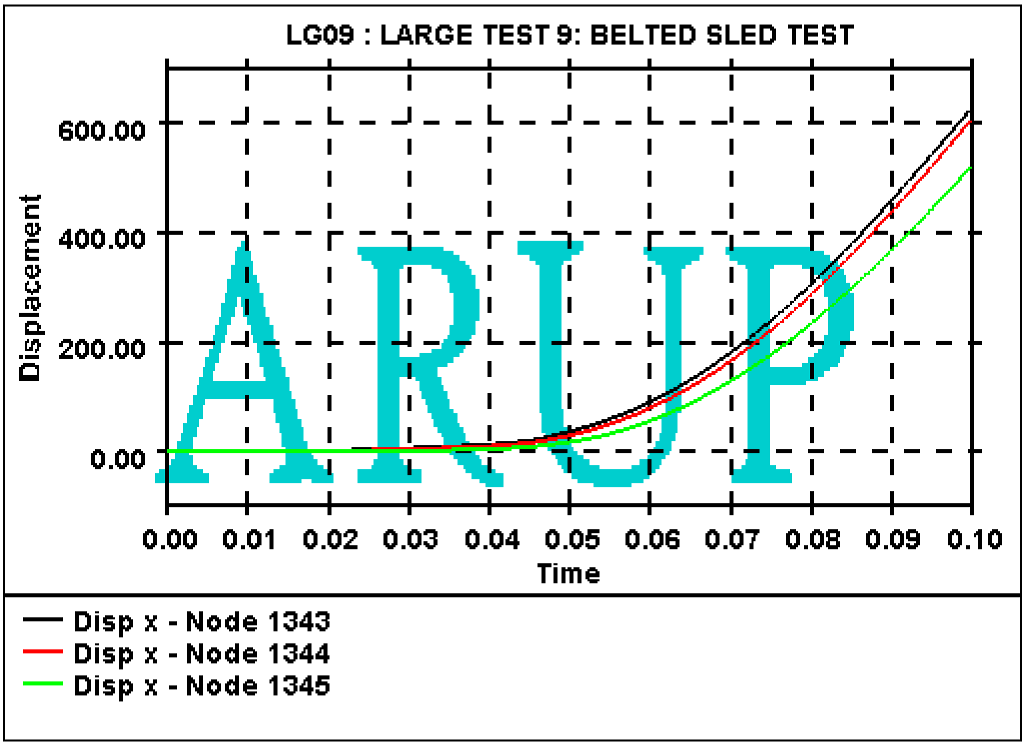
This option can be used to modify the background colour (see Section COLOUR for details on the available colours) or to set a background image. By default the background colour is set to BLACK.

Image

The IMAGE option can be used to display a background image behind a graph instead of a solid background colour.

If the image dimensions do not match the graph window dimensions then the image can be scaled to fit or it can be tiled.




LG09 : LARGE TEST 9: BELTED SLED TEST
600.00
400.00
200.00
0.00
Displacement
0.00 0.01 0.02 0.03 0.04 0.05 0.06 0.07 0.08 0.09 0.10
Time
ARUP
Disp x - Node 1343
Disp x - Node 1344
Disp x - Node 1345

8.17.9. Foreground

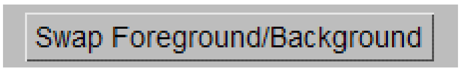
Foreground

This option can be used to modify the foreground colour (see Section COLOUR for details on the available colours). By default the background colour is set to BLACK and the foreground colour is set to WHITE.

Initially the grid, axes, border and labels are all set to the foreground colour.

8.17.10. Swap Foreground/Background

Swap Foreground/Background



This option can be used to swap the currently defined foreground and background colours.

8.17.11. Display Max/Min

Display Max/Min

In versions of T/HIS prior to 9.4 the display of minimum and maximum curve values was controlled in the **DISPLAY** menu . In versions since 9.4 these options have been moved to the **PROPERTIES** menu (see section PROPERTIES).

8.17.12. Save Preference to File

Save Preference to file

Launches a popup to quickly save preferences to the oa_pref file. See Save Preferences Popup .

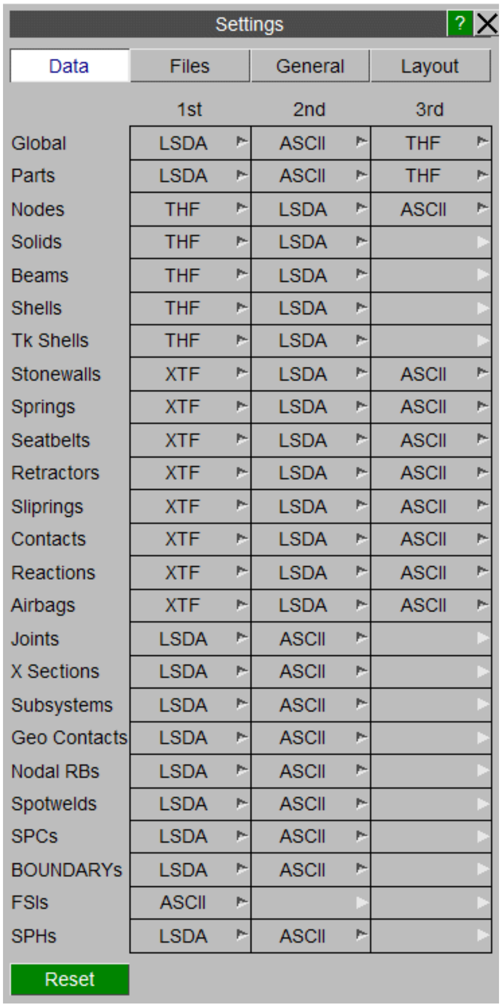
8.18. SETTINGS

8.18.1. Data Sources

Data Sources

This menu allows the user to specify their preferred order of data sources for the different data types. Upon reading in models T/HIS will read all files regardless of these preferences. When T/HIS extracts data for plotting the source is dependent on that currently set in this menu. If you select a data component or entity that is not available in the first data source T/HIS will automatically try the other data sources in order until the combination is found.

The **Model Manager** can be used to see what source has been used for each item for models already read into T/HIS.



8.18.2. Files

Files

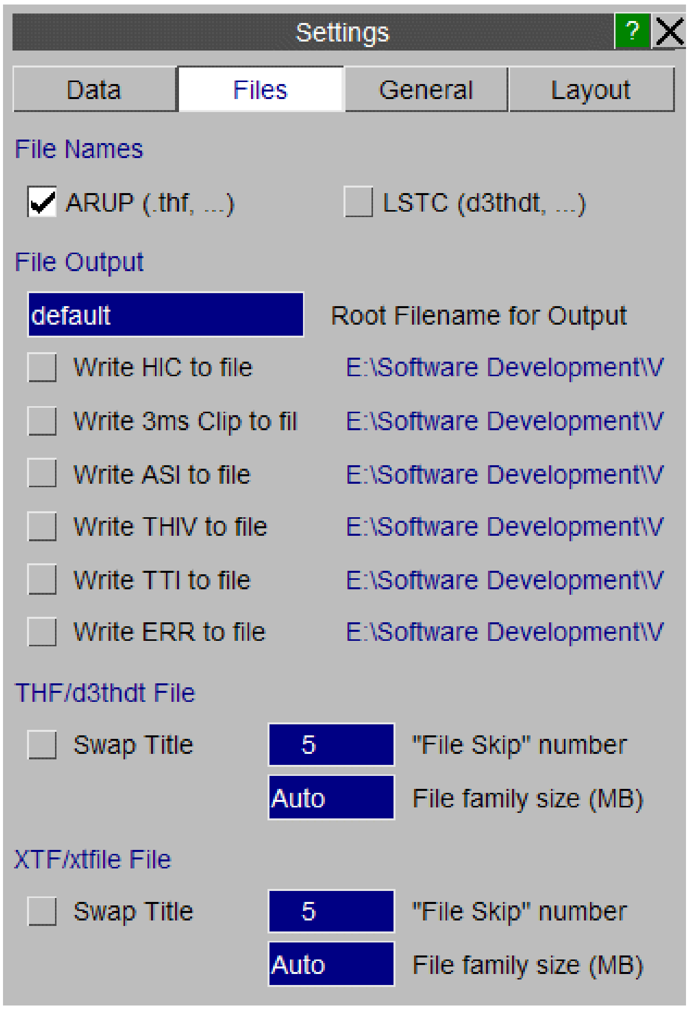
File Names

By default the file filters in T/HIS are set to look for the file naming convention set for the LS-DYNA output files by the SHELL. This option can be used to swap the file filters back to the default Ansys LST naming convention. This option can be set in the Preference File.

File type	ARUP	Ansys LST
Time history	"job".thf	d3thdt
Extra Time history	"job".xtf	xtfile

File Output

The HIC , 3ms Clip, ASI , THIV, TTI Automotive injury criteria functions, ERR operator function and COR1 and COR2 correlation functions can all send their output to a file as well as to the screen. These options can be used to select which functions send output to a file and to specify a Root Filename that is used for all of the output files. The Root Filename can be set in the Preference File.


Settings
?
Data
Files
General
Layout
File Names
ARUP (.thf, ...)
LSTC (d3thdt, ...)
File Output
default
Root Filename for Output
Write HIC to file
E:\Software Development\V
Write 3ms Clip to fil
E:\Software Development\V
Write ASI to file
E:\Software Development\V
Write THIV to file
E:\Software Development\V
Write TTI to file
E:\Software Development\V
Write ERR to file
E:\Software Development\V
THF/d3thdt File
Swap Title
5
"File Skip" number
Auto
File family size (MB)
XTF/xtfile File
Swap Title
5
"File Skip" number
Auto
File family size (MB)

8.18.3. General

General

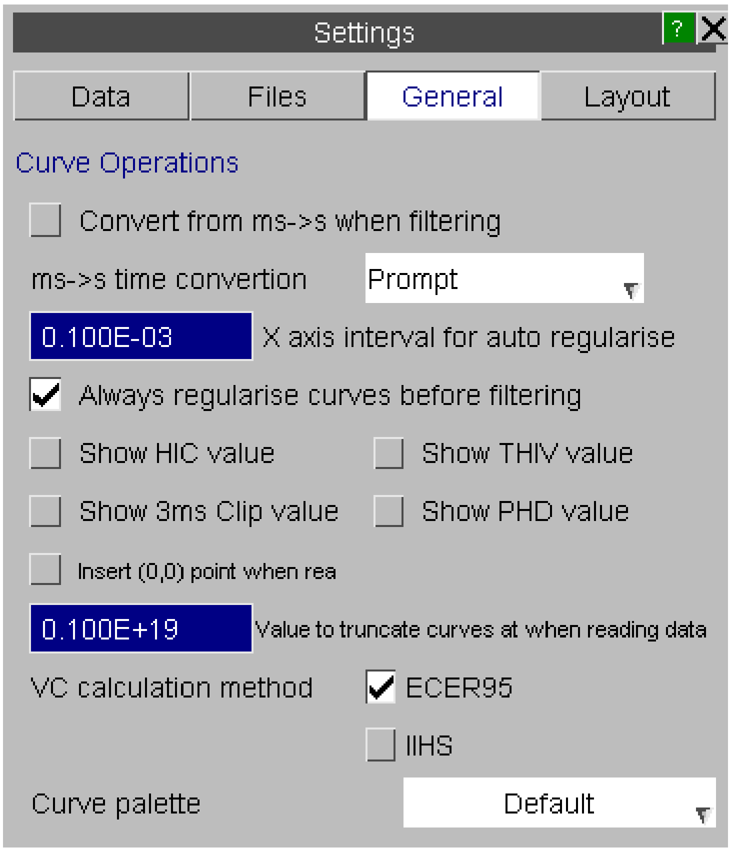
Curve Operations

All of the AUTOMOTIVE filters are designed to filter curves using seconds as the time unit. This option can be used to automatically convert the x-axis values of any curves from milliseconds to seconds before applying one of the filters. If a curve is automatically converted then the output curve is also automatically converted back into milliseconds. This option can be set in the Preference File.

All of the AUTOMOTIVE filters require curves with constant time intervals. This option can be used to specify a default time interval that will be used to automatically regularise a curve before it is filtered.

By default the HIC and 3ms Clip functions calculate and report a value to the screen. These options can be used to make T/HIS display the peak values and the time widows they occur over. These options can be set in the Preference File.

At present 2 different methods are used to calculate the VC injury criteria for the ECER95 and IIHS regulations (see Viscous Criteria for more details). This option can be used to set the default value. This option can be set in the Preference File.



By default T/HIS uses 6 colours (White, Red, Green, Blue, Cyan and Magenta) for any curves that have not had a colour explicitly defined for them. Curves 1,7,13... will be White, 2,8,14... will be Red.

This option can be used to change the default number of colours T/HIS uses.

Default	Use the default 6 colours
Extended	Use the first 13 colours
No Grey	Use all 30 predefined colours except the 3 grey ones
Full	Use all 30 predefined colours plus any user defined ones.

The default value for the curve palette can also be specified in the preferences file (see Appendix H for more details).

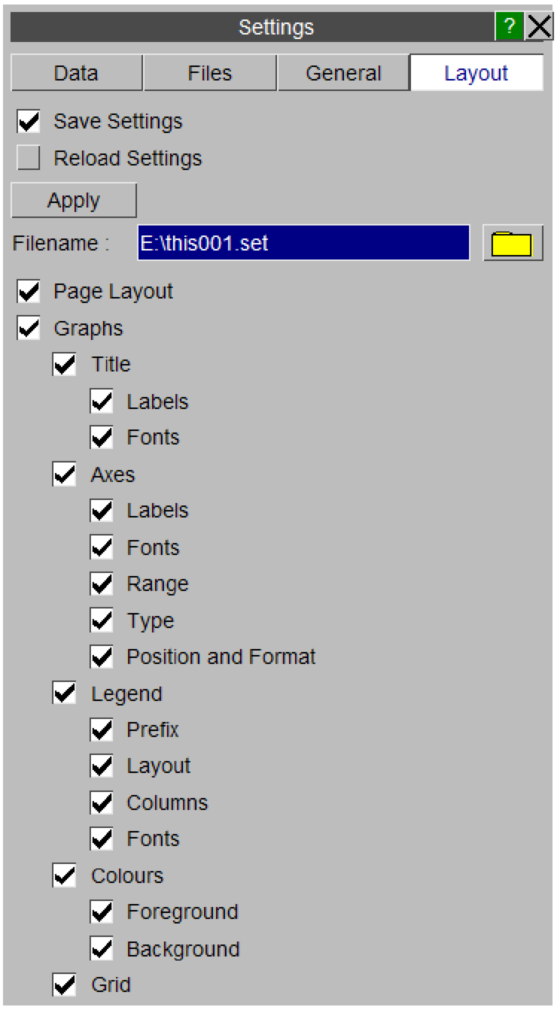
8.18.4. Layout

Layout

Save Settings

This option can be used to save a T/HIS settings file which can be reloaded later. The settings file uses the same syntax as a FAST-TCF script except it only contains **layout** and **setup** commands.

The settings file can contain all of the commands required to reproduce the current page and graph layout or a subset of the commands.



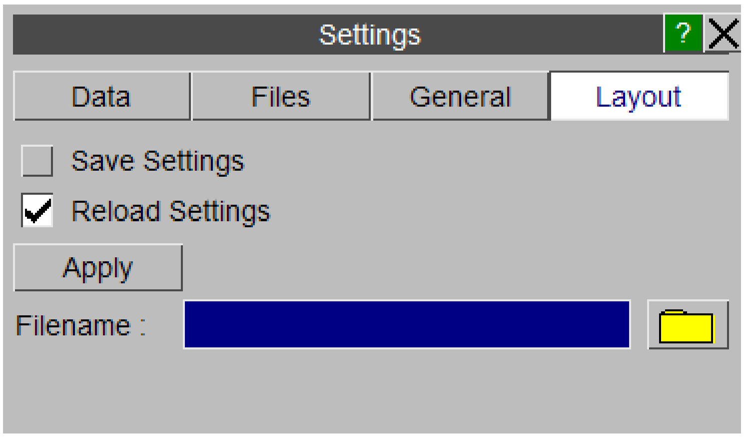
Reload Settings

This option can be used to reload a previously saved settings file. In addition to reloading a file interactively a settings file can also be specified on the command line

```
-set=<filename>
```

or via the [Preference File]

```
this*settings_file: <filename>
```



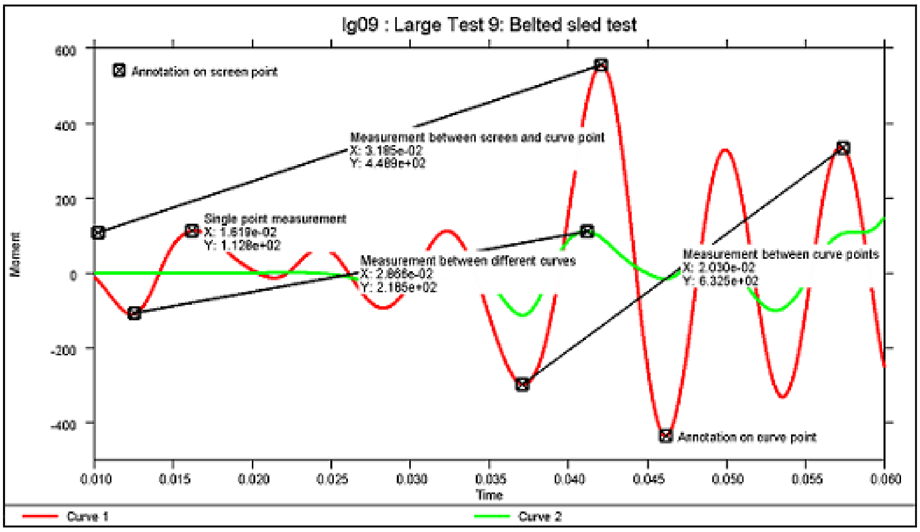
8.19. MEASURE

MEASURE

This menu can be used to make measurements between curve points and/or screen points. It can also be used to annotate graphs.

Each graph can contain multiple measurements and annotations.

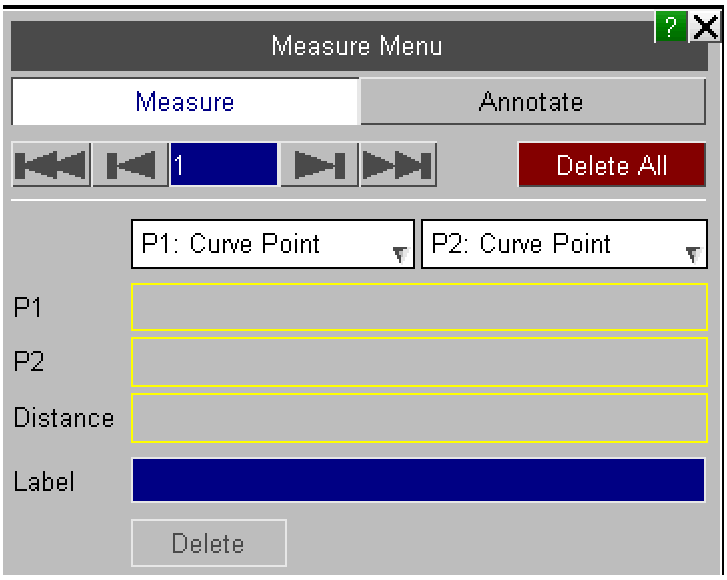
Measurements can be made between curve and/or screen points and can be made between different curves. Single points can be measured too.



8.19.1. Measure Menu

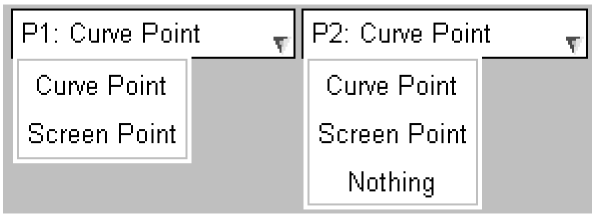
Measure Menu

Use this option to pick points on the graph to measure between.



Point Types

Use the popups to select the point type to measure to/from.



Label

If you specify a label this will be displayed on the measurement.

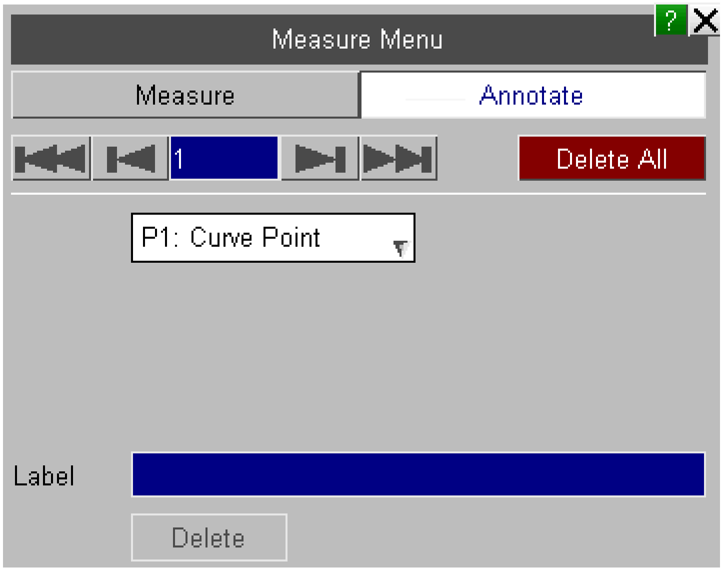
Delete

This will delete the current measurement.

8.19.2. Annotate

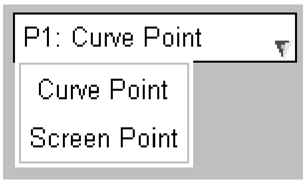
Annotate

Use this option to make annotations on the graph.



Point Type

Use the popup to select the point type to annotate on.



Label

This is the annotation that will be displayed on the graph.

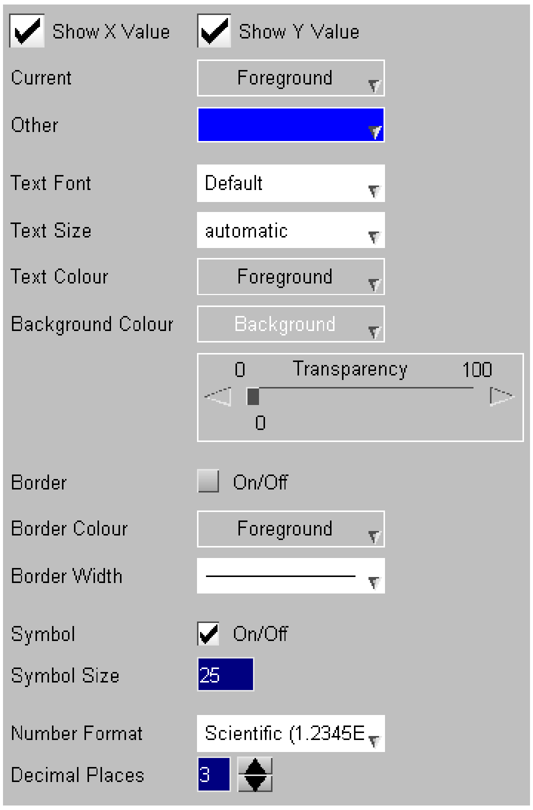
Delete

This will delete the current annotation.

8.19.3. Format

Format

These options can be used to control the display of the measurements and annotations.



Values

Measurements can be displayed with both the X and Y values, just the X value, just the Y value or neither.

If only one of the values is shown the line between the two points will be drawn like so:

Text

The font, font size and colour of the values can be selected.

Background

To make it easier to read the values a background can also be specified. In addition to specifying the background colour a transparency value can be used to control the visibility of curves under the text.

Border and Border Colour

Specify a border and border colour to be added around the value.

Symbols

The symbols drawn on the measurement points can be turned on/off. The size of the symbol can also be specified.

Number Format

Specify the format of the values displayed on the graph.

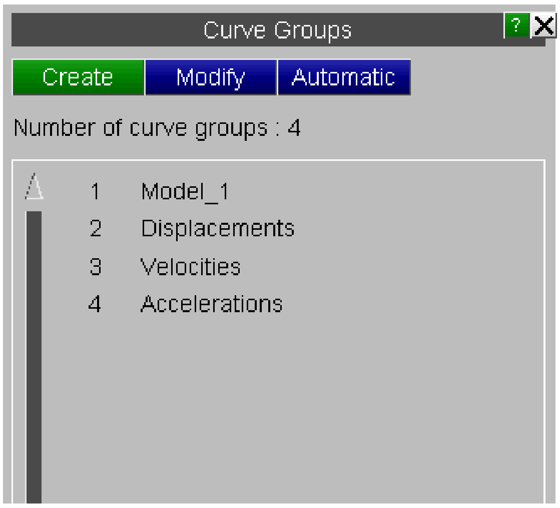
8.20. Curve Groups

Curve Groups

This panel can be used to create and modify curve groups. T/HIS can contain an unlimited number of curve groups each of which can contain any curve.

Curve groups can be used as input to most T/HIS functions that require one or more input curves (see Selecting Curves for more details)

Each curve group should be given a unique name.

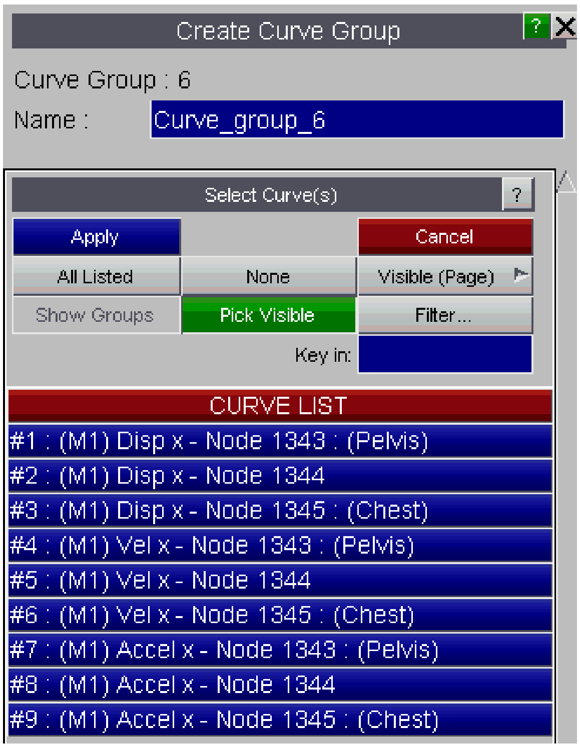


8.20.1. Create

Create

This option can be used to create a new curve group.

By default the group will be called "Curve_group_#" where "#" is the curve group number if an alternate name is not specified.



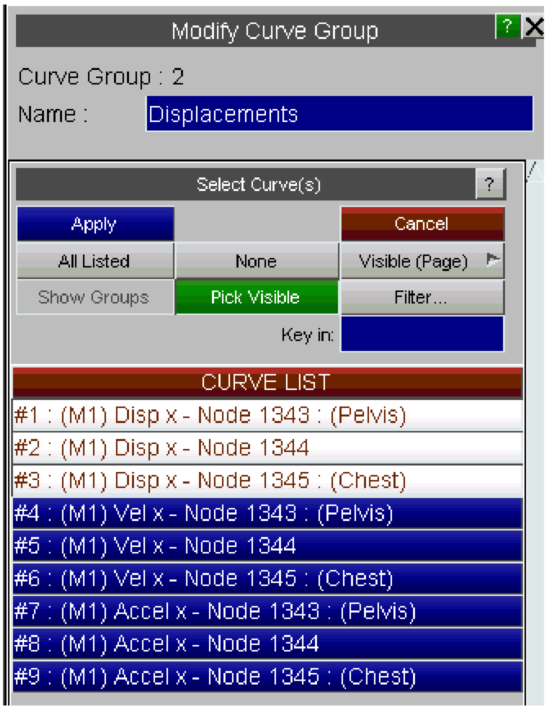
T/HIS

8.20.2. Modify

Modify

This option can be used to modify the contents of an existing curve group or its name.

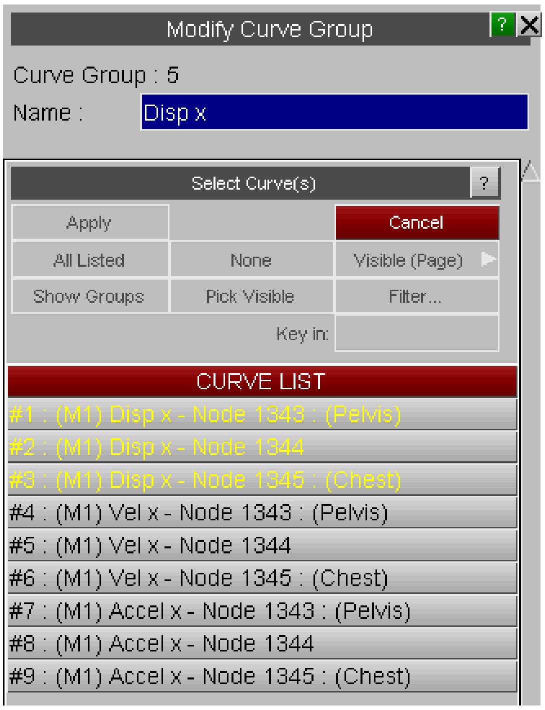
When a curve group is selected any curves that are already defined in the group are highlighted in the curve list.



The contents of Automatic curves groups can not be modified as T/HIS automatically adds and removes curves from automatic groups.

Curves that belong to an automatic curve group are highlighted in yellow.

The name of an automatic curve group can be changed.

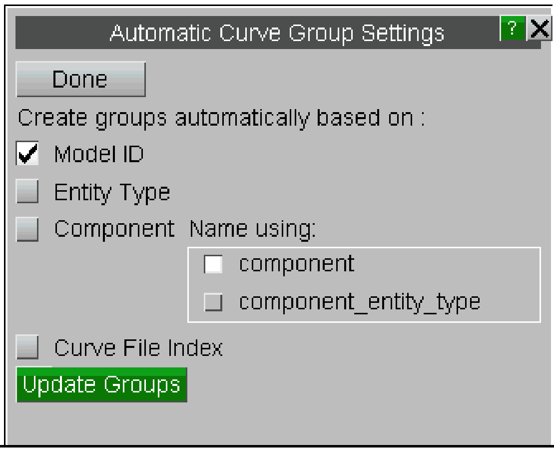

Modify Curve Group
Curve Group : 5
Name :
Disp x
Select Curve(s)
Apply
Cancel
All Listed
None
Visible (Page)
Show Groups
Pick Visible
Filter...
Key in:
CURVE LIST
#1 : (M1) Disp x - Node 1343 : (Pelvis)
#2 : (M1) Disp x - Node 1344
#3 : (M1) Disp x - Node 1345 : (Chest)
#4 : (M1) Vel x - Node 1343 : (Pelvis)
#5 : (M1) Vel x - Node 1344
#6 : (M1) Vel x - Node 1345 : (Chest)
#7 : (M1) Accel x - Node 1343 : (Pelvis)
#8 : (M1) Accel x - Node 1344
#9 : (M1) Accel x - Node 1345 : (Chest)

8.20.3. Automatic

Automatic

By default T/HIS will automatically create a curve group for each model that is read in and will add any curves read in from that model into the curve group.

This option can be used to create other "automatic" curve groups.



Model ID	The default - one group is created for each model.
Entity Type	This option will create one group for each Entity type (Modal, Node, Solid etc) that data is read from. If data is read from multiple models then a single group for each entity type will be created containing curves from multiple models.
Component Name	This option will create one group for each component (Node X displacement, Contact X Force etc), that data is read from. If data is read from multiple models then a single group for each component will be created containing curves from multiple models. The component groups can be named using either the component name (Disp X, Vel X...) or the component name and the entity type (Disp X - Node, Vel X - Node)
Curve File Index	If curves are read in from curve files (.CUR or CSV) then this option will create one group for the 1st curves read form each file, a second group for the 2nd curve read from each file and so on.

Multiple options can be selected at the same time.

Update Groups	This option will create and update the contents of automatic curve groups if the options are changed.

The following preference options can be used to change the default options, (see Format of the oa_pref File for more details).

```
group_by_model
group_by_type
group_by_component
group_by_file_index
component_group_name
```

8.21. Graphs

GRAPHS

This panel can be used to create additional graphs within T/HIS.

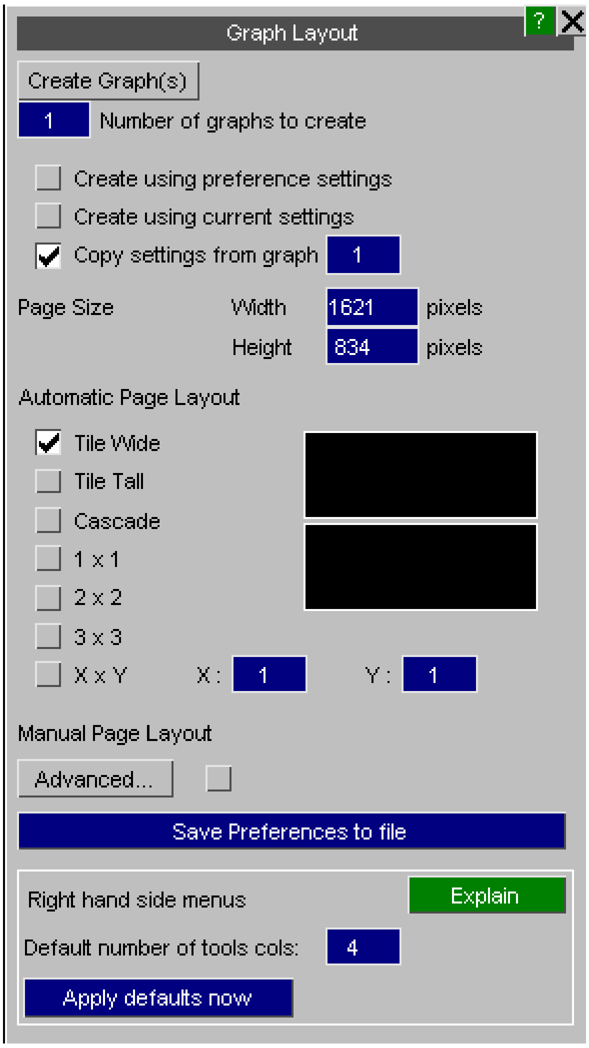
In addition to creating graphs this menu can also be used to control the layout of the graphs and to set up pages of graphs within T/HIS.

See Graphs and Pages for more details.

8.21.1. Save Preferences to File

Save Preferences to File

Launches a popup to quickly save preferences to the oa_pref file. See Save Preferences Popup.

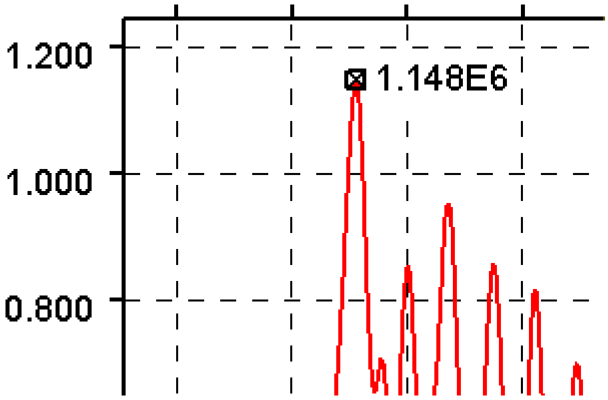


8.22. PROPERTIES

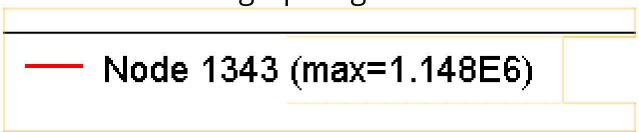
PROPERTIES

This menu can be used to display additional curves properties.

Minimum and maximum curve values can be highlighted for each curve and the value can also be displayed.



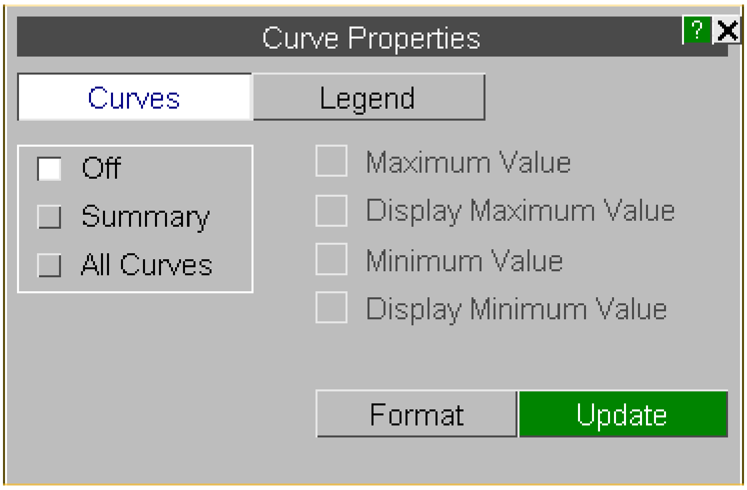
In addition to displaying the value on the curve the values can also be added automatically to the curve label in the graph legend.



8.22.1. Curves (Off)

Curves (Off)

This option will turn off the display of all minimum and maximum values.



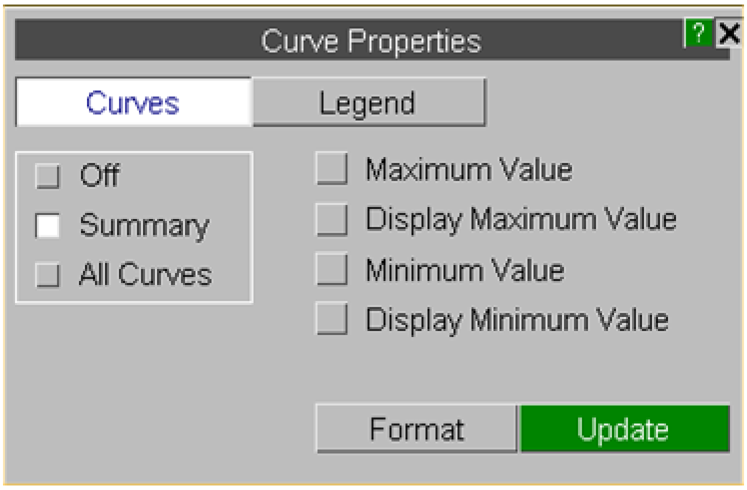
8.22.2. Curves (Summary)

Curves (Summary)

This option will display a single minimum/maximum value from all curves currently displayed..

The following properties can be displayed

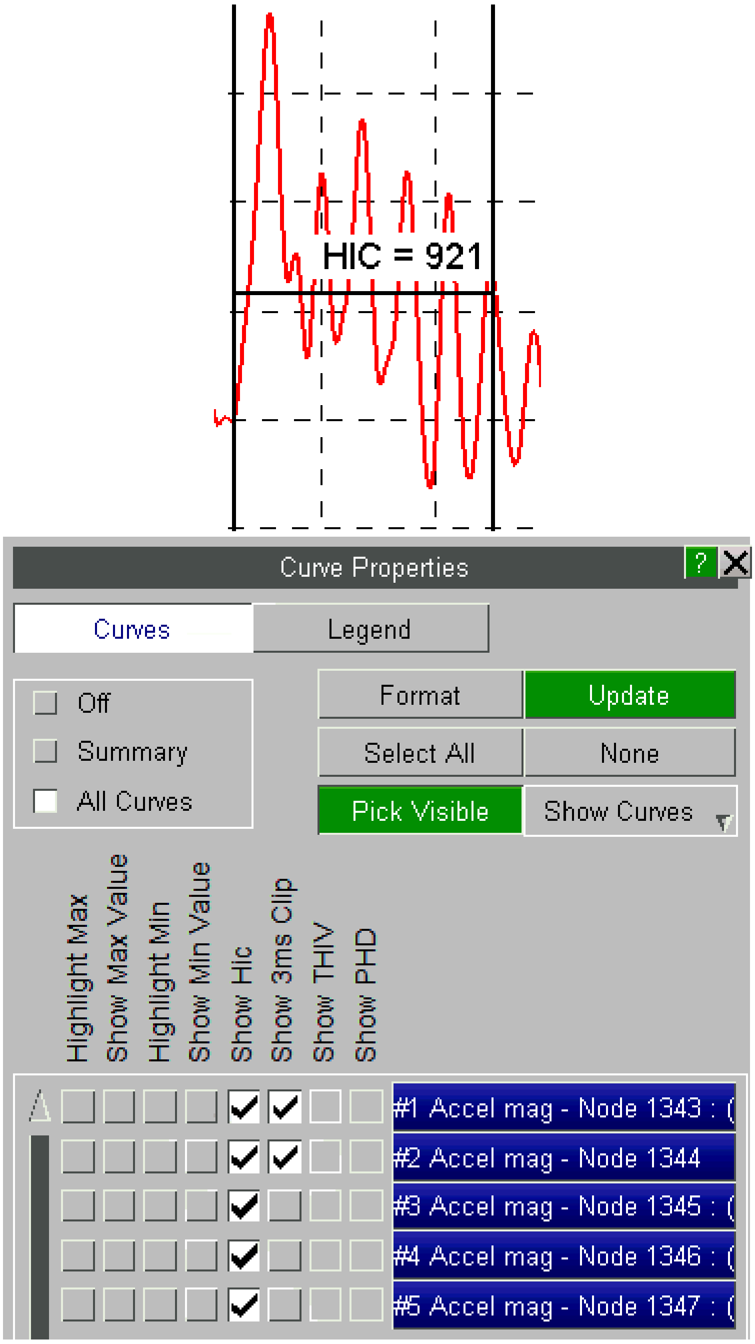
Maximum value	Mark the maximum value with a cross
Display Maximum	Display the maximum value
Minimum value	Mark the minimum value with a cross
Display Minimum	Display the minimum value



Curves (All curves)

This option can be used to select the properties that are displayed for each individual curve.

When this option is selected the display of injury criteria (HIC,HICd etc) for curves can also be selected.


HIC = 921
Curve Properties
Curves
Legend
Off
Summary
All Curves
Format
Update
Select All
None
Pick Visible
Show Curves
Highlight Max
Show Max Value
Highlight Min
Show Min Value
Show Hic
Show 3ms Clip
Show THIV
Show PHD
#1 Accel mag - Node 1343 : (
#2 Accel mag - Node 1344
#3 Accel mag - Node 1345 : (
#4 Accel mag - Node 1346 : (
#5 Accel mag - Node 1347 : (

8.22.3. Format

Format

This option can be used to control the display of the minimum/maximum values on the screen.

Text

The font, font size and colour of the values can be selected. Either a single colour can be used for all the values or the values for each curve can be coloured using the same colour as the curve.

Background

To make it easier to read the values a background can also be specified. In addition to specifying the background colour a transparency value can be used to control the visibility of curves under the text.

Border and Border Colour

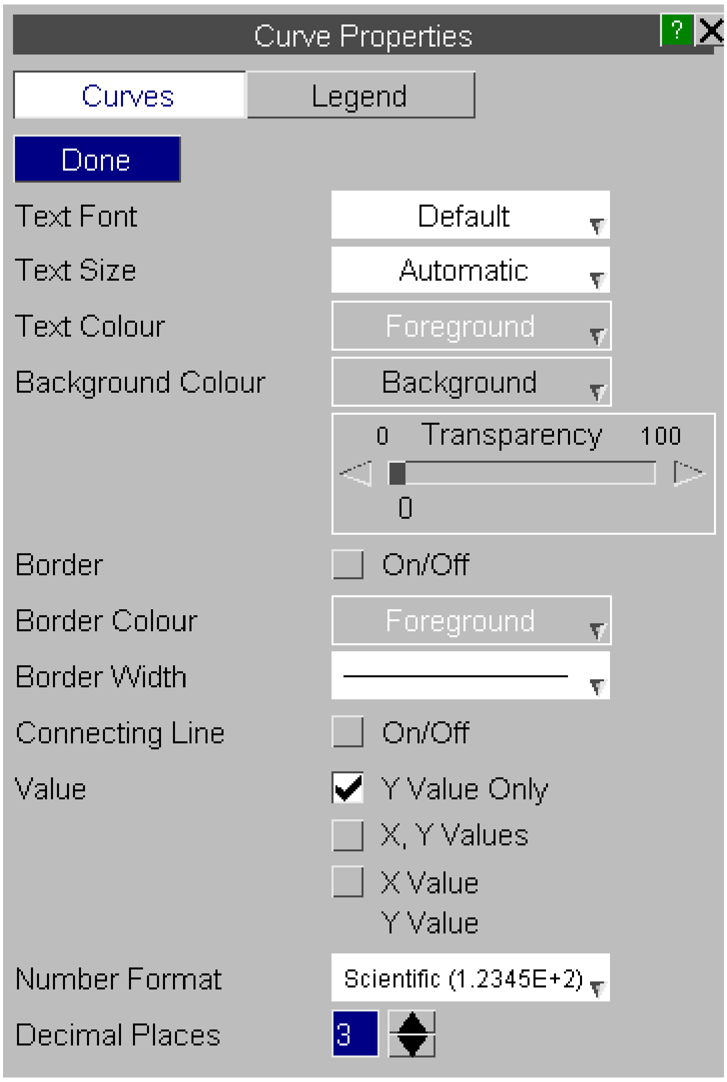
Specify a border and border colour to be added around the value.

Connecting Line

This option will draw a line connecting the value with the point it relates to on the curve. The connecting line is drawn using the same colour as the border.

Value

The values can be displayed showing just the Y axis value or with both the X and Y axis values. If both values are displayed they can either be displayed separated by a comma or one above the other.



Number Format

The values can be displayed using 3 different formats

Automatic	Values are displayed using exponential format, all values are displayed as values of E0, E3, E6 etc. e.g 11.234E+03
Scientific	Values are displayed using exponential format. e.g 1.123E+04
General	Values are displayed as real numbers. e.g 11234.000

Decimal Places

In addition to specifying the format, the number of decimal places can also be set between 0 and 9.

8.22.4. Legend

Legend

This option can be used to automatically added curve properties to the curve labels in the legend area.

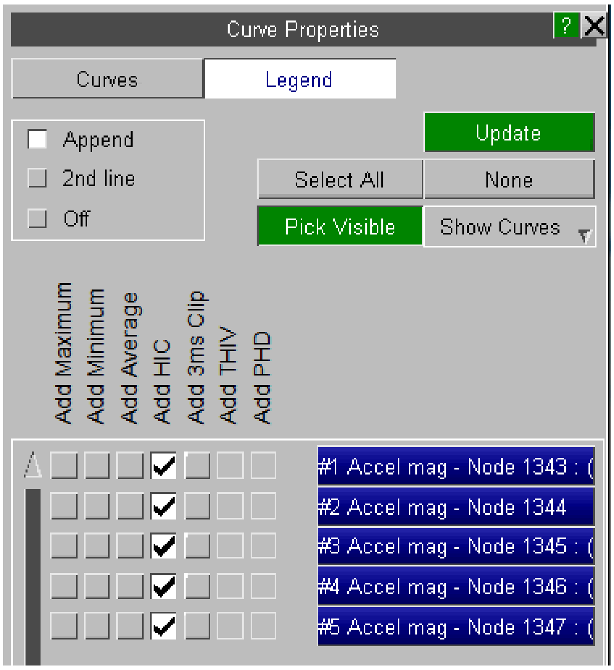
The following curve properties can be added to each curve label

Maximum value
Minimum value
Average value
Injury Criteria (HIC, HICd etc)

Other options

Off	Turns off the display of curve properties in the legend
Append	Add the values to the same line as the curve labels in the legend
2nd Line	Display the values using a second line for each curve in the legend

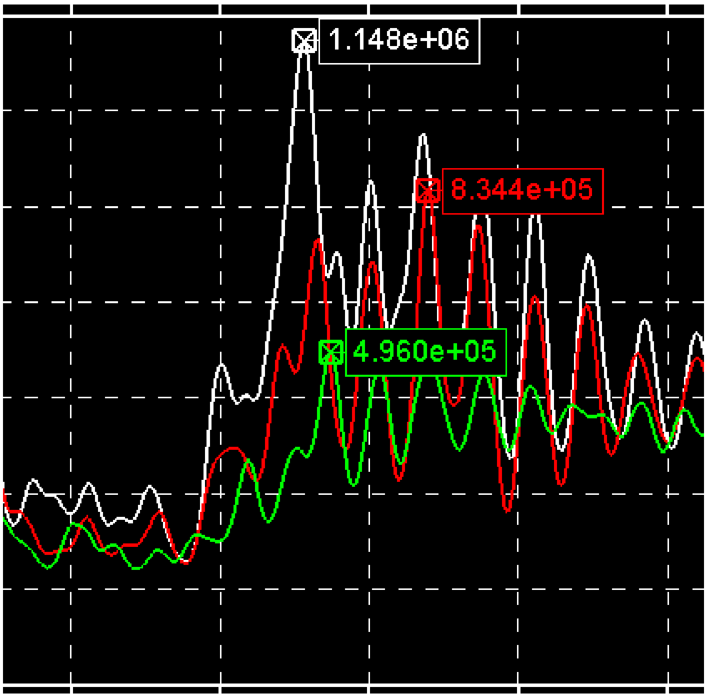
The format of the numbers added to the curve labels is the same as that used to display values on the curves.



8.22.5. Positioning Values

Positioning Values

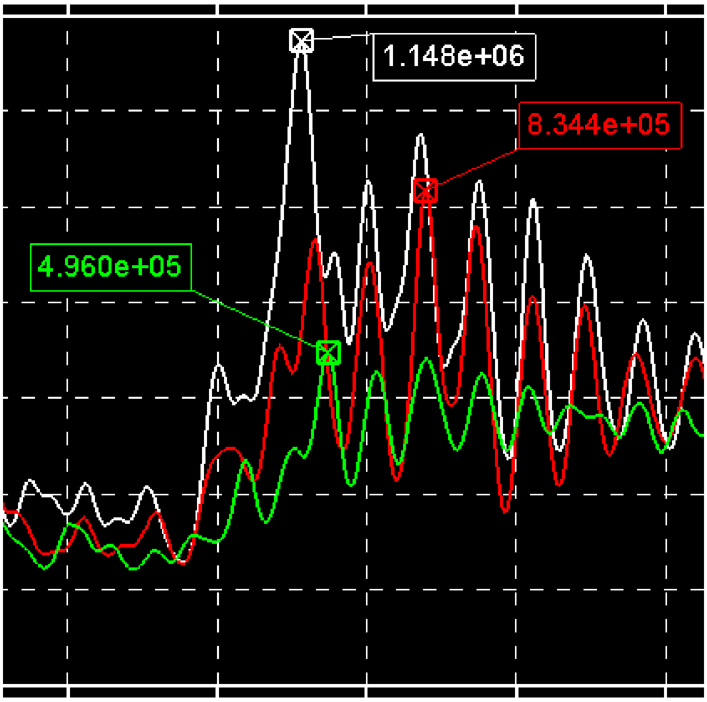
By default T/HIS will automatically position minimum and maximum values to the right of the point they apply to.



If the default location of the text obscures other curves then the position can be changed by clicking on the value with the left mouse button and then dragging the value to a new position.

If dynamic viewing is used to either zoom in or translate the curves after a value has been moved to a new position then it will maintain it's new position relative to the minimum/maximum value location.

As well as moving the minimum/maximum values the values used to display injury criteria like HIC and HIC(d) can also be moved.


1.148e+06
8.344e+05
4.960e+05

8.23. UNITS

UNITS

From version 9.4 onwards, T/HIS tries to keep track of the units for each curve's X and Y axis. For every data component that T/HIS can read from an LS-DYNA results file, one of the following basic units is stored for the curve's X and Y axis.

Time	**Rotation**	**Momentum**	**Energy Density**
Energy	**Rotational Velocity**	**Density**	**Mass Flow**
Work	**Rotational Acceleration**	**Stress**	**Frequency**
Temperature	**Length**	**Strain**	**Power**
Displacement	**Area**	**Force**	**Thermal Flux**
Velocity	**Volume**	**Moment**	**Force per unit width**
Acceleration	**Mass**	**Pressure**	**Moment per unit width**
Viscosity	**Thermal Diffusivity**	**Vorticity**	**Q Criterion**
Current	**Vector Potential**	**Magnetic Flux Vector**	**Electric Field Vector**
Conductivity			

When a curve operation is carried out on curve which has either the X or Y axis unit defined, the units for the output curve(s) are also calculated. If a curve operation is carried out using 2 or more input curves with different units and the result is a curve with inconsistent units, then the units are set to zero.

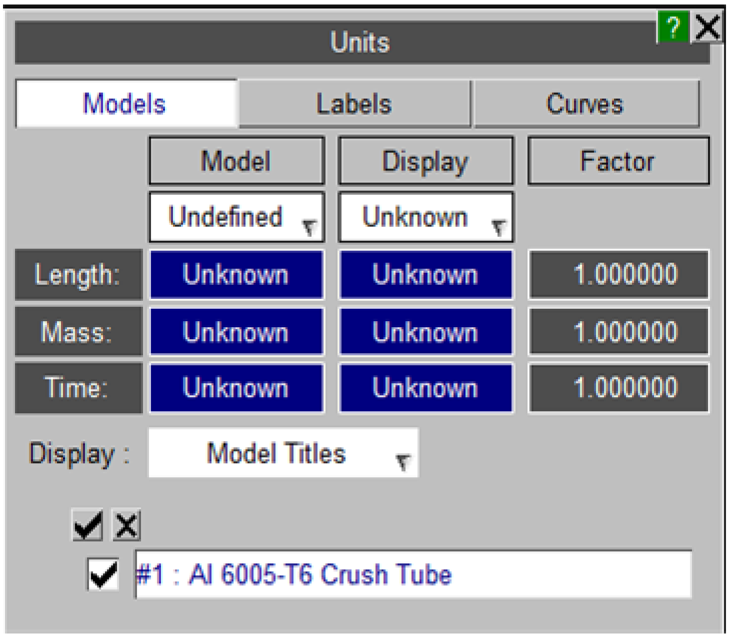
If one of the inputs is a constant then it assumed to be unitless.

Input 1	Input 2	Operation	Output
Velocity (m/s)	Velocity (m/s)	Add	Velocity (m/s)
Velocity (m/s)	Displacement (m)	Add	Unknown
Velocity (m/s)	Velocity (m/s)	Divide	Constant
Velocity (m/s)	Displacement (m)	Divide	Frequency (1/s)
Velocity (m/s)	Constant	Add	Velocity (m/s)
Velocity (m/s)	Constant	Divide	Velocity (m/s)
Velocity (m/s)	-	Differentiate	Acceleration (m/s^2)

8.23.1. Models

Models

By keeping track of the X and Y axis units for each curve T/HIS can now convert results from one unit system to another.

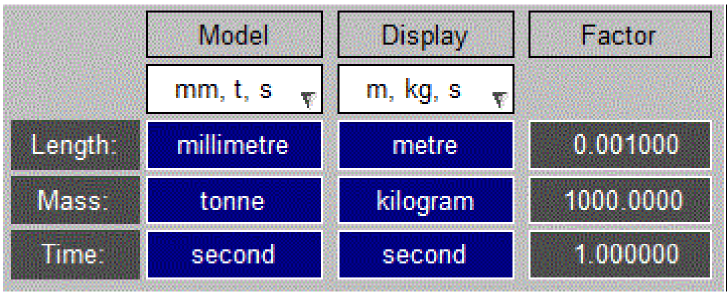


For each model one of the following 6 unit systems can be defined.

Name	Units
U1	metre, kilogram, second (SI)
U2	millimetre, tonne, second
U3	millimetre, kilogram, millisecond
U4	millimetre, gram, millisecond
U5	foot, slug, second
U6	metre, tonne, second

In addition to specifying a unit system for each model a separate unit system can also be selected to use to display results.

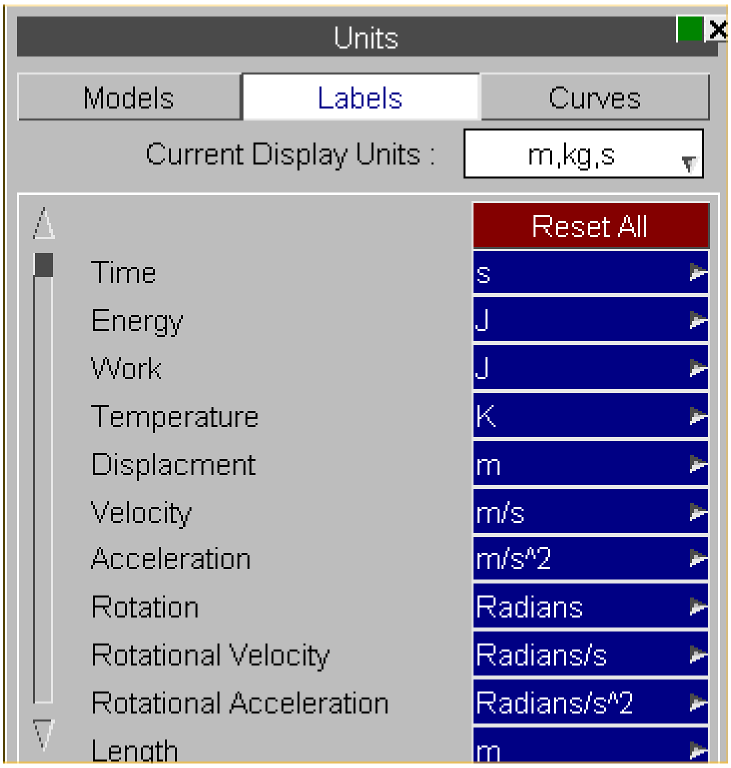
If the model unit system and the display unit system are different then T/HIS will automatically calculate the correct factors to apply to the X and Y axis as the curve data is read from the file (All curves are stored inside T/HIS using the currently defined Display unit system).


Model
Display
Factor
mm, t, s
m, kg, s
Length:
millimetre
metre
0.001000
Mass:
tonne
kilogram
1000.0000
Time:
second
second
1.000000

8.23.2. Labels

Labels

This option will display the labels that will be used for each of the built-in units. Each Unit System has its own set of labels which can be modified if required.



The default labels for each unit system are shown below.

	U1: m,kg,s	U2: mm,T,s	U3: mm,kg,ms	U4: mm,gm,ms	U5: ft,slug,s	U6: m,T,s
Time	s	s	ms	ms	s	s
Energy	J	mJ	J	mJ	ft lbf	kJ
Work	J	mJ	J	mJ	ft lbf	kJ
Temperature	K	K	K	K	K	K
Displacement	m	mm	mm	mm	ft	m
Velocity	m/s	mm/s	mm/ms	mm/ms	ft/s	m/s
Acceleration	m/s^2	mm/s^2	mm/ms^2	mm/ms^2	ft/s^2	m/s^2
Rotation	Radians	Radians	Radians	Radians	Radians	Radians
Rotational Velocity	Radians/s	Radians/s	Radians/s	Radians/s	Radians/s	Radians/s
Rotational Acceleration	Radians/s^2	Radians/s^2	Radians/s^2	Radians/s^2	Radians/s^2	Radians/s^2
Length	m	mm	mm	mm	ft	m

Area	m^2	mm^2	mm^2	mm^2	sq ft	m^2
Volume	m^3	mm^3	mm^3	mm^3	cu ft	m^3
Mass	kg	T	kg	gm	slug	T
Momentum	kg m/s	T mm/s	kg mm/ms	gm mm/ms	ft slug/s	T m/s
Density	kg/m^3	T/mm^3	kg/mm^3	gm/mm^3	slug/cu ft	T/m^3
Stress	N/m^2	N/mm^2	kN/mm^2	N/mm^2	lbf/sq ft	kN/m^2
Strain	-	-	-	-	-	-
Force	N	N	kN	N	lbf	kN
Moment	Nm	Nmm	kNmm	Nmm	ft lbf	kNm
Pressure	N/m^2	N/mm^2	kN/mm^2	N/mm^2	lbf/sq ft	kN/m^2
Energy Density	J/m^3	mJ/mm^3	J/mm^3	mJ/mm^3	ft lbf/cu ft	kJ/mm^3
Mass FLow	kg/s	T/s	kg/ms	gm/ms	slug/s	T/s
Frequency	Hz	Hz	kHz	kHz	Hz	Hz
Power	W	mW	kW	W	ft lbf/s	kW
Thermal Flux	W/m^2	mW/mm^2	kW/mm^2	W/mm^2	lbf/ft	kW/m^2
Force per unit width	N/m	N/mm	kN/mm	N/mm	lbf/ft	kN/m
Moment per unit width	Nm/m	Nmm/mm	kNmm/mm	Nmm/mm	ft lbf/ft	kNm/m
Viscosity	kg/m s	T/mm s	kg/mm ms	gm/mm ms	slug/ft s	T/m s
Thermal Diffusivity	m^2/s	mm^2/s	mm^2/ms	mm^2/ms	ft^2/s	m^2/s
Vorticity	Radians/s	Radians/s	Radians/ms	Radians/ms	Radians/s	Radians/s
Q Criterion	1/s	1/s	1/ms	1/ms	1/s	1/s
Current	A	A	A	A	A	A
Vector Potential	kg m/A s^2	T mm/A s^2	kg mm/A ms^2	gm mm/A ms^2	slug ft/A s^2	T m/A s^2
Magnetic Flux Vector	kg/A s^2	T/A s^2	kg/A ms^2	gm/A ms^2	slug/A s^2	T/A s^2
Electric Field Vector	kg m/A s^3	T mm/A s^3	kg mm/A ms^3	gm mm/A ms^3	slug ft/A s^3	T m/A s^3
Conductivity	A^2 s^3/kg m^3	A^2 s^3/T mm^3	A^2 ms^3/kg mm^3	A^2 ms^3/gm mm^3	A^2 s^3/slug ft^3	A^2 s^3/T m^3

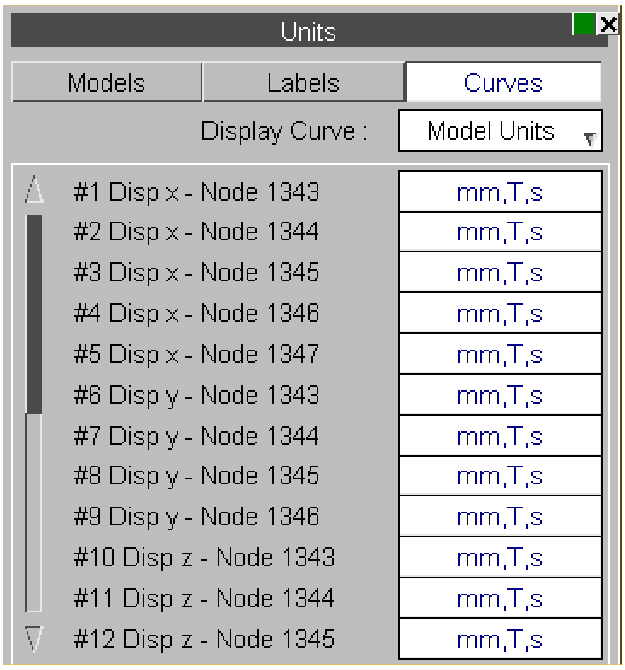
If a curve has a user defined unit or if after a curve operation one of the curve axis units is not one of the basic units that T/HIS knows about then T/HIS will build a label from the currently defined length,mass,time,temperature and angle labels.

If for example a velocity/time curve is multiplied by another velocity time curve then the Y axis will have units of Velocity^2. If the current display unit system is U1 (m,kg,seconds) then the unit label for the curves y axis will be "m^2/s^2".

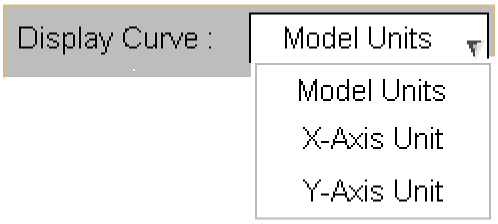
8.23.3. Curves

Curves

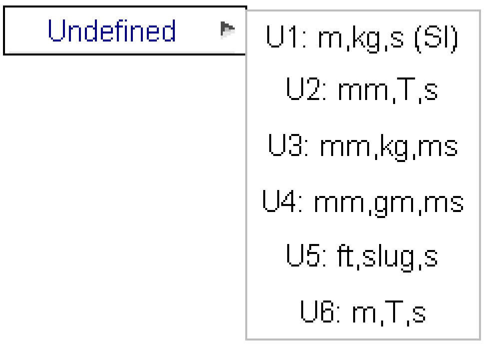
This option can be used to display the unit information for each curve.



By default, the unit system for each curve is displayed, but this can be changed to show either the X or Y axis unit using the popup menu.



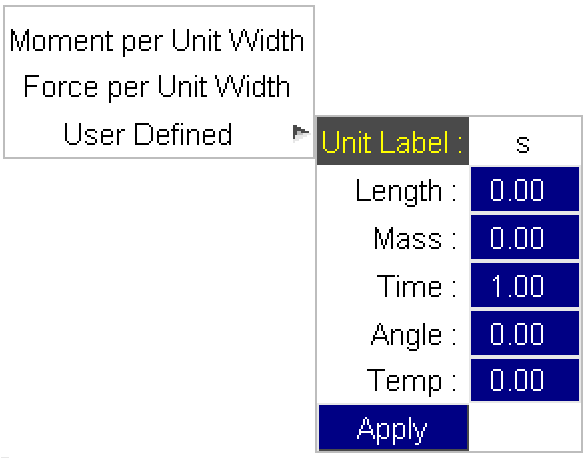
Setting the Unit System for a Curve



If the unit system for a curve has not been defined then it will be displayed as "Undefined" and a popup menu will be available that can be used to select the correct unit system. If the selected unit system is different to the unit system currently being used to display results then the curve values will automatically be converted to the current display unit system.

Note : Once the unit system for a curve has been defined it can not be changed.

Setting the Axis Units for a Curve



The X and Y axis units of a curve can be defined or changed at any time. The popup menu contains all of the basic Unit types that T/HIS knows about plus an option to setup a user defined unit.

To create a user defined unit for a curve, the unit should be defined in terms of its basic properties. The values for ***mass, length, time, angle, tempetrature*** and ***current*** should be the powers that are used to describe the unit in terms of it's fundamental dimensions.

Some examples of common units defined using this method are shown below.

Unit	Mass	Length	Time	Angle	Temperature	Current
Time	0.0	0.0	1.0	0.0	0.0	0.0
Displacement	0.0	1.0	0.0	0.0	0.0	0.0
Velocity	0.0	1.0	-1.0	0.0	0.0	0.0
Acceleration	0.0	1.0	-2.0	0.0	0.0	0.0
Stress	1.0	-1.0	-2.0	0.0	0.0	0.0

8.24. The JavaScript Interface

8.24.1. Introduction

Introduction

JavaScript is a freely available scripting language that is normally found performing the "work" behind interactive web pages, however its syntax and structure also make it an excellent tool for providing an externally programmable interface to programmes in general.

Within T/HIS it is implemented as an Application Programming Interface (API) which provides a range of functions that allow you to edit and create curves, open windows, generate plots, and so on. This is written in a very simple and non-intimidating way, with relatively few functions, that should be easy for non-programmers to use.

Anyone familiar with C or shell script programming will find existing JavaScripts are instantly readable, and can be given minor edits without further ado. For those who are more ambitious a good guide to the language is " **JavaScript, A definitive Guide** " by David Flanagan, published by O'Reilly, ISBN 0596101996.

In T/HIS 17.0 and earlier the implementation supported ECMAScript 5 features of JavaScript. In T/HIS 18.0 the implementation has been upgraded to support ECMAScript 6 (and newer) features of JavaScript.

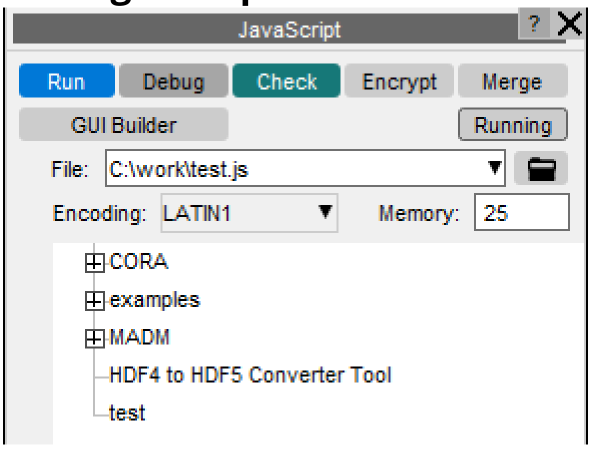
The sections below describe how to run JavaScripts in T/HIS, and summarise its JavaScript API. For details of the API and its functions, and also some examples, see the JavaScript API Reference Manual

8.24.2. Using JavaScript in T/HIS

Using JavaScript in T/HIS.

Human-readable JavaScripts need to be ***compiled*** , meaning turned from something human-readable into a set of instructions that a computer can understand; and then ***run*** in their compiled form. They can be changed and rerun in their modified form at any time without having to exit and re-enter T/HIS, making the "write, test, modify, re-test" development cycle very quick and easy.

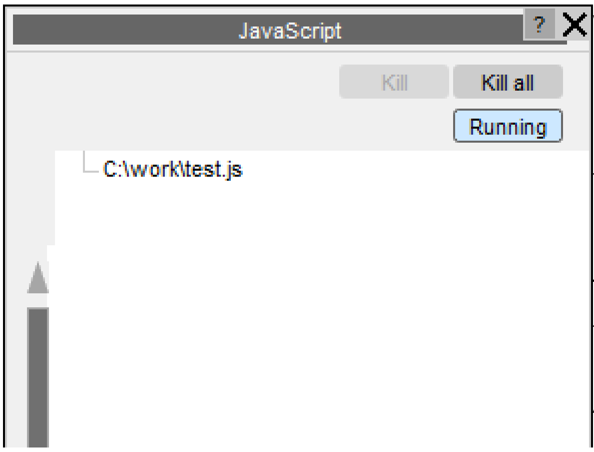
Compiling and Running a script



Run	Will both compile and run the script unless it contains syntax errors, in which case it stops with an error message when compilation fails.
Debug	Starts the JavaScript debugger, JaDe to debug the script.
Check	Only compiles the script, reporting any errors found, and does not run it.
Encrypt	A script can be encrypted so that the source code is hidden but the script can still be run (when compiling and running the script T/HIS decrypts the file in memory). Once encrypted, the source code cannot be retrieved by an ordinary user, so make sure that you keep the original file somewhere safe. As a last resort contact Oasys Ltd who can decrypt the script if required. If a script is split up into separate files by Use, the files are all combined together into the main file before encrypting.
Merge	If a script is split up into separate files by Use, the files are all combined together into a single file. This may be useful if you want to give the script to someone else and you do not want to have to give lots of different files.
GUI Builder	Opens the GUI Builder to interactively build GUIs for your script.
Running	Enables you to view (and kill) any scripts that are currently running

Memory size is the memory allocated for garbage collection in the JavaScript engine. Please see the garbage collection section for more details.

Knowing which scripts are running



When scripts are running in T/HIS, they are shown in the Running submenu.

Use the **Running** button to toggle between the list of scripts that are currently running, and the tree of available scripts.

To kill/terminate a running script, highlight the script(s) in the tree and press **Kill**.

Alternatively, to kill all the scripts that are currently running use **Kill all**.

File encodings for scripts

Version 10.0 of T/HIS introduced the ability for unicode text to be used on widgets created in a script. Previous versions of T/HIS only supported English text so the default ASCII encoding was used for script files (this is still the default encoding for script files).

If you want to use unicode text in widgets then you must use a file encoding that is capable to representing the unicode 'characters' you require. The **File encoding** popup allows you to change the file encoding used when reading the script file. T/HIS supports the following file encodings:

Encoding	Description
LATIN-1	Default 'ASCII' encoding
BIG5	Taiwan/Hong Kong (traditional)
EUC-CN	Extended unix code (Simplified Chinese)
EUC-JP	Extended unix code (Japanese)
EUC-KR	Extended unix code (Korean)
GB	Chinese (simplified)
GBK	Chinese

ISO-2022-CN	Chinese
ISO-2022-CN-EXT	Chinese (extended)
ISO-2022-JP	Japanese
ISO-2022-JP-2	Japanese (extended)
ISO-2022-KR	Korean
JOHAB	Korean
SHIFT-JIS	Japanese
UTF-8	Should NOT have a byte order mark (BOM).
UTF-16	Should have a byte order mark (BOM). If not present assumes big endian
UTF-16LE	Little endian with or without byte order mark (BOM)
UTF-16BE	Big endian with or without byte order mark (BOM)
UTF-32	Should have a byte order mark (BOM). If not present assumes big endian
UTF-32LE	Little endian with or without byte order mark (BOM)
UTF-32BE	Big endian with or without byte order mark (BOM)

Please contact Oasys Ltd if you have problems or require another encoding to be supported.

To show the unicode text the appropriate font must be used. This can be set using the preferences `this*cjk_unix_font` and `this*cjk_windows_font` .

Dealing with errors in scripts

Script errors come in two forms:

`Syntax errors`
: Are mistakes of JavaScript grammar or spelling, resulting in error messages during compilation.

  These are easy to detect and correct since the line number and offending syntax are both described by the compiler. The script needs to be edited to correct the problem and then recompiled. Sometimes several iterations of the compile/edit cycle are required to eliminate all errors from a script.

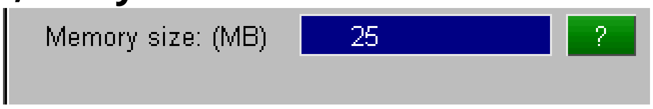
Run-time errors

Are errors of context or logic in scripts that are syntactically correct, and thus have compiled, but which fail at some stage when being run.

A typical example of a run-time error is an attempt to divide a value by zero, yielding the illegal result infinity. More subtle errors involve passing an invalid value to a function, accessing an array subscript that is out of range, and so on.

The JavaScript API Reference Manual has been written in such a way that it handles "harmless" run-time errors by issuing a warning and continuing execution, but that more serious errors which could result in the wrong answers being generated issue an error message and terminate.

Setting the Garbage Collection Threshold Size (This is an advanced topic, and you don't need to understand it.)



JavaScripts execute inside a memory "arena", allocated dynamically from the operating system, which grows in size as storage is requested within the script. This growth occurs due to requests for "new" variables within the script and also when API functions allocate and return values and objects, and it is limited only by what the operating system can deliver.

The nature of JavaScript means that objects frequently become redundant, and it is wasteful not to reuse the storage that they occupy, therefore there is a "Garbage Collection" process running behind the scenes which periodically checks storage and releases that which is no longer needed. This process is automatic and hidden from the user, it just "happens".

However Garbage Collection is quite a CPU-hungry process, so it is only carried out periodically when a certain threshold is reached. This can sometimes be observed during script execution as a periodic "pause for thought", and if you are monitoring memory usage with a system tool you may see it drop during these pauses.

Clearly this threshold value must be large enough not to trigger excessively frequent (and costly) garbage collections, while at the same time not being so large that scripts build up large amounts of excess memory to the detriment of the rest of the programme.

The **Memory size** value in the JavaScript panel is the amount of memory allocated for garbage collection. Every time a new object, array, string or double precision number is used a garbage collection 'thing' is also allocated. The Memory size is the total memory for these 'garbage collection things', **NOT** the total memory for the script. The total memory for the script could be significantly higher than this value. e.g the memory required for a Model object could be several kbytes but the memory for the 'garbage collection thing' for the Model object will something like 10 bytes for a 64bit operating system.

When the memory used for garbage collection 'things' reaches a significant proportion of **Memory Size** (normally about 2/3) then garbage collection will take place to try to reclaim memory. If no memory can be reclaimed and the total memory used for garbage collection reaches **Memory size** then the script will terminate with an error.

If your script has to retain a large number of objects, arrays, strings etc in memory then you may have to increase the value for **Memory size** . This can also be done using the `this*javascript_memory_size` preference or adding a special memory comment at the top of the script.

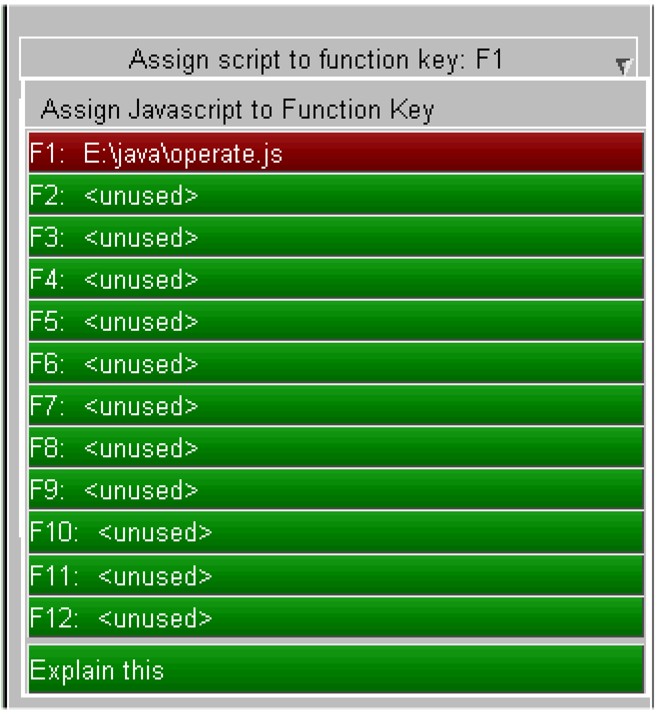
To recap:

- This threshold does ***not*** limit the memory the script can use, that is limited only by the operating system.
- It sets the memory for Garbage Collection 'objects'.
- Scripts which allocate a lot of memory, and which exhibit frequent pauses, ***may*** run faster with a larger value.
- ... and finally:

  If you don't understand this topic don't worry. Most scripts will run quite happily with the default value, and you can ignore this setting unless they appear to be struggling, in which case try raising it. (As good an approach as any is to keep on doubling this value until the script works, but don't use very large sizes unnecessarily.)

Assigning Scripts to Shortcut Keys

If a script is to be run repeatedly, it can be convenient to set up a shortcut to it. From within the JavaScript menu the script can be assigned to one of the 12 function keys. Alternatively, the JavaScript can be assigned to any key using the Shortcut menu.



Maintaining a library of JavaScripts

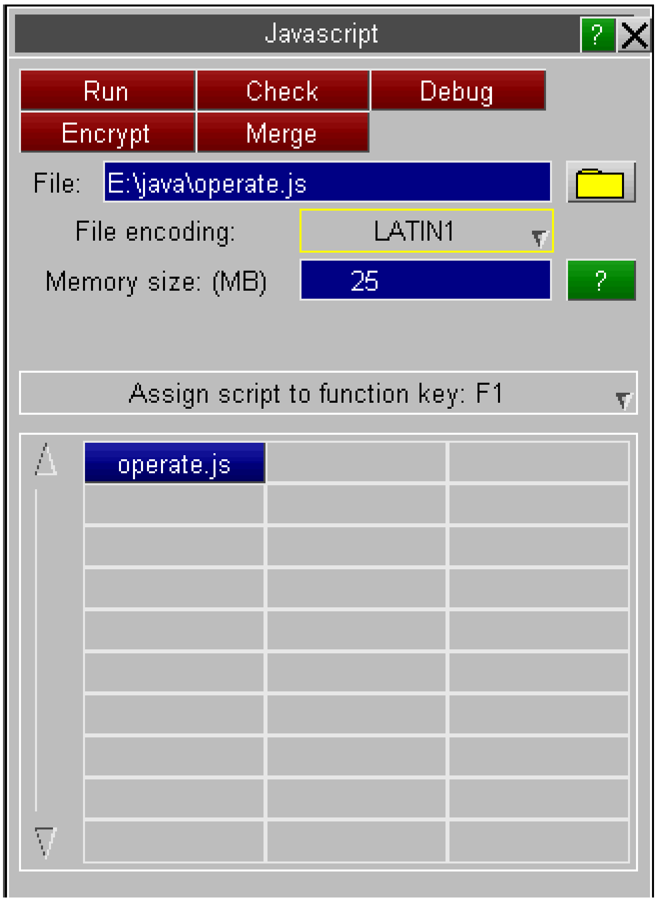
It is also convenient to have a library of scripts in a defined location.

By default T/HIS looks in **`$OA_INSTALL/this_library/scripts`** , but you can define a different directory by setting the preference:

`this*script_directory:` *`some_different_directory_name`*

in your oa_pref file.

All scripts found in the relevant directory will be listed in the JavaScript panel, as shown in this example.



Using the "description:" comment at the top of a script to identify its purpose.

To help to identify scripts special comments are searched for in the top 10 lines of each script, and if **`description:`** is found, for example the comment line:

`// description:` *`Some description of the script's purpose`*

Then the description line is shown as hover text when the mouse is placed over that filename. For example the line:

`// description: Colour curve by model number`

Will result in the hover text " **`Colour curve by model number`** " appearing when the mouse hovers over the button to launch the script.

Using the "name:" comment at the top of a script to change its name

Normally the name shown for a script will be its filename, stripped of any leading pathname and trailing ".js" extension.

However if the string **`name:`** is found in the first ten lines of the script, then the following name will be used instead. For example the line:

```
// name: Colour By Model
```

Will result in the script appearing with the name " **`Colour By Model`** " in tha JavaScript panel. This does not affect the actual name of the script, only the name on its library button.

Using the "memory:" comment at the top of a script to change the required memory

Sometimes the memory required for garbage collection needs to be changed.

If the string **`memory:`** is found in the first ten lines of the script, then the size given will be used for the memory (unless the size in the memory textbox is larger than this value). For example the line:

```
// memory: 50
```

Will result in the script using 50Mb for garbage collection memory.

Using the "encoding:" comment at the top of a script to change the encoding

By default the encoding used for scripts is LATIN1

If the string **`encoding:`** is found in a comment on the first twenty lines of the script, then the encoding will automatically be used for the script. The allowed values are **`UTF8`** or **`UTF-8`** for UTF- 8 encoding and **`ShiftJIS`** , **`Shift-JIS`** or **`sjis`** for Shift-JIS encoding.

For example the line:

```
// encoding: UTF8
```

Will result in the UTF-8 encoding being used for the script.

Using the "module:" comment at the top of a script for ES6 modules

T/HIS has to compile scripts that use ES6 modules differently to 'normal' scripts. If a script has the extension `.mjs` then T/HIS will automatically compile the script to use ES6 modules. Alternatively, if the file has a different extension, the `module` comment can be used to tell T/HIS that this file needs to be compiled to support ES6 modules.

If the string **`module: TRUE`** is found in a comment on the first twenty lines of the script, then the script will be compiled with ES6 module support.

For example the line:

```
// module: TRUE
```

Will result in the script being compiled with ES6 module support.

8.24.3. Running a JavaScript in "batch" mode

Running a JavaScript in "batch" mode.

All the above assumes that JavaScripts will be run interactively from the user interface, however it is also possible to run a script in "batch" mode using the command line interface. The relevant command-line commands are:

/JAVASCRIPT -	**+-**	**COMPILE**	Compiles and checks the script, but does not run it.
	+-	**EXECUTE**	(Re)compiles and runs the script
	+-	**MEMORY <nnn>**	Resets the Garbage Collection threshold to <nnn> MBytes

To run a JavaScript from batch these commands need to be placed in a command file and run using the command line " **-cf=** *command filename* " option. For example the command file might be:

```
... some other commands
/JAVA EXEC my_script.js
...some further commands
```

And the command line required to run T/HIS might be something like:

$OASYS/this10.exe -d=default -cf= *command_file* **-exit** *analysis_name*

Obviously multiple script invocations may be placed in a command file. For more information see:

Command and Session files	Describes command files, and explains how to create and use them
T/HIS command line arguments	Describes the various command line arguments, and how to use them

8.24.4. Running a JavaScript from within a FAST-TCF script

Running a JavaScript from within a FAST-TCF script

JavaScript scripts can also be run from within a FAST-TCF script using the "javascript" option

```
javascript "E:\javascripts\new_function.js"
```

Within a FAST-TCF script curves are usually accessed via curve tags. If a JavaScript is used within a FAST-TCF script it is recommended that the **`Curve.GetFromTag()`** function is used to access existing curves. If a new curve is created by a JavaScript within a FAST-TCF script then the new curve can be accessed within the FAST-TCF script using the "tag" parameter of the curve creation function

```
new_curve = new Curve(id,tag,label,x-axis label,y-axis label);
```

If a tag is not specified in the curve creation function

```
new_curve = new Curve(id );
```

then a curve tag will be generated automatically for the curve. The 1st curve created within the script will be tagged " **`curve_js_1`** ", the 2nd " **`curve_js_2`** " ...

8.24.5. ECMAScript 6 Modules

ECMAScript 6 modules

T/HIS 21.0 supports ES6 modules. For more information on ES6 modules, please refer to https://developer.mozilla.org/en-US/docs/Web/JavaScript/Guide/Modules.

Prior to support for ES6 modules, the only way to split up a script in T/HIS was to use the non-standard Use() functionality in the Oasys Ltd software. ES6 modules now give JavaScript built-in support for modular programming using the `import` and `export` keywords. T/HIS supports both static and dynamic imports for modules and this section gives a brief guide to how T/HIS locates modules.

To be able to support ES6 modules, T/HIS has to compile the script in a different way to a 'normal' script that does not use modules. So that T/HIS can tell how to compile the script we use a different extension `.mjs` for scripts that use modules. This follows the convention used by V8 and Node.js. Alternatively, if you prefer, you can put a special `module` comment at the top of the script and continue to use the extension `.js` (or whatever other extension you prefer).

When importing modules using import then if the module filename is an absolute filename T/HIS can locate the file directly. However if a relative filename is used T/HIS will search for the file in the following order.

- Relative to the directory that the main script is in
- Relative to any parent module directory
- Relative to the script directory specified in the OA_ADMIN directory. By default this will be $OA_ADMIN/this_library/scripts but this can be changed with the script_directory preference in the OA_ADMIN oa_pref file
- Relative to the script directory specified in the OA_INSTALL directory. By default this will be $OA_INSTALL/this_library/scripts but this can be changed with the script_directory preference in the OA_INSTALL oa_pref file
- Relative to the script directory specified in the HOME directory. By default this will be $HOME/this_library/scripts but this can be changed with the script_directory preference in the HOME oa_pref file
- Relative to the current directory
- Relative to any script_directory preference specified in a preference file given by a `-pref=xxxx` command line option.

Note that the non-standard Use() functionality and ES6 modules cannot both be used when compiling a script. You must use one or the other. Where possible you should now use ES6 modules in preference as they are now part of core JavaScript.

Individual module files can be encrypted if required so if you want to protect only some parts of your code/process and leave the rest of it open/visible this can easily be done.

One difference between using the non-standard Use() method and ES6 modules is that with the Use() method T/HIS could merge all of the individual files back into a single file using the **Merge** command which could then be encrypted if required to only have to give out a single file instead of a 'package'. For ES6 modules an external tool such as rollup.js or Webpack is required to merge the files. Once combined into a single file, T/HIS can encrypt it.

8.24.6. Scripts using GUIs

Scripts using GUIs

Scripts that create a graphical user interface (GUI) have to keep running so that the windows for the user interface remain visible. The way that this is done has changed in version 21.

All programs that have a graphical user interface (GUI) use an "event loop" to process any mouse/keyboard events. T/HIS has a main "event loop" to process all of the program's events.
In version 20 and before, if a script created and showed a window, T/HIS would start a new "event loop" to manage and process that JavaScript window.
The script would not return from the window Show() call until the window was hidden/closed.

i.e. showing the window would "block" execution of the script until the window was closed.

When the window is closed, the script continues.
When execution reaches the end of the script, the script is terminated

For example, in version 20 and earlier, in the following script, "Hello, world!" will not be printed until the window is closed because the call to `w.Show()` will not return until the window is closed.

```
// Create a window with a widget
var w = new Window("Test", 0.5, 0.6, 0.5, 0.6);
var l = new Widget(w, Widget.LABEL, 0, 50, 0, 6, "Press X to close
the window");

// Show the window and start event loop
w.Show();

// Print message
Message("Hello, world!")
```

When the window is closed the message is printed and the script will then terminate as execution has reached the end of the script.

In version 21 the behaviour has changed. If a script creates and shows a window, T/HIS will ***not*** start a new "event loop" to manage and process that JavaScript window.
The window will now be processed from the main event loop in T/HIS.

The script now returns from the window Show() as soon as the window is shown, and execution of the script continues.
i.e. showing the window no longer "blocks" execution of the script until the window is closed.

When execution reaches the end of the script, the script is ***not*** terminated.

The script continues running "in the background" as the script has shown a window.

For example, in version 21, in the same script "Hello, world!" will be printed immediately after the window is shown, because the call to `w.Show()` returns after the window is shown.

When the message is printed, the script will ***not*** terminate when execution reaches the end of the script. The script will continue to run "in the background". You can use the Running button in the script menu to see which scripts are running at any time.

If a script that shows windows continues running "in the background", and does not terminate when execution reaches the end of the script, how/when does the script terminate?

A script that uses windows ***must*** now call **`Exit`** to terminate the script

```
// Create a window with a widget
var w = new Window("Test", 0.5, 0.6, 0.5, 0.6);
var l = new Widget(w, Widget.LABEL, 0, 50, 0, 6, "Press X to close
the window");

// Exit when window closed
w.onClose = Exit;

// Show the window
w.Show();

// Print message
Message("Hello, world!");
```

If the script does not call Exit then the script will continue to run in the background, even if no windows are being shown. If necessary the Running menu can be used to terminate the script.

This change was primarily required for running multiple scripts in PRIMER, but the behaviour in T/HIS is the same. See Why the change has been made in version 21 in the PRIMER manual for details.

For simple scripts, the only change that should be required is to make sure that the script exits when the main window is closed by calling `Exit()`. The Window onClose event can be used to do this.

If you have specifically relied on script execution "blocking" when the Window `Show` method is used, then your script may need to be slightly modified. Please contact our support team if you have any questions or need assistance.

8.24.7. Examples

Examples

By far the easiest way to learn JavaScript is by example and, more specifically by modifing existing scripts to do what you want.

The software comes supplied with examples in the **$OASYS/** *programme_* **library/examples** directory (for T/HIS **$OASYS/this_library/examples**) and you are free to use and modify these files for your own purposes.

8.24.8. MADM Correlation Tool

MADM Correlation Tool

Included in T/HIS as a JavaScript is the MADM Correlation Tool. The minimum area discrepancy method (MADM) is ideal for correlation between LS-DYNA simulations and physical tests when force versus deflection is the relationship of interest, and offers benefits over other correlation methods that focus on parameters versus time. To run the tool, open the JavaScript panel, and select **MADM → MADM Correlation Tool**. For more details, see Appendix F – MADM Correlation tool.

8.24.9. The T/HIS JavaScript API

The T/HIS JavaScript API

The API is documented in the JavaScript API Reference Manual

The global class section lists all the functions available in the global scope. To make them easier to locate, they are also presented in different categories in the tree on the left hand side.

8.24.10. CORA (CORrelation and Analysis) Tool

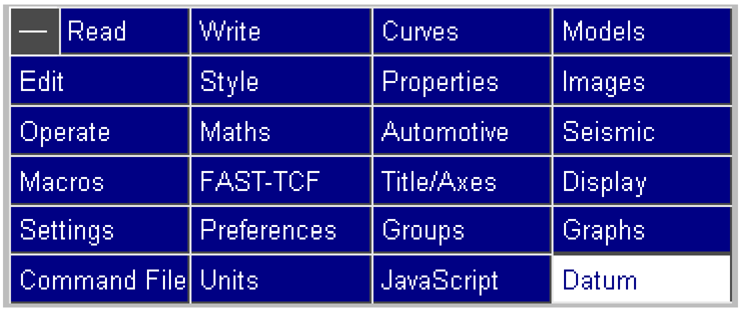
CORA (CORrelation and Analysis) Tool

Included in T/HIS as a JavaScript is the CORA (**COR**relation and **A**nalysis) tool, an implementation of the methodology used by the Partnership for Dummy Technology and Biomechanics (PDB) software CORA. To run the tool, open the JavaScript panel, and select **CORA → CORA (CORrelation and Analysis)**. For more details, see Appendix F – CORA implementation.

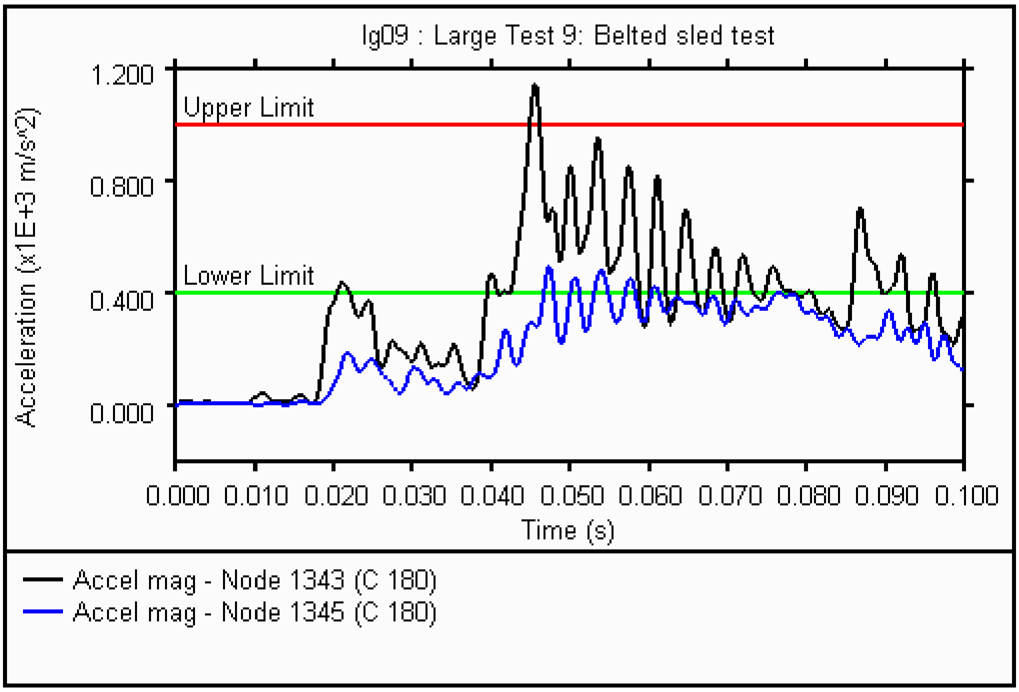
8.25. Datum Lines

Datum Lines

Datum lines can be added to graphs to show limits and reference curves. Unlike normal curves DATUM lines are not used to calculate graph limits when auto scaling and are not shown in the curve legend.



Each graph can contain multiple DATUM lines, all DATUM lines are drawn in the order they have been defined before any curves are plotted.



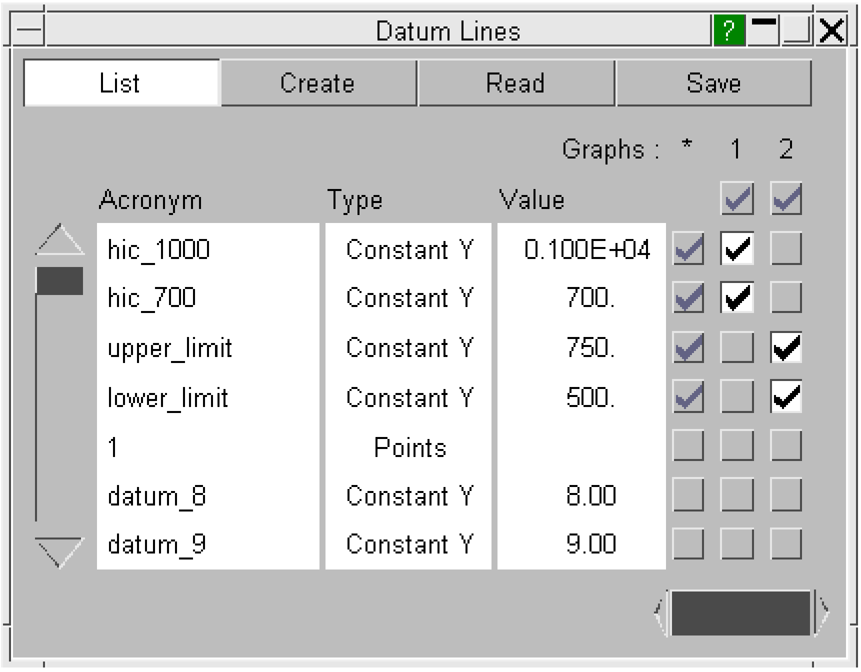
8.25.1. List

List

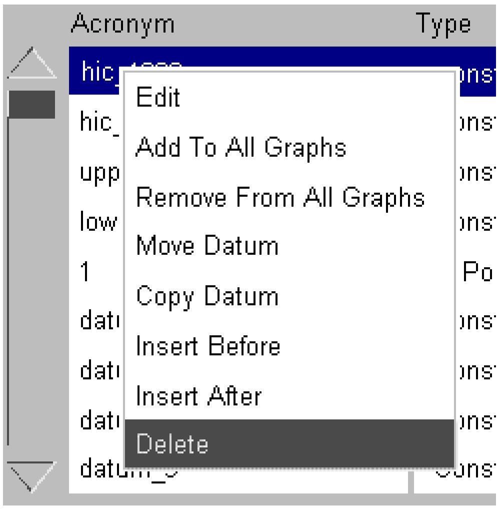
This option lists any DATUM line definitions that have been created.

This menu can also be used to select which DATUM lines appear on each graph. Each DATUM line can appear on more than one graph.

A range of DATUM lines can be added/removed from graphs by selecting the first line/graph combination and then holding down SHIFT while selecting the second line/graph.



Clicking on any of the DATUM line definitions will highlight it in blue and display a popup menu containing the following options.



Edit	Edit the selected DATUM line definition. This option will display the CREATE/EDIT menu.
Add to All Graphs	Add the selected DATUM line definition to all the currently defined graphs
Remove From All Graphs	Remove the selected DATUM line definition from all the currently defined graphs
Move Datum	Make a copy of the selected DATUM line, the original definition will be deleted when the copy in inserted.
Copy Datum	Make a copy of the selected DATUM line.
Insert Before	Insert the previously copied/moved DATUM line definition before the selected DATUM line.
Insert After	Insert the previously copied/moved DATUM line definition after selected DATUM line.
Delete	This will delete the selected DATUM line.

8.25.2. Create/Edit

Create/Edit

Each DATUM line must be defined with a unique acronym that is used to identify it in FAST-TCF scripts. The acronym shouldn't contain any spaces.

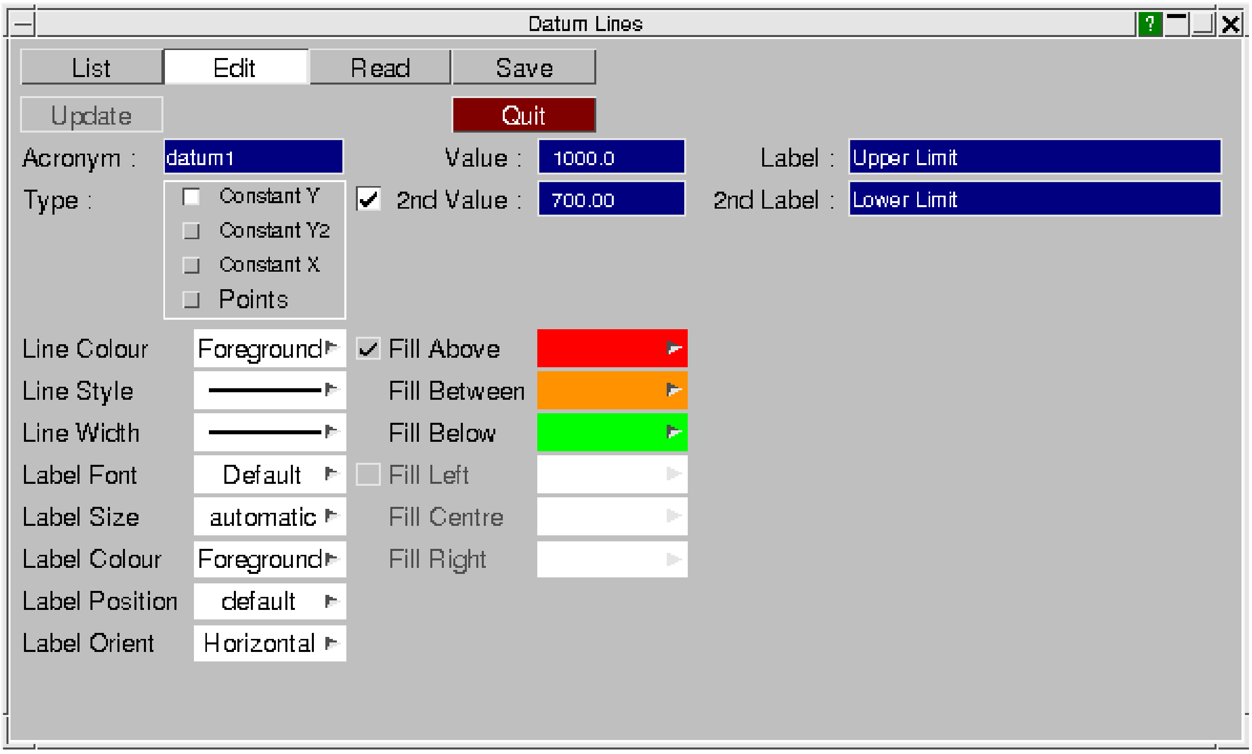
An optional label that is displayed on the graph next to the DATUM line can also be defined. The font, size and colour for the label can be defined, as well as the orientation and position of the label relative to the DATUM line.

DATUM lines can be defined as

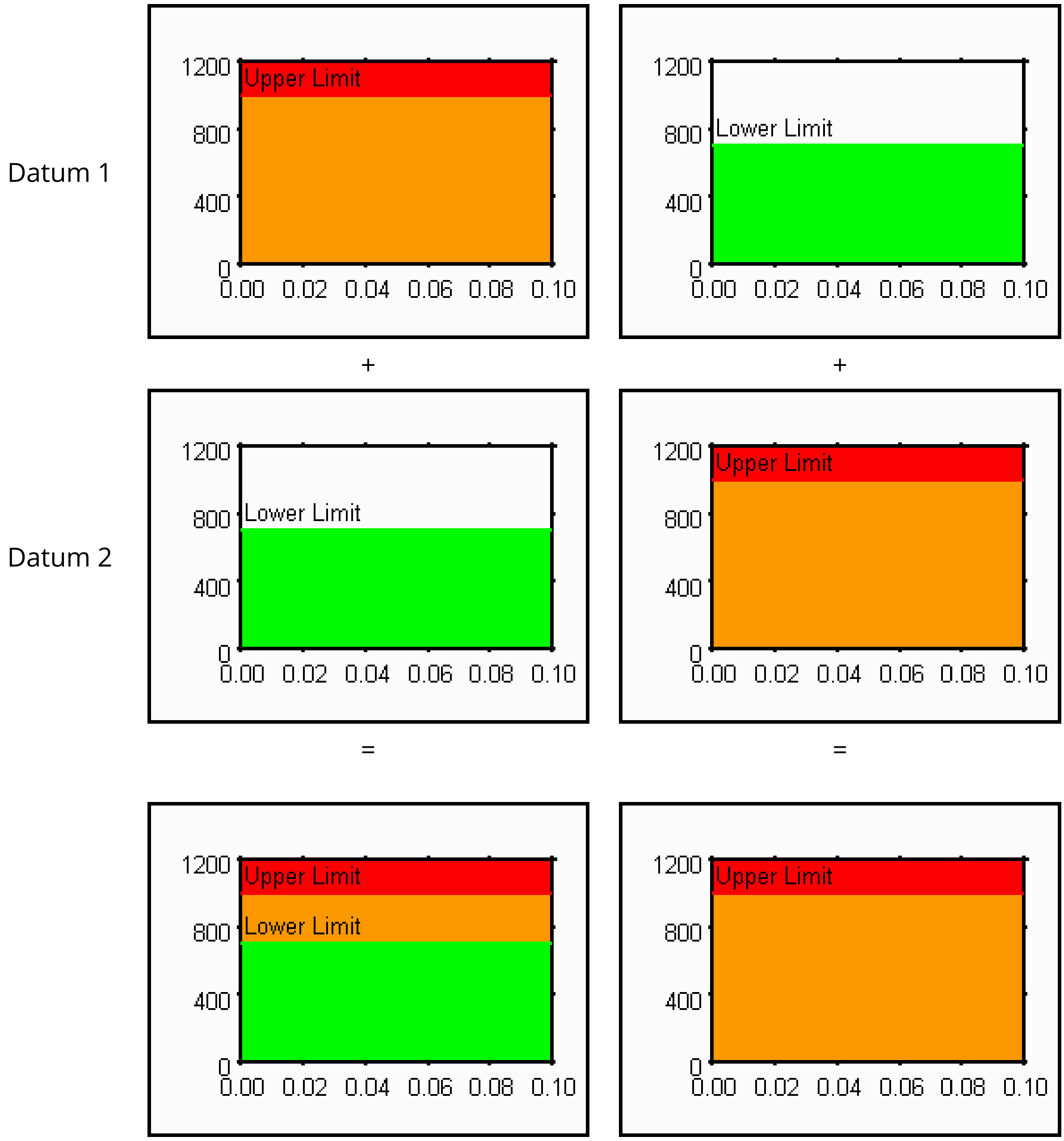
- Constant Y values
- Constant Y2 values
- Constant X values
- Curves of X,Y points

For constant X, Y or Y2 DATUMs, the line will automatically extend to the edges of the graph and the areas either side of the line can be filled using any of the standard T/HIS colours.

For constant DATUMs, an optional second value can be defined, along with a corresponding second label. A third fill colour can be used to fill in between the two DATUM lines, reducing the need for multiple DATUMs which rely on being drawn in the correct order.

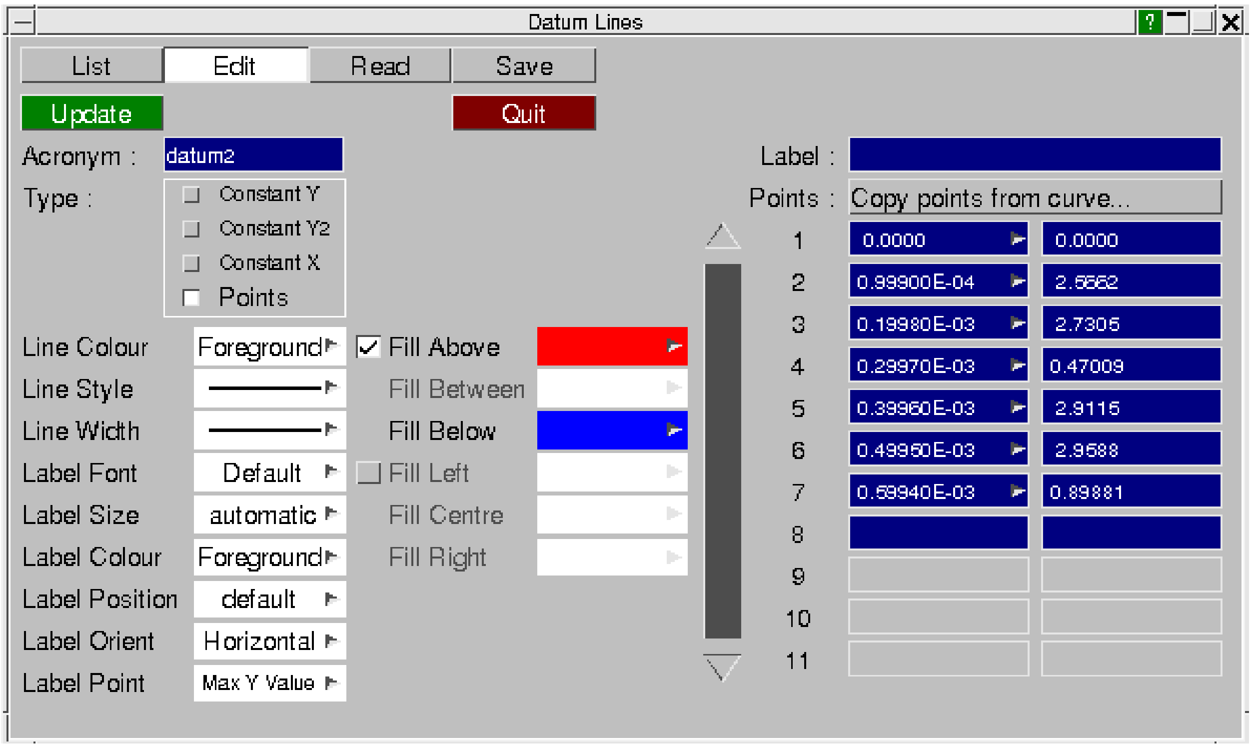


As the DATUM lines are drawn in the order they are defined, care must be taken when applying fill colours. The following example shows the effect of defining the DATUMs in a different order. Using the optional second value, the following example can actually be made into a single DATUM with two values, two labels and three fill colours, as in the above image.



In order to define a DATUM using X,Y points, either each point can be manually added or the points can be copied into the DATUM from a curve, using the **Copy points from curve...** button. This will open a list of curves and allow one to be selected or picked on the screen and the option to copy the curve label is also given. The areas between the curve and the axes can be filled, either above and below or left and right. Note that

DATUM curves are always plotted against the left-hand y-axis scale, even when copied from a curve currently plotted against the right-hand y-axis.



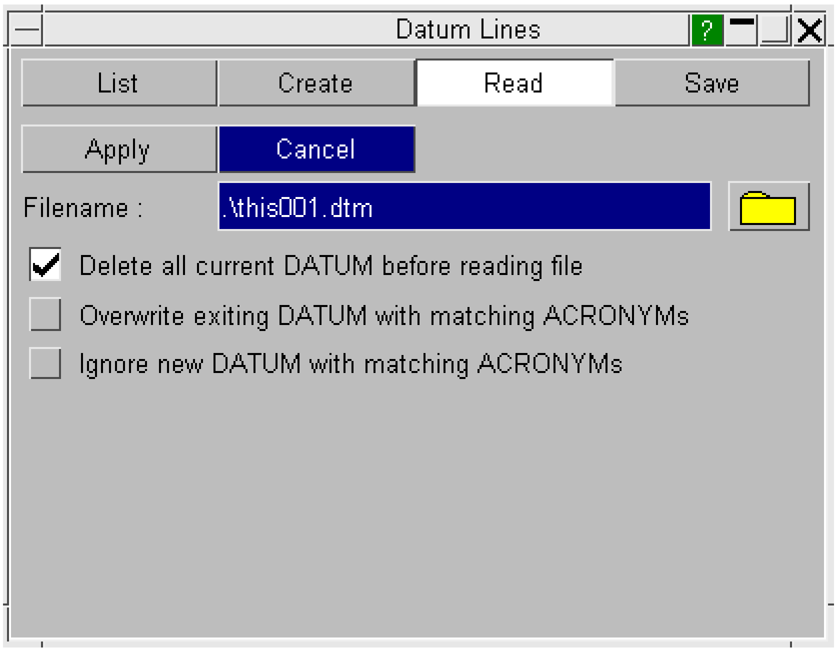
8.25.3. Read

Read

This option can be used to read in a file containing DATUM line definitions that has previously been saved.

All DATUM lines must have a unique acronym. When the file is read the user has the choice to:

1. Delete any existing DATUM line definitions before the file is read.
2. If a DATUM line in the file being read has the same acronym as an existing DATUM line then the existing definition will be overwritten.
3. If a DATUM line in the file being read has the same acronym as an existing DATUM line then the new definition in the file will be ignored.



The preference option

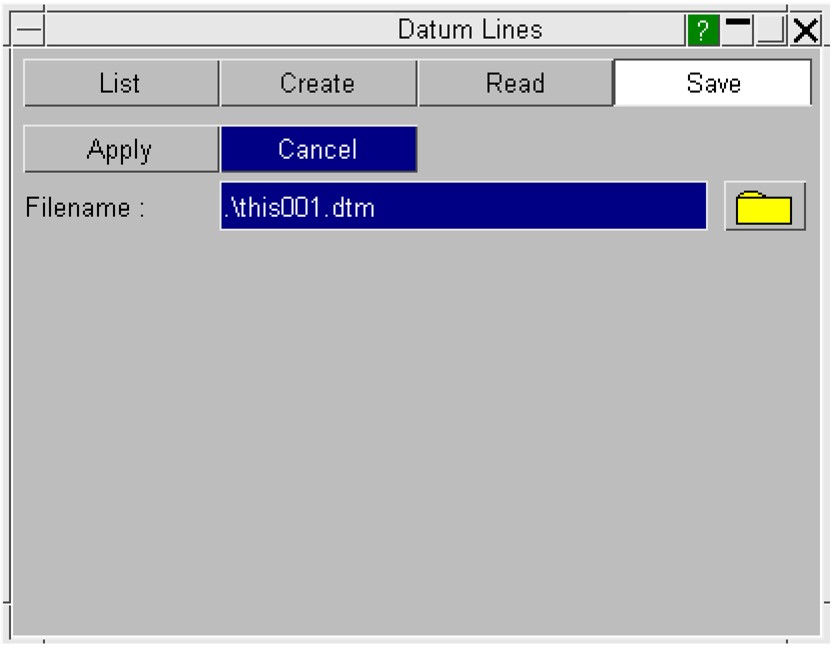
`this*datum_file : C:\datum\this001.dtm`

can also be used to define a default file containing DATUM line definitions that is read automatically when T/HIS starts (see Appendix H for more details)

8.25.4. Save

Save

This option can be used to save any DATUM line definitions to a file so that they can be reloaded and used in future T/HIS sessions.



8.26. T/HIS Session Save and Retrieve

T/HIS Session Save and Retrieve

T/HIS session save and retrieve saves the current T/HIS session as a session file of format (.tsf) onto the disk which can be read back later on to retrieve the saved T/HIS session. A T/HIS session file can also appended to or overlayed on top of an existing session.

8.26.1. Save Session

Save Session

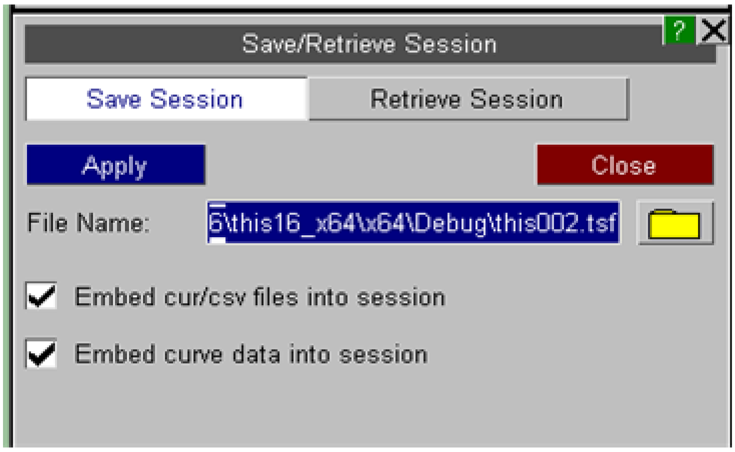
A T/HIS session can be saved either through Save/Retrieve Session panel or on Exit.

Save Session - Save/Retrieve Session Panel

To save a T/HIS session, select **File** -> **Session File** -> **Save** to open the Save/Retrieve Session panel in the menu area. Enter a name for the session file in the **File Name** textbox and click **Apply**. A filename can also be entered using file selection browser.

It should be noted that T/HIS session file does not directly store either the LS-DYNA model results or any csv/cur files that have been used for generating the curves inside the session file by default. It will only contain the full address path to these files. As a result of this session file do not occupy much space on disk.

If users want they can embed extra information into the session file so that the saved T/HIS session can be retrieved even if LS-DYNA results/csv/cur files are deleted or lost.



Embed cur/csv files into session	This option embeds the cur or csv files that are used for creating the curves. The session file with embedded cur/csv files no longer depend on these files and the session can be retrieved even if these files are deleted or lost. If you want this option to be enabled always, you can set the preference `this*session_embed_cur_csv_files: ON` (see Format of the oa_pref File for more details).
Embed curve data into session	This option embeds the curve xy coordinate data for all curves into the session file. A session file with embedded curve data can be retrieved even if the model files are missing. However, a session retrieved using embedded curve data loses information such as curve ID and graph properties. If you want this option to be enabled always, you can set the

preference: `this*session_embed_curve_data: ON` (see Format of the oa_pref File for more details).

Save Session - On Exit

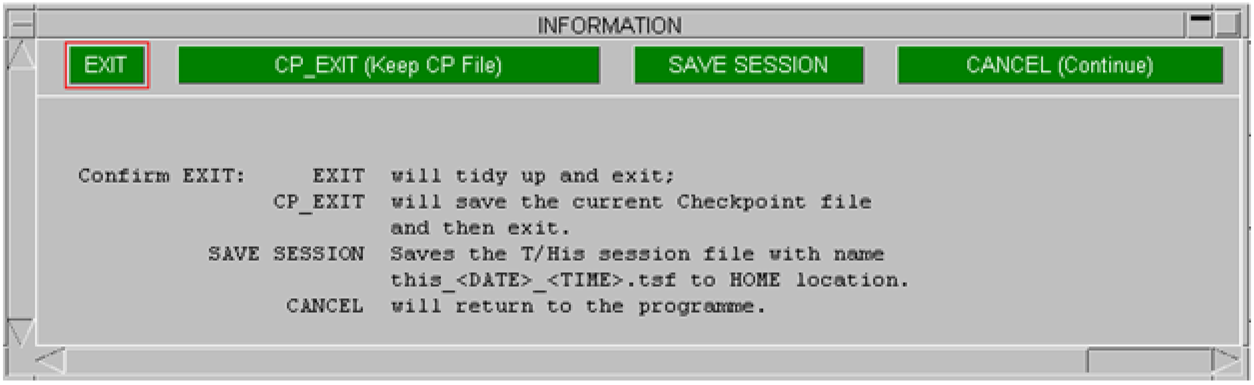
T/HIS session can be saved on exit from T/HIS by selecting **SAVE SESSION** button on the exit INFORMATION panel. The session file saved would have a name of the format **`this_<DATE>_<TIME>.tsf`**. The file would be by saved to the location defined in preference file (see Format of the oa_pref File for more details).

`this*session_save_option:`

`this*session_save_dir:`

The session file can also saved automatically every time T/HIS exits by defining in the preference file (see Format of the oa_pref File for more details).

`this*session_auto_save:`



8.26.2. Retrieve Session

Retrieve Session

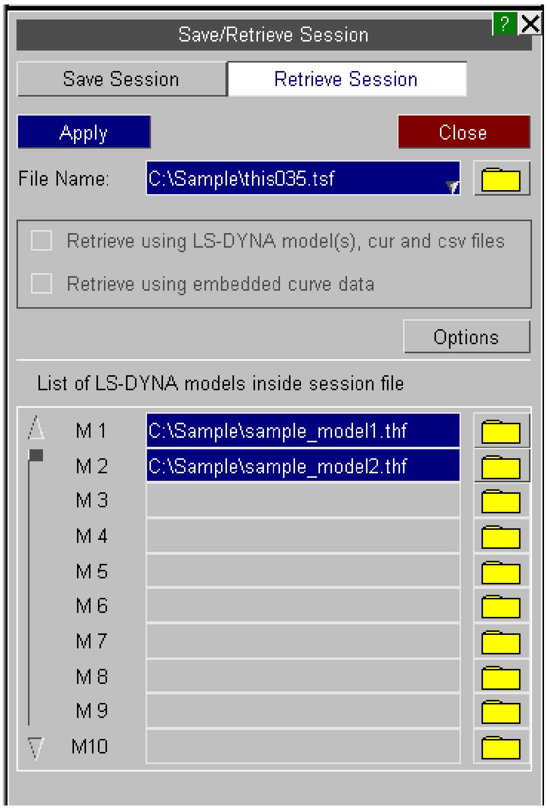
A session file (.tsf) that has been saved onto the disk can be opened by simply double-clicking on it.

To open a session file (.tsf) from inside T/HIS, select **File** -> **Session File** -> **Retrieve** to open the Save/Retrieve Session panel in the menu area. Enter the name of the session file which needs to be opened in the **File Name** textbox and click **Apply**. The session file can also be selected using file selection browser. The Retrieve Session panel can pop-up on the screen every time T/HIS is launched by setting the preference `this*show_session_retrieve_on_start: ON` (see Format of the oa_pref File for more details).

A typical T/HIS session can be retrieved in two possible ways depending on the data saved in session file:

1. Using LS-DYNA model(s), cur and csv files
2. Using embedded curve xy data

Note: An option to select either of the above types is provided only if the session file contains embedded curve xy data. By default session is retrieved using LS-DYNA model(s), cur and csv files option.

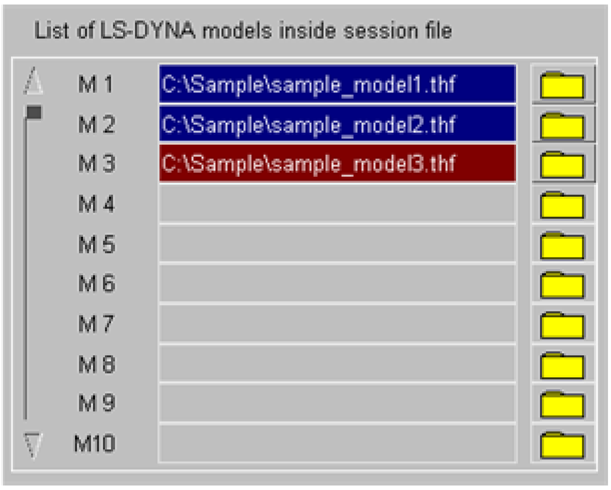


Using LS-DYNA model(s), cur and csv files

The session retrieved using LS-DYNA models, cur and csv files can restore all the curve information such as ID, blanking status, curve history, curve and graph properties such as color, curve symbol, line width, line style and captions. This option needs LS-DYNA model(s), cur and csv files that are not embedded in the session files to be present in the locations defined inside the session.

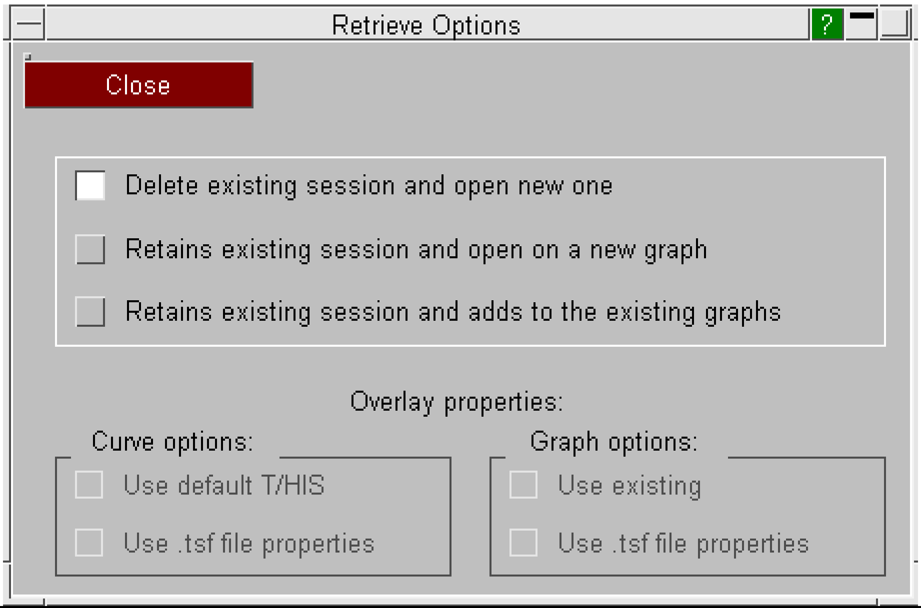
If the session files uses any LS-DYNA models then the path of models used by the session file can be found in **List of LS-DYNA models inside session file**. If the session file contains any model which is not found in the path defined inside the session file, the background color of the textbox turns red from blue.

The path of the models used by the session files can be modified either by modifying the path of the file in the textbox or by select the file using the file selection browser.



A T/HIS session file can be opened even if some of the models are missing. The retrieved session will only contain the curves from the models that are present.

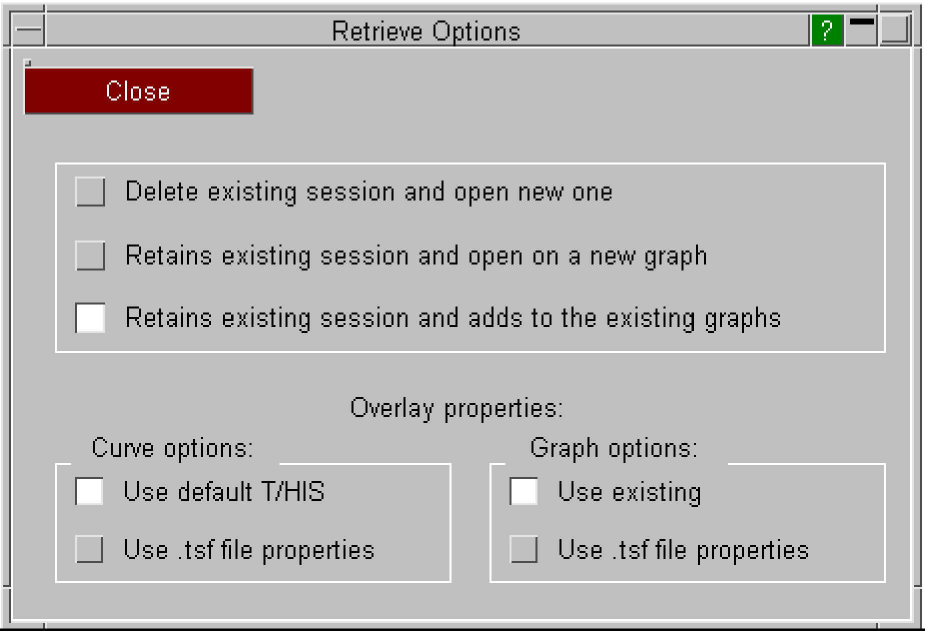
When trying to open a new session file with T/HIS already containing model(s) or curve(s), T/HIS session retrieve offers three different options:



Delete existing session and open new one **(New Session)**	This option deletes the current existing session and opens the selected session file as a fresh new T/HIS session. It should noted that graphs and curves, once deleted, cannot be retrieved unless they are saved beforehand.
Retains existing session and open on a new graph **(Append session)**	This option retains the current existing session as it is and the graphs from the session file will start from the highest available graph id. Total number graphs in the combined session is limited to 32.
Retains existing session and adds to the existing graphs **(Overlay session)**	This option retains the current existing session as it is and the curves from session file will be added to their corresponding graph in the current session.

The above Retrieve Options can be set by clicking on the **Options** button in the Save/Retrieve Session panel. This button becomes available only if T/HIS already contains model(s) or curve(s). Clicking on this button will open up the Retrieve Options pop-up.

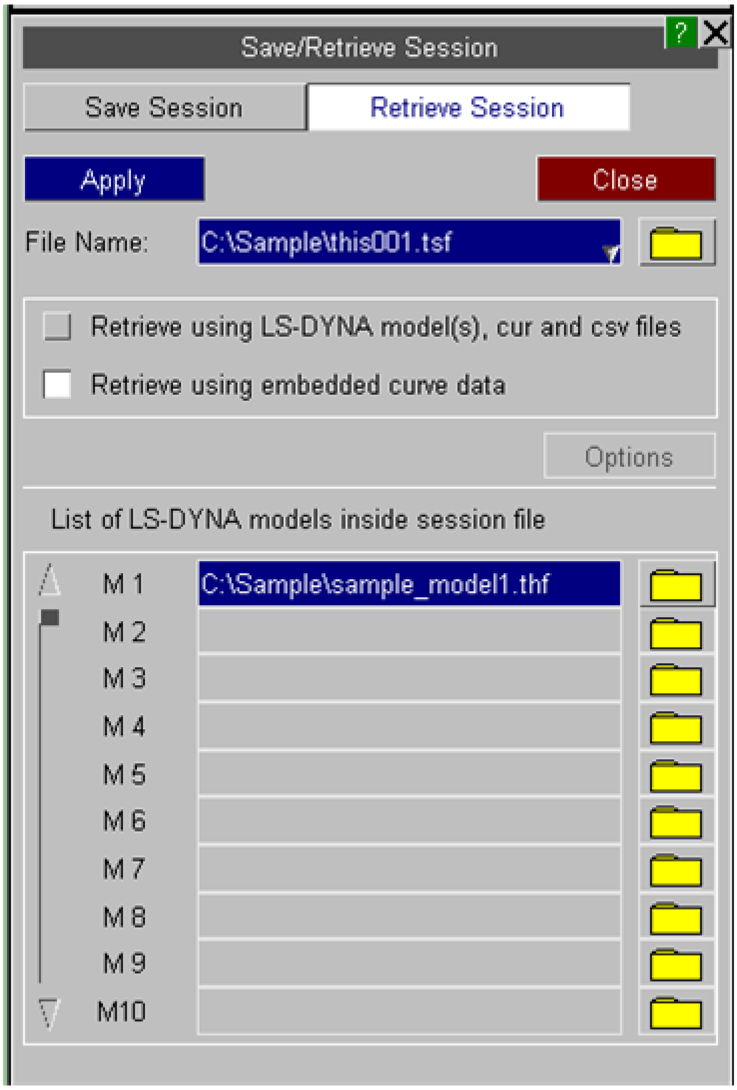
The Overlay Properties become available when **Retains existing session and adds to the existing graphs** option is selected. The Overlay Properties will help in defining the curve style for the curves/graphs properties such as curve line type, width, graph title, x-axis and y-axis labels and font that are being overlaid on top of existing session the curve and graph.



Overlay Curve Options	Use default T/HIS	This option automatically defines the curve style for the curves that are being overlaid.
	Use .tsf file properties	For the curves that are being overlaid, this option will apply the curve style defined inside the session file (.tsf).
Overlay Graph Options	Use existing	This option will retain the current existing graph properties such as x-axis and y-axis labels and font as it is.
	Use .tsf file properties	This option will apply the graph properties such as x-axis and y-axis labels and font as defined inside the session file (.tsf).

Using embedded curve xy data

If the selected session file contains embedded curve xy data, the option to select **Retrieve using embedded curve data** becomes available. This option retrieves all the curves even when the LS-DYNA model files required for session are missing/lost. However, the session retrieved using embedded curve xy data loses certain curve and graph properties. Hence, session option such as append and overlay are not applicable for a session retrieved using embedded curve data.


Save/Retrieve Session
Save Session
Retrieve Session
Apply
Close
File Name:
C:\Sample\this001.tsf
Retrieve using LS-DYNA model(s), cur and csv files
Retrieve using embedded curve data
Options
List of LS-DYNA models inside session file
M 1
C:\Sample\sample_model1.thf
M 2
M 3
M 4
M 5
M 6
M 7
M 8
M 9
M10

8.27. Workflows

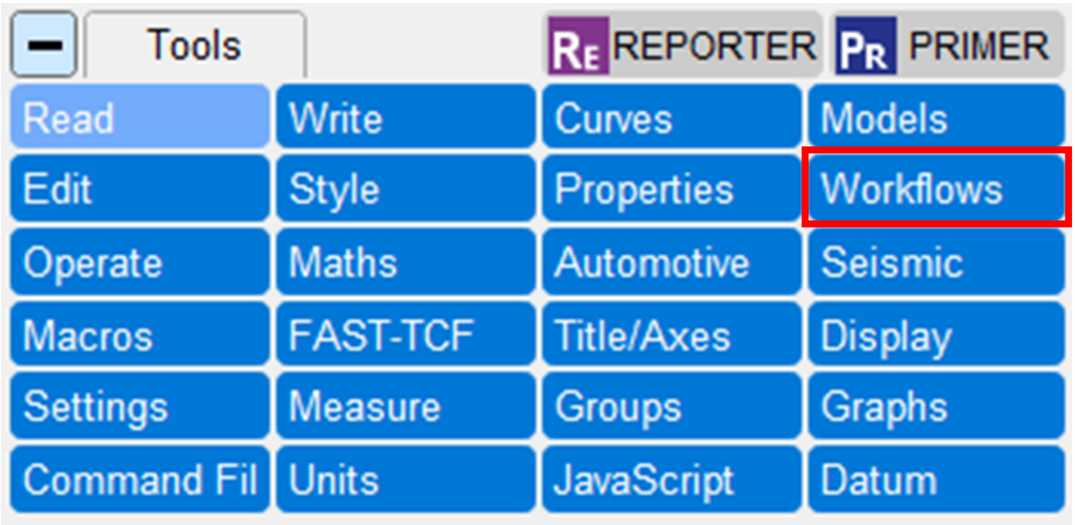
Workflows

The Oasys Suite contains powerful tools and capabilities that can be used to interrogate and debug your analysis results. However...

1. The tools are not always customised for your specific loadcases or tasks
2. You may need to manually perform a number of steps to process your results, which can be time-consuming and prone to error
3. The JavaScript API can be used to create tools to automate your post-processing workflow, but this requires time, resource and knowledge, which is not always available

To address these issues, the Workflows feature provides tools customised for specific loadcases and tasks, built upon the existing capabilities in the Oasys Suite, to make it easier to interrogate and post-process results.

The Workflows framework provides a simple structure to transfer data from PRIMER to the post-processing software. Browse the selection of already-available Workflow Tools, or read more about the Workflows menu in T/HIS.

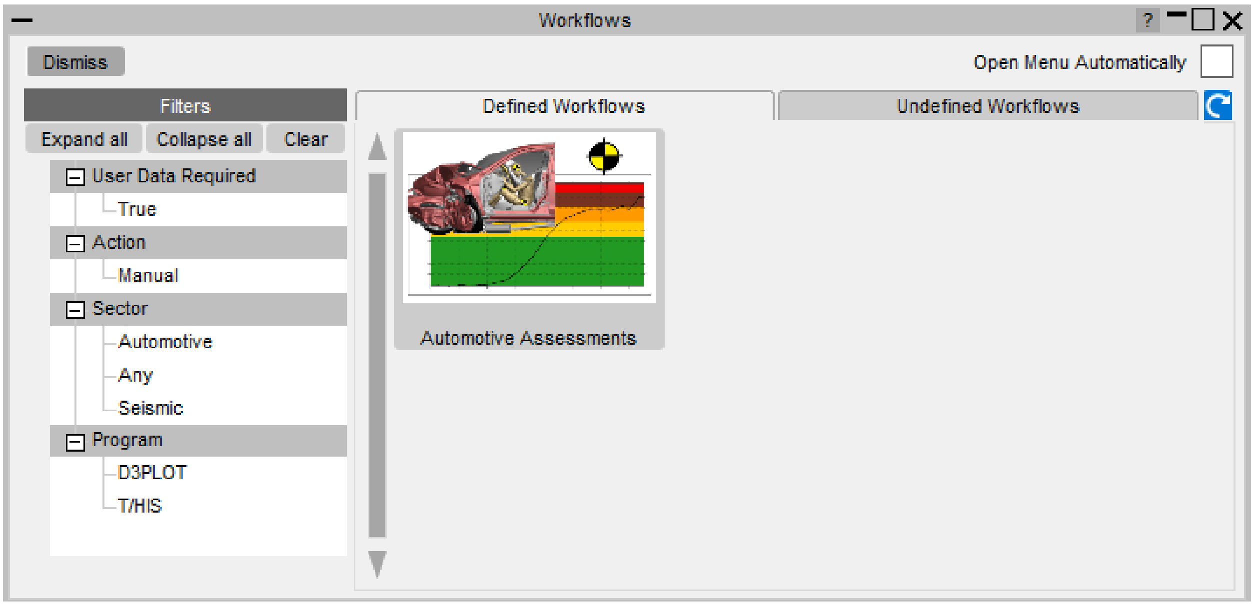


8.27.1. Workflows menu in T/HIS

Workflows menu in T/HIS

Tools → Workflows

The Workflows menu allows easy access to all the available workflows. In T/HIS, you can open the Workflows menu by selecting **Tools → Workflows**. The **Defined Workflows** tab shows all the Workflows that can be selected. Selecting any of the Workflows will run the JavaScript defined in the Workflow Definition. Running a Workflow will minimise the Workflows menu and open the script's user-interface. The Workflows menu will maximise again after closing the script.



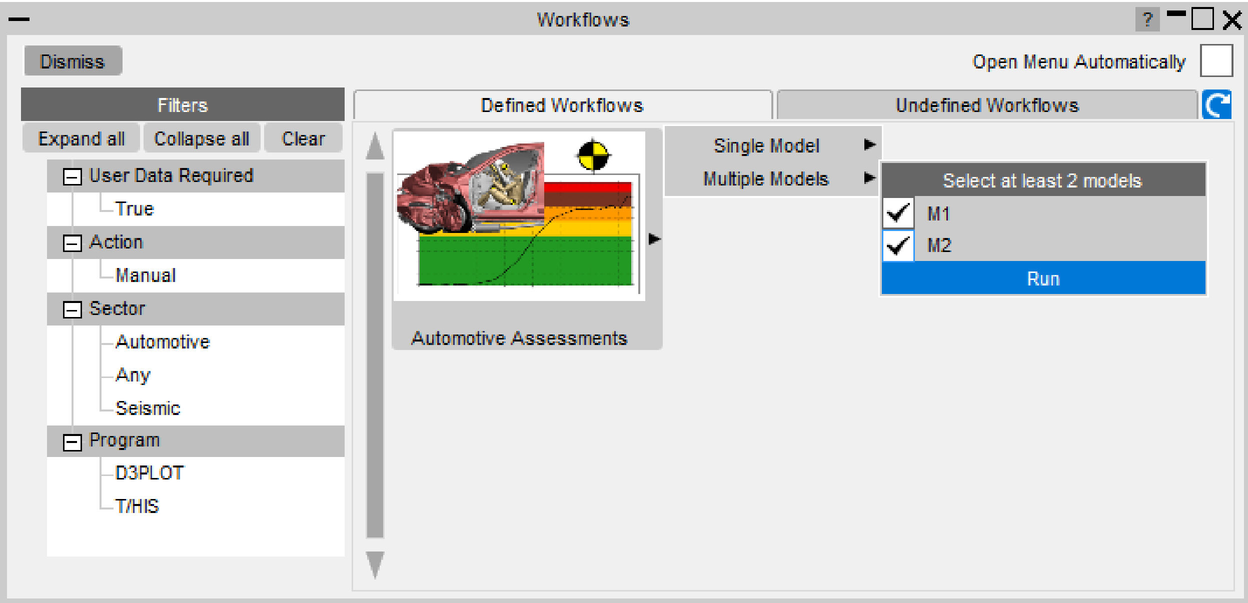
When the Workflows menu is initially opened, it shows all the available Workflows, but you can filter the Workflows by using the **Filters** tree. You can select multiple categories to filter the available Workflows displayed. Individual categories can be selected with single clicks; pressing the **Ctrl** and **Shift** keys while clicking on the tree will do a multi select.

Every Workflow can be tagged with Category and Value pairs, which populate the Filters tree. Selecting multiple Values within a Category will show all the Workflows tagged with any of the selected Values. However, selecting Values across Categories will show workflows which are tagged with all the selected Category/Value.

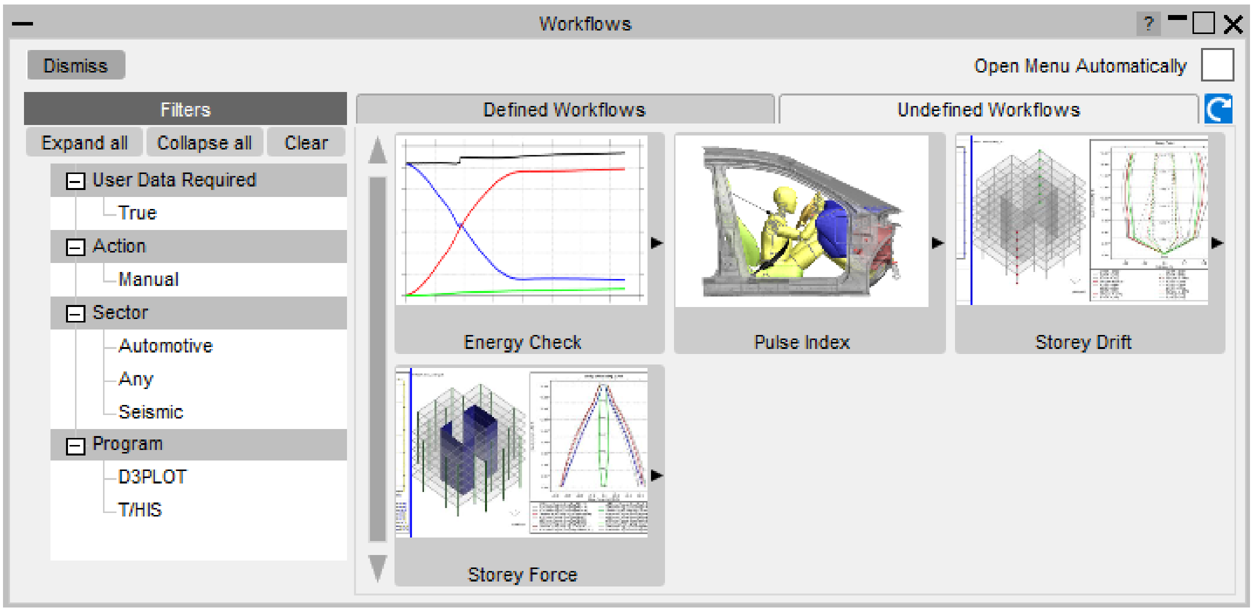
You can always refresh the Workflow user data and available Workflows shown in the menu by using the refresh button in the top-right corner of the panel. This will reread the Workflow data from the files so that the updated data is available. This is useful if you update the Workflow data from a PRIMER session and you want to use the revised user data in your already-open T/HIS session.

In both D3PLOT and T/HIS, you have the option to automatically open the Workflow menu when reading a model that has associated workflow data, by selecting the **Open Menu Automatically** tick box at the top-right corner of the panel.

If there is more than one model in memory and multiple models have the same Workflow associated with them, the individual Workflow buttons in the **Defined Workflows** pane will have a drop-down option to select for which models you wish to run the Workflow (see example below). You can select a single model or multiple models. The maximum and minimum number of models you can use in a Workflow is defined by the Minimum Multiple Models and Maximum Multiple Models parameters in the Workflow Definition.

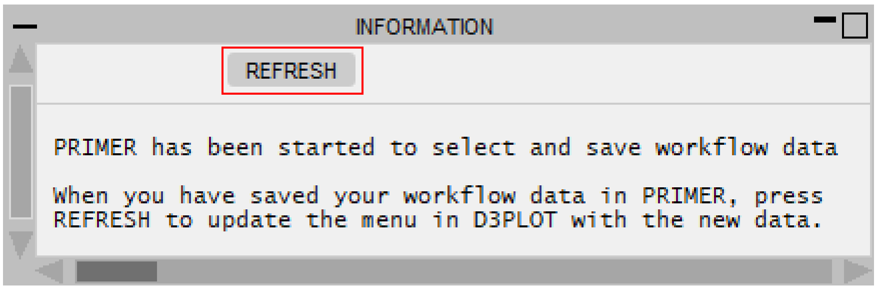


The **Undefined Workflows** tab shows all the workflows that can be run in T/HIS, but don't have the required user data to run them.



Selecting one of the workflows will open the model in PRIMER and start the workflow so you can select the required data.

In T/HIS a window will open telling you to press **REFRESH** when you have saved the data in PRIMER. This will update the workflow menu, moving the workflow to the **Defined Workflows** tab so it can be run in T/HIS.



9. Other Options

9.1. Tool Bar

Tool Bar

The tool bar is located across the top of the main T/HIS window and provides easy access to all of the main T/HIS menus from a series of drop down menus. In addition to the menus the drop down menus also allow a number of items to be changed dynamically and it provides a constant feedback of the cursor position within the graph area.

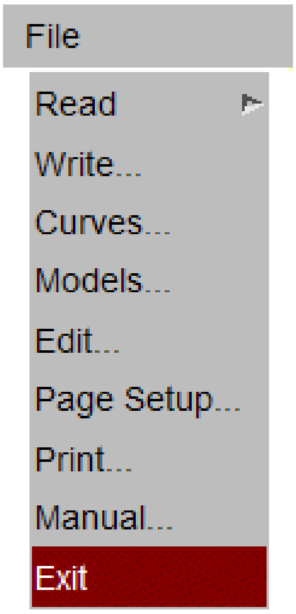
Each graph window contains its own tool bar that provides a subset of the functions in the main toolbar (see Graph Tool Bar).

9.1.1. File

File

The File drop down menu can be used to access the following menus.

Read	see READ Options for more details.
Write	see WRITE Options for more details.
Curve Manager	see Curve Manager for more details.
Model Manager	see Select for more details.
Edit	see EDIT Options for more details
Page Setup	This option is only available on PC's and can be used to access the standard Windows Page Setup menu.
Print	This option is only available on PC's and can be used to access the standard Windows Print menu.
Manual	Displays this manual.

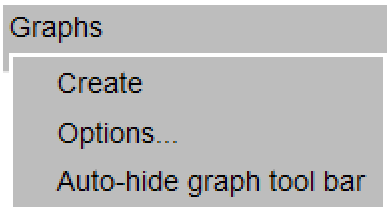


9.1.2. Graphs

Graphs

The Graphs drop down menu can be used to create new graphs and to change layout options.

Create	Create a new graph, see Creating Graphs for more details.
Options...	Modify graph layout options, see Creating Graphs for more details.
Auto-hide graph tool bar	This option can be used to automatically hide the tool bar, see Graph Tool Bar , at the top of each graph window.

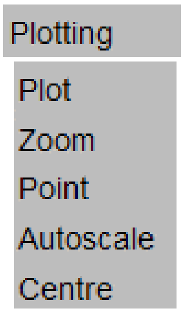


9.1.3. Plotting

Plotting

The Plotting drop down menu can be used to access the following plotting commands.

Plot	see PLOT for more details.
Zoom	see ZOOM for more details.
Point	see POINT for more details.
Autoscale	see AUTOSCALE for more details.
Centre	see CENTRE for more details.

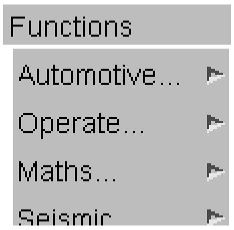


9.1.4. Functions

Functions

The Functions drop down menu can be used to access all of the curve functions.

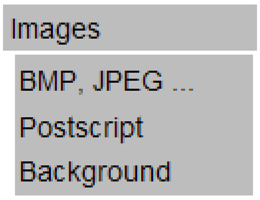
Automotive	see AUTOMOTIVE Options for more details.
Operate	see OPERATE Options for more details.
Maths	see MATHS Options for more details.
Seismic	see SEISMIC Options for more details.



9.1.5. Images

Images

The Images drop down menu can be used to save the current displayed graphs as an image in a number of formats. In addition to saving an image this menu can also be used to read in an image that is used as the background for each graph.

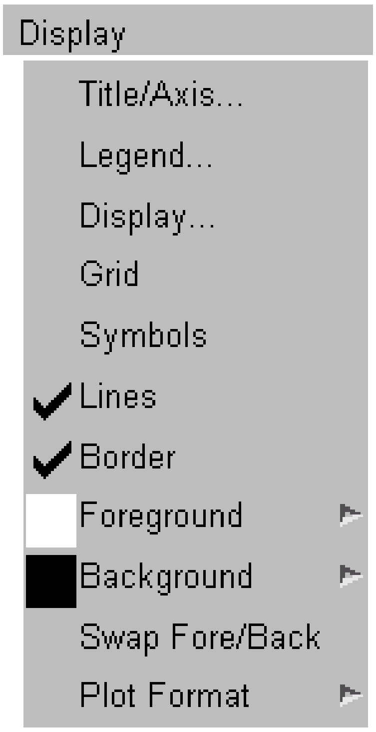


BMP, JPEG...	Capture the image as a bitmap or JPEG, see BMP, JPEG... for more details.
Postscript	Generate a Postscript or PDF image, see Postscript for more details.
Background	This option can be used to set an image as the background for each graph, see Background for more details.

9.1.6. Display

Display

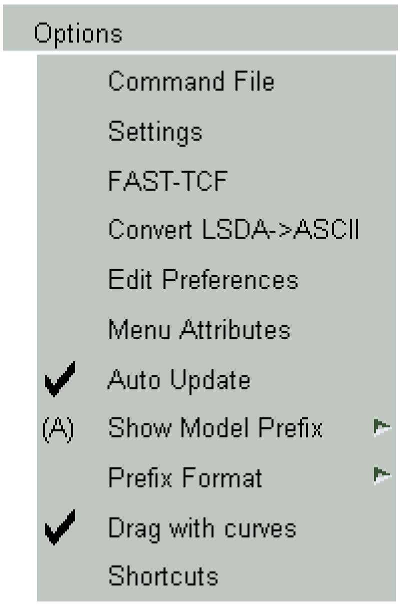
The Display drop down menu can be used to access the Title/Axis and Display menus and to dynamically modify the appearance of graphs. This menu changes all of the currently active graphs (see Active Graphs).



Title/Axis	see TITLE/AXES/LEGEND Options for more details.
Legend	see Legend for more details.
Display	see DISPLAY Options for more details.
Grid	Turns the grid on/off, see GRID for more details.
Symbols	Turns graph symbols on/off, see SYMBOLS for more details.
Lines	Turns graph lines on/off, see LINE STYLES for more details.
Border	Turns the plot border on/off, see BORDER for more details.
Foreground	Sets the foreground colour, see FOREGROUND for more details.
Background	Sets the background colour, see BACKGROUND for more details.
Swap Fore/Back	Swaps the current foreground and background colours, see Swap Foreground/Background for more details.
Plot Format	Set the current plot format, see Layout for more details.

9.1.7. Options

Options



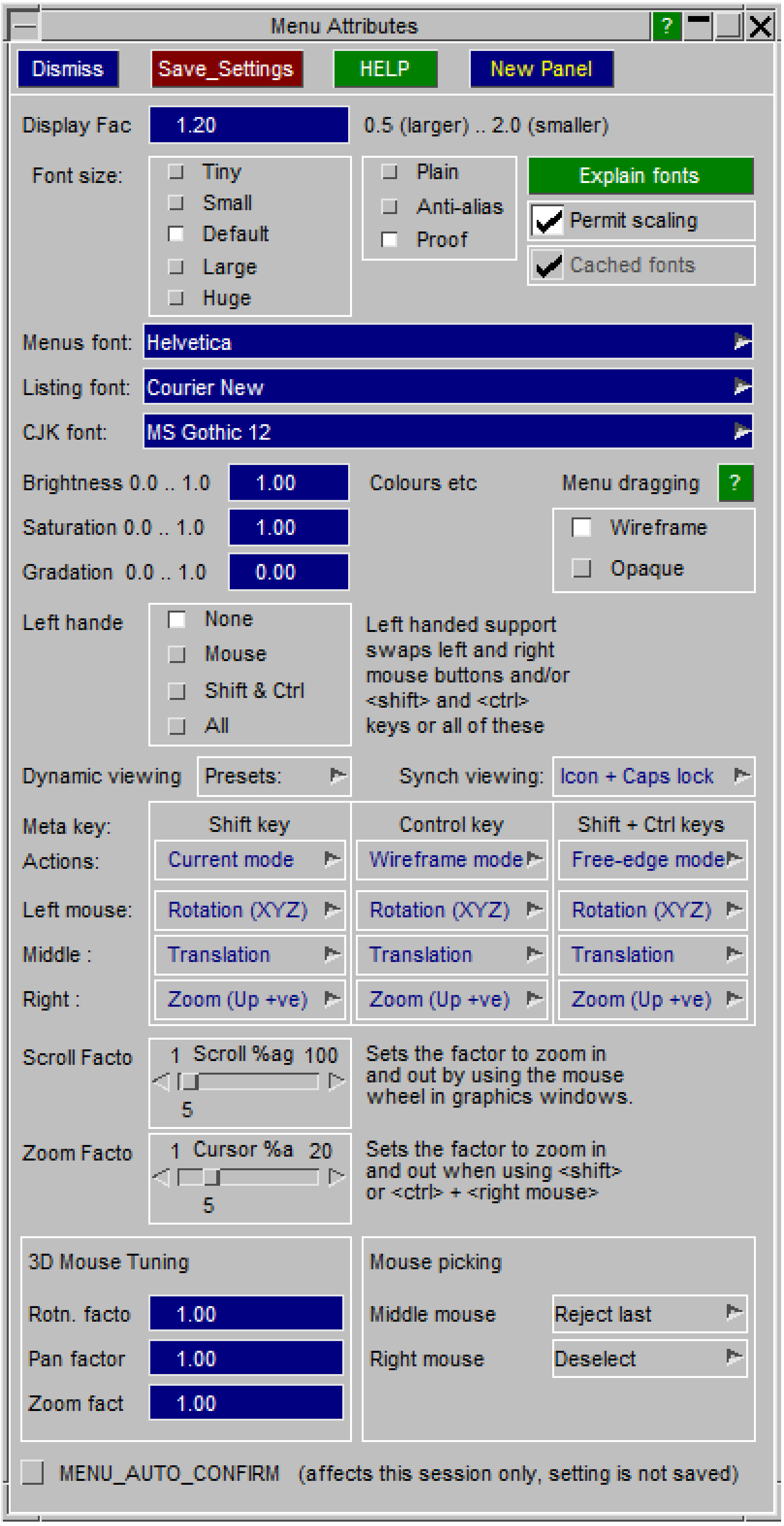
The Options drop down menu can be used to access all the following functions.

Command File	see Command / Session Files for more details.
Settings	Change data sources and other settings, see SETTINGS for more details.
FAST-TCF	Generate/playback FAST-TCF scripts, see FAST-TCF Options for more details.
Convert LSDA>ASCII	Convert a LSDA binout file to ASCII, see Convert Binout for more details.
Edit Preferences	Displays the preference editor, see Preferences for more details.
Menu Attributes	Modify menu fonts, size and colours, see MENU Attributes for more details.
Auto Update	Turn on/off automatic update.
Show Model Prefix	Turn the model prefix on/off or set it to automatic, see Curve Labels for more details.
Prefix Format	Select the prefix format displayed for each model. See Prefix Format Options for more details.

Drag with curves	Turn on/off the display of curves when dragging axis borders and legends. On some slow machines the time taken to update the display when a large number of curves is displayed makes the dragging response too slow. This option will automatically turn off the display of curves while the dragging operation is active.
Shortcuts	Setup keyboard shortcuts for commonly used function, see section Keyboard Shortcuts for more details.

MENU Attributes

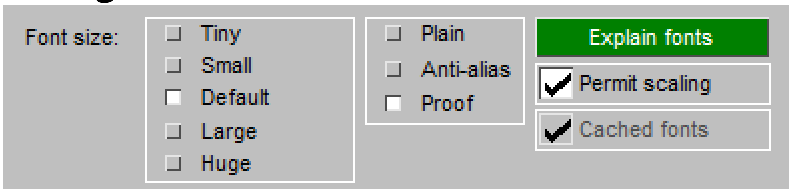
This panel allows you to tune the visual attributes of the screen menus within T/HIS and save them if you wish.



Display Factor

Lies in the range 0.5 to 2.0, default 1.0. Values < 1.0 reduce the apparent size of the screen so that menus and text become larger. Values > 1.0 act in the opposite sense. This is the simplest way of taking into account the display size.

Font Size, Quality and Scaling



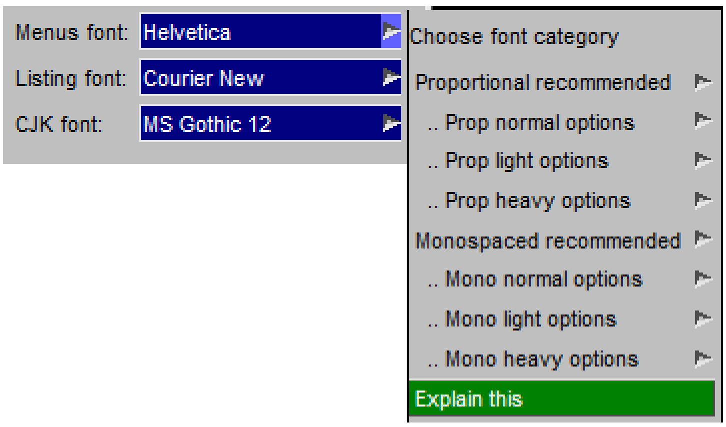
On most displays the **Default** font size will give the best appearance in menu interface panels, but occasionally **Small** or **Large** fonts may look better. It is recommended that you set the Display Factor first in order to get the best overall layout on your display, then adjust the font size if necessary.

Font quality improvements were made for T/HIS 17.0, and on most displays **Proof** quality will look best. However on low resolution displays it may look a little fuzzy due to the anti-aliasing process, and **Anti-alias** (coarser) or **Plain** (not anti-aliased) may give a crisper result.

Font scaling (**Permit scaling**) can be useful when your choice of font is a bit too large for the buttons in the user interface, since it allows the default font size to be reduced where text would overflow the space in a button. However it can result in a mixture of font sizes in a panel, which might improve legibility but looks untidy, so it is generally better to choose a Display Factor and Font Size that work well together on your display, and turn Scaling off.

Cached fonts is an obscure setting that will only apply on Linux systems where the "core" X11 font package has not been loaded, and the software reverts to cached bitmaps. If you have font problems on Linux please contact Oasys Ltd for advice and help.

Font selection



Historically T/HIS only provided Helvetica, Times and Courier fonts, but from T/HIS 17.0 a wider range of fonts has been made available.

The default for the User interface is still Helvetica for menu panels (the "Menus font") and Courier for listings (the "Listing font"), but you can use the popup menus to select from any of the fonts on your computer. The range of fonts available will depend both on the operating system and what has been installed, but typically there can be many. To try to make the choice manageable these are separated into

Proportionally spaced fonts, where character width varies. This is preferred for GUI panels with buttons.
`Monospaced fonts, where each character width is the same.` This is preferred for text listings.

Within each category fonts are also sorted by weight, with "normal" being the most commonly used. "Light" options tend to be narrower, permitting more characters to fit in a button, "Heavy" options tend to use bold text, and can be useful when using very large fonts - perhaps on a projector or when setting up the user interface for someone who is visually impaired.

Brightness

Lies in the range 0.0 to 1.0, default 1.0. Controls the brightness of the menu interface only (it will not affect displayed graphics).

Saturation

Lies in the range 0.0 to 1.0, default 1.0. Controls the colour saturation of the menu interface. (Again it will not affect displayed graphics.)

Left Handed

The software uses mouse buttons and keyboard 'meta settings keys (<shift> & <control>) in a handed way that is set up by default for right-handed use. It is possible to configure either or both for left-handed use.

Save Settings

Once you have adjusted the above to your taste you can save these settings in your 'oa_pref' file for future use with the **Save_Settings** button. If you do not save settings they will be lost when this session exits.

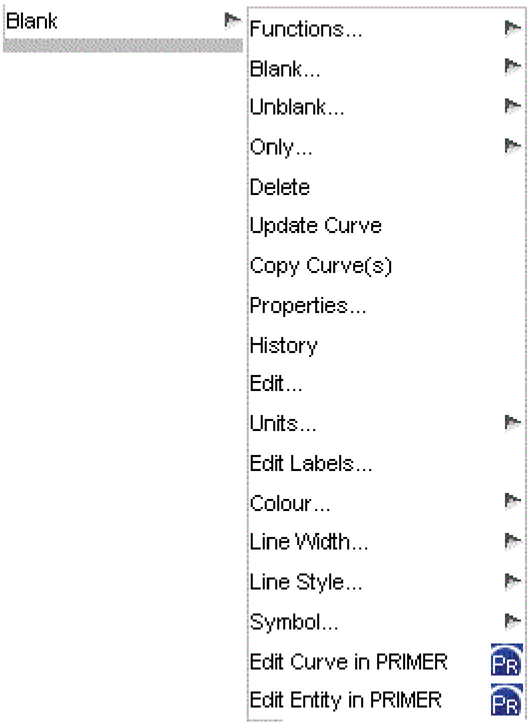
9.1.8. Quick Pick

Quick Pick

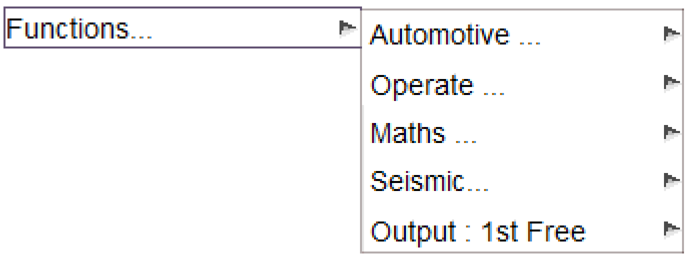
The Quick Pick menu can be used to perform many common curve operations using just the mouse. The current "Quick Pick" mode is displayed on the tool bar and can be changed using the popup menu.

The current "Quick Pick" option can be applied to a single curve by selecting the curve using the left mouse button. Multiple curves can be selected by holding down the left mouse button and dragging out an area.

Some functions can be undone using the middle mouse button.



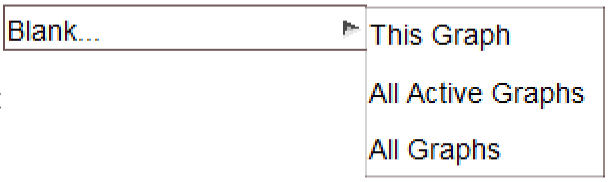
Functions...



This option can be used to select any of the curve operations (see OPERATE Options , MATHS Options , AUTOMOTIVE Options and SEISMIC Options) that have a single curve as input. In addition to selecting a curve operation this menu can also be used to set the output curve for the curve operation to either the 1st free curve or to overwrite the input curve.

This option can be applied to multiple curves but it can not be undone.

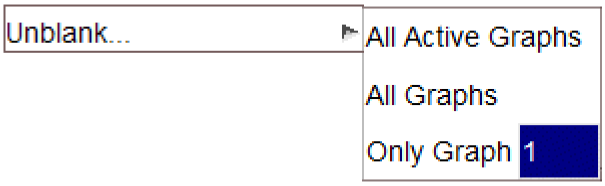
Blank...



This option can be used to blank curves. The selected curves can be blanked in just the graph they were selected in, all the currently active graphs or all graphs.

This option can be applied to multiple curves and it can be undone using the middle mouse button.

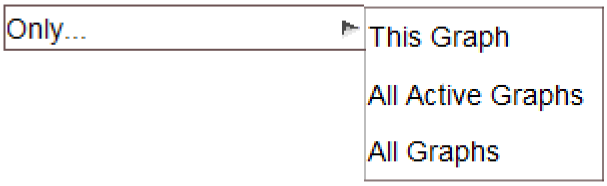
Unblank...



This option can be used to unblank curves. The selected curves can be unblanked in all the currently active graphs, all graphs or a individual graph can be specified.

This option can be applied to multiple curves and it can be undone using the middle mouse button.

Only...



This option can be used to blank all curves except for the selected ones. The selection can be applied to just the graph they were selected in, all the currently active graphs or all graphs.

This option can be applied to multiple curves and it can be undone using the middle mouse button.

Delete

This option can be used to delete curves. It can be applied to multiple curves but it can not be undone.

Properties...

This option will display the current properties for a curve (see Properties... for more details). If multiple curves are selected this option is only applied to the one with the lowest curve ID.

History...

This option can be used to view and edit the history of operations used to create a curve (see Curve Histories ... for more details).

Edit...

This option can be used to select a curve for editing (see EDIT Options for more details). If multiple curves are selected this option is only applied to the one with the lowest curve ID.

Edit Labels...

This option can be used to edit the label, title and axis labels for a curve (see Edit Labels... for more details) . If multiple curves are selected this option is only applied to the one with the lowest curve ID.

Colours...

This option can be used to change the colour of curves. This option can be applied to multiple curves and it can be undone using the middle mouse button.

Line Width...

This option can be used to change the line width of curves. This option can be applied to multiple curves and it can be undone using the middle mouse button.

Line Style...

This option can be used to change the line style of curves. This option can be applied to multiple curves and it can be undone using the middle mouse button.

Symbols...

This option can be used to change the symbol style of curves. This option can be applied to multiple curves and it can be undone using the middle mouse button.

Edit Curve in PRIMER...

This option can be used to send the load curves in the linked session of PRIMER, see PRIMER: Synchronising with ... for more details.

Edit Entity in PRIMER...

This option can be used to send the curve entities in the linked session of PRIMER, see PRIMER: Synchronising with ... for more details.

9.2. Graph Tool Bar

Graph Tool Bar

9.2.1. Graph Selection

Graph Selection

This option can be used to make a graph active or inactive, see Active Graphs for more details.

9.2.2. Plotting

Plotting

Plotting

This option provides the same functions as the Plotting menu in the main toolbar with the exception that the settings only apply to the graph in the window instead of all of the currently active graphs.

9.2.3. Display

Display

Display

This option provided the same functions as the Display menu in the main toolbar with the exception that the settings only apply to the graph in the window instead of all of the currently active graphs.

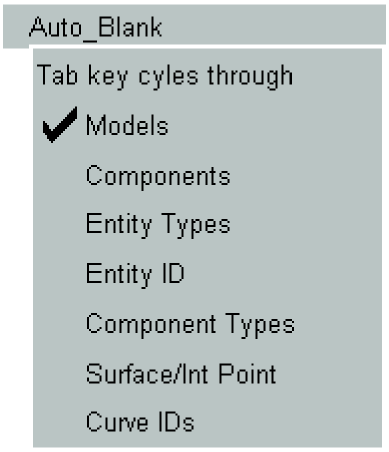
9.2.4. Auto_Blank

Auto_Blank

The **Auto_Blank** function can be used to blank and unblank curves in a graph using either the TAB key or SHIFT+TAB.

By default if you now press the TAB key in a graph T/HIS will automatically blank all the curves except for those belonging to model 1. If you press TAB a 2nd time you will just see the curve belonging to model 2, a third time model 3. When you reach the end of the models you have curves for pressing the TAB key loops back to model 1. If you press SHIFT+TAB then it goes the other way (model 3 > model 2 > model 1 > model 3 ....)

Instead of blanking curves by model the behaviour of the TAB key can be changed.



Models	Default. Blanks curves by model ID.
Components	Blanks curves by component. e.g Node X Displacement > Node Y Displacement > Node Z Displacement > ...
Entity Types	Blanks curves by entity type e.g. Whole Model > Parts > Nodes > Solids > ...
Entity ID	Blanks curves by ID. e.g Node 1 and Solid 1 > Node 2 and Solid 2 > ....
Component Types	This is similar to Component except that it lumps all the displacement curves together then velocity so you get x,y,z and magnitudes. You will also get data for different entity types. So Energy would show things like Whole Model KE and Contact Energies.
Surface/Int Point	Blanks curves by surface or through thickness integration point. e.g Top > Middle > Bottom > Layer 1 > ...
Curves ID's	Blanks curves by ID

The default **Auto_Blank** mode can be modified using the preference file (see Appendix H for more details)

`this*auto_blank_mode:`

9.2.5. Curve Locking

Curve Locking

The **Lock** option works in a similar way to locking in PRIMER and D3PLOT. When the **Lock** button is pressed at the top of a graph, all currently blanked curves in that graph are locked from becoming visible, until they are unlocked. This allows the remaining unlocked curves to be manipulated without unblanking any of the locked curves. This includes the use of the shortcut keys 'u', 'r' and ,'b'. Locking can also be set via the Curve Table using the graph buttons (see Table for more details).

Once the **Lock** button is pressed, a popup is attached, providing the option to either unlock the curves in that graph or **Unfreeze All**. 'Freezing' is the equivalent of locking for visible curves, so once a curve is frozen, it will stay visible until unfrozen. This can be set using Quick Pick, the Curve Manager or the Curve Table.

9.2.6. AB

AB

This option can be used to turn and off the Auto Blank option. The default setting for this option can be modified using the preference file (see Appendix H for more details)

this*auto_blank:

9.3. CURVE INFORMATION

CURVE INFORMATION

Pressing the right mouse button while in the graphics window will display a popup menu listing the ID, label and the data source of the nearest curve.

When data is read from either one of the LS-DYNA output files T/HIS will store the ID and type of the entity that the data applies to. If the curve label is modified this data will remain unchanged so that the curve source can still be identified.

If a curve has been read in from another source then T/HIS will report the data source as being UNKNOWN.

If a curve is created from another curve using one of the T/HIS curve operations then the data source for the new curve will be copied from the original curve. If the operation uses more than one curve as input then the data source information will only be copied to the new curve if all of the input curves had the same data source.

Edit Points... will open the Curve Editor for the selected curve, allowing points to be added, deleted or changed individually. If the curve is an equation curve (see Equation), then there will also be an **Edit Equation...** option present, allowing the equation to be updated and overwrite the original curve.

The **Colour**, **Line Width**, **Line Style** and **Symbol** pop-up menus

9.3.1. Properties...

Properties...

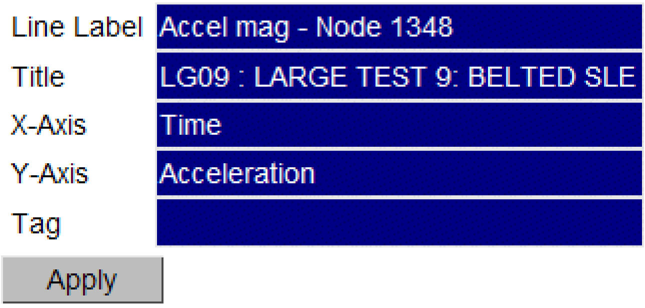
This option displays a number of properties for a curve including minimum and maximum values, average and RMS value.

Xmin	0.0000000	
Xmax	0.0998993	
Ymin	0.0000000	@ X = 0.0000000
Ymax	1217170.8	@ X = 0.0479982
RMS	343394.38	
Average	271682.56	

9.3.2. Edit Labels...

Edit Labels...

This option can be used to change the title, tag, line label and axis labels for a curve.

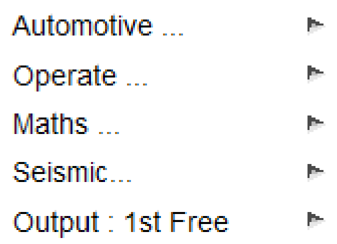


9.3.3. Functions...

Functions...

The Functions popup menu can be used to access any of the curve operations that take a single curve as the only input. As well as applying an operation to a curve this menu can also be used to select between:

- Overwriting the input curve with the output from each function
- Writing the output to the 1st unused curve



9.4. Curve Histories...

Curve Histories...

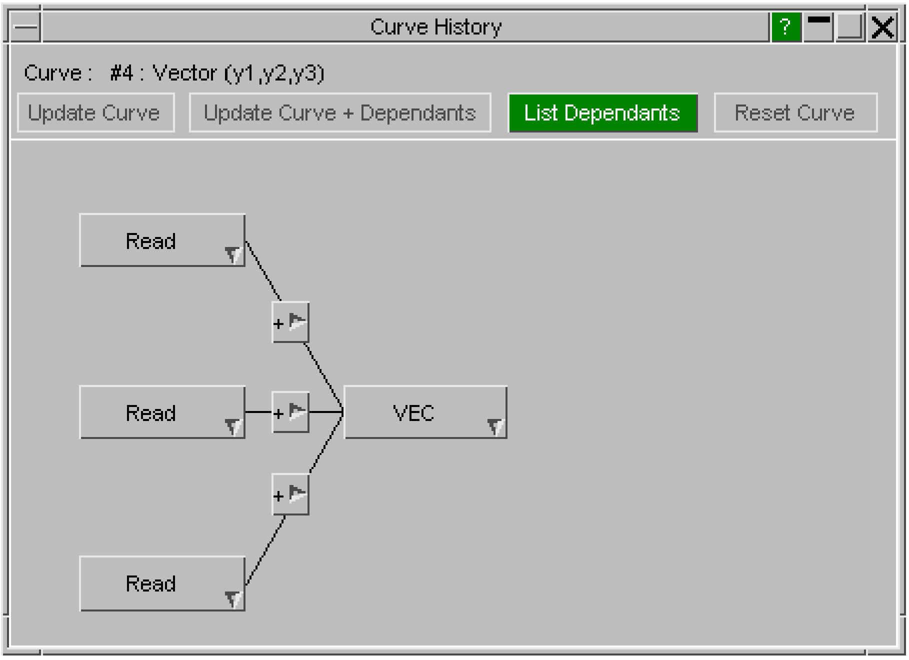
Internally T/HIS knows about all of the operations used to create a curve and the order that the operations were applied. In addition to knowing the operations used to create each curve T/HIS also knows which curves were used as inputs to operations that created other curves.

9.4.1. Viewing

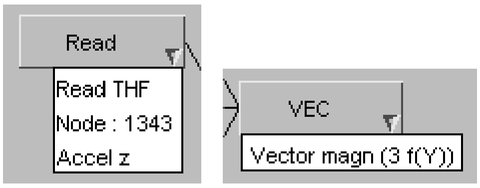
Viewing

When a curve is selected and the curve history is displayed, a floating window will be displayed that shows all of the operations used to create a curve.

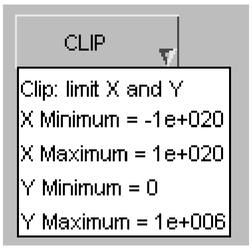
In the example below, 3 items were read in and then combined using the **VEC** operation.



More information on each part of the curve history can be obtained by moving the mouse across each operation.



If a curve operation has one or more inputs that are not curves then the hover text will display all of the inputs along with their values.

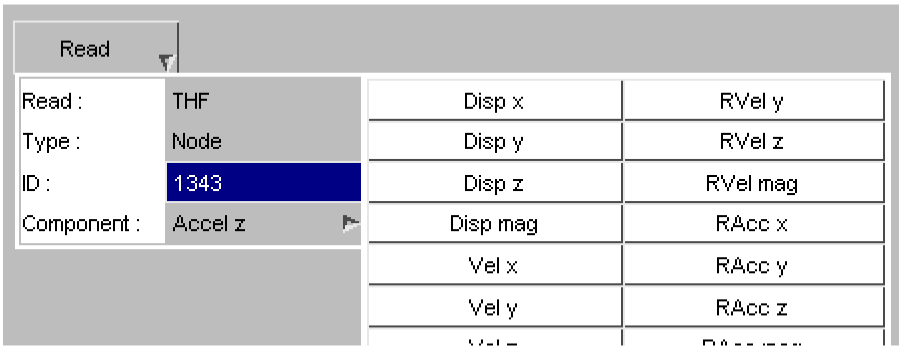


9.4.2. Modifying

Modifying

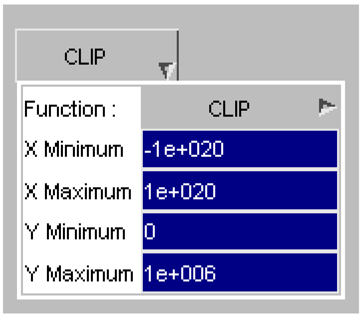
As well as viewing the operations used to create a curve the operations can also be modified by right clicking on them.

For a **READ** operation, the entity ID can be changed to any other ID of the same entity type. T/HIS will automatically check if results are available for the new ID and display a warning if they are not.

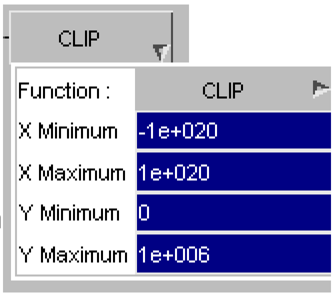


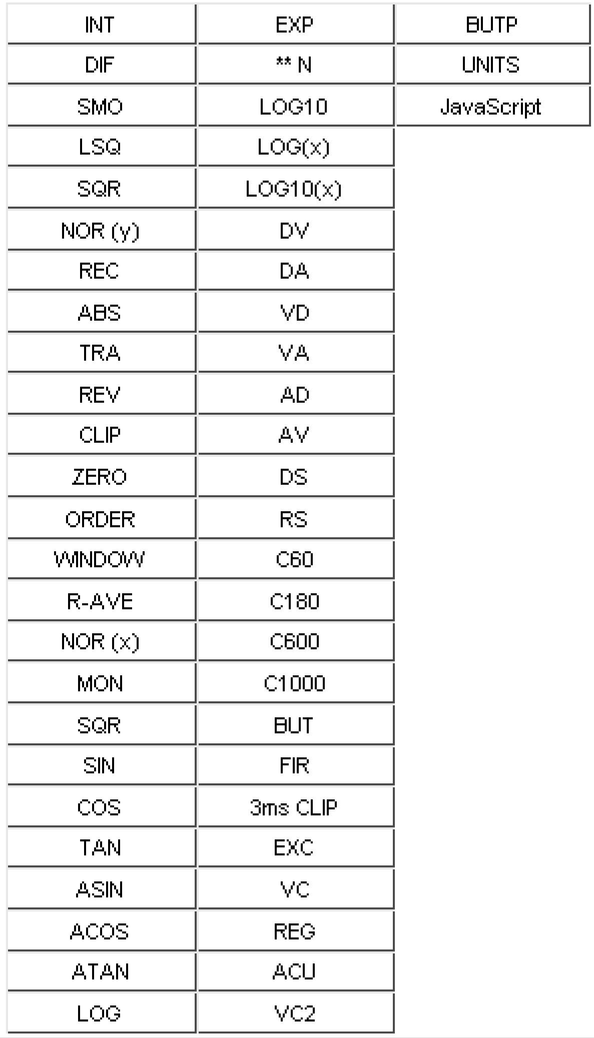
As well as changing the entity ID the data component can also be modified by selecting a different component in the popup menu.

If a curve operation has one or more inputs that are not curves then right clicking on the operation will display a popup menu that will allow all of the values to be modified.



As well as changing the inputs to existing curve operations it is also possible to change a curve operation to any other curve operation that has the same number of input curves.





Right clicking on the popup symbol next to the name of the current curve operation will display a menu containing a list of all of the curve operations that are available which have the same number of input curves.

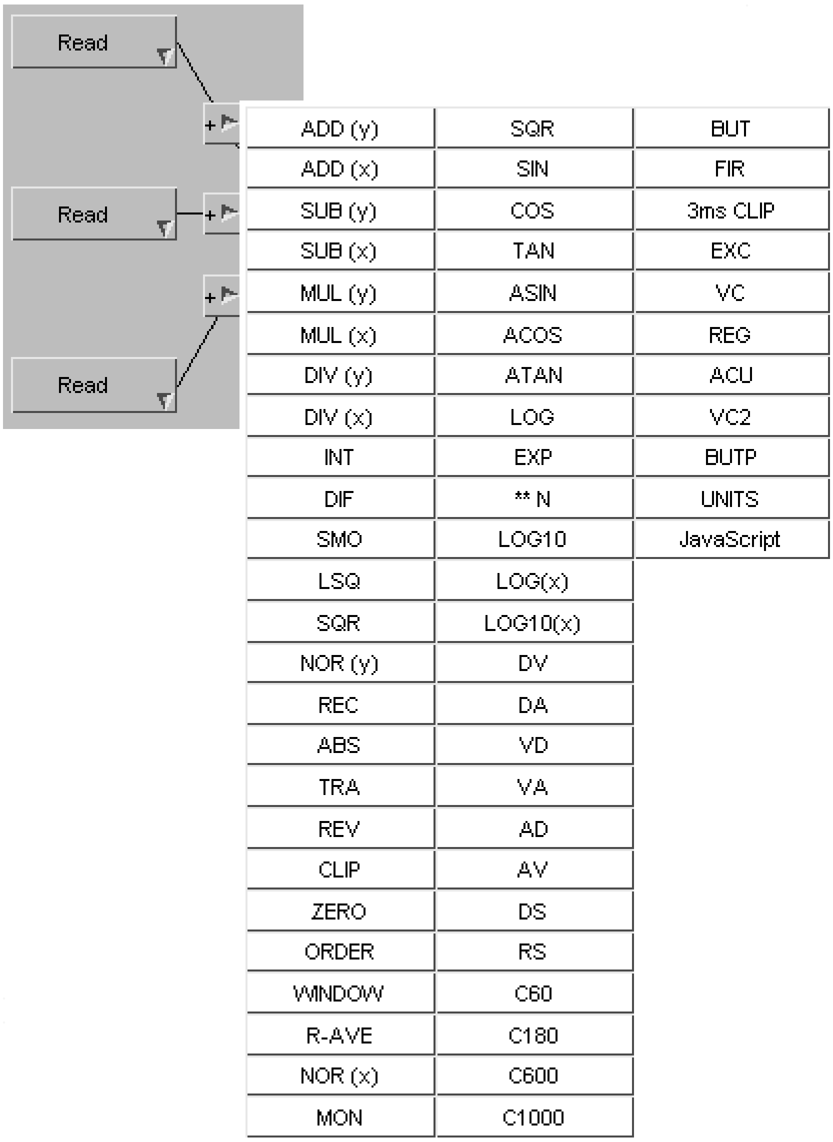
If for example the current curve operation is **CLIP**

9.4.3. Inserting New Operations

Inserting New Operations

New operations can be inserted into the chain of curve operations by right-clicking on one of the + symbols between the existing operations.

The popup menu that is displayed will contain all of the curve operations that take a single curve as input and produce a single output curve.



9.4.4. Update Curve

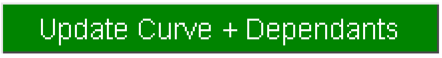
Update Curve

Update Curve

If any of the operations used to create a curve are modified or if a new operation is inserted then this option can be used to automatically update the curve. T/HIS will automatically rebuild the curve using the updated set of curve operations and will replace the old curve with the new one.

9.4.5. Update Curve Dependants

Update Curve + Dependants



This option will update the selected curve and any dependant curves. As T/HIS stores all of the operations used to create every curve, it knows if a curve has been used as an input to any other curves.

The selected curve will be automatically rebuilt and replaced with the new curve, and then any curves that use the selected curve as an input will also be rebuilt and replaced.

9.4.6. List Dependants

List Dependants

This option will display a list containing any curves that have been created which use the currently selected curve as an input somewhere in their chain of curve operations.

9.4.7. Reset Curve

Reset Curve

This option can be used to reset all of the curve operations used to create a curve if any of them have been modified.

9.5. Keyboard Shortcuts

Keyboard Shortcuts

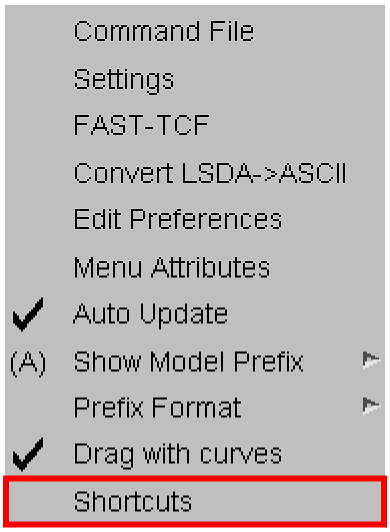
Some panels and actions can be accessed through pre-programmed shortcuts and from T/HIS 9.4 onwards the keys they are assigned to are customisable.

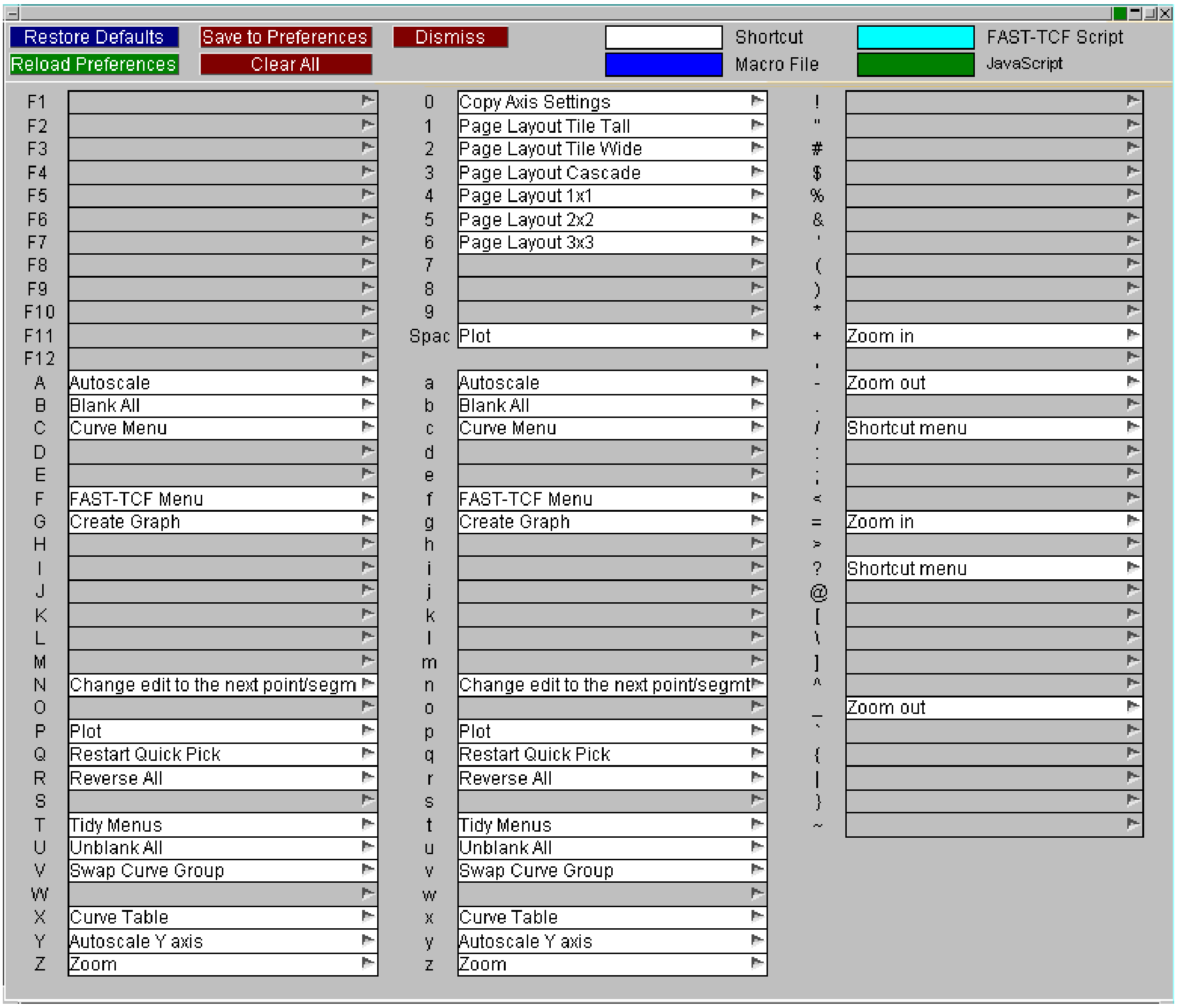
From T/HIS 9.4 onwards a number of new pre-programmed shortcuts have been added, including the top menu panels and window layout options. In addition to these pre-programmed shortcuts Macros and FAST-TCF scripts can also be assigned to a key.

A listing of the available shortcuts and the keys they are assigned to can be brought up by pressing the '?' key (by default) or accessing it through the Options top menu.

This will bring up a panel, from which you may assign the shortcuts, Macros and FAST-TCF scripts to the keys. Note that upper and lower case letters can be assigned different shortcuts.

A list of all the available pre-programmed shortcuts is given at the end of this section with their default key(s) if assigned.





At the top of the panel you will see the following buttons.

Restores the shortcuts to their default keys, removing any shortcuts assigned by the user.

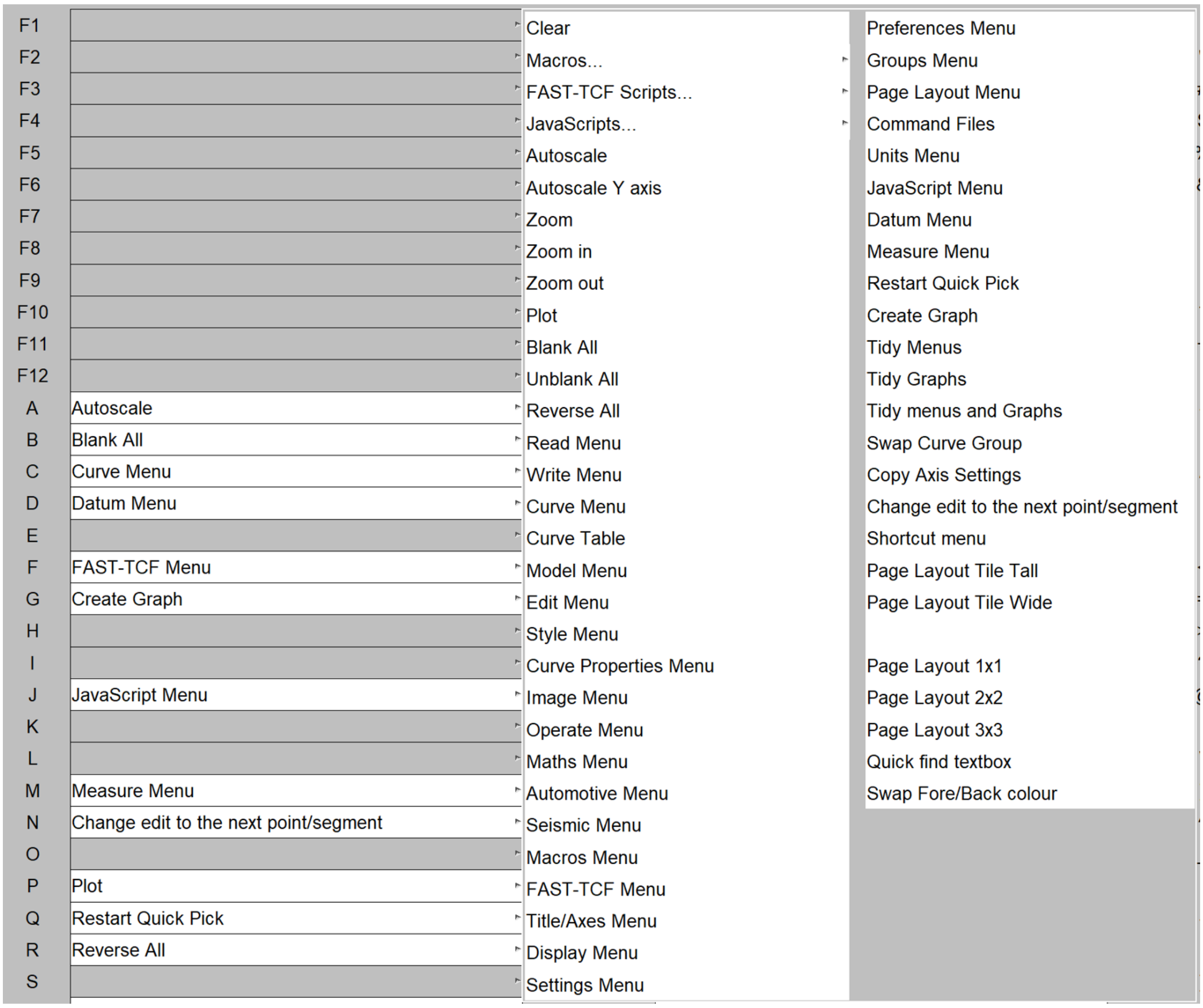
Saves the shortcuts to the oa_pref file in the home directory. They are saved in the format "this*A_key: AUTOSCALE" where the first part defines which key the shortcut is assigned to and the second part is the shortcut being assigned. Each shortcut has a specific name to use in the oa_pref file, and a list is given below.

When T/HIS is started this is read and the saved shortcuts are restored.

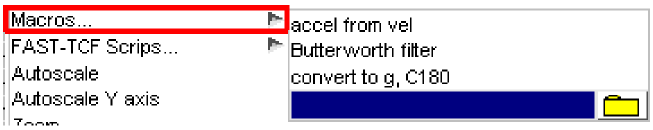
Reloads the shortcuts from the oa_pref file in the home directory.

Clears all the shortcuts on the panel.

To assign a shortcut, click on the popup next to the key you want to assign it to. This will bring up a list of all available shortcuts in T/HIS as well as the option to assign Macros or FAST-TCF scripts.



To assign a Macro, FAST-TCF script or JavaScript a to a key, click on **Macros...**,**FAST-TCF Scripts...** or **JavaScripts...**.



This will bring up another popup from which you can select the Macro or script. The popup will contain a list of scripts that T/HIS has picked up from the $OA_INSTALL and $OA_HOME directory. If the script you want is not in this list you can browse for it by clicking on the folder icon.

The listing of assigned keys is colour coded to easily distinguish between pre-programmed shortcuts (white), FAST-TCF scripts (light-blue), Macros (dark-blue) and JavaScripts (dark-green)

Pre-programmed Shortcuts:

Defaults shown in bold, oa_pref name shown in brackets

View Controls	
A/a – Autoscale (AUTOSCALE)	Autoscale Y axis (Y_AUTOSCALE)
P/p – Plot (PLOT)	**[SPACE]** – Plot (PLOT)
Z/z – Zoom (ZOOM)	**"+"/"="** – Zoom in (ZOOM_IN)
"-"/"_" – Zoom out (ZOOM_OUT)	
Blanking	
B/b – Blank All (BLANK)	**R/r** – Reverse curve blanking (REVERSE)
U/u – Unblank all curves (UNBLANK)	
Menus	
Automotive Menu (AUTOMOTIVE_MENU)	Command Files Menu (CFILE_MENU)
C/c – Curve Menu (CURVE_MENU)	Curve Properties Menu (PROP_MENU)
Curve Table (CURVE_TABLE)	Display Menu (DISPLAY_MENU)
Edit Menu (EDIT_MENU)	Groups Menu (GROUPS_MENU)
Image Menu (IMAGE_MENU)	**F/f** – FAST-TCF Menu (FAST_TCF_MENU)
Macros Menu (MACROS_MENU)	Maths Menu (MATHS_MENU)
Model Menu (MODEL_MENU)	Operate Menu (OPERATE_MENU)
Page Layout Menu (PAGE_MENU)	Preferences Menu (PREF_MENU)
Read Menu (READ_MENU)	Shortcut Menu (SHORTCUT)
Seismic Menu (SEISMIC_MENU)	Settings Menu (SETTINGS_MENU)
Style Menu (STYLE_MENU)	Title/Axes Menu (TITLE_MENU)

Units Menu (UNITS_MENU)	Write Menu (WRITE_MENU)
Page Layout	
1 – Page Layout Tile Tall (TILE_TALL)	**2** – Page Layout Tile Wide (TILE_WIDE)
3 – Page Layout Tile Cascade (CASCADE)	**4** – Page Layout Tile 1x1 (LAYOUT_1X1)
5 – Page Layout Tile 2x2 (LAYOUT_2X2)	**6** – Page Layout Tile 3x3 (LAYOUT_3X3)
Miscellaneous	
G/g – Create a new graph Window (NEW_WINDOW)	**T/t** – Tidy Menus (TIDY_MENUS)
V/v – Change Curve Picking Group (CURVE_GROUP)	**Q/q** – Swap to Quick Pick (QUICK_PICK)
PAGE UP – Next Page	**PAGE DOWN** – Previous Page
HOME – First Page	**END** – Last Page
Change edit to next point (EDIT_NEXT)	**0** – Copy Axis Settings (COPY_AXIS)

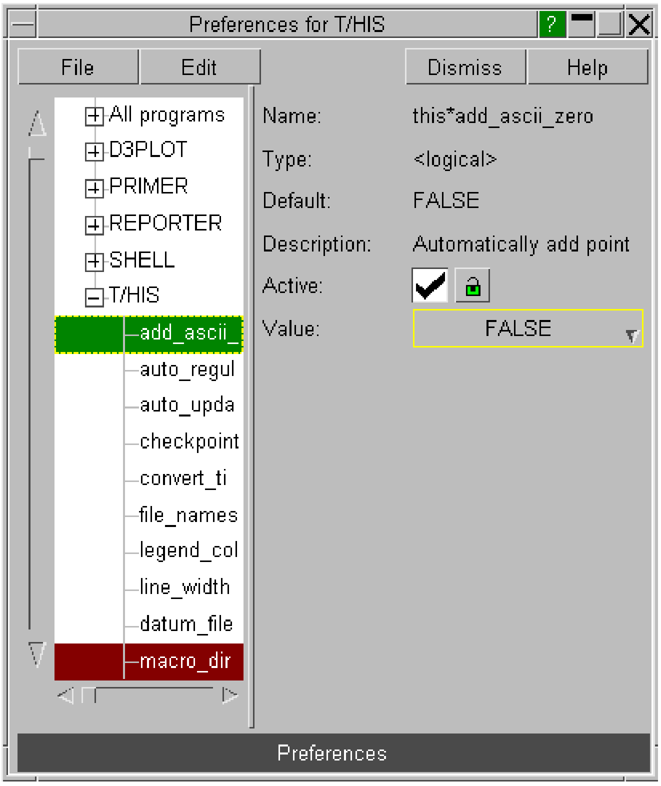
9.6. Preferences

Preferences

The Preference menu provides an interactive editor for setting options for T/HIS in the oa_pref preference file (see Appendix H for more details on the oa_pref file/options).

The preferences editor reads an XML file that contains all possible preferences and their valid options, and allows you to change them at will. In this example the user is changing the background colour in T/HIS.

Note that changes made in the Preferences editor will not affect the current session of T/HIS, they will only take effect the next time it is run.



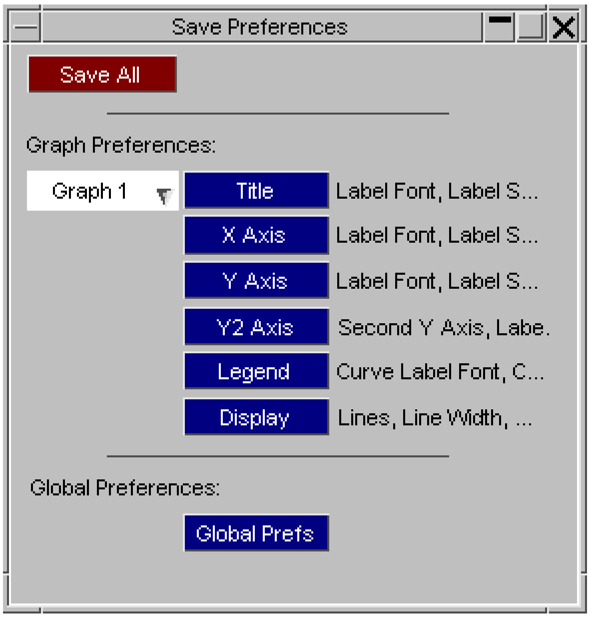
9.6.1. Save Preferences Popup

Save Preferences Popup

The Save Preferences menu allows a means to quickly save graph properties straight to the oa_pref file.

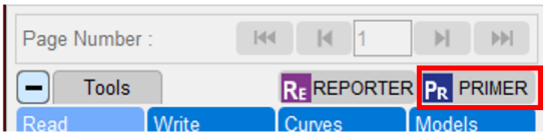
The popup works by reading the graph (defaults to Graph 1) properties to take preferences from. When a save button is pressed (**Title**, **X Axis**, **Y Axis**, etc..), the menu will look for changes to the relevant preferences and print those preferences to the oa_pref file.

For example, by pressing on **Display** will save (most of) the options setup in the Display menu from the specified graph into your oa_pref file.



9.7. PRIMER: Synchronising with PRIMER

PRIMER: Synchronising with PRIMER



T/HIS can be synchronised with PRIMER using a shared memory link. This means that a post-processing model that is open in T/HIS can have its corresponding keyword file open in PRIMER, and information can be exchanged between the two programs.

By default, no link takes place, but it can be opened in any of the following ways:

- A running T/HIS session starts a new PRIMER session using the stipulated model.
- A running PRIMER session starts a new T/HIS session using the stipulated model. and
- Once a link is established, in either of the modes above, further models can be opened and linked at will.

The link is symmetrical and bi-directional, with no concept of parent or child, and it can be closed at any time leaving both programs running autonomously. What you ***can't*** do at present is to link an autonomous, already running, T/HIS or PRIMER session with another autonomous session.

9.7.1. The PRE panel

The PRE panel

When running linked with PRIMER the Pre panel (invoked by pressing the PRIMER button) shows the current status of the link. In this example we have four models open in T/HIS, and in this example:

- Models 1 and 2 are currently open in PRIMER
- Model 3 is not open in PRIMER, but a keyword file has been found automatically.
- Model 4 is also not open in PRIMER, and T/HIS has not found a keyword file automatically.

The file open/close options are

Option	**Status of model**	**Action performed**
Add to PRIMER	Not linked	T/HIS has found a keyword file automatically, add this model to PRIMER
Find KW file	Not linked	T/HIS cannot find a keyword file, browse for a filename manually
Disconnect	Linked	Model is linked with PRIMER session, disconnect it

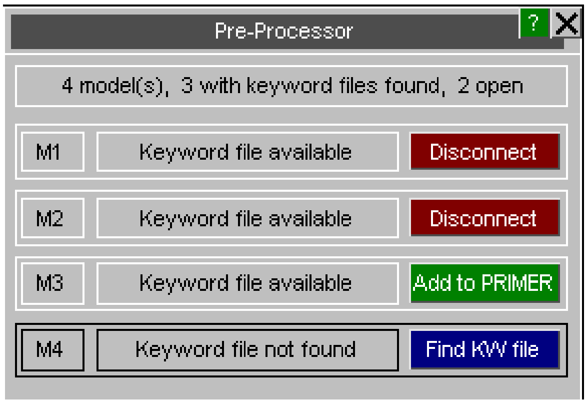
There is a corresponding Post panel in PRIMER, with the same layout and functionality.

Effects of linking and unlinking models

In all cases:

- Linking or disconnecting a model does not affect that model's status in either programme, both T/HIS and PRIMER will continue to run normally.
- Models may be disconnected and reconnected at will.
- When a model is deleted in T/HIS it is implicitly disconnected in PRIMER, but will not be deleted from PRIMER. Similarly, if a model is deleted in PRIMER is will be disconnected from T/HIS, but not deleted.
- The link logic attempts to keep model numbers the same in both PRIMER and T/HIS, however it is possible to defeat this by opening additional models in one programme but not the other. Doing so may cause the link to fail in some respects (so it is not recommended!).

The PRE panel can be opened or closed at will without affecting the status of linked models, it simply provides feedback about the current status and attributes of linked models.


Pre-Processor
?
4 model(s), 3 with keyword files found, 2 open
M1
Keyword file available
Disconnect
M2
Keyword file available
Disconnect
M3
Keyword file available
Add to PRIMER
M4
Keyword file not found
Find KW file

9.7.2. Locating keyword deck in T/HIS

Locating keyword deck in T/HIS

T/HIS can automatically locate an associated LS-DYNA keyword file to load in a linked PRIMER session. Model name is written to the ztf file and if this model exists then it is auto-loaded in PRIMER. If the .ztf file is missing, the approach depends on filename convention:

If you use the Ansys / LST results filename convention (d3thdt, xtfile):

- T/HIS looks to see if there is a single LS-DYNA keyword file (.key/.k/.kby or a .gz/.zip variant thereof) in the working directory and auto-loads it in PRIMER.

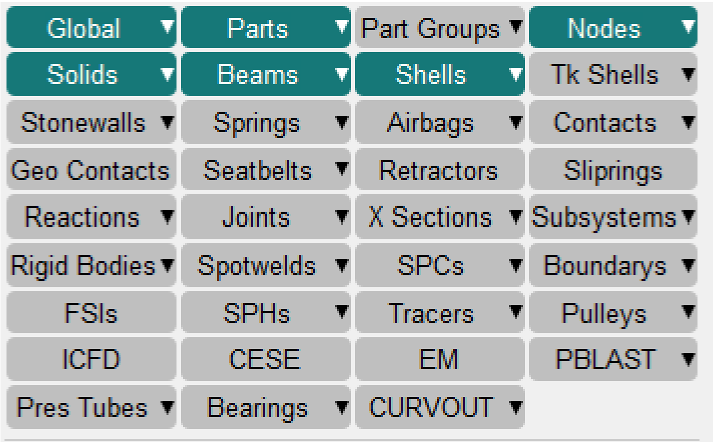
If you use the Oasys results filename convention (.thf, .xtf):

- T/HIS looks to see if a [job].thf has been loaded. If yes, T/HIS looks for a matching [job].key/.k/.kby or a .gz/.zip variant thereof.
- If a [job].thf is not found, T/HIS tries a similar logic with a potential [job].xtf file.
- Failing all of that, T/HIS looks to see if there is a single LS-DYNA keyword file in the working directory.
- The final fall-back, as always, is for you to manually select an input deck to load in PRIMER.

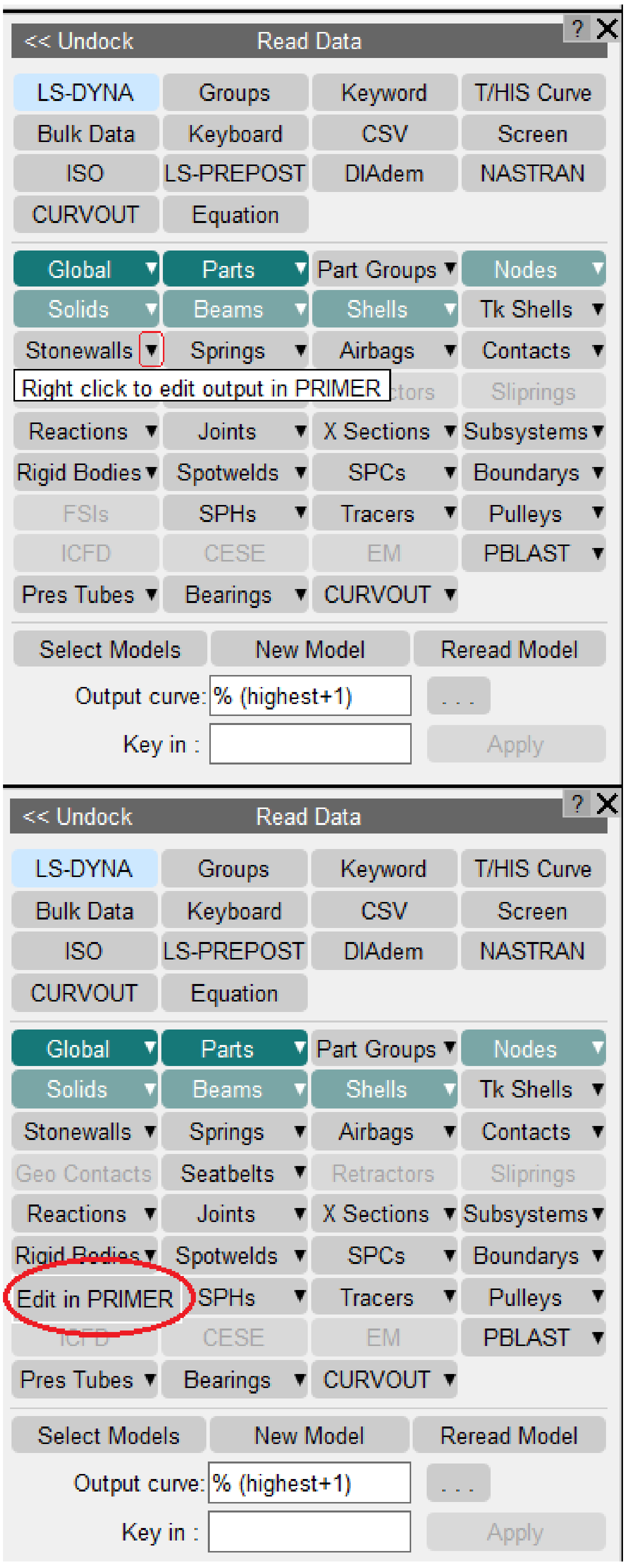
9.7.3. Highlight output database cards in PRIMER

Highlight output database cards in PRIMER

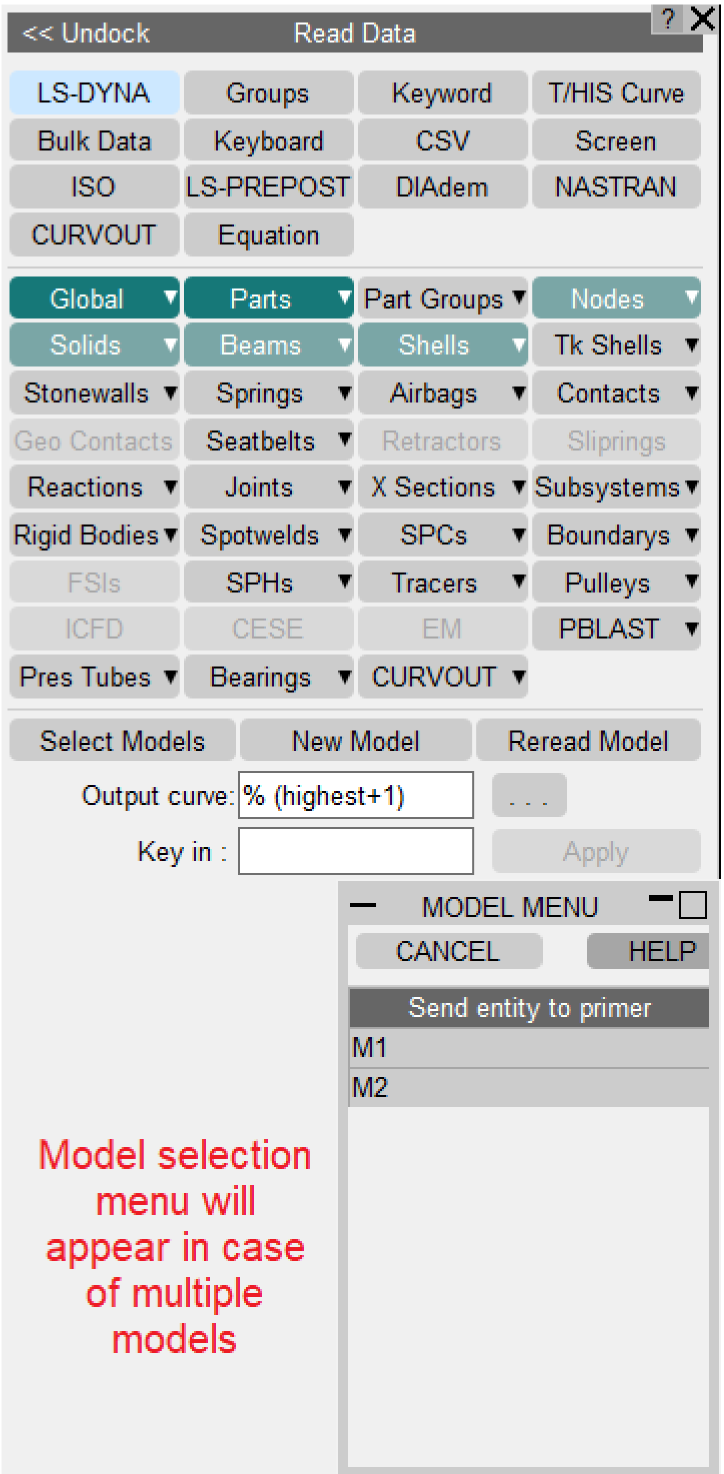
Most of the entity buttons in the Read Data panel have a dropdown option. If you click on an entity with right mouse button, an **Edit in PRIMER** option will be displayed. This option will open PRIMER (if it is not linked already) and will highlight the database cards required for this specific entity. You can turn on the required database cards with relevant values and use them in your next LS-DYNA run to get results related to that specific entity in T/HIS. PRIMER will always highlight the database cards for entities in the same order as they are clicked in T/HIS. For more reference, see the images mentioned below.



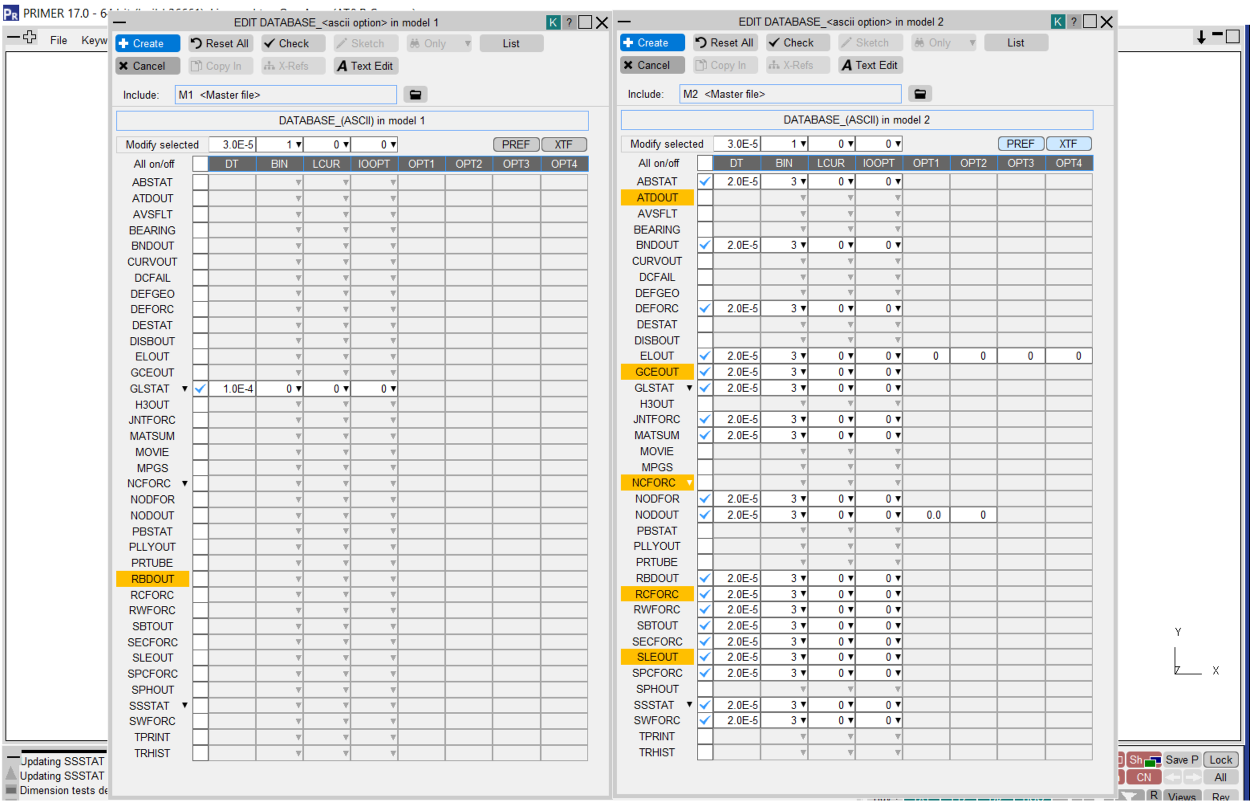
Hover text and Popup option are shown in the below images.



This feature is also available for multiple models. In case of multiple models, after you click on "Edit in PRIMER", a model selection menu will be mapped which will give you a choice to select the model for which you want database cards to be highlighted. The model menu is shown in the image below.



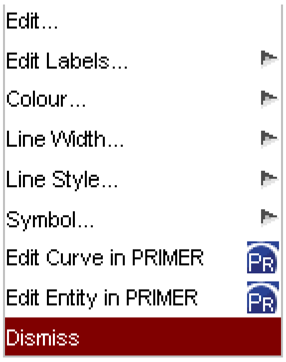
After model selection, PRIMER will open and highlight the required database cards. You need to turn on the database cards with relevant values so that these values can be used in the next LS-DYNA run.


PRIMER 17.0
EDIT DATABASE_<ascii option> in model 1
Create
Reset All
Check
Sketch
Only
List
Cancel
Copy In
X-Refs
Text Edit
Include: M1 <Master file>
DATABASE_(ASCII) in model 1
Modify selected
PREF
XTF
All on/off
DT
BIN
LCUR
IOOPT
OPT1
OPT2
OPT3
OPT4
ABSTAT
ATDOUT
AVSFLT
BEARING
BNDOUT
CURVOUT
DCFAIL
DEFGEO
DEFORC
DESTAT
DISBOUT
ELOUT
GCEOUT
GLSTAT
H3OUT
JNTFORC
MATSUM
MOVIE
MPGS
NCFORC
NODFOR
NODOUT
PBSTAT
PLLYOUT
PRTUBE
RBDOUT
RCFORC
RWFORC
SBTOUT
SECFORC
SLEOUT
SPCFORC
SPHOUT
SSSTAT
SWFORC
TPRINT
TRHIST
EDIT DATABASE_<ascii option> in model 2
Include: M2 <Master file>
DATABASE_(ASCII) in model 2
Updating SSSTAT
Updating SSSTAT
Dimension tests de
Save P
Lock
All
Views

9.7.4. Synchronising Operations

Synchronising Operations

Load curves and data can be exchanged across the link using the following methods. Like other Quick Pick commands this may be set as the current operations, or selected from a menu of choices (as shown here) in response to a right-click.

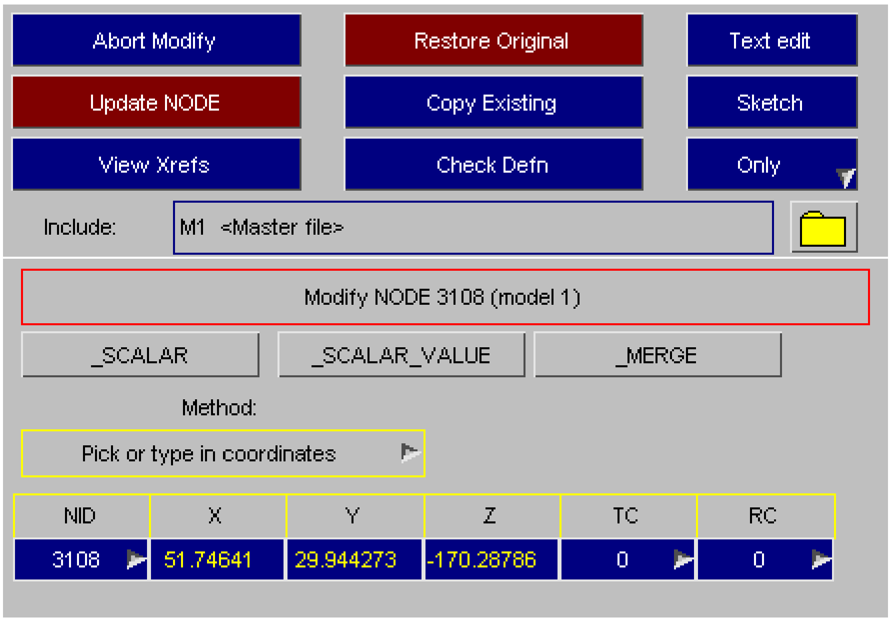


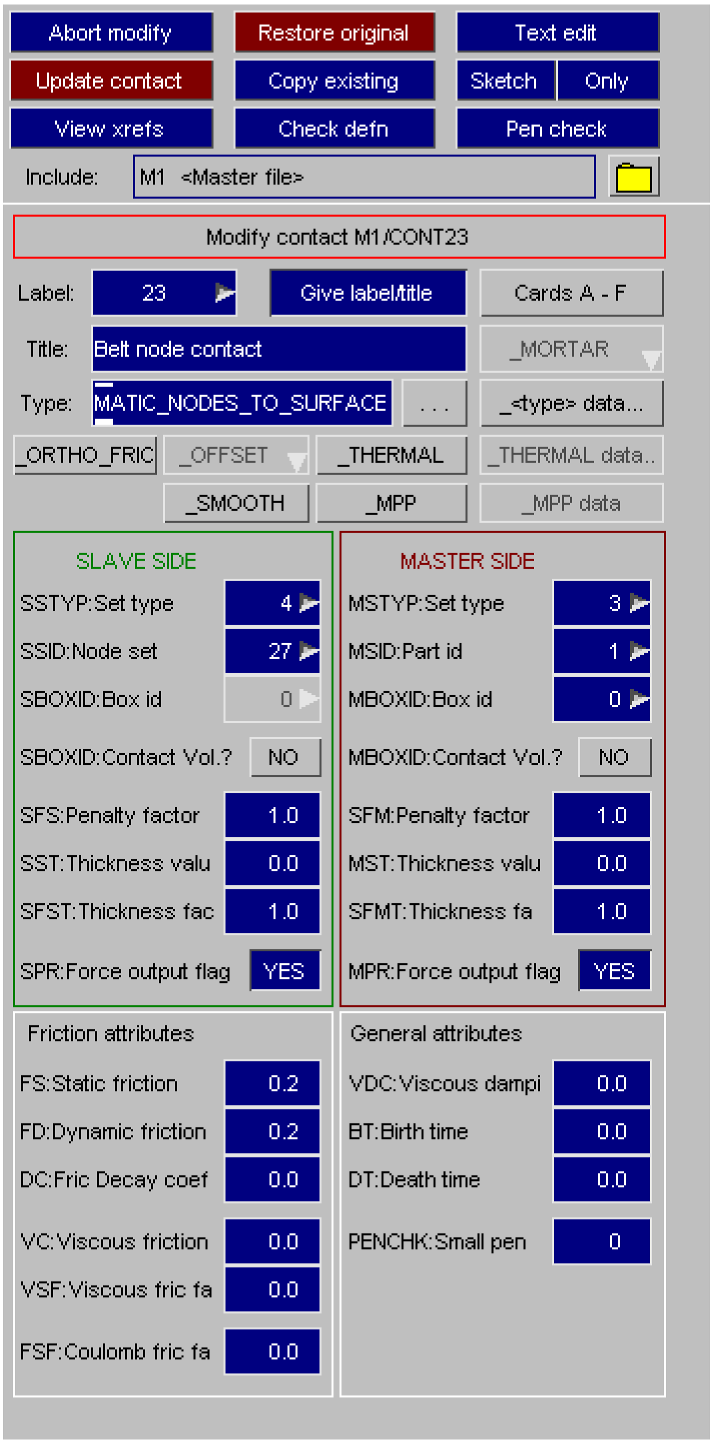
Edit Load Curve in PRIMER

Select one or more curve to send to PRIMER. The DEFINE_CURVE Edit panel is launched in the linked PRIMER session.

Edit Curve Entity in PRIMER

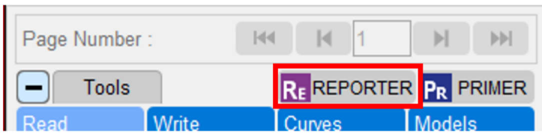
Select one or more curves to call up the corresponding Edit panel in PRIMER, so if for example you clicked on a curve that was an acceleration for a node you would get the NODE Editor in PRIMER or if you clicked on a contact force curve you would get the CONTACT Editor in PRIMER.




Abort modify
Restore original
Text edit
Update contact
Copy existing
Sketch
Only
View xrefs
Check defn
Pen check
Include:
M1 <Master file>
Modify contact M1/CONT23
Label:
23
Give label/title
Cards A - F
Title:
Belt node contact
_MORTAR
Type:
MATIC_NODES_TO_SURFACE
. . .
_<type> data...
_ORTHO_FRIC
_OFFSET
_THERMAL
_THERMAL data..
_SMOOTH
_MPP
_MPP data
SLAVE SIDE
SSTYP:Set type
4
SSID:Node set
27
SBOXID:Box id
0
SBOXID:Contact Vol.?
NO
SFS:Penalty factor
1.0
SST:Thickness valu
0.0
SFST:Thickness fac
1.0
SPR:Force output flag
YES
MASTER SIDE
MSTYP:Set type
3
MSID:Part id
1
MBOXID:Box id
0
MBOXID:Contact Vol.?
NO
SFM:Penalty factor
1.0
MST:Thickness valu
0.0
SFMT:Thickness fa
1.0
MPR:Force output flag
YES
Friction attributes
FS:Static friction
0.2
FD:Dynamic friction
0.2
DC:Fric Decay coef
0.0
VC:Viscous friction
0.0
VSF:Viscous fric fa
0.0
FSF:Coulomb fric fa
0.0
General attributes
VDC:Viscous dampi
0.0
BT:Birth time
0.0
DT:Death time
0.0
PENCHK:Small pen
0

9.8. REPORTER: Integrating with REPORTER

REPORTER: Integrating with REPORTER



T/HIS 17.0 onwards can be linked to REPORTER with a shared memory link, allowing reports to be interactively created and edited. For more information on this, see REPORTER Integration .

10. FAST-TCF

10.1. FAST-TCF Overview

FAST-TCF OVERVIEW

FAST-TCF is a scripting language for T/HIS. It is designed to be editable and backward-compatible with previous versions of T/HIS. From T/HIS 9.2, FAST-TCF scripts can be recorded and played back. The FAST-TCF scripts are plain text files, and are therefore easy to edit and manipulate.

10.1.1. New Features

NEW FEATURES

New Features for FAST-TCF version 11.0

Version 11 of T/HIS contains the following new FAST-TCF commands

- Support for DISBOUT data component
- Support for PLLYOUT data components
- "style_m" command for setting curve styles by model

New Features for FAST-TCF version 10.0

Version 10 of T/HIS contains the following new FAST-TCF commands

- Support for TRHIST data components
- Support for CPM_SENSOR data components
- New wildcard options for specifying curve tags
- Ouputting a range of curves to curve file

New Features for FAST-TCF version 9.4

Version 9.4 of T/HIS contains the following new FAST-TCF commands

- Support for DBFSI data components
- Support for TPRINT data components
- New "plot setup" commands
- New curve style options

New Features for FAST-TCF version 9.3

Because of the multiple graphs and pages available in T/HIS 9.3 additional commands have been added to FAST-TCF 93 to define and position graphs and to generate multiple images containing one or more graphs. Because of these new commands version 9.3 FAST-TCF scripts generated by T/HIS can not be used in previous releases.

- New commands have been added for generating and positioning multiple graphs and pages.
- New commands for generating images containing multiple graphs and pages.
- New variables have been added for accessing the output values of the ERR command.
- New built in variables "$run_nameN", "$run_titleN" and "$run_dirN" for multiple models.
- New built in variable"$FTCF_PATH"

New Features for FAST-TCF version 9.2

FAST-TCF has been extensively revised to include almost all of the T/HIS commands. The improved functionality does mean that old scripts may have to be changed to meet the new standards.

NOTE: FAST-TCF is not 100% compatible with pre-version 9.1 input scripts:

- Variables have changed to allow more flexibility, but the old rule for filenames (word1 + word2) has now been discontinued, filenames must all be one word
- Rigidwall command must now have "n" for the xtf file output (rather than nothing at all)
- Shell and Solid effective strain must have the fourth word "eff" to distinguish them from other types of strain that have been added
- No FAST-TCF defaults for plot setup - defaults are now the T/HIS standard ones

New features since version 9.1:

- Reading of keyword, csv, csv2, and bulk data files, keyboard entry
- Operation commands "order", "cat", "r_ave", "stress", "logx", "logx10", "translate", "vector2D", "window"
- Variables are processed on a line by line basis
- Variables can be defined using curve properties - for example a variable could be set to equal max of a curve, and then used to divide another curve
- Continuation lines added - defined using a "\" at the very end of a line
- Tabulation commands "yatmax" and "yatmin" added for Y values at maximum and minimum X
- All extraction commands are supported: Boundary, Geo contacts, FSI, Joints, SPH, Thick shells and so on
- All the missing components for previous data types are now supported
- Multiple data extraction on one line e.g. "node 100:last acc X"
- Multiple generic tagging and labeling of output curves using wildcard "*"
- Multiple curves can be operated upon in one line e.g. "oper ADD acc_* 10.0"
- Multiple curves can be plotted using wildcards "*" in tag names
- Integration point output can be changed
- Multiple models supported
- Extended plotting syntax for setting up plot defaults (grid colours, offsets, fonts and so on)
- "Tabc" command for writing out tabulation data to a csv file
- "plot" and "auto" commands added for use in interactive playback mode
- macro support for running FAST-TCF files on specific curves

10.2. FAST-TCF Introduction

10.2.1. General Rules

General Rules

1. **Each line** in the input file defines **one** data extraction or plot request
2. Long lines can be split into shorter ones using a continuation character "\" at the end of each line
3. **Space characters** are used to **divide the line into 'words'**
4. The input script is NOT case-sensitive.
5. Unless detailed elsewhere in this manual, the first few (usually three) characters of the first word on the line discriminate the request of a particular entity, and the syntax which applies to reading in the remaining words on the line
6. If the first word on the line is not recognised, the program ignores it - it is treated as a comment
7. The last words on the data extraction request lines allow options for filtering, Y-axis scaling, HIC, average and a short reference tag (The tags may be used for operation and plotting requests)
8. The last words on the plotting request line allow options for title, line style and axis changes
9. A successful data extraction always has a curve outputted, if there is no output (e.g. HIC, ERR) then a duplicate curve is outputted. This helps with tagging output curves

10.2.2. Running FAST-TCF

Running FAST-TCF

Automatic running

FAST-TCF is integrated into the T/HIS executable and can be accessed from the command line or SHELL.

Command line syntax:

```
<this executable> -tcf=<FAST-TCF input file> -start_in=<start
directory> -exit-batch <thf file name>
```

e.g. `this93.exe -tcf=side_impact.tcf -start_in=e:\side_impact\run1 -exit run1.thf`

The `<thf file file>`, `-start_in`, `-exit` and `-batch` syntax are all optional.

NOTES:

- If no THF file is specified then T/HIS will search the directory for the latest one (*.thf).
- If no THF file exists, then T/HIS will look for a d3thdt file (xtf file = xtfile).
- If this does not exist then no thf or xtf input filename is passed to FAST-TCF, and the input file is defaulted to ASCII
- The program runs in any directory you like (via the `-start_in` command line option). The FAST-TCF output files are created in that directory, and files written out are relative to that directory.

Instead of opening a single model multiple models can be read using the command line option

```
<this executable> -tcf=<FAST-TCF input file> -start_in=<start
directory> -exit-batch -model_list=<file name>
```

The `-model_list` expects a text file with a list of filenames (1 per line) to read into model slots within T/HIS.

e.g

```
e:\side_imnpact\run1\run1.thf
e:\side_imnpact\run2\run2.thf
e:\side_imnpact\run3\run3.thf
e:\side_imnpact\run4\run4.thf
```

SHELL operation:

In SHELL, click the **Options...** button below the T/HIS icon. Select the input data file, FAST-TCF input script and any other required options as necessary. Press **Run** at the bottom of the panel to launch T/HIS and automatically run the FAST-TCF script.

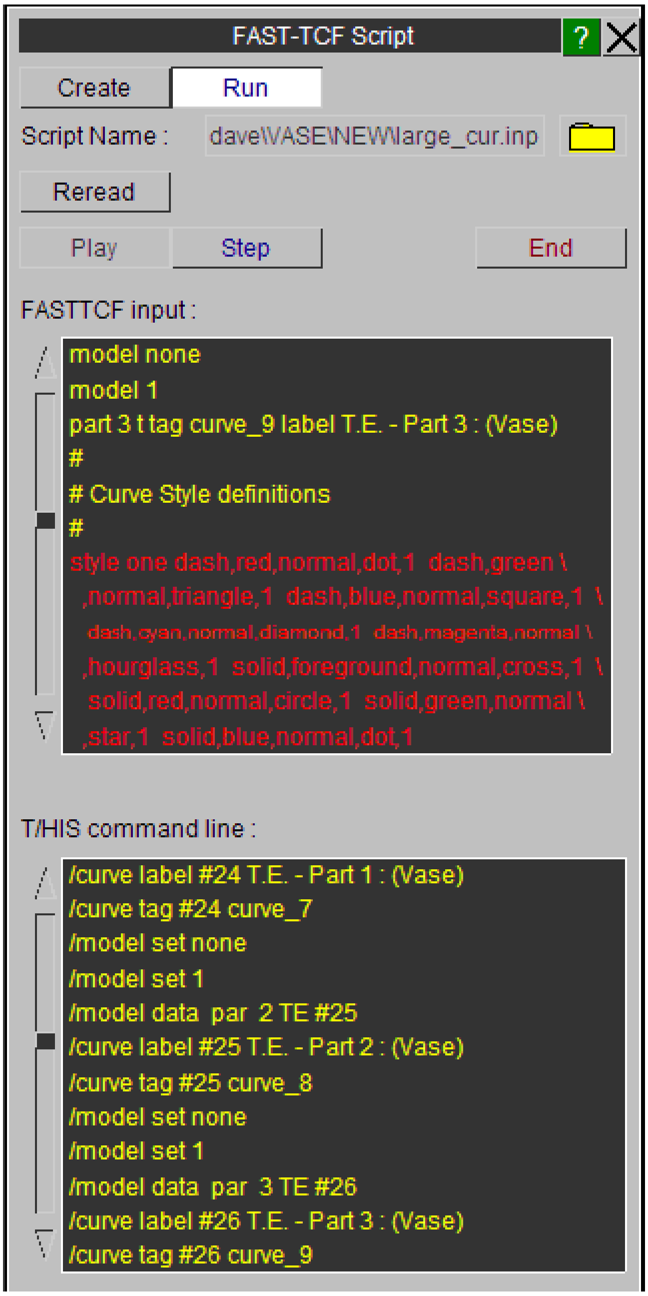
Interactive running

In the Tools menu within T/HIS, select the FAST-TCF option, then click on the **Run** tab in the sub-menu that appears. This brings up the following menu:

The user can select the script file then with play the whole file through, or step through each command one by one.

The FAST-TCF line appears in the top dialogue box, and the translated T/HIS line appears in the bottom box. The line about to be sent to T/HIS appears in red text.

To end the script prematurely, hit the **End** button.



10.2.3. Input Files Needed, and Output and Intermediate Files Created

Input Files Needed, and Output and Intermediate Files Created

1. ***input_script*** is **required** at the start.
2. *input_script* .output is a file that contains the concatenated output from FAST-TCF.
3. *input_script* .tmp is a temporary file that FAST-TCF creates for translation. This is merged after completion into *input_script* .output so if you see this file then FAST-TCF didn't finish cleanly.
4. *input_script* .rep is a temporary report file of the FAST-TCF run. This is merged after completion into *input_script* .output so if you see this file then FAST-TCF didn't finish cleanly.
5. *input_script* .tcf are the commands passed to T/HIS from FAST-TCF. This is merged after completion into *input_script* .output so if you see this file then FAST-TCF didn't finish cleanly. The command lines contain special characters such as \r, \m and \l. These are used internally in T/HIS and should be ignored by the user.
6. *input_script* .sngval contains summaries of every curve outputted.

Other files will be made, such as postscript or bitmap plots, but these will have names specified by the user.

10.2.4. Debugging FAST-TCF Files

Debugging FAST-TCF files

Complicated FAST-TCF files will inevitably go wrong. There are a number of things the user can do to help identify where it is going wrong. Assuming the command line syntax is correct and the correct files are in the run directory, these typical procedures are as follows:

Identifying errors using the interactive playback option in T/HIS:

- Read the model(s) into T/HIS.
- Read the FAST-TCF script into T/HIS under the "FAST-TCF > Run" sub menu.
- Step through the FAST-TCF script manually, keeping an eye on how FAST-TCF is translating the lines, and the output T/HIS is producing.

Identifying if FAST-TCF has found an error:

- If FAST-TCF finds an error, then it is stored and T/HIS then resets the command line and continues to translate the input file. If 10 errors are found then T/HIS will stop at this line. You can set this error amount internally within FAST-TCF.
- Once T/HIS has stopped, the errors are summarised in the command line box and the terminal that T/HIS was run from. The number of warnings found is also printed.
- It should be obvious what is wrong, FAST-TCF checks numerous things, including:
    - Whether T/HIS created the curve from the previous line.
    - That the syntax is correct for all the data input lines (the data extraction requests have additional checking to check the combinations of words inputted is right).
    - If the syntax is correct, whether it applies to the file being requested for output.
    - The output file exists in the directory for the data extraction.
- Correct the input line error utilising the reference tables in this document if applicable.

Identifying what errors T/HIS is giving:

- Identify how many curves were outputted into T/HIS before things went wrong (run T/HIS in graphical mode).
- Place an exit keyword **after the next** input line. This should stop T/HIS just after the line which is causing the file to fail.
- Check what errors T/HIS is giving out. If it's not obvious what went wrong then try another procedure.

Identifying if there are warnings or errors from FAST-TCF:

- The errors are summarised once T/HIS has finished. They are printed in the command line box and the terminal which T/HIS was run from.
- There will be a *input_file* .rep or *input_file* .output file in the directory which contains any warnings or errors that FAST-TCF has detected. Make sure nothing is obviously wrong with the input file using this report file.
- The *input_file.* tmp or *input_file* .output file contains the actual file inputted into FAST-TCF after includes have been found and special characters removed. Check this is correct and all the include files have been accounted for.

Identifying if FAST-TCF is processing the line correctly:

- It's possible that FAST-TCF has processed the line incorrectly. If so, open the *input_file* .tcf or *input_file* .output file to investigate what FAST-TCF is asking T/HIS to do.
- Identify which line is going wrong using the above procedure, and then find this section in the .tcf file. Input the entire tcf request for the line into the T/HIS command box to step through what is being asked from T/HIS. This may highlight where things are going wrong. The command lines contain special characters such as \r, \m and \l. These are used internally in T/HIS and should be ignored by the user.

Using PRIMER to check a FAST-TCF file:

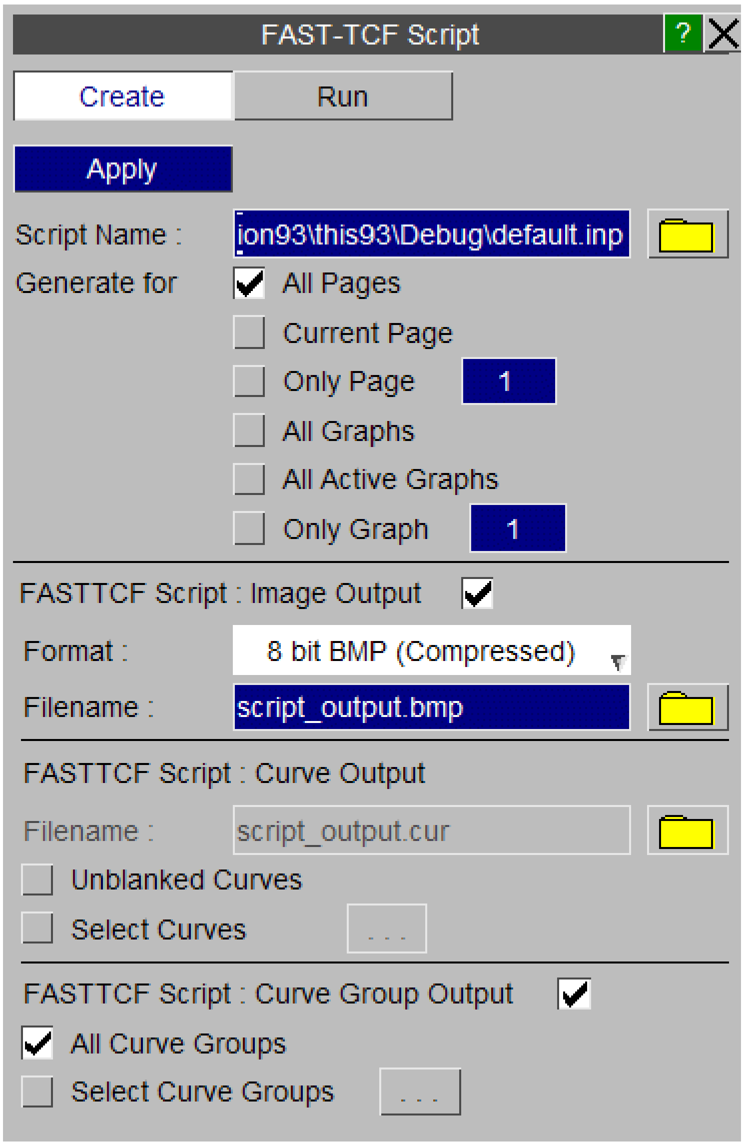
- PRIMER has a FAST-TCF Check menu under the main Check menu. This can be used to check the FAST-TCF file data requests against a certain keyword deck.
- Read the deck into PRIMER, and select **Model** > **Check** > **FAST-TCF**. Select the FAST-TCF file and press **Apply**. Details can be found in the PRIMER manual.
- PRIMER will highlight any errors that have occurred with the input file with regards to the keyword deck.
- The main PRIMER checks are if the line syntax is valid, whether the correct file is being outputted, whether the relevant DATABASE_HISTORY is present and whether the id. actually exists.
- Any errors will have to be corrected manually in PRIMER.

NOTE: If FAST-TCF has completed, then it may be necessary to open the *input_file* .output file which has the all the output files concatenated together in different sections.

10.2.5. Creating FAST-TCF Files

Creating FAST-TCF files

The most obvious option is to generate a FAST-TCF script using a text editor such as vim or wordpad. However, an easier option is to use T/HIS as normal, then generate a FAST-TCF script to recreate the curves currently displayed on the screen from within T/HIS.



It involves a single button click to produce a FAST-TCF script that can recreate the plot on the screen.

T/HIS internally stores the history behind each curve; noting which curves, operations and data requests were used to create each curve. This means that the user **does not** have to start recording a command file, and carefully record a script. Instead the user can work for as long as they like as normal, then choose to generate a FAST-TCF script to recreate the plot on the screen by using the FAST-TCF Create menu.

By default the FAST-TCF script that is generated will contain commands to reproduce all of the graphs that are currently defined in T/HIS. Instead of reproducing all of the graphs the FAST-TCF script can also contain the commands to generate a subset of pages or graphs.

FAST-TCF Script : Image Output

This option can be used to add the commands to the FAST-TCF script to generate an image of each graph/page that is selected for output. In addition to selecting the image format a filename can also be specified that is used in the FAST-TCF script as the output filename for images.

FAST-TCF Script : Curve Output

This option can be used to add commands to the FAST-TCF script to write curves out to a T/HIS curve file. By default this option will add commands to the FAST-TCF script write any curves that are unblanked in a graph to a curve file. Instead of writing all of the unblanked curves out to a file the **Select Curves** option can be used to select a subset of curves.

FAST-TCF Script : Curve Group Output

This option can be used to select additional curves for output to the FAST-TCF script by curve group. If a curve is selected that is also unblanked in one of the graphs the command to regenerate it are only added to the FAST-TCF script once. This option will also add the commands to regenerate the selected curve groups to the FAST-TCF script.

T/HIS

10.3. Page/Graph Layout and Selection

PAGE / GRAPH LAYOUT AND SELECTION

FAST-TCF scripts can contain commands to create and position multiple graphs. T/HIS Pages can also be created and graphs moved between pages. By default T/HIS will automatically create a single graph on the 1st 'Page' when it starts. If a single graph is required then the script does not need to contain any of the commands in this section. If additional graphs are required then by default they will be created on the 1st Page unless multiple pages have been selected.

Keyword	2nd word	3rd word	4th word	5th word	6th word	7th word	8th word	9th word	notes
Layout	page	wide	-	-	-	-	-	-	Set the page layout to tile wide
		tall	-	-	-	-	-	-	Set the page layout to tile tall
		cascade	-	-	-	-	-	-	Set the page layout to cascade
		1x1	-	-	-	-	-	-	Set the page layout to 1 by 1 graphs per page
		2x2	-	-	-	-	-	-	Set the page layout to 2 by 2 graphs per page
		3x3	-	-	-	-	-	-	Set the page layout to 3 by 3 graphs per page
		XY	m	n	-	-	-	-	Set the page layout to (m) by (n) graphs per page
		custom	-	-	-	-	-	-	Set the page layout to custom
		n	all	-	-	-	-	-	Add all graphs to page (n)

		n	none	-	-	-	-	-	Remove all graphs from page (n)
		n	add	graph	ID	-	-	-	Add graph (ID) to page (n)
		n	remove	graph	ID	-	-	-	Remove graph (ID) from page (n)
		size	m	n	-	-	-	-	Set the page size to m by n pixels
		size	auto	-	-	-	-	-	Set the page size to automatic
	graph	total	n	-	-	-	-	-	Set the total number of graphs to (n)
		create	-	-	-	-	-	-	Create a new graph
		delete	all	-	-	-	-	-	Deletes all graphs except the first one.
		delete	n	-	-	-	-	-	Delete graph (n)
		position	n	x1,y1	x2,y2	-	-	-	Position graph (n) with the bottom left hand corner at screen location (x1,y1) and the top right hand corner at (x2,y2). All coordinates should be in the range 0.0 to 1.0.
		select	all	-	-	-	-	-	Select all graphs
		select	n	-	-	-	-	-	Select graph (n)
		select	none	-	-	-	-	-	Deselect all graphs

		n	axes	position	left	right	top	bottom	Set the position of the left, right, top and bottom axis for graph (n). The positions given should be in the range 0.0 to 1.0 or the word 'Auto'.
		n / all	legend	position	left	right	top	bottom	Set the position of the left, right, top and bottom of the legend for graph (n) or all graphs. The positions given should be in the range 0.0 to 1.0 or the word 'Auto'
		n / all	legend	format	*<type>*	-	-	-	Set the legend format to one of *column/default, full/off, automatic, floating* for graph (n)
		n / all	legend	columns	*m*	-	-	-	Set the number of columns in the legend to m (1 to 3) for graph (n) or all graphs
		n / all	legend	background	standard colour	-	-	-	Set a background colour for the floating legend for

									graph (n) or all graphs
		n / all	legend	transparency	integer (0-100)	-	-	-	Set the background transparency for the floating legend for graph (n) or all graphs
		n / all	x	format	<*type*>	-	-	-	Set the x axis unit format to one of *automatic, general, scientific* for graph (n) or all graphs
		n / all	y	format	<*type*>	-	-	-	Set the y axis unit format to one of *automatic, general, scientific* for graph (n) or all graphs
		n / all	y2	format	<*type*>	-	-	-	Set the second y axis unit format to one of *automatic, general, scientific* for graph (n) or all graphs
		n / all	x	precision	m	-	-	-	Set the number of decimal places displayed for the x axis values to (m) in graph (n) or all graphs
		n / all	y	precision	m	-	-	-	Set the number of decimal places

T/HIS

									displayed for the y axis values to (m) in graph (n) or all graphs
		n / all	y2	precision	m	-	-	-	Set the number of decimal places displayed for the second y axis values to (m) in graph (n) or all graphs

10.4. Input Syntax to Load Other Files

INPUT SYNTAX TO LOAD OTHER FILES

FAST-TCF has the option of reading in curve files and other FAST-TCF files nested within the input file. T/HIS now writes out and reads in curve styles and internal tags. FAST-TCF recognizes these tags if the user wishes to refer to them later on in the input file. If they are relative then the include files must be relative to where T/HIS is running from.

Filenames can contain spaces, but if they do then they **must** be enclosed in quotes

e.g. read "c:\my documents\filename.cur".

Description	**keyword**	**second word**	**third word onwards**	**notes**
Bulk data	readb	bulk data file	-	curves will be read in at this point in the file, and will be numbered accordingly
CSV 1 (X,Y,X,Y...)	readcsv	csv file	**lr** <row number containing line labels>	Subsequent words can be any of these 2 options. If no options then assumes reading x from column 1 and no labels.
			ar <row number containing axis labels>	
CSV 2 (X,Y,Y,Y...)	readcsv2	csv file type 2	**xg** <x start value> <x interval>	Subsequent words can be any of the 3rd word options. O nly one of the options XG and XC can be used. If no options then assumes reading x from column 1 and no labels.
			xc <x values column number>	
			lr <row number containing line labels>	
			ar <row number containing axis labels>	
T/HIS Curve file	rea	curve name	-	curves will be read in at this point in the file, and will be numbered accordingly curve tags and styles are stored automatically through the $TAG and $STYLE lines NOTE: If the tag in the curve file conflicts with an existing tag, the tag is NOT read in

Keyword	readk	keyword filename	-	curves will be read in at this point in the file, and will be numbered accordingly
FAST-TCF Include	inc	include filename	-	FAST-TCF will search for includes within includes etc FAST-TCF pastes the include files into the final input file as soon as they are detected
LS-PrePost Curve file	readlspost	filename	-	Reads in curves from an LS-PREPOST curve file
LS-PrePost XY data file	readlsp_xy	filename	-	Reads in curves from an LS-PREPOST XY data file
DIAdem	read_diadem	header file filename	<channel number to read> OR "<channel name to read>" (must use quotes)	subsequent words can be either of these 2 options **xg** <x start value> <x interval> **xc** <channel containing x-axis> Only one of these 2 options can be used
JavaScript	java	JavaScript filename	-	Runs a JavaScript. If any curves created by the JavaScript are referenced by following command in the FAST-TCF script then the JavaScript should generate curves tags for the curves whih ccan then be used in the FAST-TCF script.

Keyboard entry can also be added into the FAST-TCF file, allowing for simple curves to be created in T/HIS. The keyword for this is **keyboard** . The order of the following words is important, and must be adhered to (see below). The continuation line character is useful here "\".

Keyword	following word	following word	notes
Keyboard	xaxis	x axis name	specifies the x axis label
	yaxis	y axis name	specifies the y axis label
	label	curve label	specifies the curve label

	data	xval,yval xval2,yval2 xval3, yval3 etc	no space between the x and y values, only a space between the **pairs** of values

for example, to create an acceleration curve with a straight line at value 1.0:

keyboard title straight line \ xaxis time \ yaxis accn \ label straight line at 1.0 \ data 0.000000,1.000000 \ 1.000000,1.000000

10.5. Input for Data Extraction Requests

INPUT FOR DATA EXTRACTION REQUESTS

Each data extraction request occupies one line, with the 'words' on the line separated by space characters.

The line starts with a keyword and the required arguments follow, then any optional requests can occur after the arguments (see later on in the manual).

ID can be a number **or a name** (enclosed in quotes ""), depending on whether the LS-DYNA version supports it in the relevant output file.

When writing out FAST-TCF scripts from T/HIS, there is an option in the 'Create' panel to write entity names (when they exist) in place of numeric IDs into any generated script.

Multiple data requests

T/HIS 9.2 onwards supports multiple data output syntax. T/HIS will read the data in one file pass, making it much quicker for larger runs. To use this in FAST-TCF you need to specify the range using a colon (:) and it must be in a single word. As well as the standard numbers you can use, there are some special words namely "all", "first" and "last" (see example).

e.g.	**whole_model**	**te**	**lsda**	
	(whole model)	(total energy)	(force lsda file)	
	node	**42**	**force**	**y_dir**
	(node extraction)	(i.d. 42)	(force in y-direction)	
	node	**"end of roof"**	**accel**	**z**
	(node extraction)	(i.d. "end of roof")	(z acceleration)	
	node	**100:last**	**force**	**y_dir**
	(node extraction)	all nodes from 100	(force in y-direction)	
	node	**all**	**force**	**y_dir**
	(node extraction)	all nodes	(force in y-direction)	

Potential Speedup for data extraction

In some situations, it may be possible to speedup the data extraction routines for FAST-TCF. A simple but effective change can be made to the FAST-TCF script such that it improves the efficiency of data extraction by an order of N items.

An example can be seen below trying to extract various nodes, tag them and then label it.

```
node 6600000 b y tag by6600000 label by6600000
node 6600001 b y tag by6600001 label by6600001
node 6600002 b y tag by6600002 label by6600002
node 6600003 b y tag by6600003 label by6600003
node 6600004 b y tag by6600004 label by6600004
```

While these are perfectly valid FAST-TCF lines and will parse correctly, these can be rewritten into:

```
node 6600000 b y tag by6600000
node 6600001 b y tag by6600001
node 6600002 b y tag by6600002
node 6600003 b y tag by6600003
node 6600004 b y tag by6600004
label by6600000 by6600000
label by6600002 by6600002
label by6600003 by6600003
label by6600004 by6600004
```

Externally, to a real person, these lines can be seen as equivalent to the first example. However, internally, it's another matter. In the first example, T/HIS would instead have to process the first line, come out of the reading loop and then back into it again to parse the second line and so on until completion. The change effectively allows T/HIS to bundle all of the "node" commands together, allowing them to be read in a singular, much more efficient pass and then apply the labels after this data extraction has been completed.

In this example, "label" is the additional option that has been given to the read line, however this would be the case for any other additional option. Unfortunately it's important to note that this means that the speedup will only work if the read line does not contain any additional options on it and the read commands are placed together like in the example. Any additional options that you may have must be separated from the read commands like the example above to work.

The speedup gained is directly linked with the number of items that are being read in so while you would see some gain for a small number of items, the speedup is much more noticeable when handling a large number of items.

10.5.1. Selecting Models

Selecting Models

If T/HIS contains more than one model the data extraction commands will attempt to read data from all the model that are currently selected. To specify which model to read data from the following commands can be used

Keyword	second word	notes
model	n	Select model "n" for reading data from
	all	Select all models for reading data from
	none	Unselect all models

10.5.2. Data Extraction Options

Data Extraction options

Specifying Files for data extraction

For some LS-DYNA data types results can be extracted from multiple files. By default FAST-TCF scripts will extract data from the default T/HIS file type for each entity type (see Data Sources). These defaults can be changed via the preference file.

Instead of using the default file any of the valid files types can be specified by using either the define file keyword (e.g. define file LSDA) or by adding an extra line option. When this occurs, FAST-TCF will take the extraction request from the specified type of file - **but only if T/HIS allows it.**

Keyword	second word	third word	notes
define	file	lsda	will always check that T/HIS can get the output from this file, if not then the original default file will be chosen (see data extraction table). This file can still be overwritten on the actual input line
		ascii	
		xtf	
		thf	
		default	

```
e.g. node 42 displacement x
```

(read data from default file)

```
define file LSDA
node 42 displacement x
```

(read data from LSDA file)

```
node 42 displacement x ASCII
```

(read data from ASCII file)

Specifying components for Steady State Dynamics (SSD) analysis

For a SSD analysis LS-DYNA generates 2 data values, an amplitude and an angle, for each component in the NODOUT and ELOUT parts of the LSDA (binout) file. By default FAST-TCF will extract the amplitude for each data component but this can be changed if required to extract the angle value.

Keyword	second word	third word	notes
define	ssd_comp	amplitude	selects the amplitude value for all following data requests
		angle	selects the angle value for all following data requests

e.g. define ssd_comp angle

(read angle value for all SSD analysis data components)

define ssd_comp amplitude

(read amplitude value for all SSD analysis data components)

10.5.3. Defining Groups of Parts

Defining Groups of Parts

Description	keyword	second word	following words
Group definition	gdef	group id	part ids
Add parts to group	gadd	group id	part ids

1. The line starts with `'gdef'` or `'gadd'` and is followed by an integer for the group i.d, and then part i.d. numbers separated by spaces, or for a range of parts - separated by a ':'.
2. No options should be applied to this card, because all the words on the line are written out as integers.
3. The input is on one line (which may result in a long line ...). If the line is too long (currently ~1000 characters) T/HIS will truncate the command and issue an error message. The `'gadd'` command is useful if the `'gdef'` command is too long to create a group on a single line.

e.g.	**gdef 1**	**1 2 3 4**	**10:20 30:40**
	(group define i.d. 1)	(parts 1 2 3 and 4)	(parts 10 to 20 and 30 to 40)
	gadd 1	**5 6 7 8**	**50:60**
	(group add i.d.	(parts 5 6 7 and 8)	(parts 50 to 60)

10.5.4. Specifying Surfaces, Integration Points and Nodal Locations for Data Extraction

Specifying Surfaces, Integration Points and Nodal Locations for data extraction

Specifying Surfaces and Integration Points

From T/HIS 12.0 onwards, the syntax for specifying which surface or integration point to read data from for Shells, Thick Shells and Beams has changed. These options are now appended to data extraction as follows.

Shells and Thick Shells

extra word #1	extra word #2	notes
surface	top	If no surface option is specified then the default (middle) surface will be used.
	middle	
	bottom	
	n	

e.g. **`shell 99 stress xx tag curve_1`**

(read x stress for shell 99 middle surface)

`shell 99 stress xx surface top tag curve_1`

(read x stress for shell 99 top surface)

`shell 99 stress xx surface 3 tag curve_1`

(read x stress for shell 99 layer 3)

Beams

extra word #1	extra word #2	notes
ipoint	n	Specifies the beam integration point to read data from

e.g. **`beam 99 stress x ipoint 1 tag curve_1`**

(read axial stress for beam 99 integration point 1)

Specifying in-plane integration points for Shells and Thick Shells

In recent versions of LS-DYNA it is possible to write out data at multiple in-plane integration points for fully integrated Shells and Thick Shells for each through thickness layer.

For fully integrated solid elements data can also be written out for all 8 integration points.

By default T/HIS will automatically read the average value for each element. If the element isn't fully integrated then the data for the 1st point will be used, if it is fully integrated and has multiple integration points then the average value will be calculated.

extra word #1	extra word #2	notes
ipoint	n	Specifies the in-plane integration point to read data from. If this option isn't specified then the surface centre value will be selected. If the element is fully integrated then the average value will be calculated from all 4 in-plane values

e.g. `shell 99 stress xx tag curve_1`

(read x stress for shell 99 middle surface, centre value)

`shell 99 stress xx ipoint 1 tag curve_1`

(read x stress for shell 99 middle surface in-plane integration point 1)

`shell 99 stress xx surface middle ipoint 1 tag curve_1`

(read x stress for shell 99 middle surface in-plane integration point 1)

`shell 99 stress xx surface 5 ipoint 2 tag curve_1`

(read x stress for shell 99 layer 5 in-plane integration point 2)

Specifying integration points for Solids

In recent versions of LS-DYNA it is possible to write out data at all 8 integration points or fully integrated solid elements.

By default T/HIS will automatically read the average value for each element. If the element isn't fully integrated then the data for the 1st point will be used, if it is fully integrated and has multiple integration points then the average value will be calculated.

extra word #1	extra word #2	notes
ipoint	n	Specifies the solid integration point to read data from. If this option isn't specified then the centre value will be selected. If the element is fully integrated then the average value will be calculated from all 8 values

e.g. `solid 99 stress xx tag curve_1`

(read x stress for solid 99 centre value)

```
solid 99 stress xx ipoint 1 tag curve_1
```

(read x stress for solid 99 integration point 1)

Selecting data at element nodal positions

In recent versions of LS-DYNA it is possible to write out data for Solid, Shells and Thick Shells that has been extrapolated from the integration points to the elements nodes.

For Shells the values at all through thickness layers can be extrapolated to the nodes. For Thick Shells the bottom surface values are extrapolated to nodes 1-4 and the top surface values are extrapolated to nodes 5-8.

extra word #1	extra word #2	notes
node	n	Specifies the element node number to read data for

```
e.g. shell 99 stress xx node 3
```

(read x stress for shell 99 middle surface extrapolated to node 3)

```
shell 99 stress xx surface 5 node 1 tag curve_1
```

(read x stress for shell 99 layer 5 extrapolated to node 1)

```
tshell 99 stress xx node 7 tag curve_1
```

(read x stress for thick shell 99 top surface extrapolated to node 7)

```
solid 99 stress xx node 4 tag curve_1
```

(read x stress for solid 99 extrapolated to node4)

10.5.5. Data Extraction Reference Table

Data extraction reference table

Data type	Keyword	Second word	Third word	Fourth word	Description
Airbag	air	Airbag id	[pr]essure	-	pressure
			[vo]lume	-	volume
			[ie]	-	internal energy
			[in]	-	mass flow rate in
			[ou]	-	mass flow rate out
			min	-	mass in
			mou	-	mass out
			[tm]	-	total mass
			[de]nsity	-	Density
			sa	-	Surface area
			[te]mp	-	Gas temperature
			rf	-	Reaction force
			maf	-	Mass flow rate through fabric
			mav	-	Mass flow rate through vent
			mof	-	Mass out through fabric
			mov	-	Mass out through vent
			tk	-	Translational Kinetic Energy
			dmp	-	Damping Energy
			pp	-	Average Particle Pressure
			if	-	Inflator Energy
Airbag CPM Part Data (ABSTAT_CPM)	ab_cpm_pa	Airbag id	Part id	[pr]essure	Pressure
				maf	Mass flow rate through fabric
				mav	Mass flow rate through vent
				ta	Total area
				[un]blocked	Unblocked area

				[te]mperature	Temperature
				ppr	Press s+
				npr	Press s-
				hc	Heat Convection Energy
				ev	Enhanced Vent Flag
				le	Leak Energy
				gas	Gas Flow Rate
				pvo	Por Volume
				pte	Part Temperature
Airbag CV Part Data (ABSTAT)	ab_cv_pa	Airbag id	Part id	[un]blocked	Unblocked area
				ba	Blocked area
				lk	Leakage
Airbag Chamber Data (ABSTAT_CHAMBER)	ab_chamber	Airbag id	Part id	[pr]essure	Pressure
				[vo]lume	Volume
				[de]nsity	Density
				ie	Internal Energy
				in	Mass flow rate in
				[ou]t	Mass flow rate out
				tm	Total mass
				sa	Surface area
				[te]mperature	Temperature
				rf	Reaction Force
				tr	Translational Energy
				np	Number of Particles
				pp	Average Particle Pressure
Airbag Sensors (CPM_SENSOR)	ab_sensor	Sensor id	xc	-	X coord
			yc	-	Y coord
			zc	-	Z coord
			vx	-	X Velocity
			vy	-	Y Velocity
			vz	-	Z Velocity

			vm	-	Velocity Magnitute
			[pr]essure	-	pressure
			[de]nsity	-	Density
			[te]mp	-	Gas temperature
			np	-	N Particles
Beam	Bea	Beam id	[n]ormal	x	Axial force
				y	Shear force in Y
				z	Shear force in Z
			[m]oment	y	Moment in Y
				z	Moment in Z
				x	Torsional moment
			[stra]in	-	Axial strain
			[e]nergy	p1	Bending energy: end 1
				p2	Bending energy: end 2
			[r]otation	y1	Y rotation: end 1
				y2	Y rotation: end 2
				z1	Z rotation: end 1
				z2	Z rotation: end 2
				x	Torsional rotation
			[b]ending	y1	Y Bending moment: end 1
				y2	Y Bending moment: end 2
				z1	Z Bending moment: end 1
				z2	Z Bending moment: end 2
			[e]nergy	a	Axial collapse energy
				i	Internal energy
			[stre]ss	x	Axial stress
				xy	XY Shear stress
				zx	ZX Shear stress
			[eff]	-	Effective plastic strain
			[exx]	-	Axial strain
			[e]xtra	##	Extra data ##
			[di]screte	dx	Relative Axial displacement

				dy	Relative S - Displacement
				dz	Relative T - Displacement
				rx	Axial rotation
				ry	Rotation in S
				rz	Rotation in T
				na	Relative Axial force
				ns	Resultant S - Force
				nt	Resultant T - Force
				ma	Axial moment
				ms	Moment in S
				mt	Moment in T
				axx	Axial Direction X
				axy	Axial Direction Y
				axz	Axial Direction Z
				sx	S - Direction X
				sy	S - Direction Y
				sz	S - Direction Z
				tx	T - Direction X
				ty	T - Direction Y
				tz	T - Direction Z
Bearing	bear	Bearing id	[fx]	-	X Force
			[fy]	-	Y Force
			[fz]	-	Z Force
			[mx]	-	X Moment
			[my]	-	Y Moment
			[mz]	-	Z Moment
			[dx]	-	X Displacement
			[dy]	-	Y Displacement
			[dz]	-	Z Displacement
			[ax]	-	X Angle
			[ay]	-	Y Angle
			[az]	-	Z Angle
			[lfx]	-	Local X Force
			[lfy]	-	Local Y Force
			[lfz]	-	Local Z Force
			[lmx]	-	Local X Moment
			[lmy]	-	Local Y Moment
			[lmz]	-	Local Z Moment

			[ldx]	-	Local X Displacement
			[ldy]	-	Local Y Displacement
			[ldz]	-	Local Z Displacement
			[lax]	-	Local X Angle
			[lay]	-	Local Y Angle
			[laz]	-	Local Z Angle
Boundary	Bou	Boundary id	[n]odal loads	fx	Applied X Force
				fy	Applied Y Force
				fz	Applied Z Force
				fm	Applied Resultant force
				e	Energy from applied force
			[ri]gid body loads	fx	Applied X Force
				fy	Applied Y Force
				fz	Applied Z Force
				fm	Applied Resultant force
				e	Energy from applied force
			[p]ressure nodal loads	fx	Applied X Force
				fy	Applied Y Force
				fz	Applied Z Force
				fm	Applied Resultant force
				e	Energy from applied force
			[rv]elocity r-body loads	fx	BC motion X Force
				fy	BC motion Y Force
				fz	BC motion Z Force
				fm	Resultant BC motion force
				en	Energy from BC motion
				mx	BC motion X Moment
				my	BC motion Y Moment
				mz	BC motion Z Moment

				mm	BC Moment Magnitude
			[v]elocity nodal loads	fx	BC motion X Force
				fy	BC motion Y Force
				fz	BC motion Z Force
				fm	Resultant BC motion force
				e	Energy from BC motion
CESE Element or Point	cese_el / cese_pt	Element / Point id	[c]oord	x	Current X coord
				y	Current Y coord
				z	Current Z coord
				m	Current Vector
			[ve]locity	x	X Velocity
				y	X Velocity
				z	Z Velocity
				m	Velocity Magnitude
			[vo]rticity	x	X Vorticity
				y	Y Vorticity
				z	Z Vorticity
				m	Vorticity Magnitude
			[d]ensity	-	Density
			[pr]essure	-	Pressure
			[t]emperature	-	Temperature
CESE FSI Drag	cese_fs	1=solid, 2=shell, 3=sol2D, 4=beam	[dr]ag	px	X Pressure Force
				py	Y Pressure Force
				pz	Z Pressure Force
				pm	Pressure Force Magnitude
CESE Segment Set Drag	cese_ss	Part id (0 if only one part requested)	[dr]ag	px	X Pressure Force
				py	Y Pressure Force
				pz	Z Pressure Force
				pm	Pressure Force Magnitude
				vx	X Viscous Force
				vy	Y Viscous Force
				vz	Z Viscous Force
				vm	Viscous Force Magnitude
				area	Total Area
Contact	Con / Sli	Contact id	[f]orce	xa	A Surface X force

				ya	A Surface Y force
				za	A Surface Z force
				ma	A Surface Force Magnitude
				xb	B Surface X force
				yb	B Surface Y force
				zb	B Surface Z force
				mb	B Surface Force Magnitude
			[mo]ment	xa	A Surface X moment
				ya	A Surface Y moment
				za	A Surface Z moment
				xb	B Surface X moment
				yb	B Surface Y moment
				zb	B Surface Z moment
			[ma]ss	a	A Surface Mass
				b	B Surface Mass
			[e]nergy	t	Total energy (A + B surface)
				a	A Surface side energy
				b	B Surface side energy
				f	Frictional energy
			[g]eometric	fx	X force
				fy	Y force
				fz	Z force
				fm	Force Magnitude
				mx	Moment in X
				my	Moment in Y
				mz	Moment in Z
				mm	Moment Magnitude
Cross section	Cro / Sec	Section id	[f]orce	x	X force
				y	Y force
				z	Z force

				m	Force Magnitude
			[m]oment	x	Moment in X
				y	Moment in Y
				z	Moment in Z
				m	Moment Magnitude
			[c]entroid	x	X centroid coord
				y	Y centroid coord
				z	Z centroid coord
			[a]rea	-	Area of section
EM Element, Node or Point	em_el / em_nd / em_pt	Element / Node / Point id	[co]ord	x	Current X coord
				y	Current Y coord
				z	Current Z coord
				m	Current Vector
			[cu]rrent	x	X Current
				y	X Current
				z	Z Current
				m	Current Magnitude
			[a]field	x	X AField
				y	Y AField
				z	Z AField
				m	AField Magnitude
			[b]field	x	X BField
				y	Y BField
				z	Z BField
				m	BField Magnitude
			[e]field	x	X EField
				y	Y EField
				z	Z EField
				m	EField Magnitude
			[l]force	x	X Lorentz Force
				y	Y Lorentz Force
				z	Z Lorentz Force
				m	Lorentz Force Magnitude
			[s]igma	-	Sigma
			[m]ur	-	Mu-R
			[j]hrate	-	JHRate
EM Circuit	em_ci	Circuit id	[v]oltage	-	voltage

			[ch]arge	-	charge
			[cu]rrent	-	current
			[d]resist	-	Circuit Resistance
			[j]resist	-	Equivalent Resistance
			[i]nduct	-	Inductance
			[mi1]	-	Mutual Inductance 1
			[mi2]	-	Mutual Inductance 2
			[mi3]	-	Mutual Inductance 3
EM Circuit0D	em_cd	Circuit0D id	[dv]oltage	-	voltage
			[dch]arge	-	charge
			[dcu]rrent	-	current
			[de]nergy	-	Total Energy
EM PartData	em_pd	PartData id	[x]lf	-	X Lorentz Force
			[y]lf	-	Y Lorentz Force
			[z]lf	-	Z Lorentz Force
			[ml]f	-	Lorentz Force Magnitude
			[j]he	-	Joule Heating Energy
			[mg]e	-	Magnetic Energy
			[k]te	-	Kinetic Energy
			[p]te	-	Plastic Energy
EM IsoPotOut	em_ip	IsoPotOut id	[v]oltage	-	voltage
			[c]urrent	-	current

EM CircuitRes	em_cr	CircuitRes id	[c]urc	-	Contact Current
			[r]esc	-	Contact Resistance
			[j]hrc	-	Contact Joule heat rate
			[a]reac	-	Contact Area
EM BoundaryOut	em_bo	BoundaryOut id	[v]oltage	-	Voltage
			[c]urrent	-	Current
			[a]rea	-	Area
EM IsoPotConnOut	em_ic	IsoPotConnOut id	[v]oltage	-	Voltage
			[ch]arge	-	Charge
			[cu]rrent	-	Current
			[r]esd	-	Contact Resistance
			[p]ower	-	Power
			[e]nergy	-	Energy
EM RandlesCell	em_rc	RandlesCell id	[to]tvoltage	-	TotVoltage
			[o]cv	-	OCV
			[d]ampvoltage	-	DampVoltage
			[cu]rrent	-	Current
			[so]c	-	SOC
			[f]uncsoc	-	SOCFunc
			[sh]iftsoc	-	SOCShift
			[su]msoc	-	SOCSum
			[r0]	-	R0

			[r1]0	-	R10
			[c1]0	-	C10
			[te]mp	-	Temp
			[ck]t_Number	-	Ckt Number
EM RandlesIntshortCell	em_ri	RandlesIntshortCell id	[m]xr	-	Maximum resistance
			[s]hc	-	Short circuits
			[toc]	-	Total circuits
			[tor]	-	Total resistance
			[a]rs	-	Area short
EM RogoCoil	em_ro	RogoCoil id	[v]c	-	Volume Current
			[s]c	-	Surface Current
			[m]f	-	Magnetic Field
EM Global	em_gl	Timestep id	[ru]n	-	Run timestep
			[cf]l	-	Condition timestep
			[rb]c	-	Ratio
		RandlesCellTot id	[to]tvoltage	-	TotVoltage
			[o]cv	-	OCV
			[d]ampvoltage	-	DampVoltage
			[cu]rrent	-	Current
			[so]c	-	SOC
			[f]uncsoc	-	SOCFunc
			[sh]iftsoc	-	SOCShift

			[su]msoc	-	SOCSum
			[r0]	-	R0
			[r1]0	-	R10
			[c1]0	-	C10
			[te]mp	-	Temp
			[vc2]	-	VC2
			[vc3]	-	VC3
			[r2]0	-	R20
			[r3]0	-	R30
			[c2]0	-	C20
			[c3]0	-	C30
		randlesCellTotEn id	[ohp]	-	Ohm Heat Power
			[rhp]	-	Reversible Heat Power
			[ecp]	-	Equivalent Capacity Power
			[ohe]	-	Ohm heat energy
			[rhe]	-	Reversible heat energy
			[ece]	-	Equivalent Capacity energy
			[es]e	-	Equivalent storage energy
		GlobEnergy id	[ecj]h	-	Ext ckt Joule Heating
			[ecm]e	-	Ext ckt Magnetic Energy
			[ecc]e	-	Ext ckt Capacitor Energy
			[mj]h	-	Mesh conductor Joule Heating
			[mm]e	-	Mesh conductor Mag Energy

			[a]me	-	Air Magnetic Energy
			[te]e	-	Total EM Energy
			[tp]e	-	Total Plastic Energy
			[tk]e	-	Total kinetic Energy
		RandlesIntshort id	[ms]r	-	Maximum short resistance
			[n]sc	-	Number of short circuits
			[tn]c	-	Total number of circuits
			[tsr]	-	Total short resistance
		RandlesIntshortTot id	[tm]r	-	Maximum resistance
			[tsc]	-	Short circuits
			[ttc]	-	Total circuits
			[ttr]	-	Total resistance
			[ta]s	-	Area short
FSI	FSI	FSI id	[pr]essure	-	pressure
			[f]orce	x	X force
				y	Y force
				z	Z force
				m	Force Magnitude
			[po]rous	-	Porous Leakage
			[m]ass	-	Mass Flux
ICFD Drag	icfd_dr	Part id / -1 for average / 0 for	[dra]g	px	X Pressure Drag
				py	Y Pressure Drag
				pz	Z Pressure Drag

		sum or if only one part		pm	Pressure Drag Magnitude
				vx	X Viscous Drag
				vy	Y Viscous Drag
				vz	Z Viscous Drag
				vm	Viscous Drag Magnitude
				pmx	MX Pressure Drag
				pmy	MY Pressure Drag
				pmz	MZ Pressure Drag
				pmm	Pressure Drag Moment Magnitude
				vmx	MX Viscous Drag
				vmy	MY Viscous Drag
				vmz	MZ Viscous Drag
				vmm	Viscous Drag Moment Magnitude
ICFD Node or Point	icfd_nd / icfd_pt	Node / Point id	[c]oord	x	Current X coord
				y	Current Y coord
				z	Current Z coord
				m	Current Vector
			[ve]locity	x	X Velocity
				y	X Velocity
				z	Z Velocity
				m	Velocity Magnitude
			[vo]rticity	x	X Vorticity
				y	Y Vorticity
				z	Z Vorticity
				m	Vorticity Magnitude
			[d]ensity	-	Density
			[pr]essure	-	Pressure
			[t]emperature	-	Temperature
ICFD Temp	icfd_th	Part id / 0 for sum or if only one part	[t_a]rea	average	Temperature Area Average
			[t_s]um	average	Temperature Sum Average
			[he]at	flux	Average Heat Flux
			[tot]al	area	Total Area

			[ht]c	-	Heat Transfer Coefficient
Joint	Joi	Joint id	[f]orce	x	X force
				y	Y force
				z	Z force
				m	Force Magnitude
			[m]oment	x	Moment in X
				y	Moment in Y
				z	Moment in Z
				m	Moment Magnitude
			***CONSTRAINED_JOINT_STIFFNESS_GENERALIZED**		
			[ph]i	an	Phi angle
				dt	d(Phi)/dt
				st	Phi stiffness moment
				da	Phi damping moment
				to	Phi total moment
			[th]eta	an	Theta angle
				dt	d(Theta)/dt
				st	Theta stiffness moment
				da	Theta damping moment
				to	Theta total moment
			[ps]i	an	Psi angle
				dt	d(Psi)/dt

				st	Psi stiffness moment
				da	Psi damping moment
				to	Psi total moment
			[ge]neralized	en	Total joint energy
			***CONSTRAINED_JOINT_STIFFNESS_FLEXION-TORSION**		
			[al]pha	an	Alpha angle
				dt	d(Alpha)/dt
				st	Alpha stiffness moment
				da	Alpha damping moment
				to	Alpha total moment
			[be]ta	an	Beta angle
				dt	d(Beta)/dt
				st	Beta stiffness moment
				da	Beta damping moment
				to	Beta total moment
			[ga]mma	an	Gamma angle
				dt	d(Gamma)/dt
				fa	Gamma scale factor
			[fl]exion	en	Total joint energy
Joint	Joi	Joint id	***CONSTRAINED_JOINT_STIFFNESS_TRANSLATIONAL**		
			[tr]anslational	xd	X displacement
				dxd	d(X)/dt
				yd	Y displacement
				dyd	d(Y)/dt
				zd	Z displacement

				dzd	d(Z)/dt
				xsf	X stiffness
				xdf	X damping
				xtf	X total
				ysf	Y stiffness
				ydf	Y damping
				ytf	Y total
				zsf	Z stiffness
				zdf	Z damping
				ztf	Z total
				en	Total joint energy
			***CONSTRAINED_JOINT_STIFFNESS_CYLINDRICAL**		
			[cy]lindrical	pd	P displacement
				dpd	d(P)/dt
				rd	R displacement
				dyd	d(Y)/dt
				zd	Z displacement
				dzd	d(Z)/dt
				psf	P stiffness
				pdf	P damping
				ptf	X total
				rsf	R stiffness
				rdf	R damping
				rtf	R total
				zsf	Z stiffness
				zdf	Z damping
				ztf	Z total
				en	Total joint energy
Node	No	Node id	[te]mperature	x	Temperature
			[to]p	temperature	Top Surface Temperature
			[bo]ttom	temperature	Bottom Surface Temperature
			[d]isplacement	x	X Displacement
				y	Y Displacement
				z	Z Displacement
				m	Displacement Magnitute
			[v]elocity	x	X Velocity
				y	Y Velocity

				z	Z Velocity
				m	Velocity Magnitute
			[a]cceleration	x	X Acceleration
				y	Y Acceleration
				z	Z Acceleration
				m	Acceleration Magnitude
			[c]oord	x	Current X coord
				y	Current Y coord
				z	Current Z coord
				m	Current Vector
			[b]asic	x	Basic X coord
				y	Basic Y coord
				z	Basic Z coord
				m	Basic Vector
			[r]otation	x	X rotation
				y	Y rotation
				z	Z rotation
				m	Rotation Magnitude
				vx	X rotational velocity
				vy	Y rotational velocity
				vz	Z rotational velocity
				vm	Rotation Vel Magnitude
				ax	X rotational acceleration
				ay	Y rotational acceleration
				az	Z rotational acceleration
				am	Rotation Accel Magnitude
			force	x	X force
				y	Y force
				z	Z force
				m	Force Magnitude
			[e]nergy	-	Energy
Node Group	Ng	Group id	force	x	X force

T/HIS

				y	Y force
				z	Z force
				m	Force Magnitude
Part	Pa	Part id	[k]inetic e	-	Kinetic energy
			[i]nternal e	-	Internal energy
			[h]ourglass e	-	Hourglass energy
			[t]otal e	-	Total energy
			[mx]	-	X momentum
			[my]	-	Y momentum
			[mz]	-	Z momentum
			[x] velocity	-	Average X velocity
			[y] velocity	-	Average Y velocity
			[z] velocity	-	Average Z velocity
			[am]	-	Added mass
			[ma]	-	Mass
			[ek]	-	Eroded Kinetic energy
			[ei]	-	Eroded Internal energy
Part group	Gro	Group id	[k]inetic e	-	Kinetic energy
			[i]nternal e	-	Internal energy
			[h]ourglass e	-	Hourglass energy
			[t]otal e	-	Total energy
			[am]	-	Added mass
Particle Blast	Pbl	PBlast id	[a]ir	ie	Air Internal Energy
			[d]etprod	ie	Detn Product Internal Energy
			[o]ut	ie	Outside Domain Internal Energy
			[a]ir	te	Air Translational Energy
			[d]etprod	te	Detn Product Translational Energy
			[o]ut	te	Outside Domain Translational Energy

Particle Blast Part	Pbp	Part id	[a]ir	pr	Air Pressure
			[d]etprod	pr	Detn Product Pressure
			[r]es	pr	Resultant Pressure
			[ar]ea	-	Surface Area
			[a]ir	x	Air X Force
			[a]ir	y	Air Y Force
			[a]ir	z	Air Z Force
			[d]etprod	x	Detn Product X Force
			[d]etprod	y	Detn Product Y Force
			[d]etprod	z	Detn Product Z Force
			[r]es	x	Resultant X Force
			[r]es	y	Resultant Y Force
			[r]es	z	Resultant Z Force
Pressure Tube	Prt	Node id	[ar]ea	-	Cross Section Area
			[de]nsity	-	Density
			[pr]essure	-	Pressure
			[v]elocity	-	Velocity
Pulleys	Pul	Pulley id	[fo]rce	-	Force
			[sl]ip	-	Slip
			[ra]te	-	Slip Rate
			[an]gle	-	Wrap Angle
Retractor	Ret	Retractor id	[fo]rce	-	Force
			[p]ullout	-	Pullout
			[fvp]	-	Force v Pullout
Rigid wall	Rig / Wall	Wall id	[n]ormal force	-	Normal force
			[x] force	-	Global X force
			[y] force	-	Global Y force
			[z] force	-	Global Z force
			[e]nergy	-	Energy
	Rigid_seg	Wall id	Segment id	[x] force	Global X force

Rigid wall Segment				[y] force	Global Y force
				[z] force	Global Z force
Rigid part / NRB	rpa / nrb	Part id	[d]isplacement	x	X Displacement
				y	Y Displacement
				z	Z Displacement
				m	Displacement Magnitude
			[v]elocity	x	X Velocity
				y	Y Velocity
				z	Z Velocity
				m	Velocity Magnitude
			[a]cceleration	x	X Acceleration
				y	Y Acceleration
				z	Z Acceleration
				m	Acceleration Magnitude
			[c]oord	x	X coord
				y	Y coord
				z	Z coord
			[r]otation	x	X rotation
				y	Y rotation
				z	Z rotation
				m	Rotation Magnitude
				vx	X rotational velocity
				vy	Y rotational velocity
				vz	Z rotational velocity
				vm	Rotation Vel Magnitude
				ax	X rotational acceleration
				ay	Y rotational acceleration
				az	Z rotational acceleration
				am	Rotation Accel Magnitude
	rpa / nrb	Part id	[dc]os	11	Direction Cosine 11

Rigid part / NRB				12	Direction Cosine 12
				13	Direction Cosine 13
				21	Direction Cosine 21
				22	Direction Cosine 22
				23	Direction Cosine 23
				31	Direction Cosine 31
				32	Direction Cosine 32
				33	Direction Cosine 33
			[ld]isplacement (local)	x	Local X Displacement
				y	Local Y Displacement
				z	Local Z Displacement
			[lv]elocity (local)	x	Local X Velocity
				y	Local Y Velocity
				z	Local Z Velocity
			[la]cceleration (local)	x	Local X Acceleration
				y	Local Y Acceleration
				z	Local Z Acceleration
			[lr]otation (local)	x	Local X rotation
				y	Local Y rotation
				z	Local Z rotation
				vx	Local X rotational vel
				vy	Local Y rotational vel
				vz	Local Z rotational vel
				ax	Local X rotational accel
				ay	Local Y rotational accel
				az	Local Z rotational accel
Seat belt	Sea / Bel	Belt id	[fo]rce	-	Force
			[s]train	-	Strain

			[fvs]	-	Force v Strain
			[l]ength	-	Current Length
Shell	Sh	Shell id	[stre]ss	xx	Stress in XX
				yy	Stress in YY
				zz	Stress in ZZ
				xy	Stress in XY
				yz	Stress in YZ
				zx	Stress in ZX
				mx	MAX principal stress
				mn	MIN principal stress
				ms	MAX shear stress
				vm	von Mises stress
				av	Average stress (Pressure)
			[stra]in	xx	Strain in XX
				yy	Strain in YY
				zz	Strain in ZZ
				xy	Strain in XY
				yz	Strain in YZ
				zx	Strain in ZX
				ma	MAX principal strain
				mi	MIN principal strain
				sh	MAX shear strain
				vm	von Mises strain
				av	Average strain
			[pla]stic	ef	Effective plastic strain
			[m]oment	x	Moment in X
				y	Moment in Y
				xy	Moment in XY
			[f]orce	sx	Shear force in X
				sy	Shear force in Y
				nx	Normal force in X
				ny	Normal force in Y
				nxy	Normal force in XY
			[t]hickness	-	Thickness
			[i]nternal	-	Internal energy density

			[e]xtra	##	Extra data ##
Slipring	Slp	Slipring id	[p]ullout	-	Pull through
			[w]arp	-	Warp Angle
			[s]kew	-	Skew Angle
			[f]riction	-	Friction Coefficient
			[n]ormal	-	Normal Force
			belt1	-	Side 1 Belt Force
			belt2	-	Side 2 Belt Force
Solid	So	Solid id	[stre]ss	xx	Stress in XX
				yy	Stress in YY
				zz	Stress in ZZ
				xy	Stress in XY
				yz	Stress in YZ
				zx	Stress in ZX
				mx	MAX principal stress
				mn	MIN principal stress
				ms	MAX shear stress
				vm	von Mises stress
				av	Average stress (Pressure)
			[stra]in	xx	Strain in XX
				yy	Strain in YY
				zz	Strain in ZZ
				xy	Strain in XY
				yz	Strain in YZ
				zx	Strain in ZX
				ma	MAX principal strain
				mi	MIN principal strain
				sh	MAX shear strain
				vm	von Mises strain
				av	Average strain
			[pla]stic	ef	Effective plastic strain
			[e]xtra	##	Extra data ##
SPC	SPC	SPC id	[f]orce	x	X force

				y	Y force
				z	Z force
				m	Force Magnitude
			[m]oment	x	Moment in X
				y	Moment in Y
				z	Moment in Z
				m	Moment Magnitude
SPH	SPH	SPH id	[d]ensity	-	Density
			[stra]in	xx	Strain in XX
				yy	Strain in YY
				zz	Strain in ZZ
				xy	Strain in XY
				yz	Strain in YZ
				zx	Strain in ZX
			[stre]ss	ef	Effective Stress
				xx	Stress in XX
				yy	Stress in YY
				zz	Stress in ZZ
				xy	Stress in XY
				yz	Stress in YZ
				zx	Stress in ZX
			[l]ength	-	Smoothing Length
Spotweld	Sw	Spotweld id	[co]nstrained	[a]xial	Axial force
				[s]hear	Shear force
				[l]ength	Length
				[f]ailure	Failure (failed if > 1.0)
				[ma]ximum	Maximum failure value
				[t]ime	Failure Time
			[ge]neralised	[a]xial	Axial force
				[s]hear	Shear force
				[l]ength	Length
				[f]ailure	Failure (failed if > 1.0)
				[ma]ximum	Maximum failure value
				[t]ime	Failure Time
			[sp]otweld	[a]xial	Axial force

				[s]hear	Shear force
				[l]ength	Length
				[f]ailure	Failure (failed if > 1.0)
				[ma]ximum	Maximum failure value
				[t]ime	Failure Time
				[m]oment	Resultant Moment
				[to]rsion	Torsion
			[so]lid	[a]xial	Axial force
				[s]hear	Shear force
				[l]ength	Length
				[f]ailure	Failure (failed if > 1.0)
				[ma]ximum	Maximum failure value
				[t]ime	Failure Time
				[m]oment	Resultant Moment
				[to]rsion	Torsion
				ff	DC Failure Function
				nf	Normal Failure Term
				sf	Shear Failure Trem
				bf	Bending Failure Term
				[ar]ea	Spotweld Area
			[no]n-local	[a]xial	Axial force
				[s]hear	Shear force
				[l]ength	Length
				[f]ailure	Failure (failed if > 1.0)
				[ma]ximum	Maximum failure value
				[t]ime	Failure Time
			[ass]embly	[a]xial	Axial force
				[s]hear	Shear force
				[l]ength	Length
				[f]ailure	Failure (failed if > 1.0)
				[m]oment	Resultant Moment
				[to]rsion	Torsion
				[t]ime	Failure Time

				ff	DC Failure Function
				nf	Normal Failure Term
				sf	Shear Failure Trem
				bf	Bending Failure Term
				[ar]ea	Spotweld Area
Spring	Sp / Da	Spring id	[f]orce	-	Force
			[e]longation	-	Elongation
			[fve]	-	Force v Elongation
			[en]ergy	-	Energy
			[m]oment	-	Moment
			[r]otation	-	Rotation
			[mvr]	-	Moment v Rotation
			[x] force	-	Global X force
			[y] force	-	Global Y force
			[z] force	-	Global Z force
			[mx]	-	Moment in X
			[my]	-	Moment in Y
			[mz]	-	Moment in Z
			[re]nergy	-	Rotational Energy
Subsystem	Ss	Subsystem id	[k]inetic e	-	Kinetic Energy
			[i]nternal e	-	Internal Energy
			[h]ourglass e	-	Hourglass energy
			[[kr]	-	Kinetic Energy Ratio
			[ir]	-	Internal Energy Ratio
			[mx]	-	X Momentum
			[my]	-	Y Momentum
			[mz]	-	Z Momentum
			[masst]	-	Total Mass
			[massc]	-	Center of Mass
			[massx]	-	X Center of Mass
			[massy]	-	Y Center of Mass
			[massz]	-	Z Center of Mass
			[it11]	-	Inertia Tensor Row11
			[it12]	-	Inertia Tensor Row12
			[it13]	-	Inertia Tensor Row13

			[it21]	-	Inertia Tensor Row21
			[it22]	-	Inertia Tensor Row22
			[it23]	-	Inertia Tensor Row23
			[it31]	-	Inertia Tensor Row31
			[it32]	-	Inertia Tensor Row32
			[it33]	-	Inertia Tensor Row33
			[pi1]	-	Principal Inertia I11
			[pi2]	-	Principal Inertia I22
			[pi3]	-	Principal Inertia I33
			[pd11]	-	Principal Directions Row11
			[pd12]	-	Principal Directions Row12
			[pd13]	-	Principal Directions Row13
			[pd21]	-	Principal Directions Row21
			[pd22]	-	Principal Directions Row22
			[pd23]	-	Principal Directions Row23
			[pd31]	-	Principal Directions Row31
			[pd32]	-	Principal Directions Row32
			[pd33]	-	Principal Directions Row33
Thick Shell	Thi / Tsh	Tshell id	[stre]ss	xx	Stress in XX
				yy	Stress in YY
				zz	Stress in ZZ
				xy	Stress in XY
				yz	Stress in YZ
				zx	Stress in ZX
				mx	MAX principal stress
				mn	MIN principal stress

				ms	MAX shear stress
				vm	von Mises stress
				av	Average stress (Pressure)
			[stra]in	xx	Strain in XX
				yy	Strain in YY
				zz	Strain in ZZ
				xy	Strain in XY
				yz	Strain in YZ
				zx	Strain in ZX
				ma	MAX principal strain
				mi	MIN principal strain
				sh	MAX shear strain
				vm	von Mises strain
				av	Average strain
			[pla]stic	ef	Effective plastic strain
			[e]xtra	##	Extra data ##
TRACERS	Tr	Tracer ID	[d]isplacement	x	Current X coord
				y	Current Y coord
				z	Current Z coord
				m	Current Vector
			[v]elocity	x	X Velocity
				y	Y Velocity
				z	Z Velocity
				m	Velocity Magnitute
			[stre]ss	xx	Stress in XX
				yy	Stress in YY
				zz	Stress in ZZ
				xy	Stress in XY
				yz	Stress in YZ
				zx	Stress in ZX
			EFP	-	
			(de)nsity	-	Density
			rvol	-	Relative Volume
			ac[tive]	-	Active
Whole model	Wh	-	[dt]	-	Time step
			[k]inetic e	-	Kinetic energy
			[i]nternal e	-	Internal energy
			[sw]	-	Stonewall energy

			[j]oint e	-	Joint internal energy
			[sp]ring e	-	Spring and damper energy
			[h]ourglass e	-	Hourglass energy
			[sy]stem e	-	System damping energy
			[si]	-	Sliding interface energy
			[ew]	-	External work
			[rb]	-	Rigid Body stopper energy
			[t]otal e	-	Total energy
			[er]	-	Total/initial energy
			[x] velocity	-	Average X velocity
			[y] velocity	-	Average Y velocity
			[z] velocity	-	Average Z velocity
			[cy]cle time	-	Time per zone cycle
			[am]	---	Added mass
			[pm]	-	%age Mass increase
			[ek]	-	Eroded Kinetic energy
			[ei]	-	Eroded Internal energy
			[eh]	-	Eroded Hourglass energy
			[ewoe]	-	Energy Ratio w/o Eroded
			[m]ass	-	Mass
			[mpe]	-	Mat Plastic Energy
			[mee]	-	Mat Elastic Energy
			[mde]	-	Mat Damage Energy
			[die]	-	Dissipation Internal Energy
			[dke]	-	Dissipation Kinetic Energy
			[de]	-	Drilling Energy

7.4.5.1 Defining Surfaces / Integration points for data extraction

Some data components can be written out at multiple locations.

In recent versions of LS-DYNA it is possible for each element to write out multiple values for some data components.

For fully integrated Shells and Thick Shells values can be written out for all 4 in-plane integration points in each through thickness location. In addition to the integration point values it is also possible to write out data that has been extrapolated from the integration points out to the shells nodes.

For fully integrated solid elements data can also be written out for all 8 integration points and values can also be extrapolated to the elements nodes.

To select these additional values the entity ID's specified in a FAST-TCF scripts can be modified as follows.

Solids	n	Average value for solid (default)
	n@X	Value at integration point X (0 < X < 8)
	n@-X	Value at node X (0 < X < 8)
Shells	n	Average value for shell (default)
	n@X	Value at integration point X (0 < X < 4)
	n@-X	Value at node X (0 < X < 4)
Shells	n	Average value for thick shell (default)
	n@X	Value at integration point X (0 < X < 4)
	n@-X	Value at node X (0 < X < 8)

e.g. **`solid 10`**

(solid 10 - average value)

`solid 20@5`

(solid 20 - data from 5th integration point)

`shell 20@-3`

(shell 20 - data extrapolated to shells 3rd node)

10.6. Units

UNITS

T/HIS 9.4 onwards can automatically add unit information to graph labels and it can convert results from one unit system to another.

Each model in T/HIS can have a Unit System defined for it and a separate Unit System can be defined for displaying results. T/HIS will automatically convert results from the model Unit System to the display Unit System. T/HIS has 6 built in unit systems.

Unit System name	Units
U1	m, kg, s
U2	mm, t, s
U3	mm, kg, ms
U4	mm, g, ms
U5	ft, slug, s
U6	m, t, s

Setting the unit system for a model

To set the unit system for a model

Keyword	**second word**	**third word**	**fourth word**	**notes**
unit	model	n	U1	Set the unit system for model 'n' to U1
			U2	Set the unit system for model 'n' to U2
			U3	Set the unit system for model 'n' to U3
			U4	Set the unit system for model 'n' to U4
			U5	Set the unit system for model 'n' to U5
			U6	Set the unit system for model 'n' to U6
		all	*as above*	Set the unit system for all models

Setting the DISPLAY unit system

To set the display unit system

Keyword	second word	third word	notes
unit	display	U1	Set the display unit system to U1
		U2	Set the display unit system to U2
		U3	Set the display unit system to U3
		U4	Set the display unit system to U4
		U5	Set the display unit system to U5
		U6	Set the display unit system to U6

Curve Axis units

By default T/HIS will automatically set the Unit System for any curves read from a model to those of the model. In addition to setting the curve Unit System, T/HIS will automatically set a unit type for the X and Y axis of the curve. These unit types are maintained through curve operations so that the correct units can be displayed for each curve.

The X and Y Axis units for a curve can be manually set if required.

Keyword	second word	third word	additional words			notes
unit	cx	curve #1	curve #2 to curve #n	##	*Unit name*	## ends the curve list Set the X axis unit for curves
		*	##		*Unit name*	## ends the curve list Set the X axis unit for all curves
	cy	curve #1	curve #2 to curve #n	##	*Unit name*	## ends the curve list Set the Y axis unit for curves
		*	##		*Unit name*	## ends the curve list Set the Y axis unit for all curves

The Unit name can be any of the following

Time	**Rotation**	**Momentum**	**Energy Den**
Energy	**Rot Vel**	**Density**	**Mass Flow**
Work	**Rot Accel**	**Stress**	**Frequency**
Temperature	**Length**	**Strain**	**Power**
Displacement	**Area**	**Force**	**Thermal Flux**
Velocity	**Volume**	**Moment**	**Force width**

Accel	**Mass**	**Pressure**	**Moment width**
Viscosity	**Thermal Diffusivity**	**Vorticity**	**Q Criterion**
Current	**Vec Potential**	**Magnetic Flux Vec**	**Elec Field Vec**
Conductivity			

Curve Unit Systems

If a curve has been read in from any source other than a model then the Unit System can also be set.

Keyword	**second word**	**third word**	**additional words**			**notes**
unit	cu	curve #1	curve #2 to curve #n	##	*Unit System name*	## ends the curve list Set the Unit System for curves
		*	##		*Unit System name*	## ends the curve list Set the Unit System for all curves

Other UNIT options

If a CSV file is written out from within a FAST-TCF script (see FAST-TCF CURVE OUTPUT) then by default it will contain rows containing UNIT information for the curves if UNITS have been defined.

Some third party applications and scripts can not read T/HIS CSV files containing this additional UNIT information correctly. The following option can be added to FAST-TCF scripts to turn on and off the output of this additional information.

Keyword	**second word**	**third word**	**notes**
unit	csv	on	Turns on the output of UNIT information to CSV files
		off	Turns off the output of UNIT information to CSV files

10.7. Curve Tags

CURVE TAGS

In FAST-TCF any operation that uses one or more curves as an input can reference the curve using either the curve number or a curve tag. **The use of curve Tags is strongly recommended as it enables scripts to be easily modified and sections added / deleted without having to renumber all the curve references within the script.**

Curve tags are defined for a curve by adding the keyword TAG to the data extraction command followed by the tag.

e.g.	**node**	**42**	**force y_dir**	**tag curve_1**
	(node)	(i.d. 42)	(force in y-direction)	(tag the curve as "curve_1")
	node	**"end of roof"**	**accel z**	**tag point_2**
	(node)	(i.d. "end of roof")	(z acceleration)	(tag the curve as "point_2")

Tags cannot begin with a numeric character, e.g. `tag 1_curve` is not allowed.

If a tag is not specified for a curve then FAST-TCF will automatically generate a tag for the curve using the T/HIS curve number.

The tag for a curve can be redefined at anytime within a script using the "`tag`" command (see Setting curve Labels, Titles and tags) for more details. Once a curve tag has been redefined the original definition should not be used in any following commands - a curve can only have 1 tag defined at any time.

Tagging curves from a T/HIS curve file

Curves read in from a T/HIS curve file can be tagged by refering to each curve in the file using a negative number:

e.g.	**tag**	**-1**	**curve_1**
		(1st curve in the curve file)	(tag as "curve_1")
	tag	**-2**	**curve_2**
		(2nd curve in the curve file)	(tag as "curve_2")

If curves are read in from a T/HIS curve file then then the FAST-TCF tag will be generated using the following rules.

1. If the data extraction command contains a `tag` option then that tag will be used (as above).
2. If the curve file contains curve tags then they will be used if the data extraction command DOES NOT contain a `tag` option.
3. If no tags are specified in the file or in the data extraction command then T/HIS will automatically tag each curve as '#' where # is the internal T/HIS curve number.

In the third case, if for example there are three curves already in T/HIS, the curves read in from the curve file will be tagged as '4', '5', '6', '7', etc. This limits how you can refer to these curves since would not be able to multiply two curves together. For example the command `op mul 4 5 tag new_curve` would multiply the curve tagged as '4' by the number 5, not by the curve tagged as '5'.

To avoid this limitation you will need to tag your curves using either the syntax explained above or by specifying a tag in the curve file.

Tagging multiple curve outputs

From T/HIS 9.2 onwards, multiple curve outputs can be generated from one FAST-TCF input line. Curve tags and labels can be specified for multiple curves using the following special syntax (note this only works on multiple curves):

- If the user specifies a wildcard in the tag or label (a "*"), then FAST-TCF will substitute the wildcard for the number of the curve outputted (starting from 1).
- If the user specifies a "##" then the entity ID is substituted in its place which is useful if the user knows what entities are expected on output.

e.g.	**node 5:last**	**accel mag**	**tag node_***	**lab Head Accn ***
	(node IDs. 5 to last)	(accel mag)	tags = node_1, node_2, etc	labels = Head Accn 1, Head Accn 2, etc
	node 10:20	**accel mag**	**tag node_##**	**lab Head Accn ##**
	(nodes 10 to 20)	(accel mag)	tags = node_10, node_11, etc	labels = Head Accn 10, Head Accn 11, etc

Using Wildcards

A number of T/HIS functions and operations can be applied to multiple curves in a single command by specifying multiple curve tags using wildcards.

From T/HIS 10.0 onwards the following wildcards are supported

Wildcard	Matches
*	1 or more characters

?	a single character
[a-e]	matches a single character against a range of characters , 'a','b','c','d' or 'e'
[abc]	matches a single character against a list of characters, 'a', 'b' or 'c'

e.g. `operate multiple x_disp_* 10 tag x_mul_*`

(Multiple all curves with a tag starting with "x_disp_" by 10 and tag the outputs as x_mul_1, x_mul_2 ... - see PERFORMING FAST-TCF CURVE OPERATIONS for more details)

`display x_disp_*`

(Display all curves with a tag starting with "x_disp_" - see Curve Display for more details)

`copy curve_file.cur x_disp_*`

(Write all curves with a tag starting with "x_disp_" to a file called "curve_file.cur"- see FAST-TCF CURVE OUTPUT for more details)

`csv curve_file.csv curve_1? curve_3[0-3]`

Write curves with tags curve_10, curve_11, curve_12 .... and curves with tags curve_30, curve_31, curve_32, curve_33 to a CSV file called "curve_file.csv"- see FAST-TCF CURVE OUTPUT for more details)

Using Curve Numbers

Although it is not recommended curves can be referenced using the internal curve number instead of the curve tag. If for example the 1st curve generated by a script has the tag "curve_1" then the following 2 commands are identical.

e.g.
```
operate    multiple    curve_1    10    tag    x_mul_*
operate    multiple    #1         10    tag    x_mul_*
```

If curve numbers are used within a script then T/HIS will automatically offset the curve numbers in the script by the number of curves T/HIS already has defined before the script is executed.

e.g. `operate multiple #1 10 tag x_mul_*`

would multiply internal curve number 1 by 10 if T/HIS didn't contain any curve definitions when the script was run.

If T/HIS already contained 100 curves then the same command would multiple internal curve 101 by 10.

This means it is possible to play a script containing curve numbers multiple times within a session without having to either delete all the existing curves or modify the script each time.

Tagging the most recently created or highest ID curve

The most recently created/edited curve or the curve with the highest ID can be tagged with the specific commands "`recent`" and "`highest`".

e.g.
```
tag    recent     curve_tag_1
tag    highest    curve_tag_2
```

These commands won't be written out automatically into a FAST-TCF script, so will need to be added manually. It is worth noting that if a curve tag starts with "`recent`" or "`highest`", then any command intended to change the tag would instead be interpreted as setting the tag of the most recent or highest ID curve, as in the above example. It is therefore recommended that curve tags should not start with "`recent`" or "`highest`".

10.8. Curve Groups

CURVE GROUPS

Curve groups can be defined within FAST-TCF scripts using the **cgroup** keyword. After a curve group has been defined in a FAST-TCF script it can then be used as an input to some FAST-TCF commands. Each curve group should be given a unique name within the FAST-TCF script.

Keyword	Second word	Third word	following word	notes
cgroup	create	name	-	Create a curve group called "name". If the name contains any spaces then it should be enclosed in quotes ("name with space")
	add	name	curve list	Adds a list of curves to the curve group called "name". If the name contains any spaces then it should be enclosed in quotes ("name with space"). The curve list should be a list of curve tags.
	remove	name	curve list	Removes a list of curves from the curve group called "name". If the name contains any spaces then it should be enclosed in quotes ("name with space"). The curve list should be a list of curve tags.

e.g. **cgroup create group_1**

(Create a curve group called "group_1")

cgroup add group_1 curve_1 curve_2

(Add curves with tags "curve_1" and "curve_2" to group "group_1")

cgroup create "Group 2"

(Create a curve group called "Group 2")

cgroup add "Group 2" curve_1*

(Add all curves with a curve tag containing "curve_1" to group "Group 2")

cgroup remove "Group 2" curve_11

(Remove curve with tag "curve_11" from group "Group 2")

To use a curve group as the input to another FAST-TCF command the curve group name is preceeded by an &. If a curve group name contains spaces then the name should be enclosed in double quotes and the & should be before the first ".

e.g. **operate multiple &group_1 10 tag output_***

(Multiple all curves in curve group "group_1" by 10 and tag the outputs as output_1, output_2 ..._)

operate multiple &"Group 2" 10 tag output_*

(Multiple all curves in curve group "Group 2 " by 10 and tag the outputs as output_1, output_2 ..._)

Curve Groups can currently be used

- As the first curve input in all of the **operate** commands
- Within the list of curves specified as input to curve range functions.
- To select a group of curves for the **display** command.
- When outputting curves to T/HIS curve files and CSV files.

10.9. Performing FAST-TCF Curve Operations

PERFORMING FAST-TCF CURVE OPERATIONS

Description	keyword	following words
Curve operation	oper	oper command + necessary words (depending on operation)

Many curve processing operations and functions are available. The syntax is common for all types of curve operation:

1. The first word is `oper` and is followed by:
2. The operation/function name e.g. ADD, INT.
3. The required number of arguments for the operation, e.g. ADD requires two arguments, a curve and either a curve or a value.
4. The remainder of the line may contain optional requests.
5. Any optional requests can occur after the arguments.
6. Curve numbers must be in the format: #<curve number>
7. An output curve is always needed - for operation commands such as `hic`, `hicd`, `tti`, `3ms`, `err`, the curve will be copied and the operation is executed on the copied curve.
8. A curve tag containing a wildcard or a curve group can be specified as the first curve input for any curve operation. If a curve tag contains a wildcard or if a curve group is specified then the curve operation will be repeated for each curve that either the tag matches or is in the curve group.

e.g.	**oper hic**	**node_acc**	**1.0**	**15E-3**	**label Hic-ed node accn**
	(hic)	(curve tag)	(scale=1.0)	(15ms period)	(label)

In T/HIS 9.2 onwards, the user can operate on multiple input curves (only the first curve can be multiple at the moment) using the wildcard "*". For example, to multiply all curves starting with the tag **acc** :

e.g.	**oper mul**	**acc***	**9810.0**
	(multiply)	(on all curves with tag acc*)	

10.9.1. Standard Operation Commands

Standard operation commands

Description	keyword	operation command	following word #1	following word #2	additional words	notes
Absolute value	oper	abs	curve #1	-	-	
Add Y	oper	add	curve #1	curve #2 or constant	-	
Add X	oper	adx	curve #1	curve #2 or constant	-	
Clip curve	oper	cli	curve #1	x min value / "auto"	x max value / "auto" y min value / "auto" y max value / "auto"	Input requires all 4 values, "auto" sets the value automatically
Combine	oper	com	curve #1	curve #2	-	
Concatenate	oper	cat	curve #1	curve #2	-	
Derivative	oper	dif	curve #1	-	-	
db	oper	db	curve #1	reference value		Convert a curve to dB
db(A)	oper	dba	curve #1	narrow		Apply narrow band A weighting
				octave		Apply octave band A weighting
Div Y	oper	div	curve #1	curve #2 or constant	-	
Div X	oper	dix	curve #1	curve #2 or constant	-	
Error calculation	oper	err	curve #1	curve #2	-	Value is stored with the output curve
Integral	oper	int	curve #1	-	-	
Least squares	oper	lsq	curve #1	-	-	
Map	oper	map	curve #1	curve #2	-	

<table>
<tr><td>Mul Y</td><td>oper</td><td>mul</td><td>curve #1</td><td>curve #2 or constant</td><td colspan="2">-</td><td></td></tr>
<tr><td>Mul X</td><td>oper</td><td>mux</td><td>curve #1</td><td>curve #2 or constant</td><td colspan="2">-</td><td></td></tr>
<tr><td>Normalise</td><td>oper</td><td>nor</td><td>curve #1</td><td>-</td><td colspan="2">-</td><td></td></tr>
<tr><td rowspan="8">Octave</td><td rowspan="8">oper</td><td rowspan="8">oct</td><td rowspan="8">curve #1</td><td rowspan="4">octave</td><td rowspan="2">rms</td><td>linear</td><td rowspan="8">Convert a curve from "narrow" band to either Octave or 1/3rd Octave bands.

Value for each band can be calculated using either mean or RMS values, and the input can either be linear or in dB.</td></tr>
<tr><td>db</td></tr>
<tr><td rowspan="2">mean</td><td>linear</td></tr>
<tr><td>db</td></tr>
<tr><td rowspan="4">third</td><td rowspan="2">rms</td><td>linear</td></tr>
<tr><td>db</td></tr>
<tr><td rowspan="2">mean</td><td>linear</td></tr>
<tr><td>db</td></tr>
<tr><td>Order</td><td>oper</td><td>ord</td><td>curve #1</td><td>-</td><td colspan="2">-</td><td></td></tr>
<tr><td>Reciprocal</td><td>oper</td><td>rec</td><td>curve #1</td><td>-</td><td colspan="2">-</td><td></td></tr>
<tr><td>Reverse</td><td>oper</td><td>rev</td><td>curve #1</td><td>-</td><td colspan="2">-</td><td></td></tr>
<tr><td>Rolling average</td><td>oper</td><td>r-av</td><td>curve #1</td><td>averaging window</td><td colspan="2">-</td><td>If the averaging window is undefined or set to 0.0 then the y-values at each point are calculated by averaging all of the proceeding curve points.

If the averaging window is set to T then the y-values at each point are calculated by averaging between -T/2 and +T/2.</td></tr>
</table>

Smooth	oper	smo	curve #1	smoothing factor	-	Factor must be an integer
Stress	oper	str	curve #1	"true" or "engineering"	-	
Sub Y	oper	sub	curve #1	curve #2 or constant	-	
Sub X	oper	sux	curve #1	curve #2 or constant	-	
Translate	oper	tra	curve #1	X value	Y value	
Vector 2D	oper	v2d	curve #1	curve #2	-	
Vector mag	oper	vec	curve #1	curve #2	curve #3	
Window	oper	win	curve #1	"han", "cos", "exp"	lead in (only for "cos" option)	Writes out 2 curves
Zero curve (X and Y)	oper	zer	curve #1	-	-	Shifts curve to 0,0 (X and Y values)
Zero curve (X only)	oper	zero_x	curve #1	-	-	Shift curve to 0,Y (X only)
Zero curve (Y only)	oper	zero_y	curve #1	-	-	Shift curve to X,0 (Y only)

10.9.2. Maths Operation Commands

Maths operation commands

Description	keyword	operation command	following word #1	following word #2	additional words	notes
Arc cosine	oper	acos	curve #1	-	-	
Arc sine	oper	asin	curve #1	-	-	
Arc tangent	oper	atan	curve #1	-	-	
Cosine	oper	cos	curve #1	-	-	
Log base 10	oper	log10	curve #1	-	-	
Log base 10 (X)	oper	log10x	curve #1	-	-	
Natural Exp	oper	exp	curve #1	-	-	
Natural log	oper	log	curve #1	-	-	
Natural log (X)	oper	logx	curve #1	-	-	
Power	oper	pow	curve #1	nth power	-	
Sine	oper	sin	curve #1	-	-	
Square root	oper	sqr	curve #1	-	-	
Tangent	oper	tan	curve #1	-	-	

10.9.3. Automotive Operation Commands

Automotive operation commands

Description	keyword	operation command	following word #1	following word #2	additional words	notes
Delta V	oper	acu	curve #1	offset	time period	
Acceleration severity index	oper	asi	Accn x curve #	Accn y curve #	Accn z curve #	word6 = acceleration conversion factor word7 = x limit word8 = y limit word9 = z limit
Butterworth filter	oper	but	curve #1	cut off frequency	order	
C60 filter	oper	c60	curve #1	-	-	
C180 filter	oper	c180	curve #1	-	-	
C600 filter	oper	c600	curve #1	-	-	
C1000 filter	oper	c1000	curve #1	-	-	
Clip value	oper	cva	curve #1	time window	Label displayed on screen *(optional)*	Value is stored with the output curve
Exceedence	oper	exc	curve #1	auto / pos / neg	-	
Fir filter	oper	fir	curve #1	-	-	
Hic	oper	hic	curve #1	division scale factor	time period	Value is stored with the output curve
Hicd	oper	hicd	curve #1	division scale factor	time period	Value is stored with the output curve
Neck injury criteria	oper	nij	Shear curve #	Axial curve #	Moment curve #	word6 = Fzc tension word7 = Fzc compression

						word8 = Myc flexion word9 = Myc extension word10 = Distance from joint
Regularise	oper	reg	curve #1	new dt value	-	
THIV	oper	thi	Accn x curve #	Accn y curve #	Yaw rate curve #	word6 = Horizontal distance word7 = Lateral distance word8 = Head to vehicle distance
TTI	oper	tti	Upper rib curve #	Lower rib curve #	Lower spine curve #	Value is stored with the output curve
Viscous criteria ECER95	oper	vc	curve #1	constant A	constant B	ECER95 method
Viscous criteria IIHS	oper	vc2	curve #1	constant A	constant B	IIHS method
Curve Correlation (strict)	oper	corr	strict	curve #1	curve #2	Value is stored with the output curves
Curve Correlation (loose)	oper	corr	loose	curve #1	curve #2	Value is stored with the output curves
Weighted Integrated Factor Curve Correlation	oper	wif	curve #1	curve #2	-	Value is stored with the output curve

10.9.4. Seismic Operation Commands

Seismic operation commands

Description	keyword	operation command	following word #1	following word #2	additional words	notes
Accn to disp spectra	oper	ad	curve #1	-	-	
Accn to vel spectra	oper	av	curve #1	-	-	
Disp to vel spectra	oper	dv	curve #1	-	-	
Disp to accn spectra	oper	da	curve #1	-	-	
Vel to disp spectra	oper	vd	curve #1	-	-	
Vel to accn spectra	oper	va	curve #1	-	-	
Baseline correction	oper	blc	curve #1	-	-	
Design spectrum	oper	ds	curve #1	broadening factor	-	
FFT	oper	fft	curve #1	-	-	
Non cumulative P.R.	oper	ncp	curve #1	curve #2	-	
Response spectrum	oper	rs	curve #1	damping factor	sampling factor	Sampling must be either 30 or 70

10.9.5. Range of Curve Operation Commands

Range of curve operation commands

Description	keyword	operation command	following word #1	following word #2	additional words	notes
Average	oper	ave	curve #1	curve #2 to curve #n	##	"##" ends the curve list
Envelope	oper	env	curve #1	curve #2 to curve #n	##	"##" ends the curve list
Minimum	oper	min	curve #1	curve #2 to curve #n	##	"##" ends the curve list
Maximum	oper	max	curve #1	curve #2 to curve #n	##	"##" ends the curve list
Resultant	oper	res	curve #1	curve #2 to curve #n	##	"##" ends the curve list
Sum	oper	sum	curve #1	curve #2 to curve #n	##	"##" ends the curve list
Sum	oper	sum	curve #1	curve #2 to curve #n	##	"##" ends the curve list

10.10. Applying Extra Options to Data Requests

APPLYING EXTRA OPTIONS TO DATA REQUESTS

Extra options can be used after a data component extraction, or a curve operation. After the basic request for a particuar component and particular entity have been made, the following extra data on the line is recognised to manipulate the curve further. This includes options to label a curve, scale it, write it out and so on.

Each request is executed in the order on the line, **if the curve label is used, it must be the last input on the line.**

e.g.	`no 54`	`accel mag`	`xsc 1000 ysc 0.0001`	`hic`	`lab Head Accn`
	(node i.d. 54)	(accel mag)	(scale x and y)	(obtain hic value)	(curve label)
	`no 1`	`accel mag`	`filter c60`	`append output.cur`	
	(node i.d. 1)	(accel mag)	(filter with C60)	(append the curve to a file)	
	`no 1`	`accel mag`	`tag node_1_acc`		
	(node i.d. 1)	(accel mag)	(tag the curve "node_1_acc" for ease of use later in the script)		

Description	extra option word	following word #1	following word #2	notes
3ms clip	3ms	-	-	Curve is squared and then square rooted to remove -ve values Curve is truncated around 3ms values - only 3ms part is left
Append into file	app	filename	-	Appends into curve file, if it doesn't exist - create it
Combine	com	curve #2	-	Y-value curve #1 vs X-value curve #2
Copy into file	cop	filename	-	Copy will overwrite any previous instance of the file
Error function	err	curve #2	-	
HIC	hic, hic15, hicd	-	-	Curve is squared and then square rooted to remove -ve values, an identical curve is outputted

Filtering	fil	fir c60 c180 c600 c1000	-	
X scale factor	xsc	scale factor	-	
Y scale factor	ysc	scale factor	-	
Label	lab	label word #1	label word #2 etc	Keyword and label must be at the end of the line
Reference tag	tag	tag word	-	Invalid words: "style", "xax", "yax", "title"
ASCII file request	ASC	-	-	
LSDA file request	LSD	-	-	
THF file request	THF	-	-	
XTF file request	XTF	-	-	

Using extra options on multiple curve outputs

From T/HIS 9.2 onwards, multiple curve outputs can be generated from one FAST-TCF input line. Unfortunately most of the extra options displayed below will NOT work on these multiple outputs. However, support has been added to allow tagging and labeling of all the multiple curves outputted in one go (see Tagging multiple curve outputs).

10.11. Setting Properties for Curves

Setting properties for curves

The folowing options can be used to set up properties for curves.

Setting curve Labels, Titles and tags

Description	keyword	second word	third word	fourth word	notes
Curve Label	lab	curve # or tag	label word 1	label word 2 etc	Specifies a new curve label
Curve Tag	tag	curve # or tag	tag	-	Specifies a new curve tag
Curve Title	tit	curve # or tag	label word 1	label word 2 etc	Specifies a new curve title
Curve X axis label	xla	curve # or tag	label word 1	label word 2 etc	Specifies a new x -axis label
Curve Y axis label	yla	curve # or tag	label word 1	label word 2 etc	Specifies a new y-axis label
1st Y axis	y1	curve # or tag	-	-	puts the curve on the 1st y axis
2nd Y axis	y2	curve # or tag	-	-	puts the curve on the 2nd y axis
User defined model prefix	prefix	model	model # or 'all'	*"prefix string"*	sets the user defined model prefix

From T/HIS 9.4 onwards, curve properties such as the minimum and maximum values can be displayed in the legend area as well as within the graph area.

The following commands use a new `properties` keyword and can be used to specify the font, colour and background used to display values as well as selecting which values are displayed on each curve.

Keyword	2nd word	3rd word	4th word	5th word	6th word	7th word	notes
properties	format	font	hm hb cm cb tm	8 10 12 14 18 24	standard colour	-	sets up font used to display curve properties fonts available: hm - helvetica

			tb default	defaul t			medium cb - courier bold tm - times new roman medium etc... font sizes in pt: 8, 10, 12 etc...
		background	standard colour	-	-	-	Set a background colour for the text
		transparenc y	integer (0-100)		-	-	Set the background transparency
		border	standard colour	on/off	-	-	Set a border colour round the text and turn it on/off
		arrow	on/off	-	-	-	Turn on/off a line connecting the text to the min/max value location
		num	y_only	-	-	-	Only display the y value
		num	x_y	-	-	-	Display both the x and y values on a single line
		num	xy	-	-	-	Display both the x and y values on separate lines
		value	<*type*>	-	-	-	Set the unit format to one of *automatic, general, scientific* for graph (n)
		precision	m	-	-	-	Set the number of decimal places displayed for the y axis values to (m) in graph (n)
propertie s	legen d	format	off	-	-	-	Turn off the display of curve values in the legend area
			append	-	-	-	Append curve values (min,max,average ...) to the curve labels in the legend.

			2nd	-	-	-	Add a 2nd line to the legned for each curve containing the curve values (min,max,average ...).
		curve #1	curve #2	##	maximum minimum average other	on/off	Turn on/off the display of one of the following curve properties in the legend. Input one or more curves and terminate the list with ## maximum - display curve maximum value minimum - display curve minimum value average - display curve average value other - display other curve values
properties	curves	format	off	-	-	-	Turn off the display of cuvre values in the graph area
			summary	-	-	-	Display the minimum/maximum value for all of the curves currently visible
			all	-	-	-	Display minimum/maximum values for each curve that is currently visible
		summary	smaximum sminimum lmaximum lminimum	on/of	-	-	Turns on/off the display of one of the following curve summary properties smaximum -

							highlight the maximum value for all the curves displayed sminimum - highlight the minimum value for all the curves displayed lmaximum - label the maximum vlaue for all the curves displayed lminimum - label the minimum vlaue for all the curves displayed
		curve #1	curve #2	##	smaximum sminimum lmaximum lminimum other	on/off	Turns on/off the display of one of the following curve summary properties. Input one or more curves

10.12. Defining Datums

10.12.1. Creating Datum Definitions

Creating Datum Definitions

The following options can be used to setup DATUM definitions

keyword	**second word**	**notes**
start datum		Starts a Datum definition
acronym	*acronym*	Specifies the datum acronym
label	*label*	Specifies the datum label
label	*2nd label*	Specifies the label for the optional second constant datum line
type	constant_x	Defines the datum as a constant x value
	constant_y	Defines the datum as a constant y value
	constant_y2	Defines the datum as a constant y2 value
	Points	Defined the datum as a set of x,y points
value	*value*	Specifies the value for a constant x, y or y2 datum
2nd value	*2nd value*	Specifies the optional second value for a constant x, y or y2 datum
num_points	#points x1,y1 y2,y2	Specifies the number of points used to define a datum, followed by pairs of x,y values.
line_colour	colour (see Line Colours)	Specify the line colour used to display the datum line (or none)
line_style	style (see Setting Curve Styles	Specifies the line style used to display the datum line (or none)
line_width	width (see Line Width	Specifies the line width used to display the datum line (or none)
fill_colour1	colour (see Line Colours)	Defines the colour used to fill above/right of the datum line
fill_colour2	colour (see Line Colours)	Defines the colour used to fill below/left of the datum line
fill_colour3	colour (see Line Colours)	Defines the colour used to fill between the two constant datum lines if a second value is present
label_font		Define the font used to display the label
label_size	8,10,12,14,18,24	Define the font point size used to display the label
label_colour	colour (see Line Colours)	Define the colour used to display the label
label_position	Above Centre	Position label at centre above line
	Above Left	Position label on left above line

	Above Right	Position label on right above line
	Below Centre	Position label at centre below line
	Below Left	Position label on left below line
	Below Right	Position label on right below line
	None	Turn off label display
	Middle Left	Position label on left in middle
	Top Left	Position label on left at top
	Bottom Left	Position label on left at bottom
	Middle Right	Position label on right in middle
	Top Right	Position label on right at top
	Bottom Right	Position label on right in middle
label_orientation	Horizontal or Vertical	Orientation of the datum label(s)
label_point	point number	Position label at datum point
end_datum		Ends a Datum definition.

Each DATUM definition must start with a **"start_datum"** keyword and end with a **"end_datum"** keyword. Any lines between a **"start_datum"** and **"end_datum"** keyword that do not form part of a datum definition are ignored. From T/HIS 17.0 onwards, FAST-TCF variables defined by "define var *name value* " can be used inside DATUM defitions.

The following creates a DATUM definition at Y=1000.0 with a label "Hic Limit", the area below the line is filled in GREEN and the area above is filled in RED.

```
START_DATUM
  ACRONYM           datum_1
  LABEL             Hic Limit
  TYPE              constant_y
  VALUE             1000.000000
  LINE_COLOUR       green
  LINE_STYLE        solid
  LINE_WIDTH        normal
  FILL_COLOUR_1     red
  FILL_COLOUR_2     green
  LABEL_FONT        default
  LABEL_SIZE        automatic
  LABEL_COLOUR      foreground
  LABEL_POSITION    default
  LABEL_ORIENTATION Horizontal
END_DATUM
```

Alternatively a *.dtm file can be read in using the 'inc' keyword, e.g.

```
inc C:\my_datum_file.dtm
```

10.12.2. Adding Datum Lines to Graphs

Adding Datum Lines to Graphs

Multiple DATUM definitions can be added to each graph using the datum acronym

Keyword	2nd word	3rd word	notes
datum	add	*acronym*	Adds the datum with the acronym to the currently selected graphs

`datum add maximum:` Add datum with the acronym "maximum" to all selected graphs

`layout graph select none:` Deselect all graphs (see PAGE / GRAPH LAYOUT AND SELECTION)

`layout graph select 1:` Select graph 1 (see PAGE / GRAPH LAYOUT AND SELECTION)

`datum add maximum:` Add datum with the acronym "maximum" to graph 1 (currently selected)

`datum add minimum:` Add datum with the acronym "minimum" to graph 1 (currently selected)

10.12.3. Removing Datum Lines from Graphs

Removing Datum Lines from Graphs

Multiple DATUM definitions can be added to each graph using the datum acronym

Keyword	2nd word	3rd word	notes
datum	remove	*acronym*	Removes the datum with the acronym from the currently selected graphs

`datum remove maximum`	Remove datum with the acronym "maximum" from all selected graphs
`layout graph select none`	Deselect all graphs (see PAGE / GRAPH LAYOUT AND SELECTION)
`layout graph select 1`	Select graph 1 (see PAGE / GRAPH LAYOUT AND SELECTION)
`datum remove maximum`	Remove datum with the acronym "maximum" from graph 1 (currently selected)
`datum remove minimum`	Remove datum with the acronym "minimum" from graph 1 (currently selected)

10.13. FAST-TCF Image Output Options

FAST-TCF IMAGE OUTPUT OPTIONS

The options to generate images can be split into 5 sections:

Setting Curve Style
Setting Curve Styles by Model
Plot setup
Curve Display
Image Generation

10.13.1. Setting Curve Styles

Setting Curve Styles

Description	keyword	second word	following words
Plot style setup	style	style name	style options
Individual curve style	stylec	curve number or tag	style options

This section explains how to set up the styles for the curves in a plot. The two types of syntax available in the table above effect when and how the curves are styled.

The "plot style setup" (keyword **`style`**) allows the user to define a plot-specific styling that applies the styles to the curves only when they are requested for a plot. It is independent of the curve id, but dependent on the order the curves are requested in the plot command. The style is given its own "tag" which the user can request on the image FAST-TCF line.

This is useful for producing plots from FAST-TCF that all have the same curve appearance. For the following words, each space represents a new curve style definition. The styles for each curve are defined by the type keywords below, separated by commas.

e.g.	**`style ENERGIES`**	**`solid,green,norm`**	**`dash,blue,heavy`**	**`sol,W5,yel,500`**
	(style name)	(curve #1)	(curve #2)	(curve #3)

When a plot is requested, FAST-TCF will apply the curve styles to the list of curves (in order) in the plot. So in the example above, the first curve would appear green, the second curve blue and the third yellow.

The "individual curve style" (keyword **`stylec`**) is the more traditional way of styling a curve that a T/HIS user would be more familiar with - FAST-TCF styles the single curve id instantly. The user can only define one style at a time.

e.g.	**`stylec #12`**	**`solid,green,W9`**
	(style curve number 12)	style to apply

Line Styles

Style options	word options	default
Line style	solid	solid
	dash	
	none	

Line Colours

Style options	word options	default
Line colour	white	dependent on curve #
	red	
	green	
	blue	
	cyan	
	magenta	
	yellow	
	orange	
	turquoise	
	indigo	
	lime	
	sky	
	pink	
	black	
	foreground	
	background	

Line Width

Style options	word options	default
Line width	fine	normal
	normal	
	bold	
	heavy	

Line Symbols

Style options	word options	default
Line symbols	triangle	dependent on curve #
	square	
	diamond	
	hourglass	
	cross	
	circle	
	star	
	dot	
	null	
Symbol	frequency	-

10.13.2. Setting Curve Styles by Model

Setting Curve Styles by Model

From T/HIS 11.0 onwards, a new option can be used to colour all of the curves belonging to a model in a single operation.

Description	keyword	second word	following words
Colour curves by model	style_m	model number	style options

The available style options are exactly the same as for the `stylec` command (see Setting Curve Styles)

e.g.	**`style_m 2`**	**`solid,green,norm`**
	model number 2	style to apply

would set all the curves belonging to model 2 to solid, green lines using the default line thickness.

10.13.3. Plot Setup

Plot setup

Description	keyword	following words
Plot setup	setup	plot setup words

These options set the appearance of any plots that are created afterwards. They are to do with the general appearance of the plot rather than the curve itself. The curve appearances can be set up with the style definition line and on the image plotting line . All following words must be on the same line. If the "on" or "off" is missed out from the following word (where applicable) then FAST-TCF will take the **opposite** to the default (this helps with backwards compatibility issues but can also make a script more compact).

e.g.	**setup**	**ax bold**	**grid on**	**line bold**	**reverse**
		(bold axes)	(grid on)	(bold lines)	(reverse foreground and background)
	setup	**double on**	**border off**	**show 3ms**	**size 250**
		(double axes on)	(no border)	(3ms window on)	(size = 1000 x 750 pixels)
	setup	**fonts title hb 24 red**			
		(title: helvetica bold 24pt, in red)			

Plot setup description	plot setup word	following word(s)	notes
Axis thickness	ax	fine	for colours - see standard list below
		normal	
		bold	
		heavy	
		standard colour	
Background	back	standard colour	for colours - see standard list below
Border	bo	fine	for colours - see standard list below
		normal	
		bold	
		heavy	
		standard colour	

		on or off	
Double yaxis	do	on or off	
Fix line styles	fix	on or off	this overwrites any style definitions
Fonts	fon	[xl]abel [yl]abel [y2l]abel [xu]nits [yu]nits [y2u]nits [t]itle [le]gend [all] hm hb cm cb tm tb 8 10 12 14 18 24 Colour	sets up fonts for the image: fonts available: hm - helvetica medium cb - courier bold tm - times new roman medium etc... font sizes in pt: 8, 10, 12 etc...for colours - see standard list below
Foreground	fore	standard colour	for colours - see standard list below
Format style	fo	default	
		automatic	
		full	
Grid on	gr	fine	
		normal	
		bold	
		heavy	
		on or off	
Line thickness	li	on	Turn on plotting of curve lines
		off	Turn off plotting of curve lines (symbols drawn)
		fine	set the line thickness to 1 pixel
		normal	set the line thickness to 2 pixels
		bold	set the line thickness to 4 pixels
		heavy	set the line thickness to 8 pixels
Model numbers on labels.	mn	auto	Adds a "model" prefix to the entity IS. If set to the "auto" only puts the model number on when there is more than 1 model in T/HIS
		on	
		off	
Model prefix format	prefix	id	Set the model prefix to the model ID

		dir		Set the model prefix to the job directory
		thf		Set the model prefix tot eh amne fof the base THF/model file
		user		Set the modle prefix to the user defined one.
Reverse black white	re	on or off		
Size of plot	si	integer		xsize = value x 4, ysize = value x 3 (aspect fixed)
Solid x and y axis	so	on or off		
Symbols on	sy	on or off		
X grid controls	xau	-		
	xin	x grid increment		
	xoff	x grid offset		
Y grid controls	yau	-		
	yin	y grid increment		
	yoff	y grid offset		
Axis type, Linear/Logarithmic	xlin	-		Swap the x axis to a linear scale
	xlog	-		Swap the x axis to a logarithmic scale
	ylin	-		Swap the y axis to a linear scale
	ylog	-		Swap the y axis to a logarithmic scale
	y2lin	-		Swap the second y axis to a linear scale
	y2log	-		Swap the second y axis to a logarithmic scale
Axis display	axis	top	on or off	Turns ON/OFF display of graphs TOP axis
	axis	bottom	on or off	Turns ON/OFF display of graphs RIGHT axis

Plot setup description	**plot setup word**	**following word(s)**	**notes**
Graph Ttitle	title	*"title string"*	Set the title for the graph.
	title_on	-	Turn on the display of the graph title
	title_off	-	Turn off the display of the graph title
X Axis Properties	x_lab	auto	Set the x axis label to be defined automatically
		manual	Set the x axis label to a user defined label

		"label string"	Set the user defined x axis label
		on	Turn on the display of the x axis label
		off	Turn off the display of the x axis label
	x_min	auto	Set the x axis minimum value to automatic
		numerical value	Set the x axis minimum value
	x_max	auto	Set the x axis minimum value to automatic
		numerical value	Set the x axis minimum value
	x_unit	auto	Set the x axis unit label to be defined automatically
		manual	Set the x axis unit label to a user defined label
		"unit string"	Set the user defined x axis label
		on	Turn on the display of the x axis unit label
		off	Turn off the display of the x axis unit label
Y Axis Properties	y_lab	auto	Set the y axis label to be defined automatically
		manual	Set the y axis label to a user defined label
		"label string"	Set the user defined y axis label
		on	Turn on the display of the y axis label
		off	Turn off the display of the y axis label
	y_min	auto	Set the y axis minimum value to automatic
		auto_visible	Set the y axis minimum to the automatic value based on the visible part of the x-axis.
		numerical value	Set the y axis minimum value
	y_max	auto	Set the y axis maximum value to automatic
		auto_visible	Set the y axis maximum to the automatic value based on the visible part of the x-axis.
		numerical value	Set the y axis maximum value
	y_ranges	auto	Set the y and y2 axis minimum and maximum values to automatic
		auto_visible	Set the y and y2 axis minimum and maximum to the automatic values based on the visible part of the x-axis.
		y_auto	Set the y axis minimum and maximum values to automatic

		y_auto_visible	Set the y axis minimum and maximum to the automatic values based on the visible part of the x-axis.
	y_unit	auto	Set the y axis unit label to be defined automatically
		manual	Set the y axis unit label to a user defined label
		"unit string"	Set the user defined y axis label
		on	Turn on the display of the y axis unit label
		off	Turn off the display of the y axis unit label
2nd Y Axis Properties	y2_lab	auto	Set the second y axis label to be defined automatically
		manual	Set the second y axis label to a user defined label
		"label string"	Set the user defined second y axis label
		on	Turn on the display of the second y axis label
		off	Turn off the display of th seconde y axis label
	y2_min	auto	Set the second y axis minimum value to automatic
		auto_visible	Set the second y axis minimum to the automatic value based on the visible part of the x-axis.
		numerical value	Set the second y axis minimum value
	y2_max	auto	Set the second y axis maximum value to automatic
		auto_visible	Set the second y axis maximum to the automatic value based on the visible part of the x-axis.
		numerical value	Set the second y axis maximum value
	y_ranges	auto	Set the y and y2 axis minimum and maximum values to automatic
		auto_visible	Set the y and y2 axis minimum and maximum to the automatic values based on the visible part of the x-axis.
		y2_auto	Set the second y axis minimum and maximum values to automatic
		y2_auto_visible	Set the second y axis minimum and maximum to the automatic values based on the visible part of the x-axis.
	y2_unit	auto	Set the second y axis unit label to be defined automatically

		manual	Set the second y axis unit label to a user defined label
		"unit string"	Set the user defined second y axis label
		on	Turn on the display of the second y axis unit label
		off	Turn off the display of the second y axis unit label

Deprecated Plot Setup Options

The following setup commands have been deprecated since T/HIS 9.4 and may be removed entirely in the future. They are superseded by the "`properties`" keyword (see Setting properties for curves).

Plot setup description	plot setup word	following word(s)		notes
Set colour of min/max value	min_max	standard colour		for colours - see standard list below
Show max value	show	max	on off	Turn on/off the highlight of the Maximum Value
Show max value	show	min		Turn on/off the highlight of the Maximum Value
Display X value at max	show	xmax		Display x value at Maximum
Display X value at min	show	xmin		Display x value at Maximum
Display Y value at max	show	ymax		Display y value at Maximum
Display Y value at min	show	ymin		Display y value at Maximum
Show 3ms Clip Widow	show	3ms		
Show HIC	show	hic		

10.13.4. Curve Display

Curve Display

The list of curves displayed in each graph is controlled by the `display` keyword. The list of curves can contain a mixture of curve tags, curve numbers (prefixed with #) or curve groups. If curve tags are specified in the curve list then they can contain wildcards.

keyword	second word	notes
display	curve list	The curve list can contain a mixture of curve tags, curve numbers (prefixed with #) or curve groups. If curve tags are specified in the curve list then they can contain wildcards.

The following option can be appended to the `display` keyword after the curve list.

Additional format	format word	following word #1	following word #2	notes
Style application	sty	style name	-	Curves have styles applied in the order they were defined

The following additional options that can be appended to the `display` keyword after the curve list were deprecated in T/HIS 9.4 and may be removed entirely in the future, so their use is not recommended. Equivalent commands have been added to the Plot Setup commands along with a number of new options.

Additional format	format word	following word #1	following word #2	notes
Title	tit	title word #1	title word #2 etc	Takes following words as a title until another keyword is found
X axis options	xax	if numeric #1 - xaxis min	if numeric #2 - xaxis max	Takes following words as a label until another keyword is found
		otherwise xaxis label	otherwise xaxis label	
Y axis options	yax	if numeric #1 - yaxis min	if numeric #2 - yaxis max	Takes following words as a label until another keyword is found
		otherwise yaxis label	otherwise yaxis label	
2nd Y axis options	2ya	if numeric #1 - yaxis min	if numeric #2 - yaxis max	

		otherwise yaxis label	otherwise yaxis label	Takes following words as a label until another keyword is found

e.g. **display curve_1 curve_2**

(display "curve_1" and "curve_2")

```
display     curve_2     &"Curve group 3"     title SLED TEST \
                                             xax Time \
                                             yax Displacement
```

(display "curve_2" and all the curves in "Curve group 3". Set the plot title and x and y axis labels.)

10.13.5. Image Generation

Image Generation

Many different types of image format can be outputted from FAST-TCF.

From T/HIS 17.0 onwards, the FAST-TCF image output options have been revised to allow multiple graphs and pages to be selected for output. The old pre-T/HIS 9.3 syntax (see Pre 9.3 Image Output) is still supported for existing scripts but is officially deprecated and users are strongly advised to move to the new command format where all options are prefixed with either the "display" or "image" keyword.

Description	**keyword**	**following words**
Image output	image	image options

The available image output options are

Option	**keyword**	**format word**	**second word**	**third word**	**fourth word**	**notes**
Bitmap (8 bit)	image	bit / bmp	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
Bitmap (8 bit uncompressed)	image	bit_u / bmp_u	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
Gif (8 bit)	image	gif	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
Png (8 bit)	image	png	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'

Bitmap (24 bit)	image	bit24 / bmp24	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
Pixel map (24 bit)	image	ppm / pix	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
Jpeg (24 bit)	image	jpg / jpeg	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
Png (24 bit)	image	png24	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
Postscript	image	ps	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'
PDF	image	pdf	filename	graph	all / active / 'n'	Generate an image containing all graphs / all active graphs / graph number 'n'
				page	all / current / 'n'	Generate an image for each page / the current page / page number 'n'

In addition to the image formats the following image output options can also be specified

Description	**keyword**	**second word**	**notes**

Image resolution	i_res	screen / 2x / 4x	Set the resolution to either the same as the screen or 2 or 4 times the screen resolution for image ouptut
Postscript /PDF resolution	p_res	screen / 2x / 4x	Set the resolution to either the same as the screen or 2 or 4 times the screen resolution for Postscript and PDF ouptut
Plot title	ti	title string	Specify the plot title (postscript / PDF output only)
Figure Number	fi	figure number	Specify the figure number (postscript / PDF output only)
Orientation	ori	land / port	Specify the paper orientation (postscript / PDF output only)

`image bmp` **`output1.bmp graph all`**

(generate a bitmap called output1.bmp containing all the current graphs)

`image jpeg` **`output2.jpg page 3`**

(generate a JPEG image called output2.jpg containing page 3)

`image i_res` **`2x`**

(set the resolution used for all following images to 2 x the screen resolution)

`image ti` **`Run number 2`**

(set the plot title to "Run number 2" for any following postscript or PDF images)

`image ori` **`landscape`**

(set the page layout to landscape for any following postscript or PDF images)

10.13.6. Pre 9.3 Image Output

Pre 9.3 Image Output

The following pre-T/HIS 9.3 image output commands are still supported but has been officially deprecated and users are recommended to use the new format described above. **Support for the below commands may be removed entirely in the future.**

Curve styles that have been previously defined can be applied to the curves in the plot, and various other settings can be made with regards to the axes and titles.

Images that require a second yaxis need to determine which curves go on which axis. To do this, use a "##" in the curve listing to switch to the second axis. The options are described in the tables below.

Curve files can be included within the curves to plot. FAST-TCF detects a curve file to read in using the pattern string ".cur" at the end of the name. The curves are read in, styles are applied, and the image is plotted. The curves are then deleted.

The user can use wildcards ("*") in the tag names to select multiple curves for plotting.

`bit d.bmp`	`#1 #3 CRV2 ## #2 #4 head_accn`			`Title 2nd axis example`	
	(2 curves on 1st yaxis and 3 on 2nd yaxis)			(Title)	
`bit h.bmp`	`#1 #3 CRV2`	`style ONE`	`xax 0 5E-3`	`Time`	`Title Head`
	(curves)	(style to apply)	(xaxis min and max)	(XLabel)	(Title)
`bit l.bmp`	`#1 #100 reference.cur line.cur #1000`			`style reference`	
	(curves and curve files to plot)			(style to apply)	
`bmp test.bmp`	`accn*`				
	(all curves with tags beginning with "accn")				

Description	keyword	second word	following words	following words
Bitmap	bit / bmp	filename	curve(s)	additional formatting
Bitmap (uncompressed)	bit_u / bmp_u	filename	curve(s)	additional formatting
Jpeg	jpg / jpeg	filename	curve(s)	additional formatting

Pixel map	ppm	filename	curve(s)	additional formatting
B & W postscript	post	filename	curve(s)	additional formatting
Colour postscript	cpost	filename	curve(s)	additional formatting

Additional format	**format word**	**following word #1**	**following word #2**	**notes**
Style application	sty	style name	-	Curves have styles applied in the order they were defined
Title	tit	title word #1	title word #2 etc	Takes following words as a title until another keyword is found
X axis options	xax	if numeric #1 - xaxis min	if numeric #2 - xaxis max	Takes following words as a label until another keyword is found
		otherwise xaxis label	otherwise xaxis label	
Y axis options	yax	if numeric #1 - yaxis min	if numeric #2 - yaxis max	Takes following words as a label until another keyword is found
		otherwise yaxis label	otherwise yaxis label	
2nd Y axis options	2ya	if numeric #1 - yaxis min	if numeric #2 - yaxis max	Takes following words as a label until another keyword is found
		otherwise yaxis label	otherwise yaxis label	

10.14. Outputting Curve Properties to Text Files, Variables and REPORTER

Outputting curve properties to text files, variables and REPORTER

These requests output a curve property (eg its maximum Y value) into a specified tabulation file, to a REPORTER variable in a text file, or into a variable within FAST-TCF.

Output type	keyword	2nd word	3rd word	4th word	extra words	Format (optional)	variable word	variable name	description words	Notes
Tabulation file	tab	filename	curve #	property to output	if value s needed	format	varf	variable name	description	
Tabulation file append	taba	filename	curve #	property to output	if value s needed	format	varf	variable name	description	
Tabulation file (csv)	tabc	filename	curve #	property to output	if value s needed	format	varf	variable name	description	Each output is appended to the current line in the file.
Tabulation file (csv)	tabcr	filename	curve #	property to output	if value s needed	format	varf	variable name	description	Each output is appended to the current line in the file, followed by a carrage

										return so that the next output starts a new line.
FAST-TCF variable	varf	variable name	curve #	property to output	if values needed	format	-	-	description	
REPORTER variable	var	variable name	curve #	property to output	if values needed	format	varf	variable name	description	
REPORTE	vara	variabl	curv	proper	if	format	varf	variabl	descriptio	

10.14.1. Available Curve Properties

Available Curve Properties

Various advanced requests can be performed (e.g. first non-zero Y, maximum in a window) and the table below describes them in more detail. Requests which require inputs (e.g. t1 and t2 of a window) take the default values in the table if the following words do not appear to be numbers, or if no following words exist.

Property to output	property word	value words	notes
Minimum x	minx	-	-
Maximum x	maxx	-	-
Minimum y	min	-	-
X at minimum y	xatmin	-	-
Y at minimum x	yatmin	-	-
Minimum y in window t1 t2	minw	t1 and t2	default t1=-1E19 and t2=+1E19
X at minimum y in window t1 t2	xminw	t1 and t2	default t1=-1E19 and t2=+1E19
Maximum y	max	-	-
X at maximum y	xatmax	-	-
Y at maximum x	yatmax	-	-
Maximum y in window t1 t2	maxw	t1 and t2	default t1=-1E19 and t2=+1E19
X at maximum y in window t1 t2	xmaxw	t1 and t2	default t1=-1E19 and t2=+1E19
Average in window t1 t2	ave	t1 and t2	default t1=-1E19 and t2=+1E19
Hic	hic	-	-
t1 of Hic window	hict1	-	-
t2 of Hic window	hict2	-	-
Hicd	hicd	-	-
t1 of Hicd window	hicdt1	-	-
t2 of Hicd window	hicdt2	-	-
3ms	3ms	-	-
t1 of 3ms window	3mst1	-	-
t2 of 3ms window	3mst2	-	-
Y at X	yatx	x value	default xvalue=-1E19
X when Y is passed after gate time	xygate	y value & gate time	default yvalue=-1E19, gate=+1E19
X at first non-zero Y	xnonz	-	nonzero = 1/1000000th of curve max

X at last non-zero Y	xfail	-	nonzero = 1/1000000th of curve max
Y value at last non-zero Y	yfail	-	nonzero = 1/1000000th of curve max
TTI	tti	-	-
Error Function - Max difference & time	max_err	-	-
Error Function - Difference as a %age of reference	pc_err	-	-
Error Function - Difference as a %age of peak reference	pc_max_err	-	-
Error Function - Average Difference	av_err	-	-
Error Function - Average Difference as a %age of peak reference	av_max_err	-	-
Error Function - Area weighted difference	area_err	-	-
Error Function - Max difference & time	err	-	-
Curve Correlation Function	correlate		Returns curve correlation value

The are also variables which relate to properties of all curves rather than specific curves. In this case the 3rd word "curve #" should be replaced by "all".

Property to output	property word	value words	notes
Minimum x over all curves	all_minx	-	-
Maximum x over all curves	all_maxx	-	-
Minimum y over all curves	all_miny	-	-
X at minimum y over all curves	all_xatmin	-	-
Maximum y over all curves	all_maxy	-	-
X at maximum y over all curves	all_xatmax	-	-
Curve number of curve containing minimum y over all curves	all_catmin	-	-
Curve number of curve	all_catmax	-	-

10.14.2. Writing Out Curve Properties to a Text "Tabulation" File

Writing out curve properties to a text "tabulation" file

This is achieved using the **`"tabulation"`** command. This automatically overwrites any existing file in the output directory, but only on the **first occurrence** in the input script. If this is not desired then use the **`"taba"`** command which will append an existing file on the first tab call.

The command **`"tabc"`** is available from T/HIS 9.2 onwards. This command appends the data into CSV format on the last line in the file. The first call to this command writes a **new line** to the file, and the subsequent calls append the end of this line. This enables the user to compare runs on a line by line basis in software such as Microsoft Excel.

Some examples of writing out curve properties to a text file are below:

e.g.	**`tab output.txt`**	**`#1`**	**`max`**	**`max y of curve #1`**	
	(file output.txt)	(curve number)	(maximum Y)	(description)	
	`tab output.txt`	**`node_head_accn`**	**`maxw`**	**`1.00E-03`**	**`30.00E-3`**
	(file output.txt)	(curve tag)	(max Y in window)	(window t1)	(window t2)
	`taba output.txt`	**`node_head_accn`**	**`min`**		
	(append output.txt)	(curve tag)	(minimum Y)		

Properties for multiple curves can be output by specifying either multiple "tab" commands or by using a curve tag containing wildcards or a curve group.

e.g.	**`tab output.txt`**	**`node_*`**	**`max`**	**`maximum y value`**
	(file output.txt)	(all curves with a tag starting with node_)	(maximum Y)	(description)
	`tab output.txt`	**`&group_1`**	**`max`**	**`maximum y value`**
	(file output.txt)	(all curves in group	(maximum	(description)

10.14.3. Writing Out REPORTER Variables

Writing out REPORTER variables

REPORTER can write curve properties to its reports, so FAST-TCF needs to output a text file that REPORTER can interrogate to find out the curve properties it needs. To tell FAST-TCF to output a REPORTER variable, the keyword **"varr"** is used (for backwards compatibility **"var"** is sufficient). Use **"vara"** to append to an existing file.

e.g.	**varr head_hic**	**#1**	**hic**	**hic result for head node**
	(REPORTER variable %head_hic%)	(curve number 1)	(output request)	(description)
e.g.	**vara max_y**	**#1**	**max**	**maximum y value**
	(REPORTER variable	(curve	(output	(description)

10.14.4. Setting Up New FAST-TCF Variables to Contain Curve Properties

Setting up new FAST-TCF variables to contain curve properties

If you wish to use a curve property as a new variable within FAST-TCF - there are two ways you can achieve this.

1. Use the keyword **`"varf "`**. This should be used when the user doesn't also require the value to be outputted into a text file or a REPORTER file.
2. Within a **`"tab"`** , **`"taba"`** , **`"tabc"`** or **`"varr"`** line, use the word **`"varf"`** just **before** the description words. The variable name is defined as the word after **`"varf"`** .

The variable value is equal to the return value of the request. The variable can then be used in any subsequent lines of FAST-TCF.

For instance, the simplest way to set the variable **`MAX_ACCN`** to the max of curve #1 is:

```
varf MAX_ACCN #1 max
```

However, if the user wishes to combine writing a property to a text file and defining a variable in FAST-TCF, this syntax could be used:

```
tab output.txt #1 max varf MAX_ACCN
```

10.14.5. Format

Format

From T/HIS 9.3 onwards, the format used to display the value can be controlled by adding an optional "format" keyword after the property to be output and any additional inputs that property requires. The format should be specified directly after the "format" keyword and should use standard "C" programming syntax to specify a floating point format using either f,e,E,g or G format specifiers.

e.g.	**tab output.txt**	**head**	**max**	**max y of curve #1**	
	(file output.txt)	(curve tag)	(maximum Y)	(description)	
	tab output.txt	**head**	**max**	**format %6.3f**	**max y of curve #1**
	(file output.txt)	(curve tag)	(maximum Y)	(format)	(description)
	tab output.txt	**head**	**max**	**format %.3f**	**max y of curve #1**
	(file output.txt)	(curve tag)	(maximum Y)	(format)	(description)

Example formats

Number	Format	Output
12.3456	%5.2f	12.35
12.3456	%7.3e	1.2345e+01
12.3456	%7.3E	1.2345E+01
2345678.9	%.0f	2345678
2345678.9	%6.5g	2.3457e+06
2345678.9	%6.5G	2.3457E+06
-0.000013583	%4.3e	-1.358E-05

10.14.6. Description

Description

From T/HIS 9.3 onwards, the description specified as part of the output for a curve property can contain the following keywords that will automatically be replaced with the corresponding curve property.

keyword	Curve Property
{tag}	FAST-TCF curve tag
{label}	Curve label
{id}	Entity ID that the curve was created from
{model}	Model ID curve was created from

e.g.	**`tab output.txt`**	**`head`**	**`max`**	**`Max accl of node {id}`**
	(file output.txt)	(curve tag)	(maximum Y)	(description)
	`tab output.txt`	**`head`**	**`max`**	**`Model {model} max accl of node {id}`**
	(file	(curve	(maximum	(description)

10.15. FAST-TCF Curve Output

FAST-TCF CURVE OUTPUT

Curves can be written out to either T/HIS curve files or CSV files from within a FAST-TCF script by using either the **`"app"`**, **`"cop"`**, **`"csv"`** or **`"csv2"`** keyword.

Description	**keyword**	**second word**	**third word**		**notes**
Copy into file	cop	filename	curve list	-	will overwrite any previous file
Append into file	app	filename	curve list	-	will append any previous file
CSV file TYPE 1	csv	filename	curve list	-	will overwrite any previous csv file. CSV has the format X1,Y1,X2,Y2,X3,Y3
CSV file TYPE 2	csv2	filename	curve list	last word = auto	will overwrite any previous csv file. CSV has the format X1,Y1,Y2,Y3 x axis interval is taken from curve #1 if all curves are chosen
				2nd last word= x start time last word = x axis interval	will overwrite any previous csv file. CSV has the format X1,Y1,Y2,Y3 start time and interval are defined in the line

The curve list for all of these commands can contain either curve tags (with or without wildcards), curve numbers (prefixed with #), curve groups or '*' to select all curves.

e.g.

```
copy   output_file.cur curve_1  &"group 1"
```

(Write "curve_1" and all the curves in curve group "group 1" to a new file "output_file.cur")

```
append output_file.cur curve_1  &"group 1"
```

(Append "curve_1" and all the curves in curve group "group 1" to the file "output_file.cur")

```
csv    output.csv      curve_1* curve_2*
```

(Write all curves with tags that start with "curve_1" or "curve_2" to a CSV called "output.csv")

NOTE : There is no limit to the number of curves that can be output to a file but there is a limit to the number of items that can be specified in the curve list (currently 100). If more than 100 curves are to be output to a file then a curve group containing all of the curves should be created and used within the curve list. Alternatively if the curves are being written to a T/HIS curve file then the first 100 curves can be output using the **"cop"** keyword and then additional curves can be appended to the file using the **"app"** keyword.

Specifying curves using Curve Numbers

When outputting curves to a file a range of explicit curve numbers can be specified using the syntax **#start:#end** . This option only applies to curve numbers because curve tags can be defined in an arbitrary order.

CSV files

If a CSV/CSV2 file is written out from within a FAST-TCF script then by default it will contain rows containing UNIT information for the curves if UNITS have been defined. This additional information can be turned off if it isn't required (see Curve Unit Systems).

10.16. FAST-TCF ADDITIONAL

10.16.1. T/HIS Preferences and Additional Commands

T/HIS preferences and additional commands

There are a number of additional commands that improve the functionality of FAST-TCF such as labeling, resetting values, tagging curves and defining variables. All following words must be on the same line. The variables section is explained in more detail below
.

e.g.	`report`	`3ms`	`err`	`hic`
		(FAST-TCF.clp written to)	(FAST-TCF.err written to)	(FAST-TCF.hic written to)
	`define`	`file`	`lsda`	
			(define lsda as default file)	
	`copy`	`output.cur`	`#1`	
		(file name)	(copy curve #1 into file name)	
	`define`	`var`	`date`	`30_Nov_2005`
		(define variable)	(variable name)	(variable value)

Description	keyword	second word	third word	fourth word	notes
Autoscale plot	ac	-	-	-	for use in interactive playback
Auto update plot	auto	on or off	-	-	whether to auto update the plot on data read / font updates and so on. Please note this is reset to ON after any font definition.
Plot graphs	plot	-	-	-	for use in interactive playback
Append into file	app	filename	curve name	-	will append any previous file
Condense	condense	-	-	-	Condense the curve numbers
Convert	convert	on or off			Automatically convert curves from ms units to s(econds) before applying any filters and then convert back again.

Define FAST-TCF variable	def	var	name (without "$")	value	see FAST-TCF variables section
Define error fail value	def	err	error value (integer)	-	default is 10 errors before T/HIS will stop
Define default file	def	file	lsda ascii xtf thf default	-	will always check that T/HIS can get the output from this file, if not then the original default file will be chosen (see data extraction table). This file can still be overwritten on the actual input line
Define default title	def	tit	title word 1	title word 2 etc	
Define user line	def	user	user line number (1 to 6)	font size (8 to 24)	rest of line is the label
Define surface integration	def	surf	shell / beam / thickshell	layer number	t = top, m = middle, b = bottom, or use a number for the integration point (see INPUT FOR DATA EXTRACTION REQUESTS)
Define ssd component	def	ssd_comp	amplitude / angle	-	Define which value is read for each data component in a Steady State Dynamics (ssd) analysis (see INPUT FOR DATA EXTRACTION REQUESTS)
CSV field separator	def	csv_separator	comma tab space		used to change the field separator before reading a CSV file.
Delete	del	curve #1 to curve #n	-	-	Delete curves
Exit reading file	exit	-	-	-	stops reading file here

ISO MME curve label format	isolabel	"label" or "code"	-	-	sets the curve label format when reading ISO MME data
Model set	mod	model # or "no" or "all"	-	-	sets the model number for extracting curve data
Regularise filtering	reg	time interval, or "off"	-	-	sets the auto regularise interval and turns it on, or turns it off
Report files written	rep	3ms asi err hic thiv tti	-	-	To turn off see the **reset2** keyword
Reset	reset1	-	-	-	All curves and curve tags deleted
	reset2	-	-	-	Plot setup defaulted and all style definitions removed. Report files not written
	reset3	-	-	-	Variable names and default title removed
User-defined colours	col	colour number (1-6)	RRGGBB (6-digit hexadecimal)	-	6-digit hexadecimal format for rgb colour values.
	col_rgb	colour number (1-6)	r g b (3 integers in	-	Seperate integer format for rgb colour values.

T/HIS

10.16.2. Limits

Limits

Description	limit
word limit per line	80 words
include file name	150 characters
tag length	60 characters

10.16.3. Variables

Variables

Variable names can only have "a-z", "0-9" and "_" in them. Variables can be inserted anywhere in the script, FAST-TCF will replace any variables with their corresponding values before processing the line, for example:

```
define var output displacement
define var nod_id 12345678
define var xscale 0.001

node $nod_id $output x xscale $xscale
```

converts into:

```
node 12345678 displacement x xscale 0.001
```

Variable definitions can contain several words or other varaibles, and these will be joined together to form the final variable value, for example:

```
define var day 31st
define var month january
define var year 2099
define var date $day _ $month _ $year
```

creates the variable **date** with value **31st_january_2099**

Because variables cannot have anything other than "a-z", "0-9" and "_" in them, it is possible to use variables within strings:

```
define var analysis run01_vers2

read january_$analysis.cur
```

converts into

```
read january_run01_vers2.cur
```

However, sometimes the user may want to insert a variable within other alphanumeric words, in these circumstances use a **"$$"** terminator to designate the end of the variable name:

```
define var analysis xyz_run01
read $analysis$$_x.cur
```

converts into

```
read xyz_run01_x.cur
```

There are several built in variables, and these depend on the system and command line used to run FAST-TCF, they can be checked on the dialogue T/HIS prints before starting:

$run_name	This is the basename of the key file in for the 1st model directory (should there be one). If a script refers to multiple models then **$run_name *N*** (where ***N*** is the model number) can be used for each model.
$run_dir	This is the full pathname of the directory containing the output files for a model. If a script refers to multiple models then **$run_dir *N*** (where ***N*** is the model number) can be used for each model.
$run_title	This is the title of the analysis found in the output files. If a script refers to multiple models then **$run_title *N*** (where ***N*** is the model number) can be used for each model.
$ftcf_script	This is the name of the FAST-TCF that is being run
$ftcf_script_dir	This is the name of the directory containing the FAST-TCF that is being run
$ftcf_dir	This is the name of the current working directory.
$ftcf_path	This is the full pathname of the current working directory.
$ftcf_startin_dir	This is the full pathname of the directory that T/HIS was started in.

```
FASTTCF called

=====================================================
Fasttcf script: C:\example\test.inp
tmp file:   C:\example\other_folder\test.tmp
tcf file:   C:\example\other_folder\test.tcf
report file:    C:\example\other_folder\test.rep
$FTCF_SCRIPT variable:  test.inp
$FTCF_SCRIPT_DIR variable:  C:\example\
$FTCF_DIR variable: other_folder
$FTCF_PATH variable:    C:\example\other_folder
$RUN_TITLE var: lg09 : Large Test 9: Belted sled test
$RUN_NAME var:  new_lg09
$RUN_DIR var:   E:\test\sled
$RUN_TITLE1 var:    lg09 : Large Test 9: Belted sled test
$RUN_NAME1 var: new_lg09
$RUN_DIR1 var:  E:\test\sled
```

10.16.4. Notes and Presumptions

Notes and Presumptions

- Curves must be in the format: #<curve number> to differentiate between curves and constants
- Any image will be overwritten if it already exists in the run area
- Curves are always labeled and then written to files, any other options are done in the order of input on the line
- If your line is getting too long, use a "\" to designate a continuation line - FAST-TCF will then join the lines together before processing

11. Search (Quick Find)

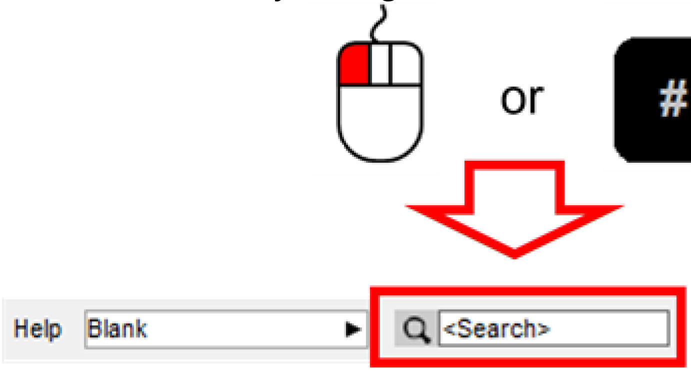
11.1. Introduction

Introduction

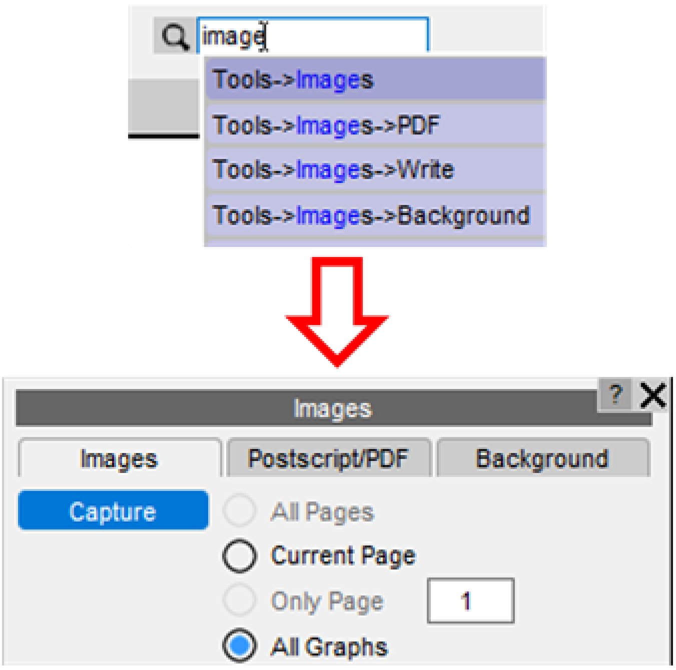
Quick Find can be used to search for and quickly:

- Go to menus / functionality
- Open tutorials

It can be accessed by clicking in the Search box in the top bar or by pressing the '#' key.



Typing in the textbox brings up a list of found items that match the entered text. Items in the list can be selected by clicking on them or by using the up and down arrow keys and pressing enter. The selected item will then perform the task, e.g. open a menu.

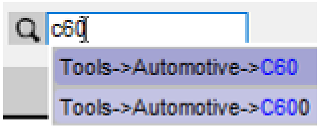


11.2. Fuzzy Matching

Fuzzy Matching

A 'fuzzy' matching method is used to match the entered text with the searchable items. It judges that something has matched when the characters of the entered text appear in the same order as the item that can be searched for.

For example if you type 'c60' then 'Tools->Automotive-> **C60** ' would be a match, but 'Tools->Automotive-> **C1000** ' wouldn't because the '6' doesn't match. (Note that the search is case insensitive).



Additionally, if the entered search pattern contains spaces and the characters do not all match in the same order then T/HIS will look to see if the words can be swapped to find a match.

For example 'back image' would find 'Image->Background' even though the words do not appear in that order.

This hopefully makes it easier to find items as you do not need to know the precise search term.

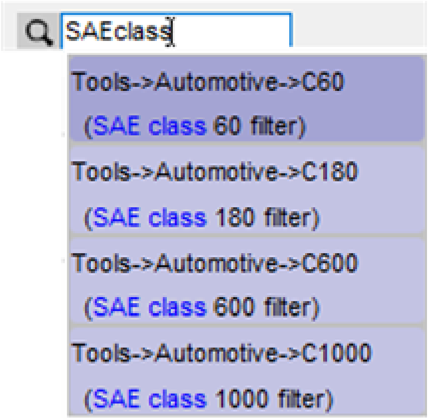
The found items are listed in order of how closely they match the entered text so items that more closely match appear nearer the top of the list. It determines this by assigning a score to each match, with higher scores given to items that contain consecutively matched characters and if the characters appear at the start of words.

11.3. Search Terms

Search Terms

The default search term associated with a menu item is the trail of menus/buttons you would need to manually open/press, e.g. to get to the C60 filter you would need to go to **Tools**, then **Automotive** then **C60**, hence the search term 'Tools->Automotive->C60'.

In addition, some menus have alternative search terms associated with them. For example C60 can also be found from the alternative text 'SAE class 60 filter':



This can be useful for cases where you don't know or can't remember under which menu some functionality lives.

Note that the alternative text appears in brackets under the default search term so you can see how you would get to the menu manually.

If you can't find menus that you know exist in T/HIS it is likely that you are using different terminology to what we expect. If so, please contact Oasys Ltd and we can add alternative text based on what you are entering as your search text.

Alternative text associated with a menu may also describe some of the features on a menu. For example the text colour is set in the Display menu, but if you didn't know this it would be hard to find.

In this case the alternative text 'Text Colour' is associated with this menu:

11.4. Tutorials

Tutorials

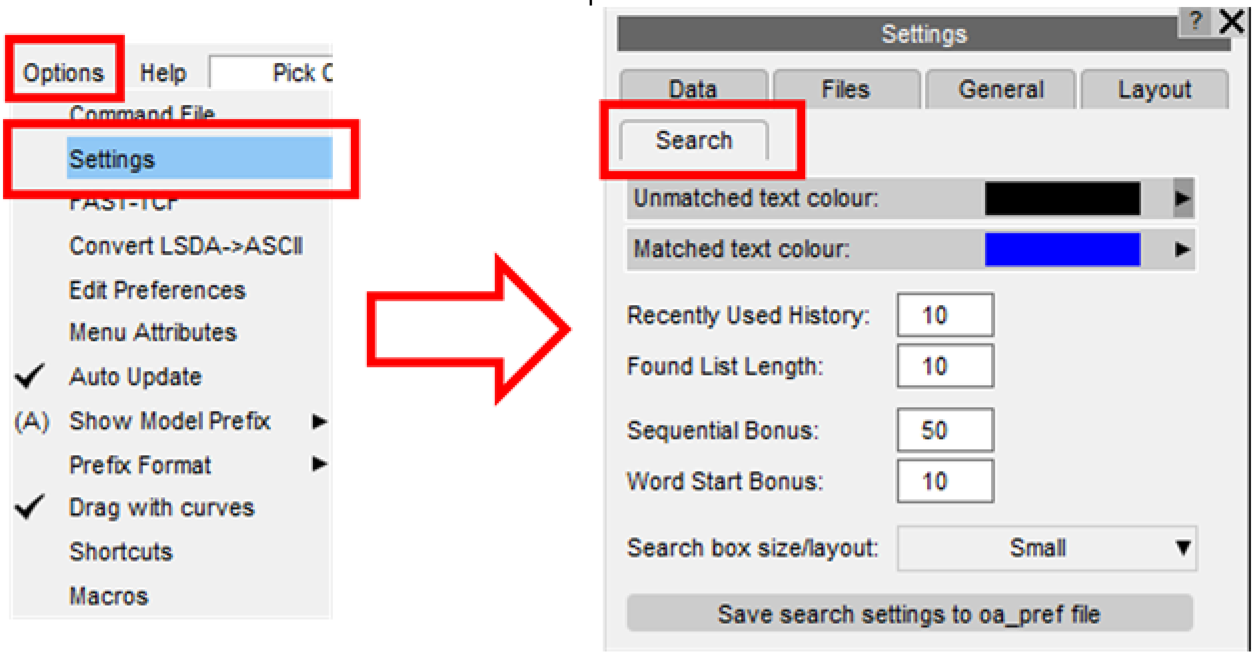
The full installation of the Oasys Ltd software contains some pdf tutorials for various features within the software. They are installed in the $OA_INSTALL/manuals/tutorials/this directory and can be found and opened using Quick Find.

11.5. Options

Options

There are a few options that can be set to alter how Quick Find works. These can be accessed by pressing the **Options** -> **Settings** button and choosing the **Search** tab.

- Save settings to the oa_pref file
- Set the text colours for matched and unmatched characters
- Recently selected items are saved by T/HIS and appear higher in the list of available options. By default the last ten selected items are saved, but this can be changed here. To turn it off set it to zero.
- Set the maximum number of found items to display in the list
- The size of the Search box on the top bar



12. REPORTER Integration

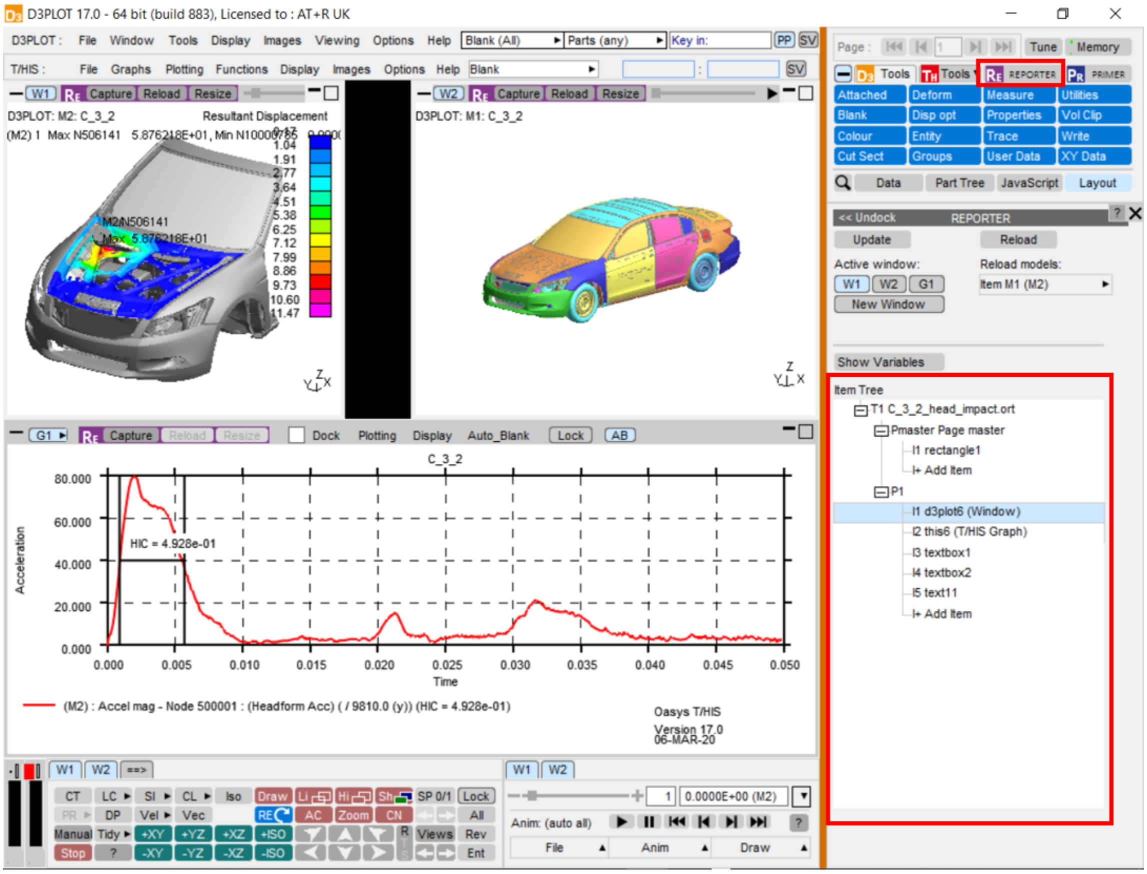
REPORTER INTEGRATION

This section describes how to work with D3PLOT, T/HIS and REPORTER to quickly and easily create reports from results.

12.1. Linking the Programs

Linking the Programs

REPORTER can be opened from D3PLOT and T/HIS using the REPORTER button in the top-right. This opens a linked session of REPORTER, allowing reports to be interactively created and edited. Both D3PLOT and T/HIS can be opened from inside REPORTER too, using the program buttons in the top bar of REPORTER. REPORTER can be connected to both D3PLOT and T/HIS at the same time and the D3PLOT->T/HIS link is also supported. Graphs in T/HIS are treated the same as graphs in a D3PLOT->T/HIS linked session.



12.2. Item Tree

Item Tree

Once a template is opened in REPORTER, all items in the template will appear in the Item Tree in the REPORTER panel in D3PLOT or T/HIS. Selecting an item in the Item Tree will select the corresponding item in REPORTER and vice-versa.

The Item Tree can include items of all types in REPORTER, such as textboxes and images, as well as D3PLOT, T/HIS and PRIMER items. Only placeholders, D3PLOT items and T/HIS items can be overwritten with new D3PLOT or T/HIS items. Placeholder items exist to allow a layout to be created for the report before populating it and can be converted into any other item type.

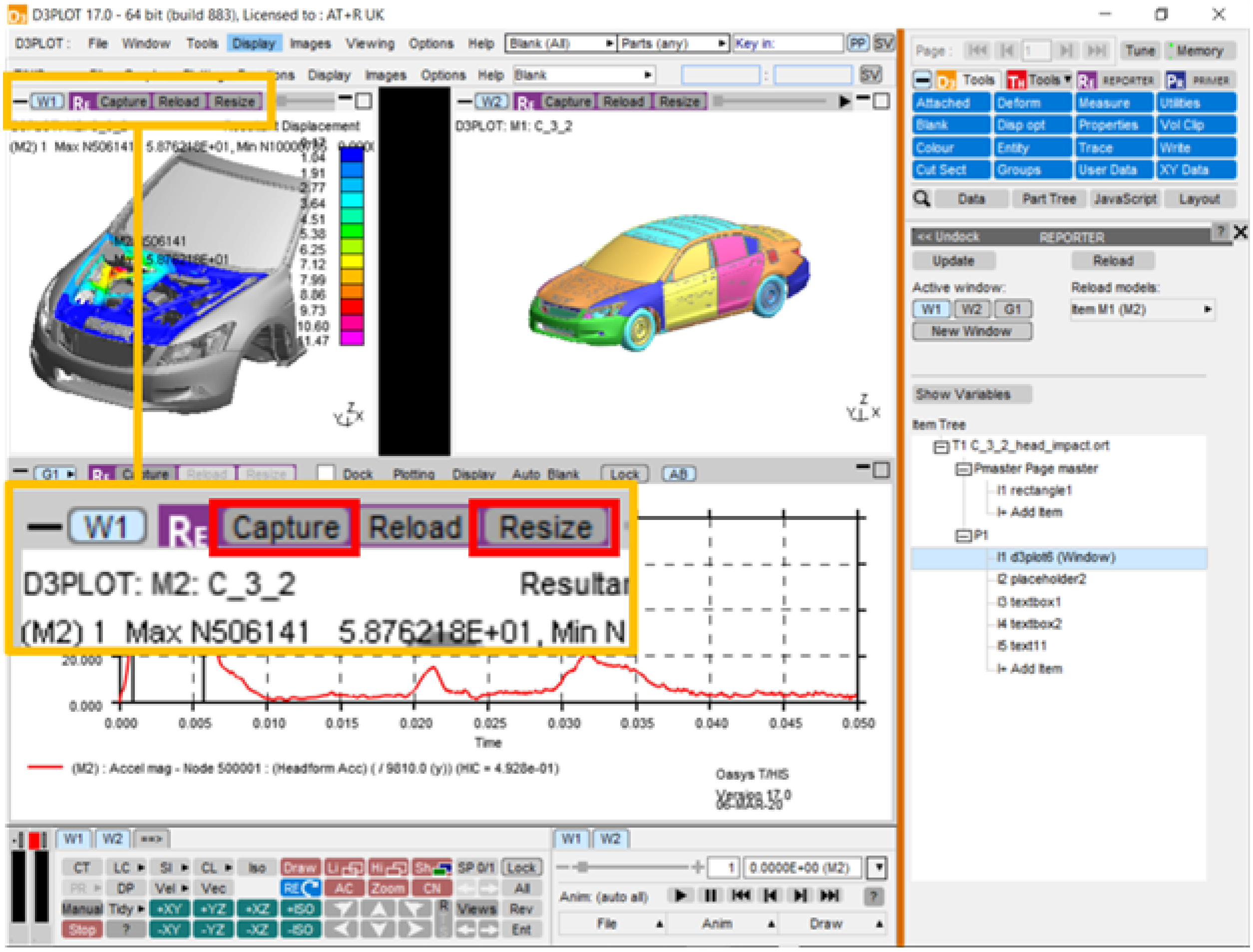
12.3. Capture

Capture

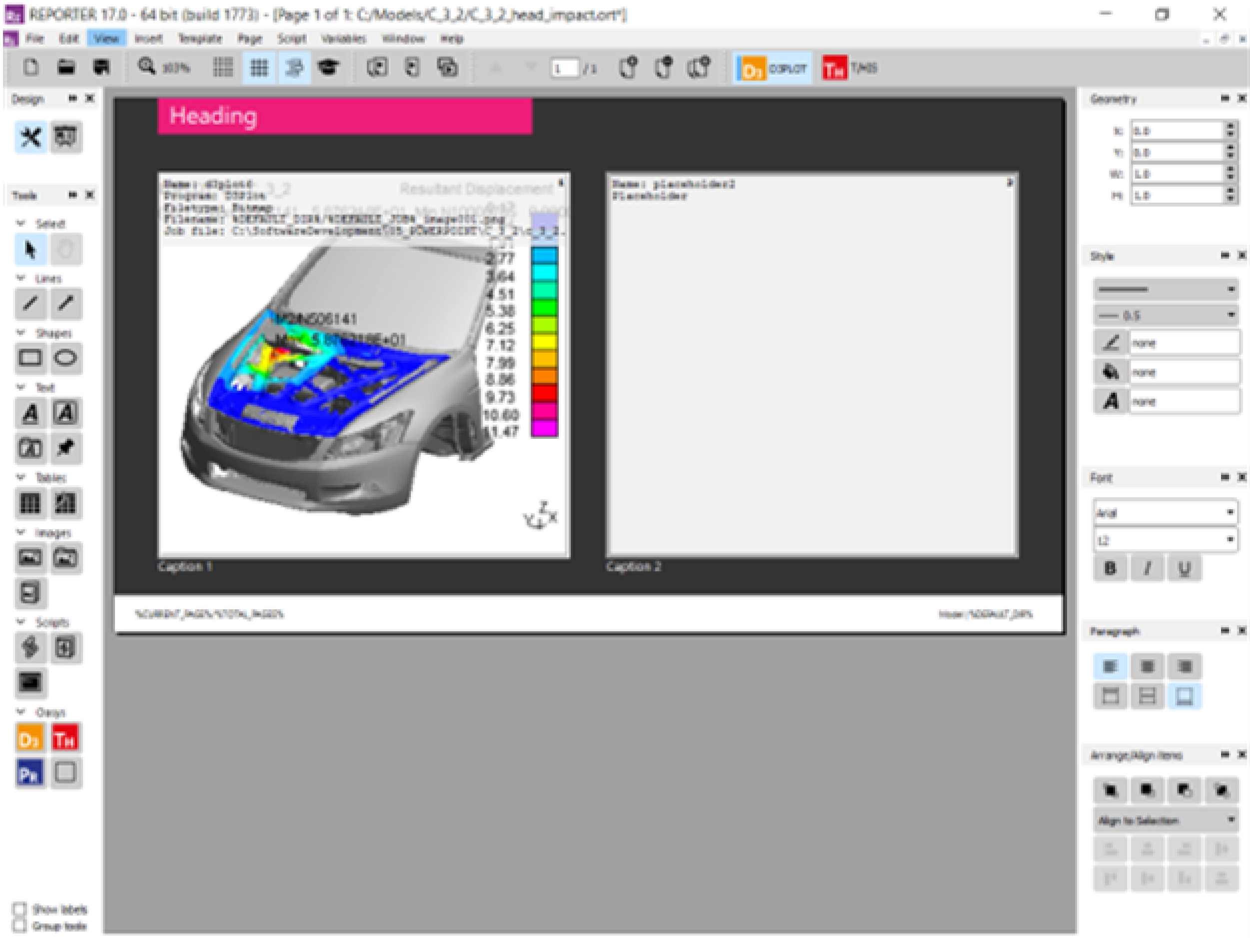
Windows and graphs can be captured into REPORTER, saving an image together with additional information to allow the capture to be reloaded later. For D3PLOT windows, this is a properties and settings file. For T/HIS graphs, this is a FAST-TCF script. Graphs captured in the D3PLOT->T/HIS link are treated exactly the same as graphs in T/HIS, so the resulting items will be identical. Variables containing useful values related to the models or curves in the captured window can be added to the item before capturing (see Variables).

Note that in the Oasys Suite 17.0 method, only single windows and graphs can be captured. The intention being that the windows and graphs are easily captured individually and laid out in REPORTER with greater flexibility.

In order to capture a window, first select the target item in REPORTER, either selecting it directly in REPORTER or using the item tree. You can capture into a new item by selecting 'I+ Add Item' in the item tree. Once the item is selected, the 'Resize' button on the top bar of the window can be used to resize the window to match whatever image size is specified on the selected REPORTER item, such as 'Fit object box'. Finally, either press 'Capture' on the top bar of the target window or select the window in the 'Active window' list in the REPORTER panel and press 'Capture' at the top of the panel.



This will send the information to REPORTER and the image will appear on the item.

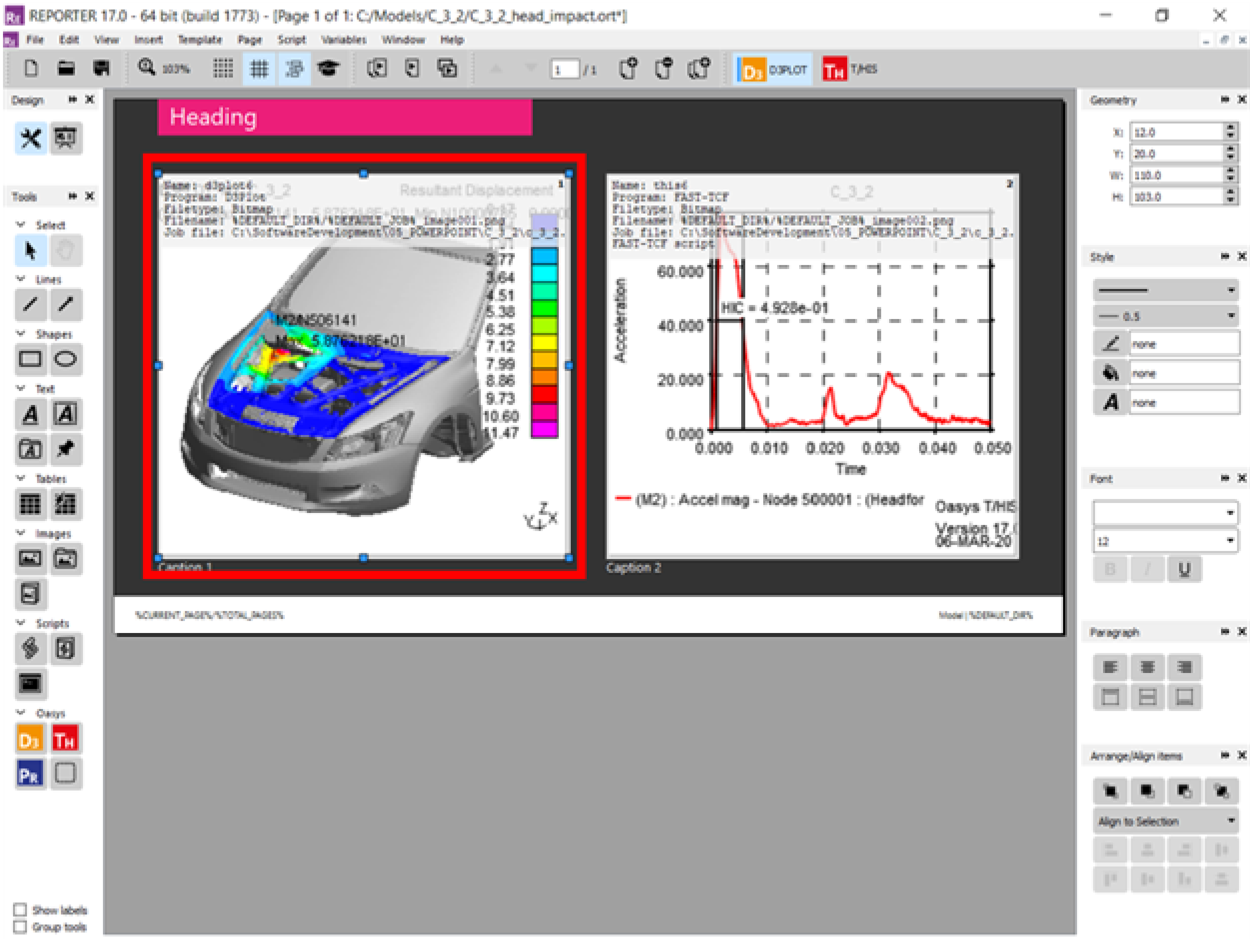


12.4. Reload

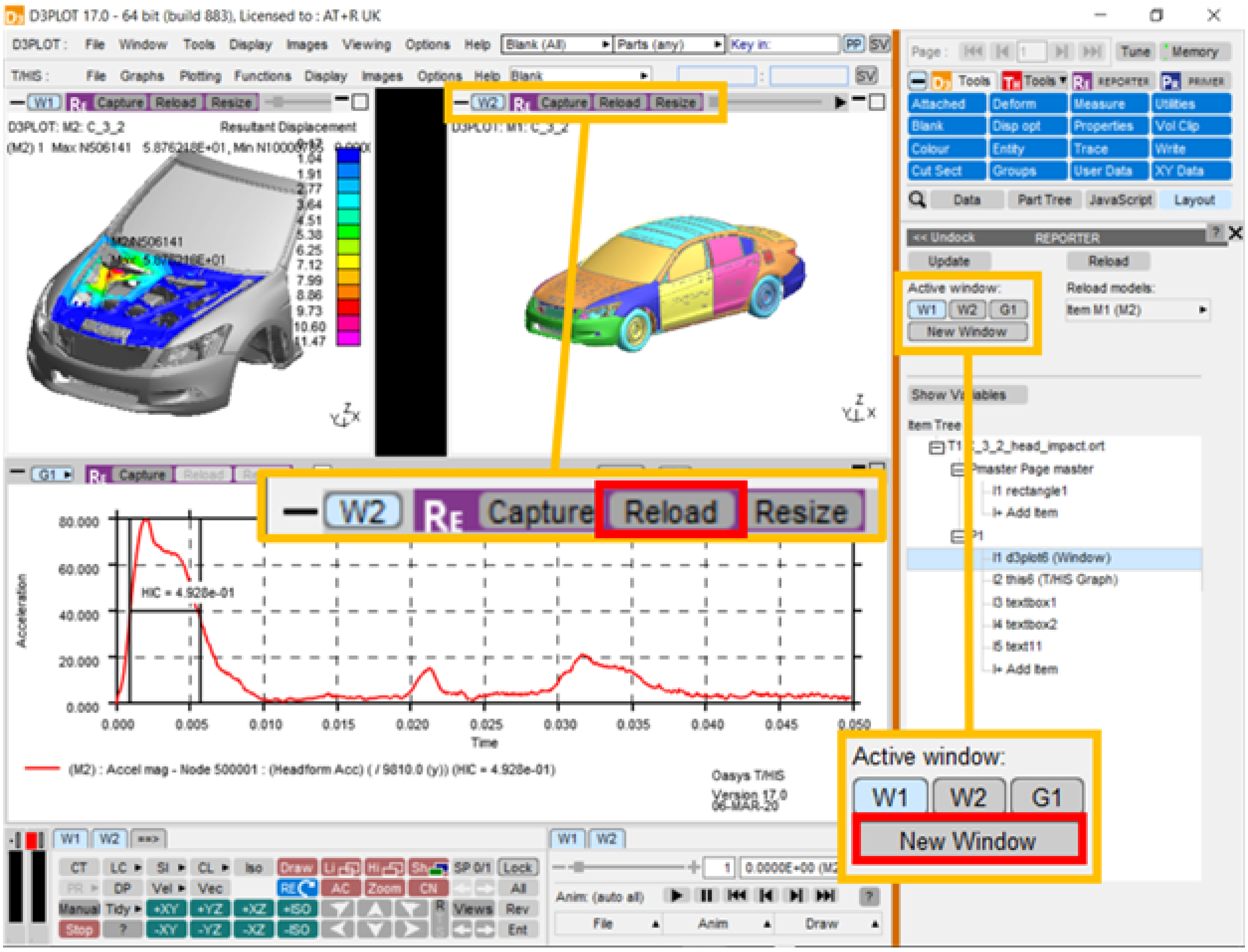
Reload

Existing REPORTER items can be reloaded back into D3PLOT or T/HIS. Items captured from graphs in the D3PLOT->T/HIS link are treated the same as items captured from standalone T/HIS. As such, they can each be reloaded either into D3PLOT or T/HIS.

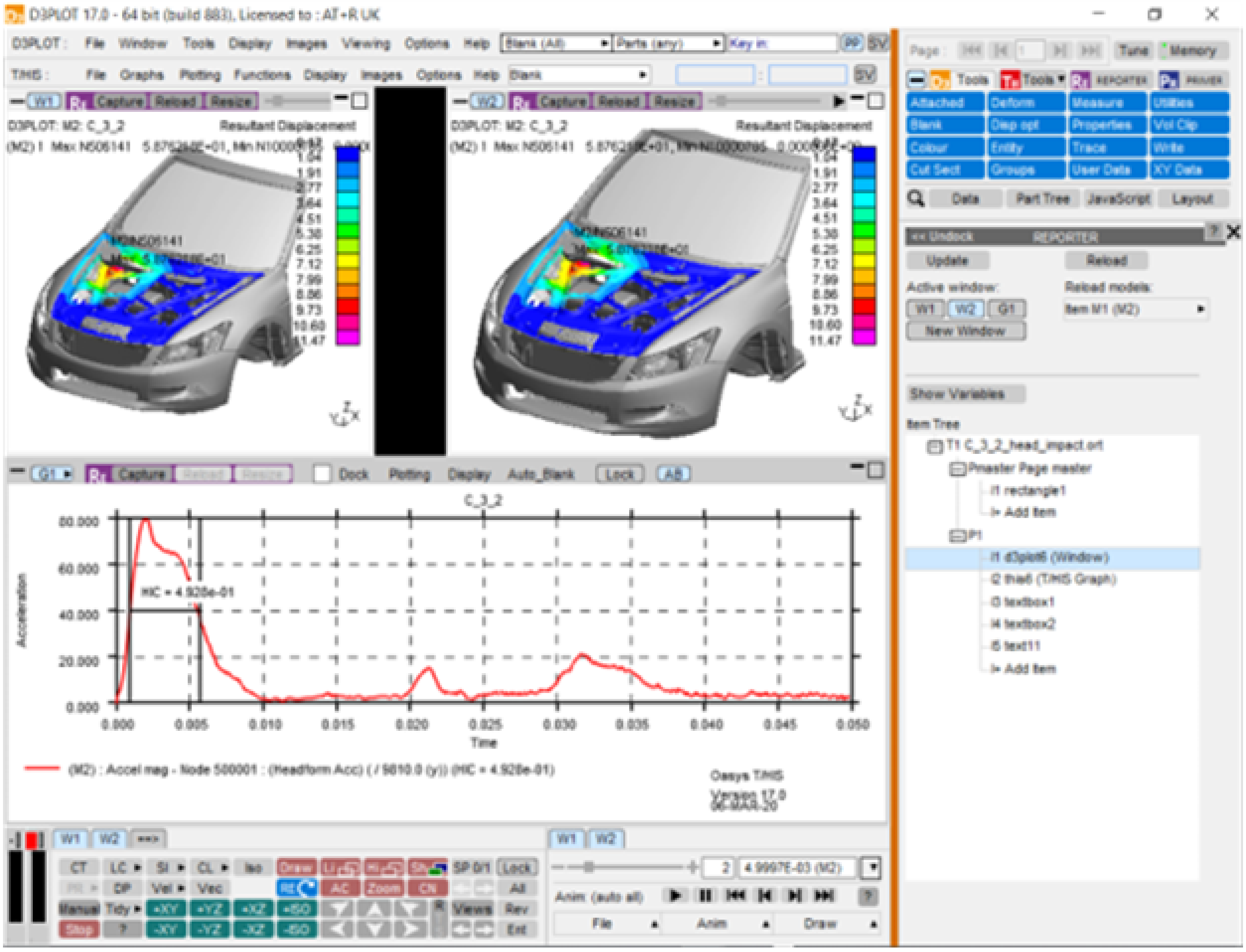
First select the item in REPORTER that you want to reload.



Then either press reload at the top of the target window, or select 'New Window' in the Active window list.



This will clear the target window, open the relevant models, not opening them again if they are already open in the session, then load the stored item information, reproducing the capture.

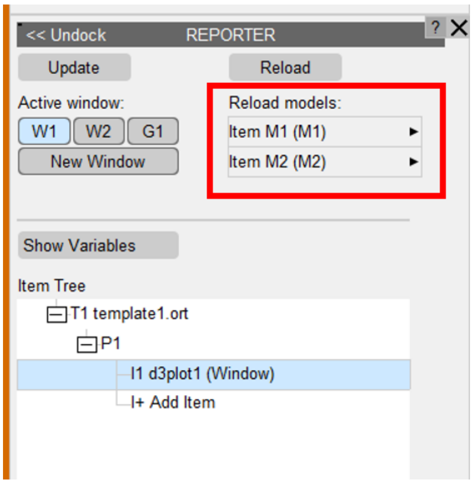

D3PLOT 17.0 - 64 bit (build 883), Licensed to : AT+R UK
D3PLOT: M2: C_3_2
Resultant Displacement
C_3_2
HIC = 4.928e-01
Acceleration
Time
(M2) : Accel mag - Node 500001 : (Headform Acc) (/ 9810.0 (y)) (HIC = 4.928e-01)
Oasys T/HIS
Version 17.0
REPORTER
Active window:
Reload models:
Show Variables
Item Tree
T1 C_3_2_head_impact.ort
Pmaster Page master
I1 rectangle1
P1
I1 d3plot6 (Window)
I2 this6 (T/HIS Graph)
I3 textbox1
I4 textbox2
I5 text11

12.4.1. Reload Models

Reload Models

The models used in an existing item are listed in the Reload models list. The models will be listed as Item Mn, where n is the index of the model in the item, not of the model in the session. If the model is also open in the current session, then the model ID in the current session will be displayed in brackets.

Each entry in the list has a popup attached, allowing the model to replaced either by a model in the current session or by browsing for a model. This will not change the models stored in the item, but instead when the item is reloaded into the current session the replacement models will be used. The resulting window will then need to be captured, either into a new item or to overwrite the original.



12.5. Variables

Variables

Variables can be added to both D3PLOT and T/HIS items, allowing data related to the capture to be made available in REPORTER. The REPORTER panel can be undocked and expanded to display the variables list by selecting Show Variables.

For T/HIS items, variables can be added containing properties of any of the curves in the selected graph or all the curves combined using the All Curves option. By default, T/HIS items will have variables for the MAX and MIN values taken over all curves in the selected graph. When selecting the curve for a newly created variable using the curve popup, curves are referred to as ICn, meaning Item Curve n, where n is the index of the curve in the selected graph. The curve label and number in the current session are also displayed in the popup.

For D3PLOT items, variables can be added for the MAX and MIN values of any of the plotted data components on any of the models. By default, D3PLOT items will have variables for the MAX and MIN values of all plotted data components for each model in the selected window.

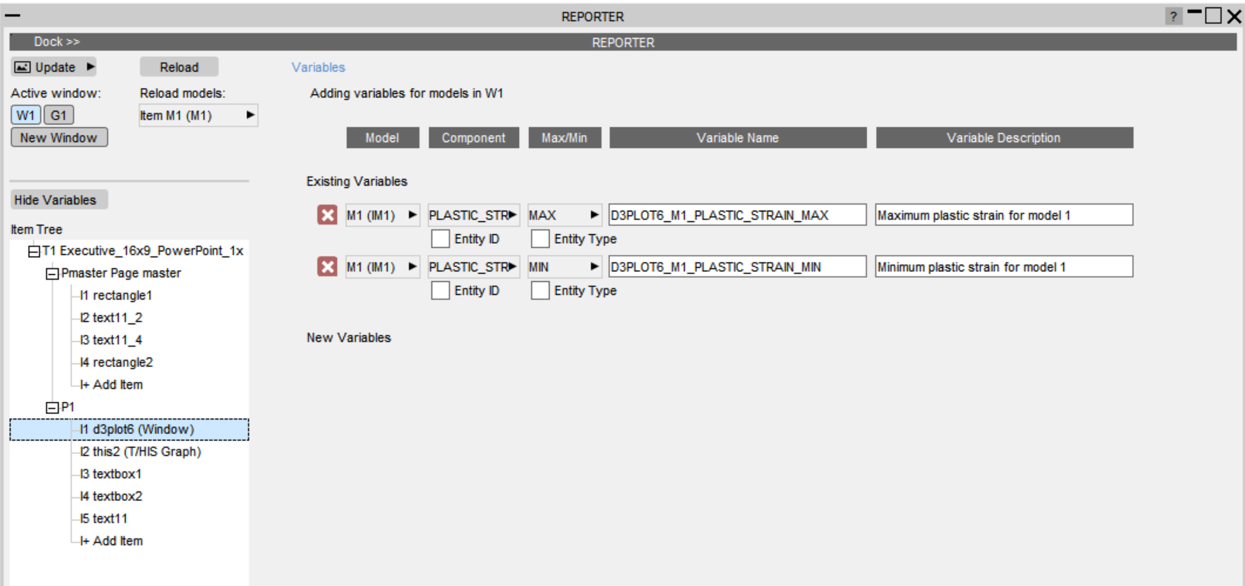
Variables can be added using the + button and deleted using the X button next to the row.

Initially, variables will appear under New Variables until the item is captured, when they will move to Existing Variables. Variables will be given default names based on their item name in REPORTER (e.g. d3plot6_1, this4), variable type and model/curve that they relate to. However, these names and descriptions can be manually edited.

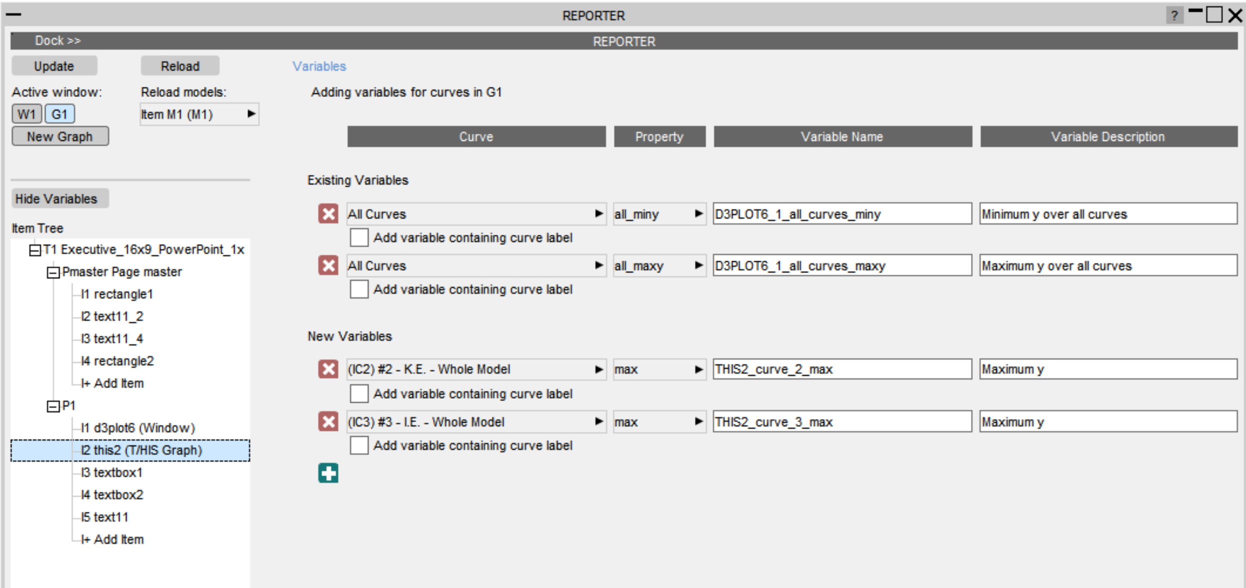
For D3PLOT items, the Entity ID and Entity Type tickboxes can be used to create additional variables to contain this information. These will have the same name as the original variable with either _ENT_ID or _ENT_TYPE appended.

For T/HIS items, the Add variables containing curve label tickbox will create an additional variable containing the curve label of the relevant curve, with _LABEL appended to the name.

Example of a D3PLOT item with two existing variables, referring to models in Window 1.



Example of a T/HIS item with two new variables and two existing variables, referring to curves in Graph 1.



12.6. Generate

Generate

Once a complete template has been created, it can be generated using File >> Generate in REPORTER. This will generate in an existing session if there is one, otherwise a new session will be started. T/HIS items will be generated in standalone T/HIS, unless the T/HIS link is already open in D3PLOT, in which case they will generate in the link. It is faster to generate in standalone T/HIS.

12.7. Exceptions to the Oasys Suite 17.0 Method and Existing Templates from Oasys Suite 16.0 and Earlier

Exceptions to the Oasys Suite 17.0 Method and Existing Templates from Oasys Suite 16.0 and Earlier

There are some item types that are not supported in the new Oasys Suite 17.0 method. In this case, the Oasys Suite 16.0 method will be used and nothing will have changed. These are:

- T/HIS JavaScript items
- Items containing multiple graphs/windows

Any item can be captured and generated using the Oasys Suite 16.0 method by selecting the Capture and generate this item using the old method option in the object information in REPORTER.

Existing Oasys Suite 16.0 and earlier templates should work exactly as they used to. All items will use the Oasys Suite 17.0 method unless they meet one of the specified exceptions above. This gives some additional benefits:

- When generating the report, all supported items will be generated in the same session, without opening the same models multiple times. This will make the process faster.
- The report can be edited interactively using all the perks of the Oasys Suite 17.0 method.

13. Appendices

13.1. APPENDIX A - LS-DYNA Data Components

13.1.1. Model Data Components

Model Data Components

The following global data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
DT	Time Step	yes		yes	yes
KE	Kinetic energy	yes		yes	yes
IE	Internal energy	yes		yes	yes
SWE	Stonewall energy			yes	yes
SPE	Spring and damper energy			yes	yes
HG	Hourglass energy			yes	yes
SDE	System damping energy			yes	yes
JE	Joint internal energy			yes	yes
SIE	Sliding interface energy			yes	yes
EW	External work		yes	yes	yes
RBE	Rigid Body stopper energy			yes	
TE	Total energy	yes		yes	yes
TER	Total/initial energy ratio			yes	yes
VX	Average X velocity	yes		yes	yes
VY	Average Y velocity	yes		yes	yes
VZ	Average Z velocity	yes		yes	yes
TZC	Time per zone cycle			yes	yes
AM	Added mass			yes	yes
PM	%age Mass increase			yes	yes
EKE	Eroded Kinetic energy			yes	yes
EIE	Eroded Internal energy			yes	yes
EHG	Eroded Hourglass energy			yes	yes

ER	Energy Ratio w/o Eroded			yes	yes
DRCE	Current Distortional Kinetic Energy				yes
DRMX	Maximum Distortional Kinetic Energy				yes
DRCO	Convergence Factor				yes
DRKE	Total Kinetic Energy				yes
MPE	Mat Plastic Energy			yes	yes
MEE	Mat Elastic Energy			yes	yes
MDE	Mat Damage Energy			yes	yes
DIE	Dissipated Internal Energy			yes	yes
DKE	Disssipated Kinetic Energy			yes	yes
DE	Drilling Energy			yes	yes

13.1.2. Part Data Components

Part Data Components

For Parts the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
KE	Kinetic energy	yes		yes	yes
IE	Internal energy	yes		yes	yes
HG	Hourglass energy			yes	yes
TE	Total energy	yes		yes	yes
XM	X momentum			yes	yes
YM	Y momentum			yes	yes
ZM	Z momentum			yes	yes
VX	Average X velocity	yes		yes	yes
VY	Average Y velocity	yes		yes	yes
VZ	Average Z velocity	yes		yes	yes
MA	Mass	yes		yes	yes
EIE	Eroded Internal energy			yes	yes
ER	Energy Ratio w/o Eroded			yes	yes
MPE	Mat Plastic Energy			yes	
MEE	Mat Elastic Energy			yes	
MDE	Mat Damage Energy			yes	

13.1.3. Part Group Data Components

Part Group Data Components

For Part Groups the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
KE	Kinetic energy	yes		yes	yes
IE	Internal energy	yes		yes	yes
HG	Hourglass energy	yes		yes	yes
TE	Total energy	yes		yes	yes
MA	Mass	yes		yes	yes

13.1.4. Nodal Data Components

Nodal Data Components

For nodes the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
TE	Temperature	yes		yes	yes
DX	X Displacement	yes		yes	yes
DY	Y Displacement	yes		yes	yes
DZ	Z Displacement	yes		yes	yes
DM	Displacement Magnitude	yes		yes	yes
VX	X Velocity	yes		yes	yes
VY	Y Velocity	yes		yes	yes
VZ	Z Velocity	yes		yes	yes
VM	Velocity Magnitude	yes		yes	yes
AX	X Acceleration	yes		yes	yes
AY	Y Acceleration	yes		yes	yes
AZ	Z Acceleration	yes		yes	yes
AM	Acceleration Magnitude	yes		yes	yes
CX	X Co-ordinate			yes	yes
CY	Y Co-ordinate			yes	yes
CZ	Z Co-ordinate			yes	yes
RX	X Rotation			yes	yes
RY	Y Rotation			yes	yes
RZ	Z Rotation			yes	yes
RM	Rotation Magnitude			yes	yes
RVX	X Rotational Velocity			yes	yes
RVY	Y Rotational Velocity			yes	yes
RVZ	Z Rotational Velocity			yes	yes
RVM	Rotational Velocity Magnitude			yes	yes

RAX	X Rotational Acceleration			yes	yes
RAY	Y Rotational Acceleration			yes	yes
RAZ	Z Rotational Acceleration			yes	yes
RAM	Rotational Acceleration Magnitude			yes	yes
FLX	X Thermal Flux			yes	yes
FLY	Y Thermal Flux			yes	yes
FLZ	Z Thermal Flux			yes	yes
FLM	Thermal Flux Magnitude			yes	yes

Frequency Domain Analysis

For a steady state dynamic analysis (SSD) the following nodal data components are available. For each data component both amplitude and phase angle data components are available.

	Component	**THF (d3thdt)**	**XTF (xtfile)**	**LSDA (binout)**	**ASCII**
DX	X Displacement			yes	
DY	Y Displacement			yes	
DZ	Z Displacement			yes	
VX	X Velocity			yes	
VY	Y Velocity			yes	
VZ	Z Velocity			yes	
AX	X Acceleration			yes	
AY	Y Acceleration			yes	
AZ	Z Acceleration			yes	

For a random vibration analysis (PSD) the following nodal data components are available.

	Component	**THF (d3thdt)**	**XTF (xtfile)**	**LSDA (binout)**	**ASCII**
DX	X Displacement	yes		yes	yes
DY	Y Displacement	yes		yes	yes
DZ	Z Displacement	yes		yes	yes
DM	Displacement Magnitude	yes		yes	yes

VX	X Velocity	yes		yes	yes
VY	Y Velocity	yes		yes	yes
VZ	Z Velocity	yes		yes	yes
VM	Velocity Magnitude	yes		yes	yes
AX	X Acceleration	yes		yes	yes
AY	Y Acceleration	yes		yes	yes
AZ	Z Acceleration	yes		yes	yes
AM	Acceleration Magnitude	yes		yes	yes

Only nodes that have been declared in "nodal time-history blocks" will be available for processing. To get a list of available node numbers in command line mode use the `M` (enu) command.

Coordinate system of results

All nodal results are in the global cartesian coordinate system **except** at nodes which have been defined as accelerometers: these report accelerations in the local coordinate system of the accelerometer subject to any rotations its "parent" rigid body has undergone.

13.1.5. Solid Data Components

Solid Data Components

For solids the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Stress components					
SXX	Stress in XX	yes		yes	
SYY	Stress in YY	yes		yes	
SZZ	Stress in ZZ	yes		yes	
SXY	Stress in XY	yes		yes	
SYZ	Stress in YZ	yes		yes	
SZX	Stress in ZX	yes		yes	
SMX	Maximum Principal Stress	yes		yes	
SMN	Minimum Principal Stress	yes		yes	
SMS	Maximum Shear Stress	yes		yes	
SVM	Von Mises Stress	yes		yes	
SAV	Average Stress (Pressure)	yes		yes	
STR	Stress Triaxiality Factor	yes		yes	
Strain components					
EFF	Effective Plastic Strain	yes		yes	
EXX	Strain in XX	yes		yes	
EYY	Strain in YY	yes		yes	
EZZ	Strain in ZZ	yes		yes	
EXY	Strain in XY	yes		yes	
EYZ	Strain in YZ	yes		yes	
EZX	Strain in ZX	yes		yes	
EMX	Maximum Principal Strain	yes		yes	
EMN	Minimum Principal Strain	yes		yes	
EMS	Maximum Shear Strain	yes		yes	
EVM	Von Mises Strain	yes		yes	

EAV	Average Strain	yes		yes	
"Extra" components					
SOEn	Extra Data Component	yes		yes	

Frequency Domain Analysis

For a steady state dynamic analysis (SSD) the following nodal data components are available. For each data component both amplitude and phase angle data components are available.

	Component	**THF (d3thdt)**	**XTF (xtfile)**	**LSDA (binout)**	**ASCII**
Stress components					
SXX	Stress in XX			yes	
SYY	Stress in YY			yes	
SZZ	Stress in ZZ			yes	
SXY	Stress in XY			yes	
SYZ	Stress in YZ			yes	
SZX	Stress in ZX			yes	
Strain components					
EXX	Strain in XX			yes	
EYY	Strain in YY			yes	
EZZ	Strain in ZZ			yes	
EXY	Strain in XY			yes	
EYZ	Strain in YZ			yes	
EZX	Strain in ZX			yes	

For a random vibration analysis (PSD) the following nodal data components are available.

	Component	**THF (d3thdt)**	**XTF (xtfile)**	**LSDA (binout)**	**ASCII**
Stress components					
SXX	Stress in XX			yes	
SYY	Stress in YY			yes	
SZZ	Stress in ZZ			yes	

SXY	Stress in XY			yes	
SYZ	Stress in YZ			yes	
SZX	Stress in ZX			yes	
SVM	Von Mises Stress			yes	
Strain components					
EXX	Strain in XX			yes	
EYY	Strain in YY			yes	
EZZ	Strain in ZZ			yes	
EXY	Strain in XY			yes	
EYZ	Strain in YZ			yes	
EZX	Strain in ZX			yes	

Coordinate systems of results

The stress and strain tensors are reported in the global cartesian system unless the option to output results in the part coordinate system has been used. Writing the directional strain tensor is optional in LS-DYNA: it will only appear in the menu if it is present.

"Extra" data components

The "extra" data components (`SOEn`) are also optional and only appear if present in the database. They are material dependent results, and are treated as scalar data of unknown type by T/HIS.

13.1.6. Beam Data Components

Beam Data Components

For beams the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Basic data components					
NX	Axial force	yes		yes	
NY	Shear force in Y	yes		yes	
NZ	Shear force in Z	yes		yes	
MY	Moment in Y	yes		yes	
MZ	Moment in Z	yes		yes	
MX	Torsional moment	yes		yes	
"Plastic" data components					
EAX	Axial strain	yes			
PE1	Plastic bending energy : end 1	yes			
PE2	Plastic bending energy : end 2	yes			
RY1	Y rotation : end 1	yes			
RY2	Y rotation : end 2	yes			
RZ1	Z rotation : end 1	yes			
RZ2	Z rotation : end 2	yes			
RX	Torsional rotation	yes			
MY1	Y bending moment : end 1	yes			
MY2	Y bending moment : end 2	yes			
MZ1	Z bending moment : end 1	yes			
MZ2	Z bending moment : end 2	yes			
ACE	Axial collapse energy	yes			
IE	Internal energy	yes			
Integration Point Data					

SXX	Axial stress	yes		yes	
SXY	XY shear stress	yes		yes	
SZX	ZX shear stress	yes		yes	
EFF	Effective plastic strain	yes			
EXX	Axial strain	yes		yes	
Discrete Beams - Only available if DISBOUT ASCII file has been written to LSDA (binout) file.					
AXD	Relative Axial displacment			yes	
SD	Relative S- Displacement			yes	
TD	Relative T- Displacement			yes	
AXR	Axial rotation			yes	
SR	Rotation in S			yes	
TR	Rotation in T			yes	
RNAX	Relative Axial force			yes	
RNS	Resultant S - Force			yes	
RNT	Resultant T - Force			yes	
MAX	Axial moment			yes	
MS	Moment in S			yes	
MT	Moment in T			yes	
AXX	Axial Direction X			yes	
AXY	Axial Direction Y			yes	
AXZ	Axial Direction Z			yes	
SX	S - Direction X			yes	
SY	S - Direction Y			yes	
SZ	S - Direction Z			yes	
TX	T - Direction X			yes	
TY	T - Direction Y			yes	
TZ	T - Direction Z			yes	

Frequency Domain Analysis

For a steady state dynamic analysis (SSD) the following nodal data components are available. For each data component both amplitude and phase angle data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Basic data components					
NX	Axial force			yes	
NY	Shear force in Y			yes	
NZ	Shear force in Z			yes	
MY	Moment in Y			yes	
MZ	Moment in Z			yes	
MX	Torsional moment			yes	
Integration point data					
SXX	Axial stress			yes	
SXY	XY shear stress			yes	
SZX	ZX shear stress			yes	
EFF	Effective plastic strain			yes	
EXX	Axial strain			yes	

For a random vibration analysis (PSD) the following nodal data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Basic data components					
NX	Axial force			yes	
NY	Shear force in Y			yes	
NZ	Shear force in Z			yes	
MY	Moment in Y			yes	
MZ	Moment in Z			yes	
MX	Torsional moment			yes	
Integration point data					
SXX	Axial stress			yes	
SXY	XY shear stress			yes	
SZX	ZX shear stress			yes	
EFF	Effective plastic strain			yes	

EXX	Axial strain			yes	

Additional Beam Results: written if requested from LS-DYNA

In addition to the basic data components additional beam results may be output to the **.THF** file for both Belytschko-Schwer and Hughes-Lui beam elements. As no indication of the element type is written to the **.THF** file it is impossible for T/HIS to work out whether a specific element is a Belytschko-Schwer or a Hughes-Liu beam. As the element type is unknown the user must know which element type a beam is in order to extract the correct results.

Belytschko-Schwer Beams

If you have used Belytschko-Schwer beams with a resultant plastic material model the following "plastic" results will also be written out to **.THF** file: (Note that these data are written even if the ***DATABASE_EXTENT_BINARY** card field < **beamip** > is not set - the presence of a resultant beam material triggers their output automatically. This is not the case for Hughes-Liu data components, for which output must be requested explicitly, see below.)

Coordinate systems of results

Beam results are always output in the element local coordinate system. Only beams declared in "beam element time-history blocks" will be available.

"Extra" data components

Where "extra" results are written, and T/HIS cannot resolve unambiguously whether they are Belytschko-Schwer plastic data, or Hughes-Liu stress/strain data, **it is your responsibility to interpret the results correctly** .

Notes on beam data

1. Hughes-Liu (integrated) beams locate their integration point(s) at mid-span, and have a constant shear force and moment along their length.

   The location and number of integration points through the thickness at mid span depends on the beam shape. See the *SECTION_BEAM keyword in the LS-DYNA manual for more information.

2. Belytschko-Schwer (resultant) beams calculate the moment variation along the beam, so may have different Myy and Mzz terms at ends one and two. This presents a problem when only the basic force and moment vector is written since only one Myy and one Mzz term are output. **These are in fact the values at end 1.** So if you have a cantilever fixed at end 2, with a point load at end 1,

you will not see any moment in it if you only plot the basic Myy or Mzz data components (although the moment will be there and it will behave correctly). Furthermore, the sign of the end 1 Myy and Mzz moments written to the extra data slots is opposite to the sign of the basic moment vector Myy and Mzz moments so care must be taken in interpreting the direction of moments when switching between basic and extra data component moments.

13.1.7. Shell Data Components

Shell Data Components

For shells the following data components are available. These combine with directions for the data component, and in some cases a location through the shell thickness.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Stress components					
SXX	Stress in XX	yes		yes	
SYY	Stress in YY	yes		yes	
SZZ	Stress in ZZ	yes		yes	
SXY	Stress in XY	yes		yes	
SYZ	Stress in YZ	yes		yes	
SZX	Stress in ZX	yes		yes	
SMX	Maximum Principal Stress	yes		yes	
SMN	Minimum Principal Stress	yes		yes	
SMS	Maximum Shear Stress	yes		yes	
SVM	Von Mises Stress	yes		yes	
SAV	Average Stress (Pressure)	yes		yes	
STR	Stress Triaxiality Factor	yes		yes	
Strain components					
EFF	Effective Plastic Strain	yes		yes	
EXX	Strain in XX	yes		yes	
EYY	Strain in YY	yes		yes	
EZZ	Strain in ZZ	yes		yes	
EXY	Strain in XY	yes		yes	
EYZ	Strain in YZ	yes		yes	
EZX	Strain in ZX	yes		yes	
EMX	Maximum Principal Strain	yes		yes	
EMN	Minimum Principal Strain	yes		yes	
EMS	Maximum Shear Strain	yes		yes	

EVM	Von Mises Strain	yes		yes	
EAV	Average Strain	yes		yes	
Force / Moment components					
MX	Moment in X	yes			
MY	Moment in Y	yes			
MXY	Moment in XY	yes			
QX	Shear force in X	yes			
QY	Shear force in Y	yes			
NX	Normal force in X	yes			
NY	Normal force in Y	yes			
NXY	Normal force in XY	yes			
Miscellaneous components					
T	Thickness	yes			
I	Internal energy density	yes			
"Extra" components					
An	Extra Data Component	yes		yes	

Frequency Domain Analysis

For a steady state dynamic analysis (SSD) the following nodal data components are available. For each data component both amplitude and phase angle data components are available.

	Component	**THF (d3thdt)**	**XTF (xtfile)**	**LSDA (binout)**	**ASCII**
Stress components					
SXX	Stress in XX			yes	
SYY	Stress in YY			yes	
SZZ	Stress in ZZ			yes	
SXY	Stress in XY			yes	
SYZ	Stress in YZ			yes	
SZX	Stress in ZX			yes	
Strain components					
EXX	Strain in XX			yes	
EYY	Strain in YY			yes	

EZZ	Strain in ZZ			yes	
EXY	Strain in XY			yes	
EYZ	Strain in YZ			yes	
EZX	Strain in ZX			yes	

For a random vibration analysis (PSD) the following nodal data components are available.

	Component	**THF (d3thdt)**	**XTF (xtfile)**	**LSDA (binout)**	**ASCII**
Stress components					
SXX	Stress in XX			yes	
SYY	Stress in YY			yes	
SZZ	Stress in ZZ			yes	
SXY	Stress in XY			yes	
SYZ	Stress in YZ			yes	
SZX	Stress in ZX			yes	
SVM	Von Mises Stress			yes	
Strain components					
EXX	Strain in XX			yes	
EYY	Strain in YY			yes	
EZZ	Strain in ZZ			yes	
EXY	Strain in XY			yes	
EYZ	Strain in YZ			yes	
EZX	Strain in ZX			yes	

THF (d3thdt) File

Stress	Stress tensors are in the global cartesian system unless the option to use material axes has been invoked for orthotropic materials (CMPFLG on *DATABASE_EXTENT_BINARY). By default results are available at top and bottom integration points and mid-surface but values can be output for all through thickness integration points by using MAXINT on *DATABASE_EXTENT_BINARY
Strain	The Strain tensors output is optional. Values are in the global cartesian system unless the option to use material axes has been invoked for orthotropic materials (CMPFLG on *DATABASE_EXTENT_BINARY). Only values at the top and bottom integration points are output. T/HIS will average these values for the mid surface.
Forces & Moments	Force and moment resultants are <data> per unit width, and are written in the element local axis system. Refer to "Theory of Plates and Shells", Timoshenko, for a precise definition of these values.
Extra	The "Extra History" data components will only appear in the menu if they have been selected for output (NEIPS on *DATABASE_EXTENT_BINARY). These are output for the same surfaces / integration points as the stress tensor values.

Through Thickness Integration Points

NOTE: The top and bottom "surfaces" are **not** the outer fibres if the default Gaussian integration rules are used, but rather the outer and inner integration points. The relationship between integration point location and shell thickness depends on the number of integration points used.

The following diagram shows locations of integration points with respect to shell half-thickness `(t/2)` assuming the default Gaussian integration rules have been used:

No of Points	Distance of outer fibres from neutral axis as a proportion of t/2
1	0.0 (membrane)
2	0.577 t/2
3	0.775
4	0.861 t/2
5	0.906

The "top" (or outer) point is on the positive local Z side of the element neutral axis. The output of shell data from LS-DYNA will fall into one of two categories, and the "surface" options available in T/HIS depend on this.

NOTE: It is possible to use non-default integration schemes in LS-DYNA which may locate the integration points at different places. This is an advanced topic: contact Oasys Ltd for advice.

Default output case: 3 "surfaces"

In this case, regardless of how many integration points the shell elements may actually have through their thickness, LS-DYNA writes out:

Top surface :	Top integration point
Centre surface :	Computed neutral axis value
Bottom surface :	Bottom integration point

Note that the "centre" surface here is the neutral axis value. For membrane elements all three sets of values will be the same.

Optional output case: user-defined number of integration points

The number of through thickness integration points written to the THF file can be modified using the value of MAXINT on the *DATABASE_EXTENT_BINARY card. If this parameter is changed then all thin and thick shell output written to the THF file will have MAXINT data slots for integration points in the file, regardless of how many integration points a given element may have through its thickness.

If MAXINT is not 3 then the order in which data is written to the THF file is controlled by the actual number of integration points of integration points in a shells formulation. The following table illustrates output for the case of MAXINT not equal to 3

<table>
<tr><th>Data slot in file</th><th>Shell with 3 Integration points</th><th>Shell with 5 Integration points</th><th>Shell with any other nunber of integration points</th></tr>
<tr><td>#1</td><td>Middle</td><td>Middle</td><td rowspan="3">Bottom
|</td></tr>
<tr><td>#2</td><td>Bottom</td><td>Bottom</td></tr>
<tr><td>#3</td><td>Top</td><td>Bottom + 1</td></tr>
</table>

<table>
<tr><td>#4</td><td>zero</td><td>Top - 1</td><td rowspan="3">|
Top</td></tr>
<tr><td>#5</td><td>zero</td><td>Top</td></tr>
<tr><td>#6</td><td>zero</td><td>zero</td></tr>
</table>

NOTE: The THF file does NOT contain any information on the number of integration points each shell was defined with.

No explicit neutral axis value is calculated or output.

The outcome of writing more integration points than have been used in a shell formulation is undefined.

There is no guarantee that the "centre" surface in this context is the neutral axis value: this will depend upon the element integration scheme. In addition where the "centre" value has been averaged from a pair of points, when the number of layers is an even number, it will definitely not be the neutral axis value: consider plastic strain in a section in pure bending!

The ZTF file generated by PRIMER can help to resolve some of these problems.

THF File + ZTF File

If a ZTF file had been generated using PRIMER then T/HIS can use additional information from the ZTF to correctly work out the number of integration points each shell element was defined with. If an attempt is made to output data for a surface that does not exist in the THF file then T/HIS will generate a warning message and a NULL curve will be generated.

In addition to working out the correct number of through thickness integration points for each element T/HIS can also use the information in the ZTF to identify models where MAXINT has been set to a -ve number in order to generate data for multiple in-plane integration points.

Effect of plotting "Top" surface on models with MAXINT = 6 and MAXINT = 9 whit and without a ZTF file.

	MAXINT = 6, no ZTF file	MAXINT = 6, ZTF file present	MAXINT = 9, no ZTF file	MAXINT = 9, ZTF file present
Shell 1 has 4 integration points	Undefined (#int points < 6)	Correct (int point #4)	Undefined (#int points < 9)	Correct (int point #4)
Shell 2 has 6 integration points	Correct (int point #6)	Correct (int point #6)	Undefined (#int points < 9)	Correct (int point #6)

Shell 3 has 9 integration points	Incorrect (6th integration point)	Warning message as #int points < 6	Correct (int point #9)	Correct (int point #9)

In-plane Integration Points

In some versions of LS-DYNA it is now possible to write out data for all 4 in-plane integration points for fully integrated shells by setting MAXINT on the *DATABASE_EXTENT_BINARY card to a -ve number. For example specifying a value of -8 will generate data for 8 layers each with 4 in-plane integration points. If this option is used then all the elements will be written out using this option regardless of whether they are fully integrated or not.

As there is no information in the THF to indicate that data for 4 in-plane integration points has been written to the file then the file format will be exactly the same as for an analysis with a +ve value of MAXINT 4 times larger. For example MAXINT = -8 and MAXINT = 32 will both produce THF files with 32 integration points worth of data and there is no way for T/HIS to know which value of MAXINT was used to generate the data. The ZTF file generated by PRIMER can help to resolve this problem.

If multiple in-plane integration points are written to the THF file then they are written in the following order.

Layer 1 - in-plane int point #1
Layer 2 - in-plane int point #1
....
Layer n - in-plane int point #1
Layer 1 - in-plane int point #2
Layer 2 - in-plane int point #2
....
Layer n - in-plane int point #2
Layer 1 - in-plane int point #3
....

NOTE: If non fully integrated shells are included in the list of elements written to the THF file then in some versions of LS-DYNA the 2nd, 3rrd and 4th in-plane values will all be zero. Care should therefore be taken if the 4 in-plane values are averaged.

In some versions of LS-DYNA the 1st in-plane integration point is correctly written out using the global axis system while the 2nd, 3rd and 4th in-plane values are written using the elements local coordinate system. Care should therefore be taken if the 4 in-plane values are averaged.

LSDA (binout) File

Stress	By default stress tensors are in the local element coordinate system. Values are written out for all the through thickness and in-plane integration points.
Strain	The Strain tensors output is optional. By defaul the values are in the local element coordinate systems and only values at the top and bottom integration points are output. T/HIS will average these values for the mid surface.
Forces & Moments	These are not written to the LSDA file.
Extra	By default "Extra" data components are not written to the LSDA file. Some recent versions of LS-DYNA can now write the"Extra" data components to the LSDA file if the parameters OPTION1, OPTION2, OPTION3 and OPTION4 are set on the *DATABASE_ELOUT card.

Global v Local coordinate system results

The LSDA file can contain both ELOUT and ELOUTDET data components. By default T/HIS uses the data from ELOUTDET as ELOUT only contains a subset of the data in ELOUDET.

In some versions of LS-DYNA it is possible to change the Shell and ThickShell data components written to the ELOUT so that they are defined using the Global coordinate system (see EOCS on *CONTROL_OUTPUT) instead of the default local element coordinate system If this option is used then only the ELOUT file is modified, the ELOUDET file is still written using the local element coordinate system.

If T/HIS detects that the LSDA file contains both ELOUT and ELOUDET and that they are using different coordinate systems then T/HIS will display an additional option can be used to force T/HIS to use the ELOUT file data instead of the ELOUTDET data.

Through Thickness Integration Points (surfaces/layers)

Unlike the THF file the LSDA file can contain different numbers of integration points for each element. This means that if "Top" surface is selected T/HIS can correctly identify which integration point it needs to read data from.

By default strain tensors are only written out for the top and bottom surfaces and T/HIS averages these for the mid surface values. In recent versions of LS-DYNA the parameter

INTOUT on the *DATABASE_EXTENT_BINARY card can change this so that strain tensor values are written out for all the through thickness integration points. T/HIS does not currently support these additional values.

In-plane Integration Points

By default the LSDA file will contain data for all 4 in-plane integration points for any fully integrated shells. As with the THF file by default there is no information in the LSDA file to tell the difference between a shell with 32 through thickness integration points and a shell with 8 through thickness layers and 4 in-plane points per layer. If a ZTF file written by PRIMER is present then T/HIS can use the extra information on the ZTF to work out which elements have multiple in-plane points.

If the parameter INTOUT on the *DATABASE_EXTENT_BINARY card is set then the format of the LSDA file is changed and the LSDA file then contains enough information for T/HIS to identify the shells with multiple in-plane integration points without the ZTF file.

In addition to changing the format of the LSDA file setting INTOUT on the *DATABASE_EXTENT_BINARY card also outputs strain tensor values at each in-plane integration point as well as all the through thickness layers. T/HIS does not currently support strain values from multiple in-plane integration points.

Extrapolated Stress / Strain Values

The parameter NODOUT on the *DATABASE_EXTENT_BINARY card "gaa" can be used to generate stress and strain values that have been extrapolated to the nodal positions instead of values at the elements integration points. T/HIS does not currently support these extrapolated values.

13.1.8. Thick Shell Data Components

Thick Shell Data Components

For thick shells the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Stress components					
SXX	Stress in XX	yes		yes	
SYY	Stress in YY	yes		yes	
SZZ	Stress in ZZ	yes		yes	
SXY	Stress in XY	yes		yes	
SYZ	Stress in YZ	yes		yes	
SZX	Stress in ZX	yes		yes	
SMX	Maximum Principal Stress	yes		yes	
SMN	Minimum Principal Stress	yes		yes	
SMS	Maximum Shear Stress	yes		yes	
SVM	Von Mises Stress	yes		yes	
SAV	Average Stress (Pressure)	yes		yes	
STR	Stress Triaxiality Factor	yes		yes	
Strain components					
EFF	Effective Plastic Strain	yes		yes	
EXX	Strain in XX	yes		yes	
EYY	Strain in YY	yes		yes	
EZZ	Strain in ZZ	yes		yes	
EXY	Strain in XY	yes		yes	
EYZ	Strain in YZ	yes		yes	
EZX	Strain in ZX	yes		yes	
EMX	Maximum Principal Strain	yes		yes	
EMN	Minimum Principal Strain	yes		yes	
EMS	Maximum Shear Strain	yes		yes	
EVM	Von Mises Strain	yes		yes	

EAV	Average Strain	yes		yes	
"Extra" components					
An	Extra Data Component	yes		yes	

Frequency Domain Analysis

For a steady state dynamic analysis (SSD) the following nodal data components are available. For each data component both amplitude and phase angle data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Stress components					
SXX	Stress in XX			yes	
SYY	Stress in YY			yes	
SZZ	Stress in ZZ			yes	
SXY	Stress in XY			yes	
SYZ	Stress in YZ			yes	
SZX	Stress in ZX			yes	
Strain components					
EXX	Strain in XX			yes	
EYY	Strain in YY			yes	
EZZ	Strain in ZZ			yes	
EXY	Strain in XY			yes	
EYZ	Strain in YZ			yes	
EZX	Strain in ZX			yes	

For a random vibration analysis (PSD) the following nodal data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Stress components					
SXX	Stress in XX			yes	
SYY	Stress in YY			yes	
SZZ	Stress in ZZ			yes	

SXY	Stress in XY			yes	
SYZ	Stress in YZ			yes	
SZX	Stress in ZX			yes	
SVM	Von Mises Stress			yes	
Strain components					
EXX	Strain in XX			yes	
EYY	Strain in YY			yes	
EZZ	Strain in ZZ			yes	
EXY	Strain in XY			yes	
EYZ	Strain in YZ			yes	
EZX	Strain in ZX			yes	

THF (d3thdt) File

Stress	Stress tensors are in the global cartesian system unless the option to use material axes has been invoked for orthotropic materials (CMPFLG on *DATABASE_EXTENT_BINARY). By default results are available at top and bottom integration points and mid-surface but values can be output for all through thickness integration points by using MAXINT on *DATABASE_EXTENT_BINARY
Strain	The Strain tensors output is optional. Values are in the global cartesian system unless the option to use material axes has been invoked for orthotropic materials (CMPFLG on *DATABASE_EXTENT_BINARY). Only values at the top and bottom integration points are output. T/HIS will average these values for the mid surface.
Extra	The "Extra Historyl" data components will only appear in the menu if they have been selected for output (NEIPS on *DATABASE_EXTENT_BINARY). These are output for the same surfaces / integration points as the stress tensor values.

Through Thickness Integration Points

NOTE: The top and bottom "surfaces" are **not** the outer fibres if the default Gaussian integration rules are used, but rather the outer and inner integration points. The relationship between integration point location and shell thickness depends on the number of integration points used.

The following diagram shows locations of integration points with respect to shell half-thickness `(t/2`) assuming the default Gaussian integration rules have been used:

No of Points	Distance of outer fibres from neutral axis as a proportion of t/2
1	0.0 (membrane)
2	0.577 t/2
3	0.775
4	0.861 t/2
5	0.906

The "top" (or outer) point is on the positive local Z side of the element neutral axis. The output of shell data from LS-DYNA will fall into one of two categories, and the "surface" options available in T/HIS depend on this.

NOTE: It is possible to use non-default integration schemes in LS-DYNA which may locate the integration points at different places. This is an advanced topic: contact Oasys Ltd for advice.

Default output case: 3 "surfaces"

In this case, regardless of how many integration points the shell elements may actually have through their thickness, LS-DYNA writes out:

Top surface :	Top integration point
Centre surface :	Computed neutral axis value
Bottom surface :	Bottom integration point

Note that the "centre" surface here is the neutral axis value. For membrane elements all three sets of values will be the same.

Optional output case: user-defined number of integration points

The number of through thickness integration points written to the THF file can be modified using the value of MAXINT on the *DATABASE_EXTENT_BINARY card. If this parameter is changed then all thin and thick shell output written to the THF file will have MAXINT data slots for integration points in the file, regardless of how many integration points a given element may have through its thickness.

If MAXINT is not 3 then the order in which data is written to the THF file is controlled by the actual number of integration points of integration points in a shells formulation. The following table illustrates output for the case of MAXINT not equal to 3

<table>
<tr><th>Data slot in file</th><th>Thick Shell with 3 Integration points</th><th>Thick Shell with any other nunber of integration points</th></tr>
<tr><td>#1</td><td>Middle</td><td rowspan="3">Bottom
|</td></tr>
<tr><td>#2</td><td>Bottom</td></tr>
<tr><td>#3</td><td>Top</td></tr>
</table>

#4	*zero*	| Top
#5	*zero*	
#6	*zero*	

NOTE: The THF file does NOT contain any information on the number of integration points each shell was defined with.

No explicit neutral axis value is calculated or output.

The outcome of writing more integration points than have been used in a shell formulation is undefined.

There is no guarantee that the "centre" surface in this context is the neutral axis value: this will depend upon the element integration scheme. In addition where the "centre" value has been averaged from a pair of points, when the number of layers is an even number, it will definitely not be the neutral axis value: consider plastic strain in a section in pure bending!

The ZTF file generated by PRIMER can help to resolve some of these problems.

THF File + ZTF File

If a ZTF file had been generated using PRIMER then T/HIS can use additional information from the ZTF to correctly work out the number of integration points each shell element was defined with. If an attempt is made to output data for a surface that does not exist in the THF file then T/HIS will generate a warning message and a NULL curve will be generated.

In addition to working out the correct number of through thickness integration points for each element T/HIS can also use the information in the ZTF to identify models where MAXINT has been set to a -ve number in order to generate data for multiple in-plane integration points.

Effect of plotting "Top" surface on models with MAXINT = 6 and MAXINT = 9 whit and without a ZTF file.

	MAXINT = 6, no ZTF file	MAXINT = 6, ZTF file present	MAXINT = 9, no ZTF file	MAXINT = 9, ZTF file present
Thick Shell 1 has 4 integration points	Undefined (#int points < 6)	Correct (int point #4)	Undefined (#int points < 9)	Correct (int point #4)
Thick Shell 2 has 6 integration points	Correct (int point #6)	Correct (int point #6)	Undefined (#int points < 9)	Correct (int point #6)

Thick Shell 3 has 9 integration points	Incorrect (6th integration point)	Warning message as #int points < 6	Correct (int point #9)	Correct (int point #9)

In-plane Integration Points

In some versions of LS-DYNA it is now possible to write out data for all 4 in-plane integration points for fully integrated shells by setting MAXINT on the *DATABASE_EXTENT_BINARY card to a -ve number. For example specifying a value of -8 will generate data for 8 layers each with 4 in-plane integration points. If this option is used then all the elements will be written out using this option regardless of whether they are fully integrated or not.

As there is no information in the THF to indicate that data for 4 in-plane integration points has been written to the file then the file format will be exactly the same as for an analysis with a +ve value of MAXINT 4 times larger. For example MAXINT = -8 and MAXINT = 32 will both produce THF files with 32 integration points worth of data and there is no way for T/HIS to know which value of MAXINT was used to generate the data. The ZTF file generated by PRIMER can help to resolve this problem.

If multiple in-plane integration points are written to the THF file then they are written in the following order.

Layer 1 - in-plane int point #1
Layer 2 - in-plane int point #1
....
Layer n - in-plane int point #1
Layer 1 - in-plane int point #2
Layer 2 - in-plane int point #2
....
Layer n - in-plane int point #2
Layer 1 - in-plane int point #3
....

NOTE: If non fully integrated shells are included in the list of elements written to the THF file then in some versions of LS-DYNA the 2nd, 3rrd and 4th in-plane values will all be zero. Care should therefore be taken if the 4 in-plane values are averaged.

In some versions of LS-DYNA the 1st in-plane integration point is correctly written out using the global axis system while the 2nd, 3rd and 4th in-plane values are written using the elements local coordinate system. Care should therefore be taken if the 4 in-plane values are averaged.

LSDA (binout) File

Stress	By default stress tensors are in the local element coordinate system. Values are written out for all the through thickness and in-plane integration points.
Strain	The Strain tensors output is optional. By default values are in the local element coordinate systems and only values at the top and bottom integration points are output. T/HIS will average these values for the mid surface.
Extra	By default "Extra" data components are not written to the LSDA file. Some recent versions of LS-DYNA can now write the"Extra" data components to the LSDA file if the parameters OPTION1, OPTION2, OPTION3 and OPTION4 are set on the *DATABASE_ELOUT card.

Global v Local coordinate system results

The LSDA file can contain both ELOUT and ELOUTDET data components. By default T/HIS uses the data from ELOUTDET as ELOUT only contains a subset of the data in ELOUDET.

In some versions of LS-DYNA it is possible to change the Shell and ThickShell data components written to the ELOUT so that they are defined using the Global coordinate system (see EOCS on *CONTROL_OUTPUT) instead of the default local element coordinate system If this option is used then only the ELOUT file is modified, the ELOUDET file is still written using the local element coordinate system.

If T/HIS detects that the LSDA file contains both ELOUT and ELOUDET and that they are using different coordinate systems then T/HIS will display an additional option can be used to force T/HIS to use the ELOUT file data instead of the ELOUTDET data.

Through Thickness Integration Points (surfaces/layers)

Unlike the THF file the LSDA file can contain different numbers of integration points for each element. This means that if "Top" surface is selected T/HIS can correctly identify which integration point it needs to read data from.

By default strain tensors are only written out for the top and bottom surfaces and T/HIS averages these for the mid surface values. In recent versions of LS-DYNA the parameter INTOUT on the *DATABASE_EXTENT_BINARY card can change this so that strain tensor values are written out for all the through thickness integration points. T/HIS does not currently support these additional values.

In-plane Integration Points

By default the LSDA file will contain data for all 4 in-plane integration points for any fully integrated shells. As with the THF file by default there is no information in the LSDA file to tell the difference between a shell with 32 through thickness integration points and a shell with 8 through thickness layers and 4 in-plane points per layer. If a ZTF file written by PRIMER is present then T/HIS can use the extra information on the ZTF to work out which elements have multiple in-plane points.

If the parameter INTOUT on the *DATABASE_EXTENT_BINARY card is set then the format of the LSDA file is changed and the LSDA file then contains enough information for T/HIS to identify the shells with multiple in-plane integration points without the ZTF file.

In addition to changing the format of the LSDA file setting INTOUT on the *DATABASE_EXTENT_BINARY card also outputs strain tensor values at each in-plane integration point as well as all the through thickness layers. T/HIS does not currently support strain values from multiple in-plane integration points.

Extrapolated Stress / Strain Values

The parameter NODOUT on the *DATABASE_EXTENT_BINARY card can be used to generate stress and strain values that have been extrapolated to the nodal positions instead of values at the elements integration points. T/HIS does not currently support these extrapolated values.

13.1.9. Rigid Wall Data Components

Rigid Wall Data Components

For rigid walls the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FN	Normal force		yes	yes	yes
FX	Global X force			yes	yes
FY	Global Y force			yes	yes
FZ	Global Z force			yes	yes

13.1.10. Discrete Element (Spring/Damper) Data Components

Discrete Element (Spring/Damper) Data Components

For springs and dampers the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FT	Force		yes	yes	yes
ET	Elongation		yes	yes	yes
FE	Force versus Elongation		yes		
EN	Energy		yes		
MT	Moment		yes	yes	yes
RT	Rotation		yes	yes	yes
MR	Moment versus Rotation		yes		
FX	Global X force			yes	yes
FY	Global Y force			yes	yes
FZ	Global Z force			yes	yes
MX	Moment in X			yes	yes
MY	Moment in Y			yes	yes
MZ	Moment in Z			yes	yes

13.1.11. Seat Belt Data Components

Seat Belt Data Components

For seat belts the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FT	Force		yes	yes	yes
ST	Strain		yes		
FS	Force versus Strain		yes		
CL	Current Length			yes	yes

13.1.12. Retractor Data Components

Retractor Data Components

For retractors the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FT	Force		yes	yes	yes
PT	Pullout		yes	yes	yes
FP	Force versus Pullout		yes		

13.1.13. Slipring Data Components

Slipring Data Components

For sliprings the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
PT	Pull through		yes	yes	yes
WA	Warp Angle			yes	yes
SK	Skew Angle			yes	yes
FR	Friction Coefficient			yes	yes
NF	Normal Force			yes	yes
SB1	Side 1 Belt Force			yes	yes
SB2	Side 2 Belt Force			yes	yes

13.1.14. Contact Data Components

Contact Data Components

For contacts the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FXA	A Surface X force		yes	yes	yes
FYA	A Surface Y force		yes	yes	yes
FZA	A Surface Z force		yes	yes	yes
FMA	A Surface Force Magnitude		yes	yes	yes
FXB	B Surface X force		yes	yes	yes
FYB	B Surface Y force		yes	yes	yes
FZB	B Surface Z force		yes	yes	yes
FMB	B Surface Force Magnitude		yes	yes	yes
TEN	Total energy (A + B surface)		yes	yes	yes
MXA	A Surface X moment			yes	yes
MYA	A Surface Y moment			yes	yes
MZA	A Surface Z moment			yes	yes
MXB	B Surface X moment			yes	yes
MYB	B Surface Y moment			yes	yes
MZB	B Surface Z moment			yes	yes
MA	A Surface Mass			yes	yes
MB	B Surface Mass			yes	yes
AEN	A Surface side energy		yes	yes	yes
BEN	B Surface side energy		yes	yes	yes
FRI	Frictional energy		yes	yes	yes

13.1.15. Nodal Reaction Force Data Components

Nodal Reaction Force Data Components

For nodal reactions the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FX	X Force		yes	yes	yes
FY	Y Force		yes	yes	yes
FZ	Z Force		yes	yes	yes
FM	Force Magnitude		yes	yes	yes
EN	Energy			yes	yes
LFX	Local X force			yes	yes
LFY	Local Y force			yes	yes
LFZ	Local Z force			yes	yes

13.1.16. Airbag Data Components

Airbag Data Components

For airbags the following data components are available. Versions of LS-DYNA 971 can also generate PART based data for AIRBAGS that use the PARTICLE airbag methods.

If *DATABASE_CPM_SENSOR has been used to define sensors then the output for the sensors will also be available under the AIRBAG data components.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Airbag components					
PR	Pressure		yes	yes	yes
VO	Volume		yes	yes	yes
IE	Internal energy		yes	yes	yes
IN	Mass flow rate in		yes	yes	yes
OU	Mass flow rate out		yes	yes	yes
MIN	Mass in			yes	yes
MOU	Mass Out			yes	yes
TM	Total mass		yes	yes	yes
DE	Density			yes	yes
SA	Surface area			yes	yes
TE	Gas temperature			yes	yes
RF	Reaction force			yes	yes
MAF	Mass flow rate through fabric			yes	yes
MAV	Mass flow rate through vent			yes	yes
MOF	Mass out through fabric			yes	yes
MOV	Mass flow through vent			yes	yes
TK	Translational Kinetic Energy			yes	
IF	Inflator Energy			yes	
DMP	Damping Energy			yes	
PP	Average Particle Pressure			yes	
Part components					
PR	Pressure			yes	

MAF	Flow rate through fabric			yes	
MAV	Flow rate through vent			yes	
TA	Total area			yes	
UN	Unblocked area			yes	
TE	Gas temperature			yes	
PPR	Pressure s+			yes	
NPR	Pressure s-			yes	
HC	Heat Convection Energy			yes	
EV	Enhanced Vent			yes	
LE	Leak Energy			yes	
PVO	Por Volume			yes	
Airbag Chamber components					
PR	Pressure			yes	
VO	Volume			yes	
IE	Internal energy			yes	
IN	Mass flow rate in			yes	
OU	Mass flow rate out			yes	
TM	Total mass			yes	
DE	Density			yes	
SA	Surface area			yes	
TE	Gas temperature			yes	
RF	Reaction force			yes	
TR	Translational Energy			yes	
NP	Number of Particle			yes	
PP	Average Particle Pressure			yes	

CPM Sensor Components (*DATABASE_CPM_SENSOR)

	Component	**THF (d3thdt)**	**XTF (xtfile)**	**LSDA (binout)**	**ASCII**
X	X Co-ordinate of Sensor			yes	yes
Y	Y Co-ordinate of Sensor			yes	yes
Z	Z Co-ordinate of Sensor			yes	yes
VX	X Velocity			yes	yes

VY	Y Velocity			yes	yes
VZ	Z Velocity			yes	yes
VM	Velocity Magnitude			yes	yes
PR	Pressure			yes	yes
DE	Density			yes	yes
TE	Temperature			yes	yes
NP	N Particles			yes	yes

13.1.17. Joint Data Components

Joint Data Components

For joints the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Basic Joints					
FX	Global X force			yes	yes
FY	Global Y force			yes	yes
FZ	Global Z force			yes	yes
FM	Force Magnitude			yes	yes
MX	Moment in X			yes	yes
MY	Moment in Y			yes	yes
MZ	Moment in Z			yes	yes
MM	Moment Magnitude			yes	yes
EN	Energy			yes	yes
General Stiffness Joints					
XD	X Displacement			yes	yes
DXD	d(X)/dt			yes	yes
XSF	X stiffness force			yes	yes
XDF	X damping force			yes	yes
XTF	X total force			yes	yes
YD	Y displacement			yes	yes
DYD	d(Y)/dt			yes	yes
YSF	Y stiffness force			yes	yes
YDF	Y damping force			yes	yes
YTF	Y total force			yes	yes
ZD	Z displacement			yes	yes
DZD	d(Z)/dt			yes	yes
ZSF	Z stiffness force			yes	yes
ZDF	Z damping force			yes	yes
ZTF	Z total force			yes	yes

EN	Total joint energy			yes	yes
Flexion Torsion Joints					
AA	Alpha angle			yes	yes
DA	d(Alpha)/dt			yes	yes
ALS	Alpha stiffness moment			yes	yes
ALD	Alpha damping moment			yes	yes
ALT	Alpha total moment			yes	yes
BA	Beta angle			yes	yes
DB	d(Beta)/dt			yes	yes
BES	Beta stiffness moment			yes	yes
BED	Beta damping moment			yes	yes
BET	Beta total moment			yes	yes
GA	Gamma angle			yes	yes
DG	d(Gamma)/dt			yes	yes
GSF	Gamma scale factor			yes	yes
EN	Total joint energy			yes	yes
Translational Joints					
XD	X displacement			yes	yes
DXD	d(X)/dt			yes	yes
YD	Y displacement			yes	yes
DYD	d(Y)/dt			yes	yes
ZD	Z displacement			yes	yes
DZD	d(Z)/dt			yes	yes
XSF	X stiffness			yes	yes
XDF	X damping			yes	yes
XTF	X total			yes	yes
YSF	Y stiffness			yes	yes
YDF	Y damping			yes	yes
YTF	Y total			yes	yes
ZSF	Z stiffness			yes	yes
ZDF	Z damping			yes	yes
ZTF	Z total			yes	yes

EN	Total joint energy			yes	yes
Cylindrical Joints					
PD	P displacement			yes	yes
DPD	d(P)/dt			yes	yes
RD	R displacement			yes	yes
DRD	d(R)/dt			yes	yes
ZD	Z displacement			yes	yes
DZD	d(Z)/dt			yes	yes
PSF	P stiffness			yes	yes
XDF	P damping			yes	yes
XTF	P total			yes	yes
RSF	R stiffness			yes	yes
RDF	R damping			yes	yes
RTF	R total			yes	yes
ZSF	Z stiffness			yes	yes
ZDF	Z damping			yes	yes
ZTF	Z total			yes	yes
EN	Total joint energy			yes	yes

13.1.18. Cross Section Data Components

Cross Section Data Components

For cross sections the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FX	X Force			yes	yes
FY	Y Force			yes	yes
FZ	Z Force			yes	yes
RM	Force Magnitude			yes	yes
MX	Moment in X			yes	yes
MY	Moment in Y			yes	yes
MZ	Moment in Z			yes	yes
MM	Moment Magnitude			yes	yes
CX	X centroid coordinate			yes	yes
CY	Y centroid coordinate			yes	yes
CZ	Z centroid coordinate			yes	yes
AR	Area of Cross Section			yes	yes

13.1.19. Subsystem Data Components

Subsystem Data Components

For subsystems the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Energy					
KE	Kinetic energy			yes	yes
IE	Internal energy			yes	yes
HG	Hourglass energy			yes	yes
KR	Kinetic energy ratio			yes	yes
IM	Internal energy ratio			yes	yes
Momentum					
XM	X momentum			yes	yes
YM	Y momentum			yes	yes
ZM	Z momentum			yes	yes
Mass					
TM	Total Mass			yes	yes
CM	Center of Mass			yes	
XCM	X Center of Mass			yes	yes
YCM	Y Center of Mass			yes	yes
ZCM	Z Center of Mass			yes	yes
Inertia Tensors					
I11	Inertia Tensor Row11			yes	yes
I12	Inertia Tensor Row12			yes	yes
I13	Inertia Tensor Row13			yes	yes
I21	Inertia Tensor Row11			yes	yes
I22	Inertia Tensor Row12			yes	yes
I23	Inertia Tensor Row13			yes	yes
I31	Inertia Tensor Row11			yes	yes
I32	Inertia Tensor Row12			yes	yes
I33	Inertia Tensor Row13			yes	yes

Principal Inertia					
I1	Principal Inertia Row11			yes	yes
I2	Principal Inertia Row22			yes	yes
I3	Principal Inertia Row33			yes	yes
Principal Directions					
P11	Principal Directions Row11			yes	yes
P12	Principal Directions Row12			yes	yes
P13	Principal Directions Row13			yes	yes
P21	Principal Directions Row11			yes	yes
P22	Principal Directions Row12			yes	yes
P23	Principal Directions Row13			yes	yes
P31	Principal Directions Row11			yes	yes
P32	Principal Directions Row12			yes	yes
P33	Principal Directions Row13			yes	yes

13.1.20. Geometric Contact Data Components

Geometric Contact Data Components

For geometric contact entities the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FX	X Force			yes	yes
FY	Y Force			yes	yes
FZ	Z Force			yes	yes
RM	Force Magnitude			yes	yes
MX	Moment in X			yes	yes
MY	Moment in Y			yes	yes
MZ	Moment in Z			yes	yes
MM	Moment Magnitude			yes	yes

13.1.21. Nodal Rigid Body Data Components

Nodal Rigid Body Data Components

For nodal rigid bodies the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
DX	X Displacement			yes	yes
DY	Y Displacement			yes	yes
DZ	Z Displacement			yes	yes
DM	Displacement Magnitude			yes	yes
VX	X Velocity			yes	yes
VY	Y Velocity			yes	yes
VZ	Z Velocity			yes	yes
VM	Velocity Magnitude			yes	yes
AX	X Acceleration			yes	yes
AY	Y Acceleration			yes	yes
AZ	Z Acceleration			yes	yes
AM	Acceleration Magnitude			yes	yes
CX	X Co-ordinate			yes	yes
CY	Y Co-ordinate			yes	yes
CZ	Z Co-ordinate			yes	yes
RX	X Rotation			yes	yes
RY	Y Rotation			yes	yes
RZ	Z Rotation			yes	yes
RM	Rotation Magnitude			yes	yes
RVX	X Rotational Velocity			yes	yes
RVY	Y Rotational Velocity			yes	yes
RVZ	Z Rotational Velocity			yes	yes
RVM	Rotational Velocity Magnitude			yes	yes
RAX	X Rotational Acceleration			yes	yes
RAY	Y Rotational Acceleration			yes	yes

RAZ	Z Rotational Acceleration			yes	yes
RAM	Rotational Acceleration Magnitude			yes	yes
D11	Direction Cosine 11			yes	
D12	Direction Cosine 12			yes	
D13	Direction Cosine 13			yes	
D21	Direction Cosine 21			yes	
D22	Direction Cosine 22			yes	
D23	Direction Cosine 23			yes	
D31	Direction Cosine 31			yes	
D32	Direction Cosine 32			yes	
D33	Direction Cosine 33			yes	
LDX	Local X Displacement			yes	yes
LDY	Local Y Displacement			yes	yes
LDZ	Local Z Displacement			yes	yes
LVX	Local X Velocity			yes	yes
LVY	Local Y Velocity			yes	yes
LVZ	Local Z Velocity			yes	yes
LAX	Local X Acceleration			yes	yes
LAY	Local Y Acceleration			yes	yes
LAZ	Local Z Acceleration			yes	yes
LRX	Local X Rotation			yes	yes
LRY	Local Y Rotation			yes	yes
LRZ	Local Z Rotation			yes	yes
LRVX	Local X Rotational Velocity			yes	yes
LRVY	Local Y Rotational Velocity			yes	yes
LRVZ	Local Z Rotational Velocity			yes	yes
LRAX	Local X Rotational Acceleration			yes	yes
LRAY	Local Y Rotational Acceleration			yes	yes
LRAZ	Local Z Rotational			yes	yes

13.1.22. Spotweld Data Components

Spotweld Data Components

For spotwelds the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
AX	Axial force			yes	yes
SH	Shear force			yes	yes
LE	Length			yes	yes
FT	Failure Time			yes	yes
FA	Failure			yes	yes
MM	Moment Magnitude			yes	yes
TO	Torsion			yes	yes
The following additional data components are also available for Solid Spotwelds and Spotweld Assemblies if the DCFAIL file is written.					
FF	DC Failure Function			yes	yes
NF	Normal Failure Term			yes	yes
SF	Shear Failure Term			yes	yes
BF	Bending Failure Term			yes	yes
AR	Spotweld Area			yes	yes

The DCFAIL file contains additional data for spotweld solids and clusters models using the _DAIMLERCHRYSLER version of *MAT_SPOTWELD (this version of the material does not support beam elements).The file contains additional failure data showing how close to failure the spotweld is in tension, shear, bending and torsion, in addition it contains another copy the normal spotweld forces written to the SWFORC file.

The new data components appear under the SOLID and ASSEMBLY sub types within the SPOTWELD menu. If the SWFORC file is also present then the normal forces and read from the SWFORC file, if the SWFORC file doesn't exist but the DCFAIL file does then the data components (Normal, shear forces etc) that are mirrored in the DCFAIL file are read from there instead.

As the DCFAIL file only contains the ID's and not the types or each connection then it is not possible to tell from the DCFAIL file alone which items are solids and which ones are spotweld clusters. If the SWFORC file is present then T/HIS used the information in this file to match up the ID's and work out the type of each item in the DCFAIL file. If the

SWFORC file isn't present then it attempts to use the data in the ZTF file to work out the types.

13.1.23. SPC Data Components

SPC Data Components

For SPC's the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FX	X Force			yes	yes
FY	Y Force			yes	yes
FZ	Z Force			yes	yes
FM	Force Magnitude			yes	yes
MX	Moment in X			yes	yes
MY	Moment in Y			yes	yes
MZ	Moment in Z			yes	yes
MM	Moment Magnitude			yes	yes

13.1.24. Boundary Condition Data Components

Boundary Condition Data Components

For SPC's the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
For Pressure and Force Boundary conditions the following components are available.					
FX	Applied X Force			yes	yes
FY	Applied Y Force			yes	yes
FZ	Applied Z Force			yes	yes
FR	Applied Resultant force			yes	yes
EN	Energy from applied force			yes	yes
For Nodal Velocity Boundary conditions the following components are available.					
FX	Boundary condition motion X Force			yes	yes
FY	Boundary condition motion Y Force			yes	yes
FZ	Boundary condition motion Z Force			yes	yes
FR	Resultant Boundary condition motion force			yes	yes
EN	Energy from Boundary condition motion			yes	yes
For Rigid Body Velocity Boundary conditions the following components are available.					
FX	Boundary condition motion X Force			yes	yes
FY	Boundary condition motion Y Force			yes	yes
FZ	Boundary condition motion Z Force			yes	yes
FR	Resultant Boundary condition motion force			yes	yes
EN	Energy from Boundary condition motion			yes	yes

MX	Boundary condition motion X Moment			yes	yes
MY	Boundary condition motion Y Moment			yes	yes
MZ	Boundary condition motion Z Moment			yes	yes
MM	Boundary condition			yes	yes

13.1.25. FSI Data Components

FSI Data Components

For Fluid structural interactions the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
PR	Pressure				yes
FX	X Force				yes
FY	Y Force				yes
FZ	Z Force				yes
FM	Force Magnitude				yes
PL	Porous Leakage				yes
MF	Mass Flux				yes
LFX	Leakage X Force				yes
LFY	Leakage X Force				yes
LFZ	Leakage X Force				yes
LFM	Leakage Force Magnitude				yes
TE	Part Temperature				yes
X	X Co-ordinate of Sensor				yes
Y	Y Co-ordinate of Sensor				yes
Z	Z Co-ordinate of Sensor				yes
PR	Pressure				yes
SO	Cpld Solid ID				yes
TE	Temperature at Sensor				yes

13.1.26. SPH Data Components

SPH Data Components

For SPH's the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
DE	Density			yes	yes
EXX	Strain in XX			yes	yes
EYY	Strain in YY			yes	yes
EZZ	Strain in ZZ			yes	yes
EXY	Strain in XY			yes	yes
EYZ	Strain in YZ			yes	yes
EZX	Strain in ZX			yes	yes
EFS	Effective Stress			yes	yes
SXX	Stress in XX			yes	yes
SYY	Stress in YY			yes	yes
SZZ	Stress in ZZ			yes	yes
SXY	Stress in XY			yes	yes
SYZ	Stress in YZ			yes	yes
SZX	Stress in ZX			yes	yes
SM	Smoothing Length			yes	yes

13.1.27. Tracer Data Components

Tracer Data Components

For tracers the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
CX	X Co-ordinate			yes	yes
CY	Y Co-ordinate			yes	yes
CZ	Z Co-ordinate			yes	yes
CV	Current vector			yes	yes
VX	X Velocity			yes	yes
VY	Y Velocity			yes	yes
VZ	Z Velocity			yes	yes
VM	Velocity Magnitude			yes	yes
SXX	Stress in XX			yes	yes
SYY	Stress in YY			yes	yes
SZZ	Stress in ZZ			yes	yes
SXY	Stress in XY			yes	yes
SYZ	Stress in YZ			yes	yes
SZX	Stress in ZX			yes	yes
EFP	Effective Plastic Strain			yes	yes
DE	Density			yes	yes
RV	Relative volume			yes	yes
AC	Active			yes	yes

13.1.28. Pulley Data Components

Pulley Data Components

For pulleys the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FT	Force			yes	yes
SL	Slip			yes	yes
SR	Slip Rate			yes	yes
AN	Warp Angle			yes	yes

13.1.29. ICFD Data Components

ICFD Data Components

For ICFD results the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
ICFD Nodes and ICFD Points					
CX	X Co-ordinate				yes
CY	Y Co-ordinate				yes
CZ	Z Co-ordinate				yes
CV	Current vector				yes
VX	X Velocity				yes
VY	Y Velocity				yes
VZ	Z Velocity				yes
VM	Velocity Magnitude				yes
AVX	X AVelocity				yes
AVY	Y AVelocity				yes
AVZ	Z AVelocity				yes
AVM	AVelocity Magnitude				yes
PR	Pressure				yes
DE	Density				yes
VC	Viscosity				yes
VTX	X Vorticity				yes
VTY	Y Vorticity				yes
VTZ	Z Vorticity				yes
VTM	Vorticity Magnitude				yes
QC	Q Critical				yes
VT	Viscous Turbulence				yes
PA	P Average				yes
LS	LSet				yes
A	Alpha				yes
TE	Temperature				yes

ICFD Drag					
FPX	X Pressure Drag				yes
FPY	Y Pressure Drag				yes
FPZ	Z Pressure Drag				yes
FPM	Pressure Drag Magnitude				yes
FVX	X Viscous Drag				yes
FVY	Y Viscous Drag				yes
FVZ	Z Viscous Drag				yes
FVM	Viscous Drag Magnitude				yes
MPX	MX Pressure Drag				yes
MPY	MY Pressure Drag				yes
MPZ	MZ Pressure Drag				yes
MPM	Pressure Drag Moment Magnitude				yes
MVX	MX Viscous Drag				yes
MVY	MY Viscous Drag				yes
MVZ	MZ Viscous Drag				yes
MVM	Viscous Drag Moment Magnitude				yes
ICFD Temperature					
TAA	Temperature Area Average				yes
TSA	Temperature Sum Average				yes
TEH	Average Heat Flux				yes
AR	Total Area				yes
HTC	Heat Transfer Coefficient				yes

13.1.30. CESE Data Components

CESE Data Components

For CESE results the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
CESE Element and CESE Points					
CX	X Co-ordinate				yes
CY	Y Co-ordinate				yes
CZ	Z Co-ordinate				yes
CV	Current vector				yes
VX	X Velocity				yes
VY	Y Velocity				yes
VZ	Z Velocity				yes
VM	Velocity Magnitude				yes
VTX	X Vorticity				yes
VTY	Y Vorticity				yes
VTZ	Z Vorticity				yes
VTM	Vorticity Magnitude				yes
DE	Density				yes
PR	Pressure				yes
TE	Temperature				yes
CESE FSI Drag					
FPX	X Pressure Force				yes
FPY	Y Pressure Force				yes
FPZ	Z Pressure Force				yes
FPM	Pressure Force Magnitude				yes
CESE Segment Set Drag					
FPX	X Pressure Force				yes
FPY	Y Pressure Force				yes
FPZ	Z Pressure Force				yes
FPM	Pressure Force Magnitude				yes

FVX	X Viscous Force				yes
FVY	Y Viscous Force				yes
FVZ	Z Viscous Force				yes
FVM	Viscous Force Magnitude				yes
AR	Total Area				yes

13.1.31. EM Data Components

EM Data Components

For EM results the following da

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
EM Element, EM Node and EM Points					
CX	X Co-ordinate				yes
CY	Y Co-ordinate				yes
CZ	Z Co-ordinate				yes
CV	Current vector				yes
ECX	X Current				yes
ECY	Y Current				yes
ECZ	Z Current				yes
ECM	Current Magnitude				yes
BFX	X BField				yes
BFY	Y BField				yes
BFZ	Z BField				yes
BFM	BField Magnitude				yes
AFX	X AField				yes
AFY	Y AField				yes
AFZ	Z AField				yes
AFM	AField Magnitude				yes
S	Sigma				yes
MUR	Mu-R				yes
JHR	JHRate				yes
LFX	X Lorentz Force				yes
LFY	Y Lorentz Force				yes
LFZ	Z Lorentz Force				yes
LFM	Lorentz Force Magnitude				yes
EFX	X EField				yes
EFY	Y EField				yes

EFZ	Z EField				yes
EFM	EField Magnitude				yes
EM Circuit					
EVO	Voltage				yes
ECH	Charge				yes
ECU	Current				yes
ECR	Circuit Resistance				yes
EER	Equivalent Resistance				yes
ECI	Inductance				yes
EM1	Mutual Inductance 1				yes
EM2	Mutual Inductance 2				yes
EM3	Mutual Inductance 3				yes
EM Circuit0D					
EVO	Voltage				yes
ECH	Charge				yes
ECU	Current				yes
ECE	Total Energy				yes
EM PartData					
LFX	X Lorentz Force				yes
LFY	Y Lorentz Force				yes
LFZ	Z Lorentz Force				yes
LFM	Lorentz Force Magnitude				yes
JHE	Joule Heating Energy				yes
MAG	Magnetic Energy				yes
KIN	Kinetic Energy				yes
PLA	Plastic Energy				yes
EM IsoPotOut					
EVO	Voltage				yes
ECU	Current				yes
EM CircuitRes					
ECV	Contact Current				yes
ECR	Contact Resistance				yes

ECJ	Contact Joule heat rate				yes
ECA	Contact Area				yes
EM BoundaryOut					
EBV	Voltage				yes
EBC	Current				yes
EBA	Area				yes
EM IsoPotConnOut					
EVO	Voltage				yes
ECH	Charge				yes
ECU	Current				yes
ECR	Contact Resistance				yes
POW	Power				yes
ENE	Energy				yes
EM RandlesCell					
TVO	TotVoltage				yes
OCV	OCV				yes
DVO	DampVoltage				yes
RCU	Current				yes
SOC	SOC				yes
SOF	SOCFunc				yes
SOS	SOCShift				yes
SOM	SOCSum				yes
RR0	R0				yes
R10	R10				yes
C10	C10				yes
TEM	Temp				yes
CNM	Ckt Number				yes
EM RandlesIntshortCell					
MXR	Maximum resistance				yes
SHC	Short circuits				yes
TOC	Total circuits				yes
TOR	Total resistance				yes

T/HIS

ARS	Area short				yes
EM RogoCoil					
RVC	Volume Current				yes
RSC	Surface Current				yes
RVM	Magnetic Field				yes
EM Global					
RUN	Run timestep				yes
CFL	Condition timestep				yes
RBC	Ratio				yes
TVO	TotVoltage				yes
OCV	OCV				yes
DVO	DampVoltage				yes
RCU	Current				yes
SOC	SOC				yes
SOF	SOCFunc				yes
SOS	SOCShift				yes
SOM	SOCSum				yes
RR0	R0				yes
R10	R10				yes
C10	C10				yes
TEM	Temp				yes
VC2	VC2				yes
VC3	VC3				yes
R20	R20				yes
R30	R30				yes
C20	C20				yes
C30	C30				yes
OHP	Ohm Heat Power				yes
RHP	Reversible Heat Power				yes
ECP	Equivalent Capacity Power				yes
OHE	Ohm heat energy				yes
RHE	Reversible heat energy				yes

ECE	Equivalent Capacity energy				yes
ESE	Equivalent storage energy				yes
ECJH	Ext ckt Joule Heating				yes
ECME	Ext ckt Magnetic Energy				yes
ECCE	Ext ckt Capacitor Energy				yes
MJH	Mesh conductor Joule Heating				yes
MME	Mesh conductor Mag Energy				yes
AME	Air Magnetic Energy				yes
TEE	Total EM Energy				yes
TPE	Total Plastic Energy				yes
TKE	Total kinetic Energy				yes
MSR	Maximum short resistance				yes
NSC	Number of short circuits				yes
TNC	Total number of circuits				yes
TSR	Total short resistance				yes
MXR	Maximum resistance				yes
SHC	Short circuits				yes
TOC	Total circuits				yes
TOR	Total resistance				yes
TRS	Area short				yes

13.1.32. Particle Blast Data Components

Particle Blast Data Components

For PBLASTs the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
Particle blast components					
AIE	Air internal energy			yes	yes
DPIE	Detonation product internal energy			yes	yes
OIE	Outside domain internal energy			yes	yes
ATE	Air translational energy			yes	yes
DPTE	Detonation product translational energy			yes	yes
OTE	Outside domain translational energy			yes	yes
Part components					
APR	Air pressure			yes	yes
DPPR	Detonation product pressure			yes	yes
RPR	Resultant pressure			yes	yes
AR	Surface Area			yes	yes
AFX	Air X Force			yes	yes
AFY	Air Y Force			yes	yes
AFZ	Air Z Force			yes	yes
DPFX	Detonation product X Force			yes	yes
DPFY	Detonation product Y Force			yes	yes
DPFZ	Detonation product Z Force			yes	yes
RFX	Resultant X Force			yes	yes
RFY	Resultant Y Force			yes	yes
RFZ	Resultant Z Force			yes	yes

13.1.33. Pressure Tube Data Components

Pressure Tube Data Components

For pressure tubes the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
AR	Cross section area			yes	yes
DE	Density			yes	yes
PR	Pressure			yes	yes
VEL	Velocity			yes	yes

13.1.34. Bearing Data Components

Bearing Data Components

For bearings the following data components are available.

	Component	THF (d3thdt)	XTF (xtfile)	LSDA (binout)	ASCII
FX	X Force				yes
FY	Y Force				yes
FZ	Z Force				yes
MX	X Moment				yes
MY	Y Moment				yes
MZ	Z Moment				yes
DX	X Displacement				yes
DY	Y Displacement				yes
DZ	Z Displacement				yes
AX	X Angle				yes
AY	Y Angle				yes
AZ	Z Angle				yes
LFX	Local X Force				yes
LFY	Local Y Force				yes
LFZ	Local Z Force				yes
LMX	Local X Moment				yes
LMY	Local Y Moment				yes
LMZ	Local Z Moment				yes
LDX	Local X Displacement				yes
LDY	Local Y Displacement				yes
LDZ	Local Z Displacement				yes
LAX	Local X Angle				yes
LAY	Local Y Angle				yes
LAZ	Local Z Angle				yes

13.2. APPENDIX B - T/HIS Curve File Format

APPENDIX B - T/HIS CURVE FILE FORMAT

A curve file is a file of x, y values which can be read into T/HIS for plotting. It can be written by T/HIS or by another program, or created using a text editor.

The format is as flexible as possible to allow many types of data to be handled.

Line 1	:	Title
Line 2	:	X axis label
Line 3	:	Y axis label
Line 4	:	Curve label
Line 5	:	X, Y point 1
Line 6	:	X, Y point 2
:		:
Line n+4	:	X, Y point n

The X and Y values can be in any format as long as the two values are separated by either a space or comma. Up to 500000 points can be input.

Several curves can be put in one file sequentially, separated by the word CONTINUE. The title and three label lines must be present for each curve.

A comment line may be included anywhere in the file by starting the line with a '$'.

Comment lines above the curve's title can contain styles and curve tags associated with the corresponding curve.

13.2.1. Curve STYLE Information

Curve STYLE Information

From version 9.1 onwards T/HIS will recognise a line starting **$ STYLE** as a style request for the following curve and the curve will be displayed with the corresponding style

A **$ STYLE** line will take the format

> **$ STYLE : LINE STYLE, LINE COLOUR, LINE WIDTH, LINE SYMBOLS, SYMBOL FREQUENCY**

The following **$ STYLE** options are available:

Style options	Available styles	Default
LINE STYLE	solid dash none	solid
LINE COLOUR	white red green blue cyan magenta yellow orange turquoise indigo lime	dependent on curve#
LINE WIDTH	fine normal bold heavy	normal
LINE SYMBOLS	triangle square diamond hourglass cross circle start dot null	dependent on curve#
SYMBOL FREQUENCY	frequency number (integer)	

13.2.2. Curve TAGs

Curve TAGs

T/HIS will recognise a line starting with **$ TAG** as a tag for the following curve and the tag can be used in T/HIS to reference the corresponding curve

a **$ TAG** line will take the format

> **$ TAG : *tag name***

13.2.3. Curve UNITs

Curve UNITs

From version 9.4 onwards a T/HIS curve file can also contain information on the Unit system and the X and Y axis units.

A unit system is defined by a line starting with **$ UNIT SYSTEM** and will take the format

$ UNIT SYSTEM : ***system name***

The following unit systems names can be specified by using either the full name or just " **Un** ."

U1: m, kg, s (SI)
U2: mm, t, s
U3: mm, kg, ms
U4: mm, g, ms
U5: ft, slug, s
U6: m, t, s

The X and Y axis units are defined by a line starting with either **$ X AXIS UNIT** or **$ Y AXIS UNIT** and take one of the 2 following formats

$ X AXIS UNIT : ***unit name***
$ X AXIS UNIT : ***mass,length,time,angle,temperature,current***

For the 1st format thefollowing predefined unit names are available.

Time	**Rotation**	**Momentum**	**Energy Den**
Energy	**Rot Vel**	**Density**	**Mass Flow**
Work	**Rot Accel**	**Stress**	**Frequency**
Temperature	**Length**	**Strain**	**Power**
Displacement	**Area**	**Force**	**Thermal Flux**
Velocity	**Volume**	**Moment**	**Force width**
Accel	**Mass**	**Pressure**	**Moment width**

If the axis units are NOT one of these predefined units then the second input format can be used to define the unit in terms of it's basic properties. The values for ***mass, length, time, angle, temperature*** and ***current*** should be the powers that are used to describe the unit in terms of it's fundamental dimensions.

Some examples of common units defined using this method are shown below.

Unit	Mass	Length	Time	Angle	Temperature	Current
Time	0.0	0.0	1.0	0.0	0.0	0.0
Displacement	0.0	1.0	0.0	0.0	0.0	0.0

Velocity	0.0	1.0	-1.0	0.0	0.0	0.0
Acceleration	0.0	1.0	-2.0	0.0	0.0	0.0
Stress	-1.0	1.0	-2.0	0.0	0.0	0.0

13.2.4. Example

Example

The following example shows a curve file containing 2 curves.

The first curve will be plotted with a bold, solid, green line with triangular symbols every other data point. The curve contains 5 data points and is given a reference tag CURVE_1

The second curve will be plotted with a dashed, white, normal line. No symbols will be displayed. The curve contains 2 data points and has no reference tag.

`$`	Comment line
`$ STYLE : solid,green,bold,triangle,2`	Style line
`$ TAG : CURVE_1`	Tag line
`$`	Comment line
`CURVE FILE EXAMPLE`	;Title
`Time`	;X axis label
`Displacement`	;Y axis label
`Curve number 1`	;Curve label
`0            2.0`	;1st data pair
`1.0          4E-3`	
`4.0,         4.7`	
`5 4`	
`10.0         8.9`	;End of 1st curve
`CONTINUE`	
`$`	Comment line
`$`	Comment line
`$ STYLE : dash,white,,,`	Style line
`CURVE FILE EXAMPLE`	;Title
`Time`	
`Displacement`	
`Curve number 2`	
`0.0          7E2`	
`2.0          8.7E-9`	

Notes:

The abscissa (x axis) values are assumed to be in the correct order.
The free format allowed for the data points.
The style line must contain 5 comma separated words in the order LINE STYLE, LINE COLOUR, LINE WIDTH, LINE SYMBOLS, SYMBOL FREQUENCY to be successfully understood by T/HIS.

If any words are unspecified in the style line, as in curve 2, T/HIS will take the default option.

13.3. APPENDIX C - T/HIS Bulk Data File Format

APPENDIX C - T/HIS BULK DATA FILE FORMAT

Format of a T/HIS Bulk Data File.

A bulk data file contains a number of curves that share the same X values.

The format of the file is as follows:

Line 1	:	Title
Line 2	:	Number of curves (maximum 12)
Line 3	:	Format, see Note 1 below
Line 4	:	Multipliers on values, see Note 2 below
Line 5	:	Axis labels, see Note 3 below
Line 6	:	Line labels, see Note 4 below
Line 7	:	X, Y1, Y2, Y3 ...... point 1
Line 8	:	X, Y1, Y2, Y3 ...... point 2
Line n+6	:	X, Y1, Y2, Y3 ...... point n

Up to 500000 points can be read in for each curve.

Note 1	The format for the point data must be given as a standard Fortran format statement, for example (F10.3, 4F10.2). The external brackets around the format must be included. If the data can be read in as a free format then type `FREE` or leave this line blank. Note however, free data is read in more slowly than formatted data.
Note 2	The multipliers are the amount by which the values read in are to be multiplied. For example you may wish to correct from ms to s or units of **G** (gravity) to mm/s ^{2}. On this line give the multipliers in the order X-value, Y1-value, Y2-value, etc. Separate each multiplier by a space or comma. A zero value is assumed to be 1. If all curves are to be read in as defined leave this line blank.
Note 3	The axis labels are character strings, separated by commas given in the following order. X-axis label, Y1-axis label, Y2-axis label, etc.
Note 4	The line labels are character strings separated by commas given in the following order. Line label 1, Line label 2, Line label 3, etc.

A comment line may be included any where in the file by starting the line with a `$` .

The following shows a bulk data file with three curves and seven points on each curve.

```
$ Comment line
Title of the curves
3
FREE
$ A multiplier of 10 on X values and 5 on Y2 values

10,,5,

x-axis,y1-axis,y2-axis,y3-axis
curve 1,curve 2,curve 3
$ Now for the data
0.0 0.0 1.0 2.0
1.0 1.0 3.0 4.0
2.0 2.0 4.0 5.0
2.4 4.4 5.5 7.4
3.3 7.8 5.8 9.2
4.4 10.0 12.0 13.0
```

13.4. APPENDIX D - Filtering

APPENDIX D - FILTERING

This Appendix describes the filtering options within T/HIS.

Curves can be filtered to remove high frequency noise. The technique is typically applied to acceleration and force traces. Options available include standard filters (Channel Frequency Classes 60, 180, 600 and 1000 as per British Standard BS AU 228: Part 1: 1989, and the USA's National Highway Traffic Safety Administration (NHTSA) FIR filter). The standard filters (except the FIR filter) are all special cases of the Butterworth filter.

13.4.1. Curve Regulation

Curve Regulation

All filtering options require the curves to have a constant time increment between points. This will generally be the case if the curves are LS-DYNA time history results. If not, the REGULARISE option will convert the curve to constant time increment.

Typically the time increment should be at least 10 times the cut-off frequency; 10kHz (a 0.0001 second interval time base) is a good choice for automotive crash applications.

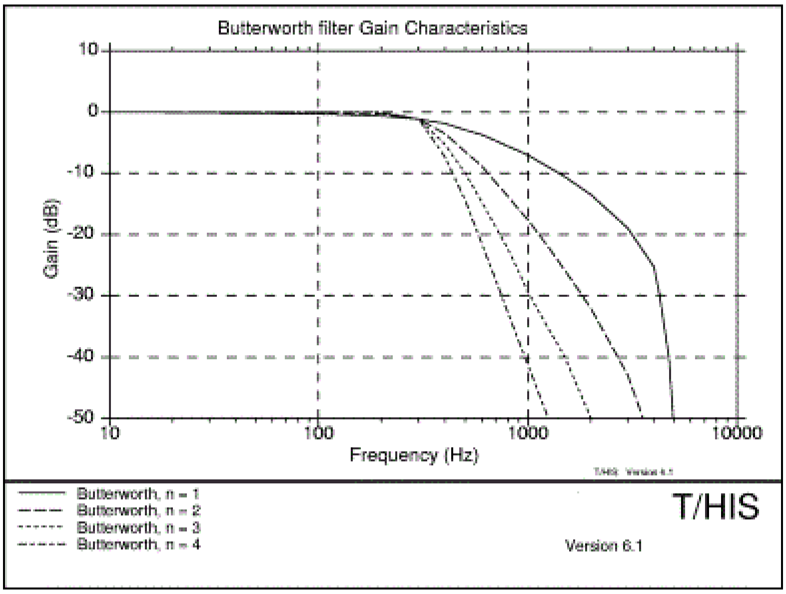
13.4.2. Use of the Butterworth Filter Option

Use of the Butterworth Filter Option

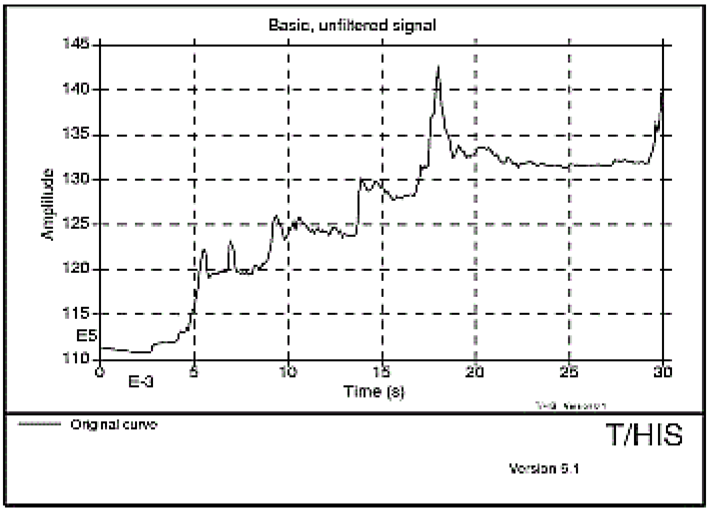
The Butterworth filter is a low pass filter with two input variables; order and cut-off frequency.

The order of the filter controls the roll-off rate, as shown here in the figure (right)

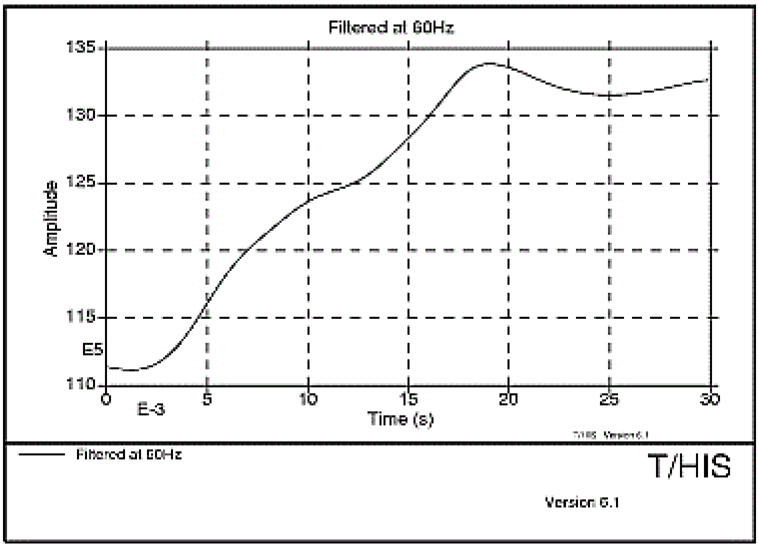
This is a 300Hz filter. It can be seen that higher orders attenuate the results more quickly: they have a higher roll-off rate.



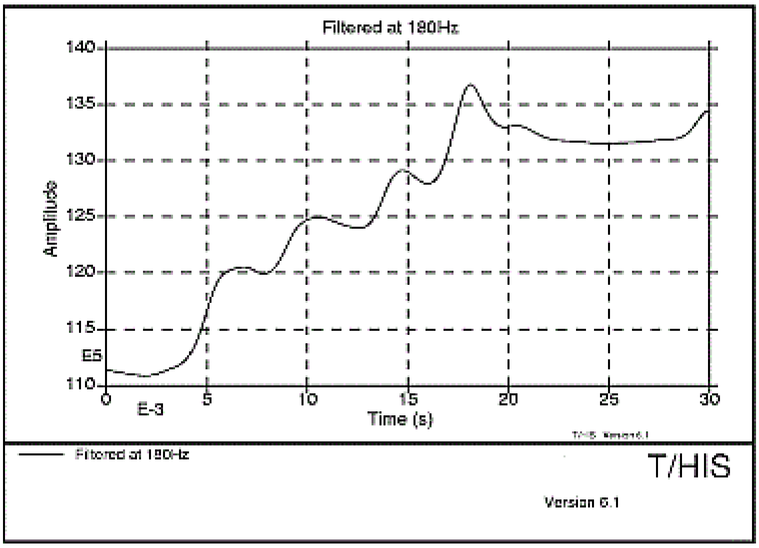
The cut-off frequency is the frequency at which the gain of the filter is -3dB (i.e. the magnitude of signals at this frequency is halved by the filter). The lower the frequency the less noise passes through; but any peaks in the signal tend to get reduced in magnitude and delayed in time.



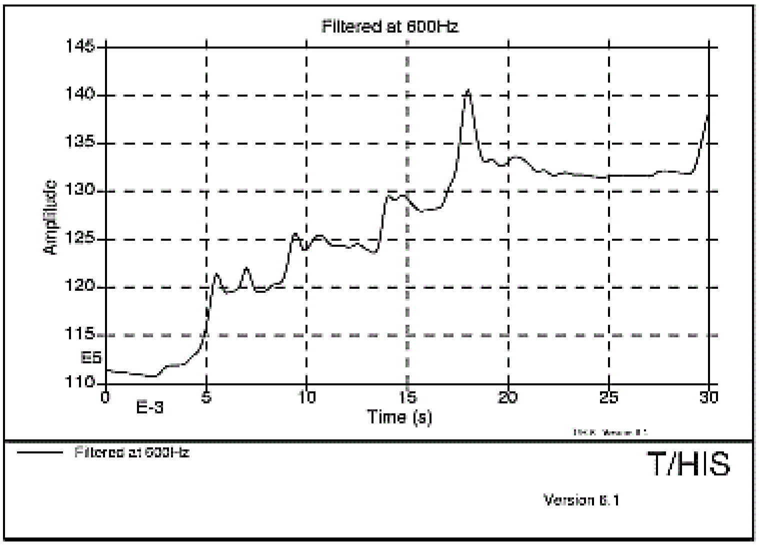
Unfiltered Signal



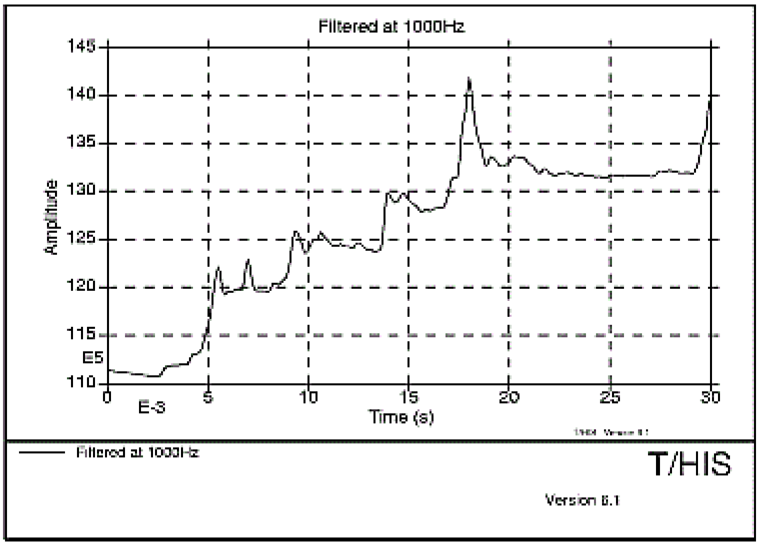
Filtered at 60Hz



Filtered at 180Hz



Filtered at 600Hz



Filtered at 1000Hz

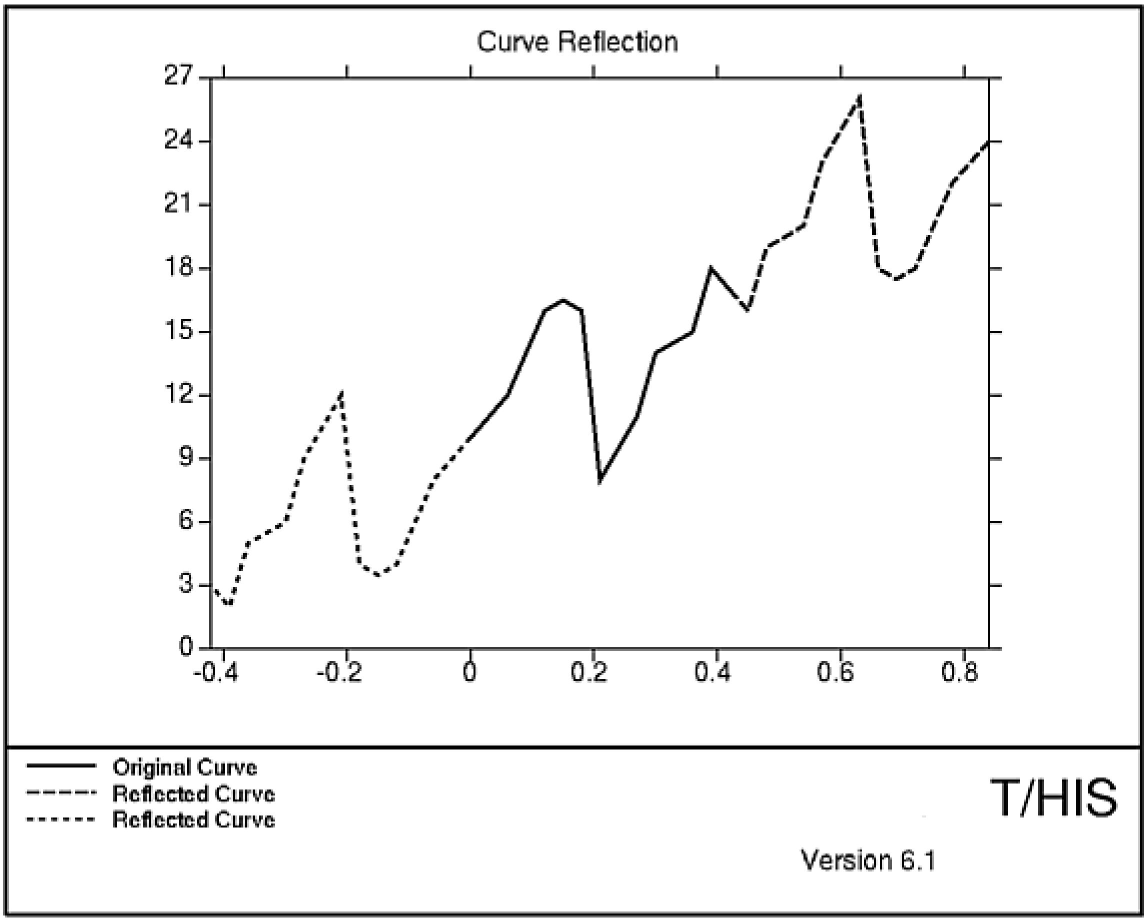
The above figures show examples of filtering frequency using the four standard SAE filters (60, 180, 600 and 1000 Hz cut-off frequencies: see below). These show clearly how the original signal is smoothed.

13.4.3. Butterworth Filter Implementation

Butterworth Filter Implementation

Two refinements have been incorporated:

- Reflection of beginning and end of curves to minimise end-effects of filtering (see the figure below).
- The curve is first passed forwards through the filter, then the resulting signal is passed through backwards. This procedure minimises phase change errors. The poles and zeros of the filter are calculated such that the desired cut-off frequency is achieved after two passes.



13.4.4. Standard SAE Filter Options

Standard SAE Filter Options

Channel Filter Classes 60, 180, 600 and 1000 are Butterworth filters with the following parameters:

Filter Class	Cut-off Frequency	Order
60	100Hz	2
180	300Hz	2
600	1000Hz	2
1000	1650Hz	2

The gain characteristics are compared with the limits given in BS AU228 in the following four figures.

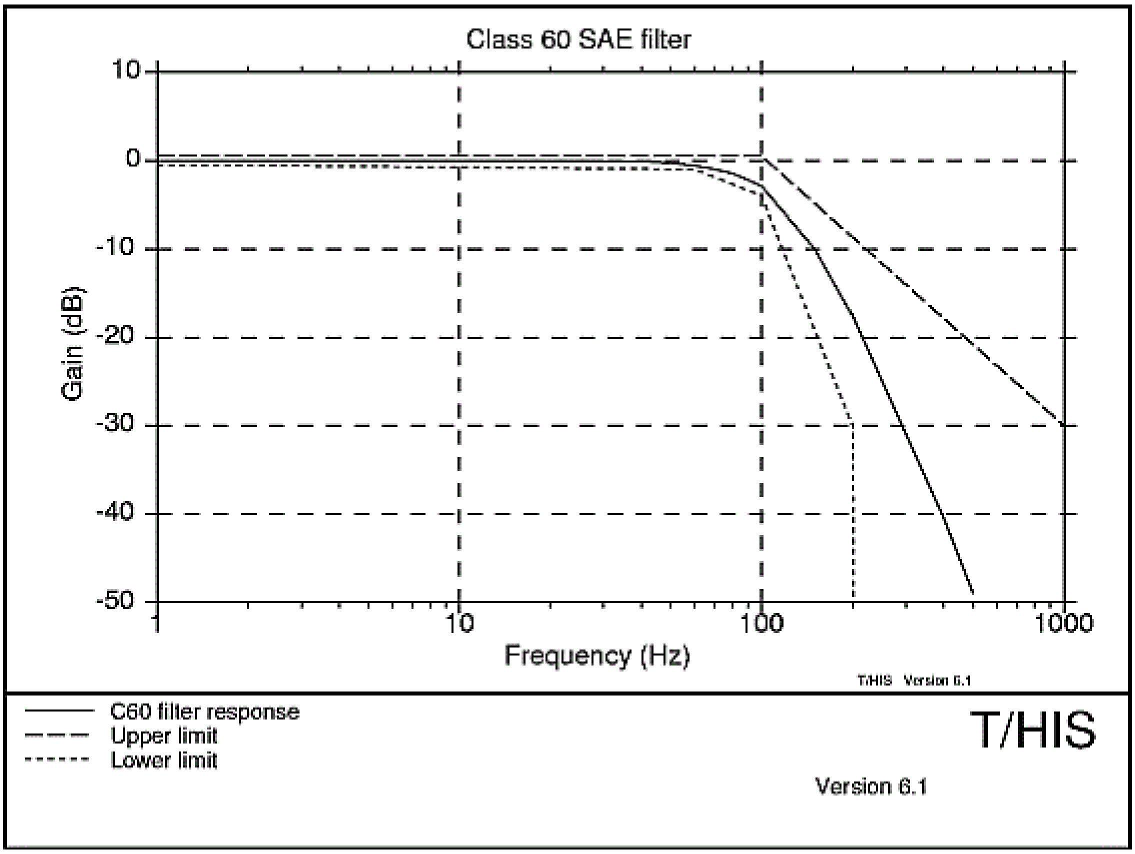
13.4.5. Standard FIR Filter Option

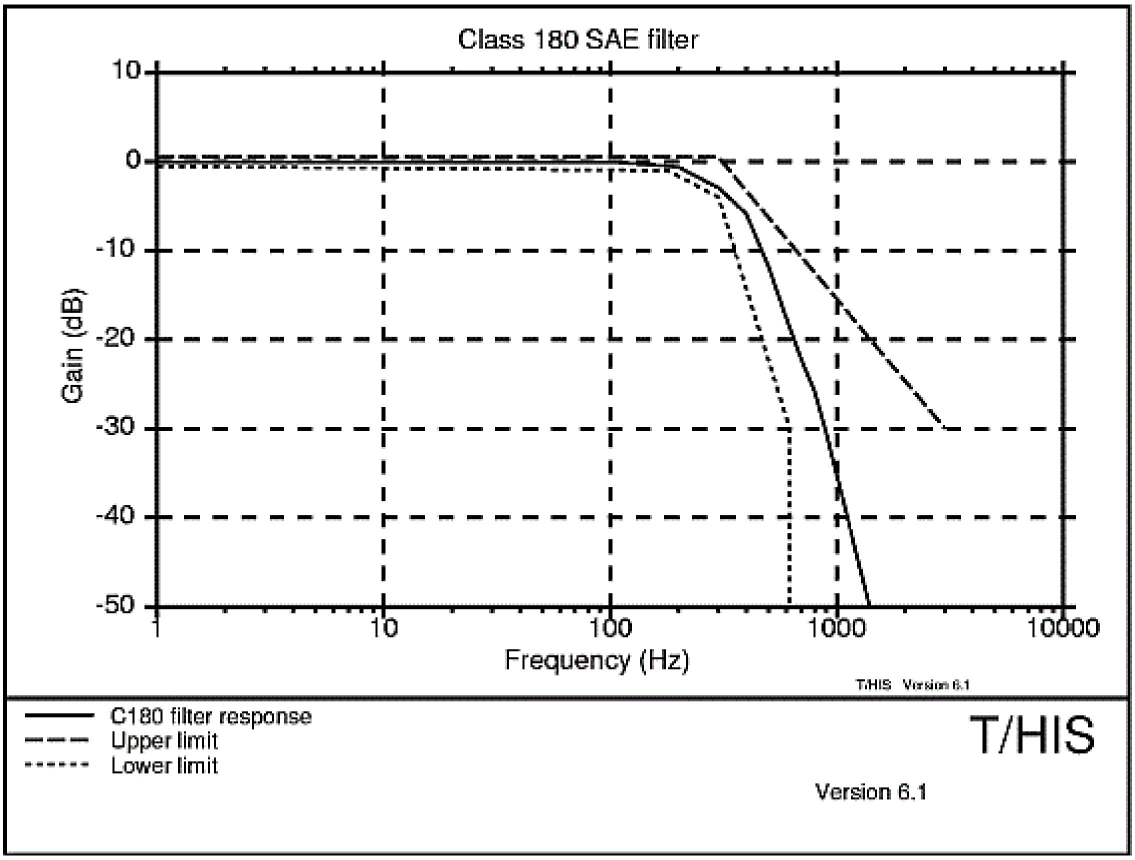
Standard FIR filter option

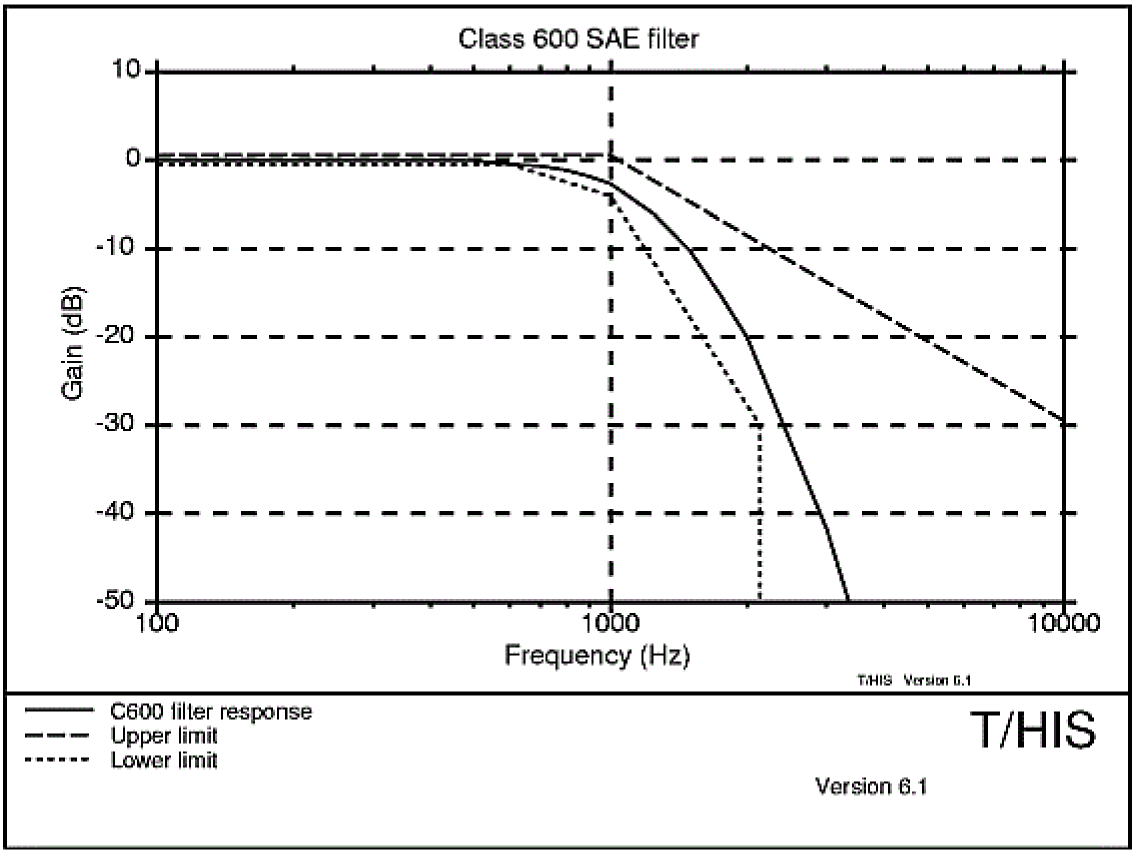
The FIR filter (Finite Impulse Response) is specified by NHTSA. It is used for filtering thoracic accelerations from side impact dummies; the filtered accelerations are then used in calculation of TTI (Thoracic Trauma Index). Its characteristics are:

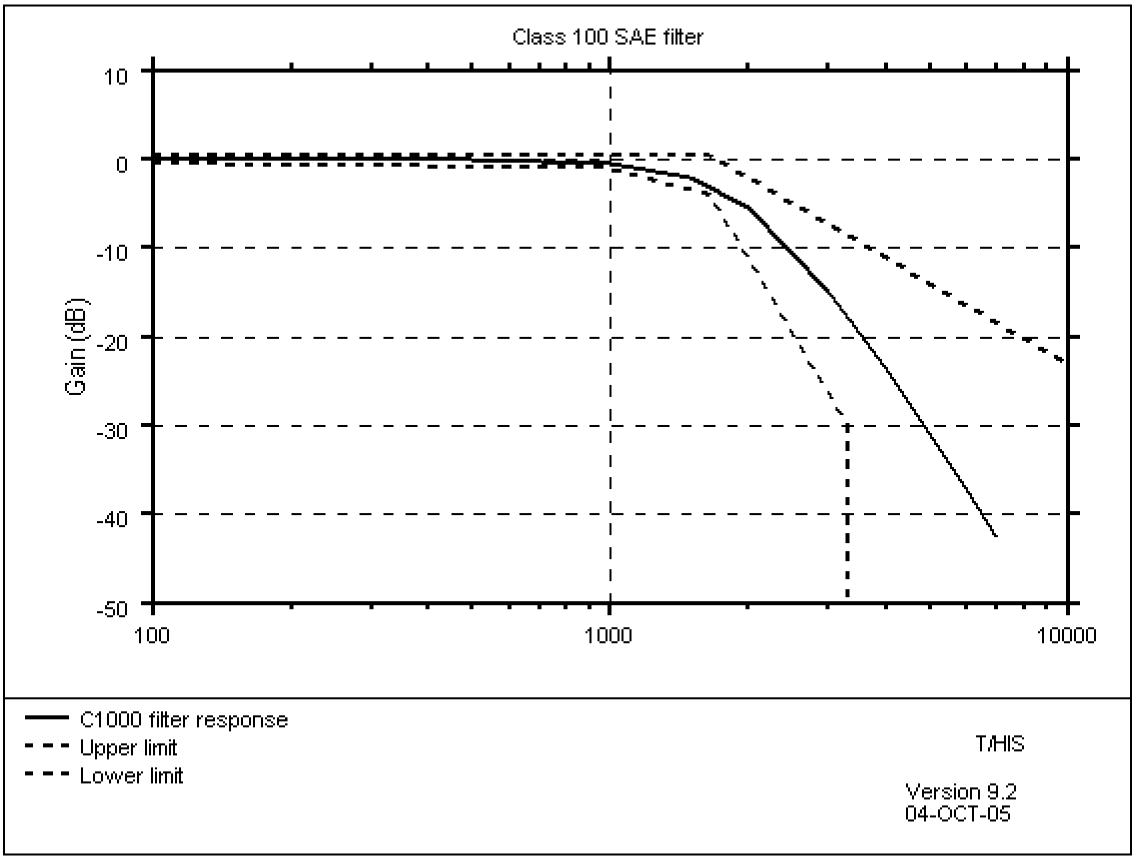
- A passband frequency of 100Hz.
- A stopband frequency of 189Hz.
- A stopband gain of -50dB.
- A passband ripple of 0.0225dB.

It is based on a standard Fortran programme available from NHTSA.




Class 180 SAE filter
10
0
-10
-20
-30
-40
-50
Gain (dB)
1
10
100
1000
10000
Frequency (Hz)
T/HIS Version 6.1
C180 filter response
Upper limit
Lower limit
T/HIS
Version 6.1


Class 600 SAE filter
10
0
-10
-20
-30
-40
-50
Gain (dB)
100
1000
10000
Frequency (Hz)
T/HIS Version 6.1
C600 filter response
Upper limit
Lower limit
T/HIS
Version 6.1

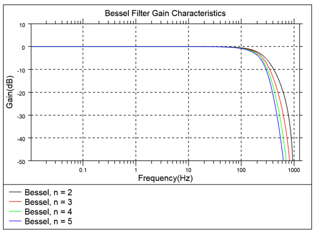

Class 100 SAE filter
10
0
-10
-20
-30
-40
-50
Gain (dB)
100
1000
10000
C1000 filter response
Upper limit
Lower limit
T/HIS
Version 9.2
04-OCT-05

13.4.6. Use of the Bessel Filter Option

Use of the Bessel Filter Option

The Bessel filter is a low pass filter with two input variables; order and cut-off frequency.

The order of the filter controls the roll-off rate, as shown here in the figure (right)



This is a 300Hz filter. It can be seen that higher orders attenuate the results more quickly: they have a higher roll-off rate.

This filter works in similar way to how the Butterworth filter works. The implementaion is the same with reflection and passing the curve through the filter forwards and backwards to minimise end-effects and phase change effects.

13.5. APPENDIX E - Injury Criteria

APPENDIX E - INJURY CRITERIA

T/HIS has the option to calculate two of the injury criteria that are used currently in occupant protection. These are the head impact criteria or HIC value and 3ms clip value. These criteria are defined as follows:

13.5.1. HIC Value

HIC Value

The HIC value is calculated from the resultant acceleration time history of the head centre of gravity filtered through a class 1000 filter. The HIC value is then calculated from;

$$\mathrm{HIC} = \left[\frac{1}{(t_2 - t_1)} \int_{t_1}^{t_2} a \, dt \right]^{2.5} (t_2 - t_1)$$

Where a is the acceleration expressed in g, and t_1 and t_2 are any two points in time. It is now usual for an upper limit on the range t_2-t_1 of 36ms to be applied.

13.5.2. 3ms Clip

3ms Clip

The 3ms clip value is the maximum value of acceleration that is exceeded for a period of not less than 3 ms. This is not an easily comprehended definition: the following may be of more use:

(1) At each time point T, take the interval (T to T+3ms);

(2) In this interval find the **lowest** acceleration value;

(3) The "3ms Clip" value is the interval (T to T+3ms) which has the **largest** "lowest" value as calculated in (2) above.

So, perhaps, a better definition might be: "the 3ms interval with the highest lowest acceleration value".

13.5.3. Viscous Criteria

Viscous Criteria

The VC value is calculated from a compression time history using the following formula (the values of the constants A and B assume the compression is in metres);

$$\mathrm{VC} = \mathrm{A}\left[\mathrm{V}_{(t)}\mathrm{C}_{(t)}\right]$$

$$\text{where} \quad \mathrm{C}_{(t)} = \frac{\mathrm{D}_{(t)}}{\mathrm{B}}$$

$$\mathrm{V}_{(t)} = \frac{8\left[\mathrm{D}_{(t+1)} - \mathrm{D}_{(t-1)}\right] - \left[\mathrm{D}_{(t+2)} - \mathrm{D}_{(t-2)}\right]}{12dt} \quad (\text{ECER95 regulations})$$

$$\mathrm{V}_{(t)} = \frac{d\mathrm{D}}{dt} \quad (\text{IIHS regulations})$$

$$\mathrm{D}_{(t)} = \text{Rib Compression}$$

$$\mathrm{A} = \text{Constant } (1.3 \text{ frontal}, 1.0 \text{ side})$$

$$\mathrm{B} = \text{Constant } (0.229 \text{ frontal}, 0.140 \text{ side})$$

13.5.4. Acceleration Severity Index

Acceleration Severity Index

The ASI value is calculated from 3 acceleration time histories using the following fomula;

$$\mathrm{ASI}_{(t)} = \left[\left(\frac{ax}{xl} \right)^2 + \left(\frac{ay}{yl} \right)^2 + \left(\frac{az}{zl} \right)^2 \right]^{0.5}$$

Where :	ax,ay,az	are the X,Y,Z accelerations of the vehicle: - for the 1998 calculation (BS EN 1317-1:1998) they are averaged over a 50ms moving interval. - for the 2010 calculation (BS EN 1317-1:2010) they are passed through a four-pole phaseless Butterworth filter with a 13Hz cut-off frequency.
	xl,yl,zl	are acceleration limits xl = 12g yl = 9g zl = 10g.

The acceleration input curves should be in units of g. If the input curves are in any other unit a conversion factor can be input to convert back to g.

When selecting input curves it is assumed that the X curve is numerically the first curve (the one with the lowest id) of the ones selected and the Z curve is the last. If they are in a different order then the acceleration limits can be modified to reflect the different order. For more information on ASI see BS EN 1317-1.

NOTE: For the BS EN 1317-1:2010 calculation T/His assumes the curves have been filtered through a Class 180 filter and padded with +/-0.5seconds of data as per the specification.

13.5.5. Theoretical Head Impact Velocity & Post Impact Head Deceleration

Theoretical Head Impact Velocity & Post Impact Head Deceleration

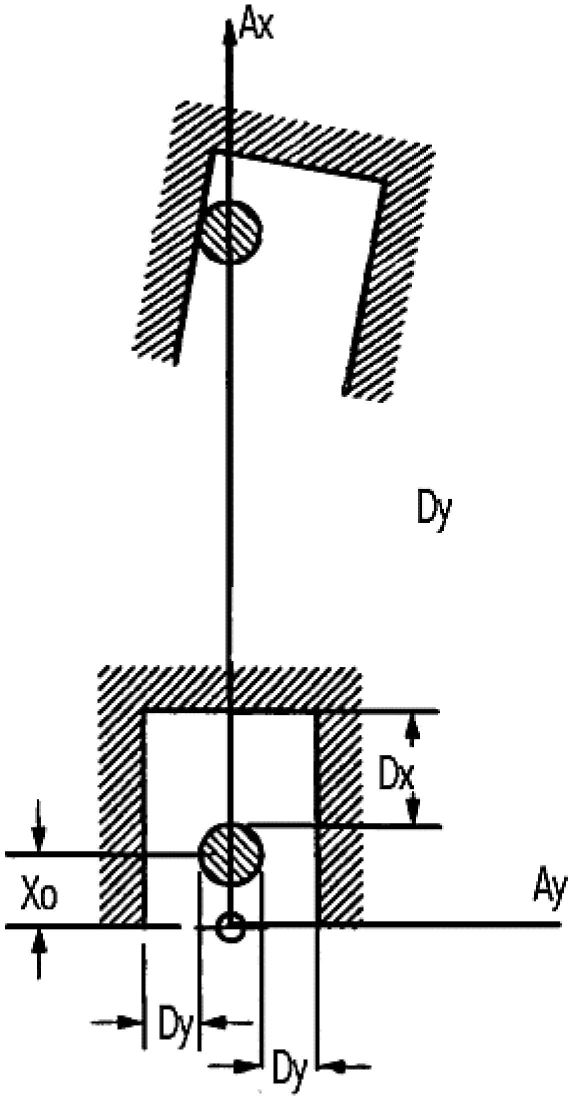
The theoretical head impact velocity concept has been developed for assessing occupant impact severity for vehicles involved in collisions with road vehicle restraint systems. The occupant inside the vehicle is considered to be a freely moving object that, as the vehicle changes its speed due to the contact with the restraint system, continues moving until it strikes the vehicle interior. The velocity magnitude at the time of impact with the vehicle interior is considered to be a measure of the vehicle to vehicle restraint system impact severity.

After impact the head is assumed to continue moving with the vehicle during the rest of the impact event. The post impact head deceleration (PHD) is calculated as the peak value using a 10ms moving average of the resultant vehicle acceleration after the THIV impact.

The THIV calculation requires the following inputs

- Horizontal Vehicle Acceleration Time History (Ax)
- Lateral Vehicle Acceleration Time History (Ay)
- Yaw Rate Time History
- Horizontal Distance from the occupants head to vehicle (Dx)
- Lateral Distance from the occupants head to vehicle (Dy)
- Initial X coordinate relative to CofG (X0)

The units for these fields should be consistent with the units that the model is in. For example, If the accelerations are in m/s^2 then values for Dx, Dy and X0 should be in metres. The Yaw Rate Time History curve is expected to be in radians/s



For more information on THIV and PHD see BS EN 1317-1.

13.5.6. Biomechanical neck injury predictor (NIJ)

Biomechanical neck injury predictor (NIJ)

The biomechanical neck injury predictor is a measure of the injury due to the load transferred through the occipital condyles. Its calculation combines the neck axial force and the flexion/extension moment about the occipital condoyles.

It is used in association with the USSID dummy for standard American frontal impact tests.

The shear force (Fx), axial force (Fz) and bending moment (My) are measured by the dummy upper neck load cell for the duration of the crash, using force and moment definitions consistent with SAE J221/1. T/HIS will caculate the bending moment using the equation:

My = My' - e Fx

Where e is the e distance specified in the input window, Fx is the shear force.

Shear force, axial force and bending moment must be filtered using an SAE Channel Frequency Class 600 filter (C600) for the purposes of calculation.

During the collision, the Axial Force (Fz) can be in either tension or compression whilst the occipital condyle bending moment (Mocy) can be in either flexion or extension. This results in 4 possible loading conditions corresponding to the 4 curves output by T/HIS; tension-extension (Nte), tension-flexion (Ntf), compression-extension (Nce), and compression-flexion (Ncf). At each point in time only one of these 4 conditions can be met, hence the NIJ value is calculated for that condition and the value for the other 3 conditions is considered a value of zero..

The expression for calculating each NIJ loading condition is given by:

NIJ = (Fz/Fzc) + (Mocy/Myc)

where Fz and Mocy are as defined above, Fzc and Myc refer to the axial force and Bending moment critical values, given below:

The values of Fzc and Myc vary depending on the occupant, the occupants position and the sign of **Fz** and **Mocy**

For the dummy to pass the test, the following conditions must be met:

(i) None of the 4 NIJ values may exceed 1.0 at any time during the event

(ii) Peak Tension Force (Fz), measured at the upper neck load cell, may not exceed the specific dummy's limit (e.g. 2070N for the Hybrid III small female) at any time

(iii) Peak Compression Force (Fz), measured at the upper neck load cell, may not exceed the specific dummy's limit (e.g. 2520N for the Hybrid III small female) at any time

For more information on the use and calculation of NIJ, refer to the FMVSS 208 document

13.5.7. The Thoracic Trauma Index (TTI)

The Thoracic Trauma Index (TTI)

The Thoracic Trauma Index is used as a predictor of thoracic injury severity in the USSID dummy in standard American Side Impact tests.

The Index considers both rib and Thorax acceleration in an impast.

The expression for calculating TTI is given by:

TTI = (G(R) + G(LS))/2

Where G(R) is the greater of the peak accelerations of either the upper or lower rib, expressed in g, and G(LS) is the peak acceleration in the lower spine (T12), expressed in g.

For the dummy to pass the test, the following conditions must be met:

(i) The TTI value must not exceed;
(a) 85g for a passenger car with 4 side doors, and for any multipurpose vehicle, truck or bus
(b) 90g for a passenger car with 2 side doors

(ii) The peak lateral acceleration of the pelvis shall not exceed 130g

(iii) Any side door, struck by the moving deformable barrier, shall not seperate totally from the car.

(iv) Any door not struck by the moving deformable barrier must meet the following requirements;
(a) The door chall not disengage from the latched position
(b) The latch shall not seperate from the striker
(c) The hinge components shall not seperate from each other or from their attachment to the vehicle
(d) Neither the latch nor the hinge systems of the door shall pull out of their anchorage

For more information on the use and calculation of TTI, refer to the FMVSS 214 document

13.6. APPENDIX F - Curve Correlation

APPENDIX F – Curve Correlation

T/HIS provides a number of ways to determine a measure for the degree to which two curves match i.e. correlation. Typically one curve is a reference curve that might come from physical testing, the other is the comparison curve that may be the result of a simulation of the scenario that generated the reference curve.

The following sections describe each of the tools and functions available in T/HIS:

- CORA An implementation of the methodology used by the Partnership for Dummy Technology and Biomechanics (PDB) software CORA (**COR**ellation and **A**nalysis) [F1], [F2]
- ISO 18571 An implementation of the calculations described in ISO 18571 [F3]
- MADM An implementation of the Minimum Area Discrepancy Method (MADM) [F4], [F5]
- COR1 and COR2 The T/HIS correlation functions COR1 and COR2
- COR3 The T/HIS correlation function COR3
- WIF The T/HIS correlation function that implements the weighted integrated factor method

CORA implementation

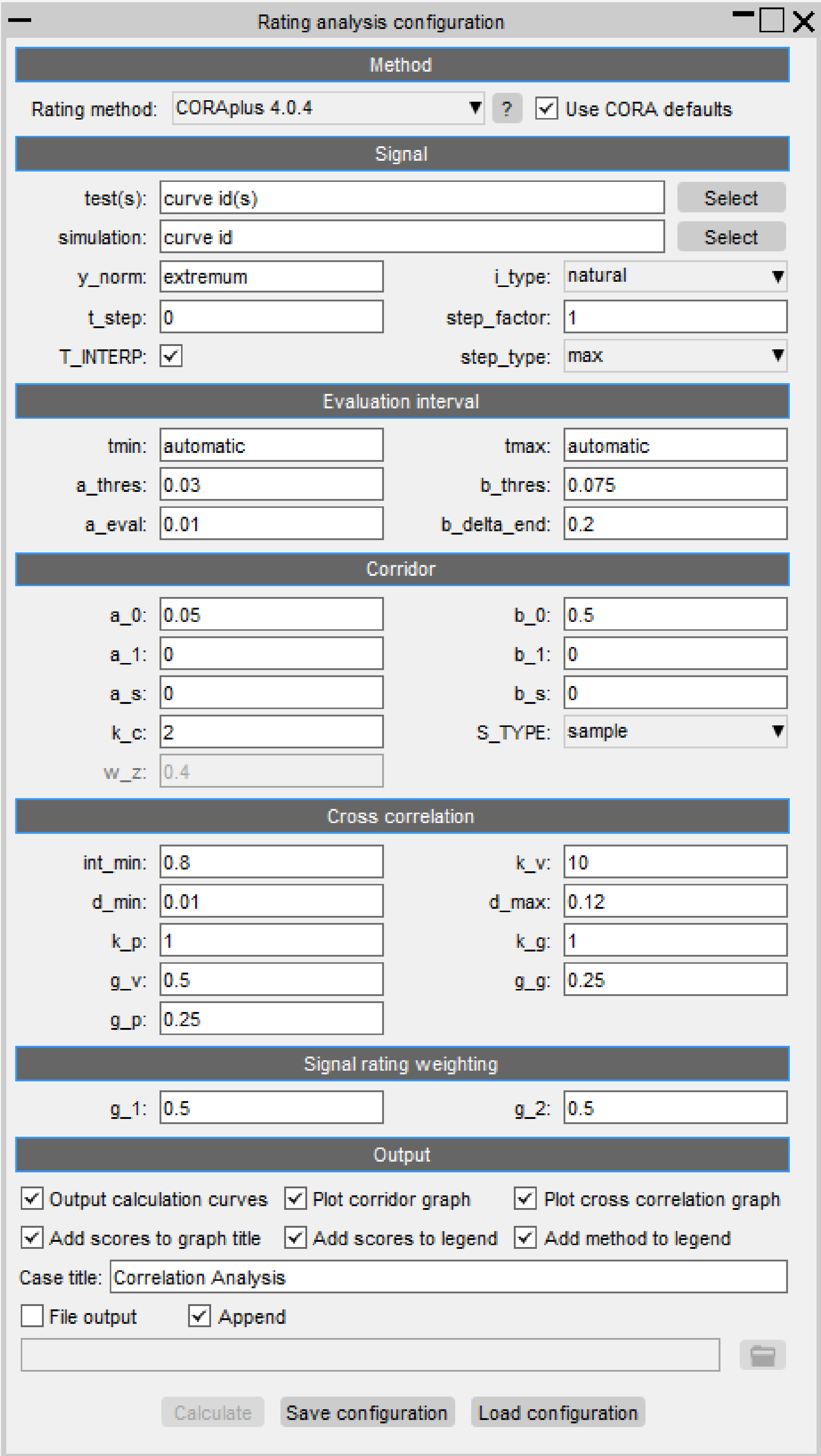
T/HIS implements the part of the PDB CORA method in which four measures of correlation are calculated and combined to give a total signal rating. One of these measures is made by examining how the curve fits within a defined corridor around the reference curve. The result is the **corridor metric**, with a value between zero and one. The remaining three measures are determined from properties related to a phase shift applied to the reference curve in order to achieve maximum cross-correlation. These three measures are each given a weighting and summed to give a **cross-correlation metric**, with a value between zero and one. The total signal rating is then determined from the weighted sum of the corridor and cross-correlation metrics.

Pre-requisites

- In order to achieve a meaningful correlation rating, the input curves should have a minimal amount of noise. This can be achieved using the filtering capabilities of T/HIS. It is assumed that any signal filtering required has been carried out prior to the use of CORA
- The curves should have x-values that increase monotonically
- The T/HIS CORA tool currently assumes the curves for analysis are available for selection within T/HIS

User interface

The T/HIS implementation of CORA is powered by JavaScript and can be accessed via **Tools → Automotive → CORA**. The options and parameters that are available for selection and modification are described in the hover text that appears when your mouse is positioned over the item of interest.


Rating analysis configuration
Method
Rating method: CORAplus 4.0.4
?
Use CORA defaults
Signal
test(s): curve id(s)
Select
simulation: curve id
Select
y_norm: extremum
i_type: natural
t_step: 0
step_factor: 1
T_INTERP:
step_type: max
Evaluation interval
tmin: automatic
tmax: automatic
a_thres: 0.03
b_thres: 0.075
a_eval: 0.01
b_delta_end: 0.2
Corridor
a_0: 0.05
b_0: 0.5
a_1: 0
b_1: 0
a_s: 0
b_s: 0
k_c: 2
S_TYPE: sample
w_z: 0.4
Cross correlation
int_min: 0.8
k_v: 10
d_min: 0.01
d_max: 0.12
k_p: 1
k_g: 1
g_v: 0.5
g_g: 0.25
g_p: 0.25
Signal rating weighting
g_1: 0.5
g_2: 0.5
Output
Output calculation curves
Plot corridor graph
Plot cross correlation graph
Add scores to graph title
Add scores to legend
Add method to legend
Case title: Correlation Analysis
File output
Append
Calculate
Save configuration
Load configuration

The features of the CORA menu control the correlation assessment as follows:

Feature	Description
Rating method	Allows the choice of CORAplus 4.0.4, ISO 18571:2014 or ISO 18571 Euro NCAP v1.0
Defaults	The default values as specified in [F1] (or [F3] for the ISO 18571 rating method)
Output calculation curves	Curves illustrating the details of the correlation calculation are created if this option is selected
Plot correlation graphs	Graph objects with respective curves for Corridor and Cross Correlation ratings. Scores and method can be automatically added to graph title or legend.
File output	A record of the calculation input and results is produced if this option is selected. The full path to the output file is required and a file dialogue can be used to select the location by using the **Save as** button. If you wish to record multiple correlation analyses in the same file, the **Append** option should be selected otherwise the output file will be overwritten.
Signal	Parameters and options related to the input signals and how they are pre-processed prior to the correlation rating being calculated
Evaluation interval	The parameters in this section allow you to specify the most relevant region of the signals for correlation analysis
Corridor	Parameters used in the calculation of the corridor metric
Cross correlation	Parameters used in the calculation of the cross-correlation metric
Signal rating weighting	The weighting factors applied to the corridor and cross-correlation metrics to obtain the overall signal rating
Save configuration	Save the current input configuration to a file (in json format) for retrieval using the **Load configuration** option.
Load configuration	To recreate a previous configuration a configuration file (in json format) can be loaded. The configuration file can be generated by using the **Save configuration** option.

Rating Methods

Three rating methods are available, described in the following sections:

1. CORAplus 4.0.4
2. ISO 18571:2014
3. ISO 18571 Euro NCAP v1.0

CORAplus 4.0.4

The T/HIS CORA tool was benchmarked against the PDB CORA software CORAplus 4.0.4. The benchmarks indicated that the results will not always be consistent, because of differences in the cross correlation shifted and truncated curve used to perform the rating. In March 2022, PDB acknowledged some issues with its implementation, and intends to make corrections once ISO 18571 is next updated.

ISO 18571:2014

The rating methods described in ISO 18751 [F3] have been implemented in T/HIS and are available by selecting the ISO 18571 rating method. This method in the T/HIS tool was also benchmarked against the PDB CORA software CORAplus 4.0.4. The benchmarks indicated that the results will not always be consistent, because of differences in the dynamic time warping algorithms used. In March 2022, PDB acknowledged these differences, which relate to issues in ISO 18571 itself. PDB intends to make corrections once ISO 15781 is next updated.

ISO 18571 Euro NCAP v1.0

The ISO 18571 Euro NCAP method is based on the ISO 18571 Validation Metrics Tool released by Euro NCAP, and aims to provide the same results. We checked the Euro NCAP Validation Metrics Tool and (as of 21 September 2023) identified two deviations from the ISO 18571 method, which we have also implemented in our "ISO 18571 Euro NCAP" method to provide consistent results:

1. Dynamic Time Warping Constraint

In the Phase score calculation, the Dynamic Time Warping (DTW) algorithm is implemented with a Sakoe-Chiba window constraint with relative window size 0.1, as opposed to the unconstrained DTW algorithm described in the ISO 18571 standard.

2. Alternative Slope Method

In the Slope score calculation, an alternative slope method is used. In the ISO 18571 standard, the gradient is calculated by dividing the signal into intervals (10 data points in length) and computing the average slope within each interval.

In the Euro NCAP method, the gradient of the original 10 kHz signal is computed using:

- forward difference for the first point (1st order accurate)

- backward difference for the last point (1st order accurate)
- central difference for all the other points (2nd order accurate)

such that the gradient curve is the same size as the input curve. The algorithm then applies a smoothing function to the gradient curve. For each point on the curve, the smoothing function takes the average of the point and the four neighbouring points on either side (reducing to zero points either side as one approaches the ends of the curve).

CORA scripting interface

A CORA or ISO 18571 assessment can be automated by importing the CORA module into your own T/HIS JavaScript. For instructions and a worked example, follow the CORA and ISO-18571 tutorial (**Help → Tutorials → CORA and ISO-18571**).

MADM Correlation tool

The MADM Correlation Tool in T/HIS is powered by JavaScript. The Minimum Area Discrepancy Method (MADM) is ideal for correlation between LS-DYNA simulations and physical tests when force versus deflection is the relationship of interest, and offers benefits over other correlation methods that focus on parameters versus time. The tool can be accessed by selecting **Tools → Automotive → MADM**.

Three input methods are available for providing average, lower, and upper curves for the MADM rating calculation:

1. **Specify average/lower/upper curves**
    - The user can directly specify average, lower, and upper curves.
    - These can be provided in the form of a CSV file, or by picking/selecting from T/HIS curves.
2. **Generate offset from average curve**
    - An average curve can be provided, and a corridor of uniform width generated around it. This width can be adjusted. Once a corridor is generated, it can be provided as the lower and upper curves.
    - The average curve can be provided in the form of a CSV file, or by picking/selecting from T/HIS curves.
3. **Generate average + corridor from dataset**
    - From provided force-time and deflection-time datasets (which should each consist of more than one curve), a mean average curve can be generated. A corridor can then be generated which varies in width (the variation depends on each point's standard deviation). The width can be controlled via the number of standard deviations used in the calculation. These curves can then be provided as the average, lower, and upper curves.

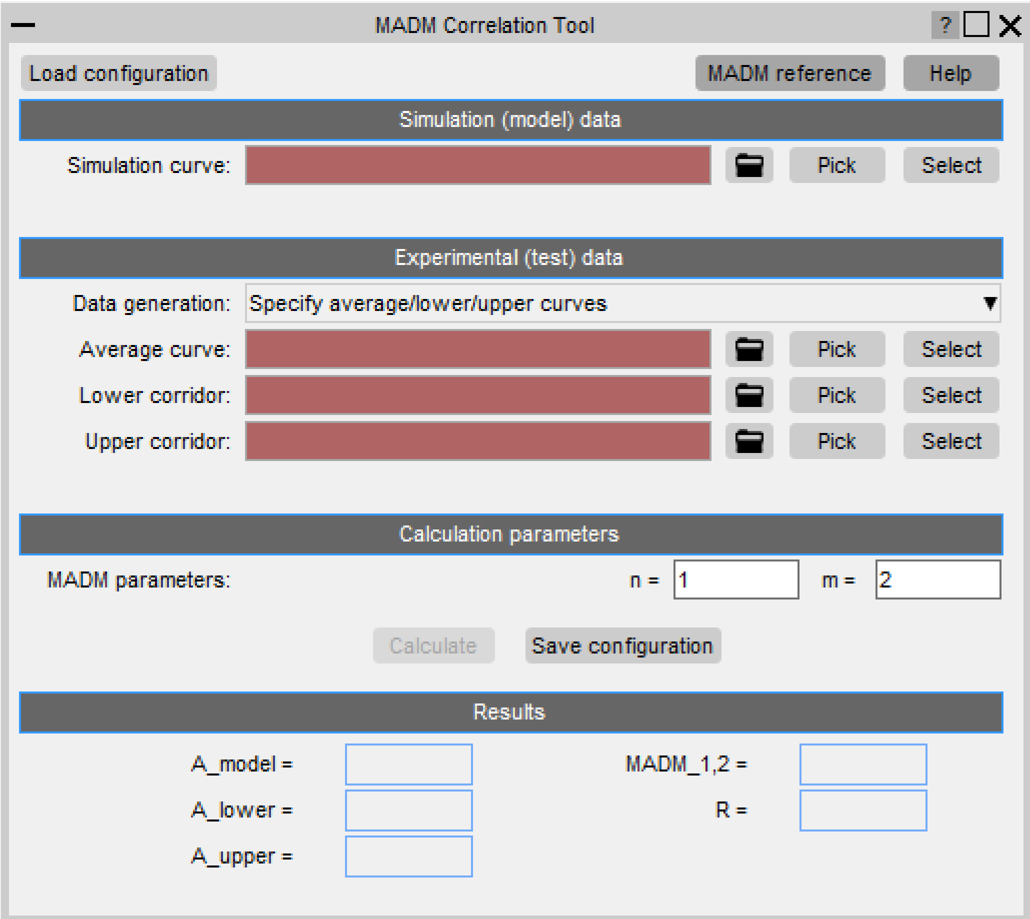
- The datasets can be provided in the form of multiple separate CSV files, or by selecting multiple T/HIS curves. Note that the dataset should have a uniform number of points, with regular time intervals throughout.

Note that all data provided should have the deflection along the x-axis, and the force along the y-axis. In addition, a corridor cannot be generated from data which self-intersects – in this case, the corridor should be generated separately by the user. Finally, a corridor cannot be generated from data with non-adjacent duplicate points.

Previously generated curves are deleted when the curves are re-generated, or when the input method is changed.

A MADM assessment can be automated by importing the MADM module into your own T/HIS JavaScript. For instructions and a worked example, follow the MADM tutorial (**Help → Tutorials → MADM Worked Example**).

For information on the Minimum Area Discrepancy Method itself, further help and references can be found in the tool menu:



MADM input configuration can be saved or loaded via the **Save configuration** and **Load configuration** buttons. The configuration is saved as a JSON file. The following table documents the configuration file properties:

Property	Description	Required?	Valid format
method	Method used for input curve generation. Will change what inputs are required.	Required for all methods.	'Can be "specifyCurves", "generateOffset", or "generateAverage".
curve_inputs.avg	Input average curve.	Required for methods "specifyCurves" and "generateOffset".	File path to a .cur or .csv file, or a valid curve ID.
curve_inputs.lower	Input lower curve.	Required for method "specifyCurves".	File path to a .cur or .csv file, or a valid curve ID.
curve_inputs.upper	Input upper curve.	Required for method "specifyCurves".	File path to a .cur or .csv file, or a valid curve ID.
curve_inputs.sim	Input simulation curve.	Required for all methods.	File path to a .cur or .csv file, or a valid curve ID.
curve_inputs.dt	Input deflection-time curves.	Required for method "generateAverage".	File path to a .cur or .csv files containing multiple curves, or an array of valid curve IDs.
curve_inputs.ft	Input force-time curves	Required for method "generateAverage".	File path to a .cur or .csv files containing multiple curves, or an array of valid curve IDs.
n	Input n value. Required for all methods. Defaults to 1.	Required for all methods.	A number or string of a number. Recommended to be between 0.5 and 3.
m	Input m value. Required for all methods. Defaults to 2.	Required for all methods.	A number or string of a number. Recommended to be between 1 and 2.
offset	Offset value of generated corridor.	Required for method "generateOffset".	A number or string of a number. Must be a real postive number.

half_width	Number of standard deviations of generated corridor half-width.	Required for method "generateAverage".	A number or string of a number. Must be a real postive number.
output_curves	File path that an output curve file will be written to.	Optional	File path to a .cur/.csv file.
output_image	File path that an output graph image will be written to.	Optional	File path to a .png file.
output_json	File path that an output results .json file will be written to.	Optional	File path to a .json file.

The MADM Correlation Tool was developed in collaboration with the University of Coventry [F4], [F5].

COR1 and COR2

The Correlation functions COR1 and COR2 provide a measure of the degree to which two curves match. When comparing curves by eye, the quality of correlation may be judged on the basis of how well matched are the patterns of peaks, the overall shapes of the curves, etc, and can allow for differences of timing as well as magnitude. Thus a simple function based on the difference of Y-values (such as T/HIS ERR function) does not measure correlation in the same way as the human eye. The T/HIS correlation function attempts to include and quantify the more subtle ways in which the correlation of two curves may be judged.

The correlation function may be applied to any two curves whose x-values increase monotonically (e.g. responses versus time). The results are independent of the units used, e.g. milliseconds or seconds are both acceptable. The sign of the y-values is not important.

Only the overlap time period is considered (i.e. the range of x-values for which both curves have a y-value). The time period (range of X-values) and maximum absolute Y-value are used to non-dimensionalise the curves such that x-values run from 0 to 1, and the maximum absolute y-value is 1.

Five measures of correlation are calculated. Each is given equal weighting. The final correlation score is given as a percentage - two identical curves would score 100%.

The first two measures require identification of peaks in the curves. An unlimited number of peaks in each curve will be considered. A peak is defined as a local maximum (or in the case of negative y-values a minimum), satisfying the following criteria:

- Absolute y-value at least 0.5
- Separated from any larger peak by a trough (local minimum) at least 0.2 deep.

Peaks of positive or negative signs are considered. Peaks are matched only against peaks of the same sign in the other curve.

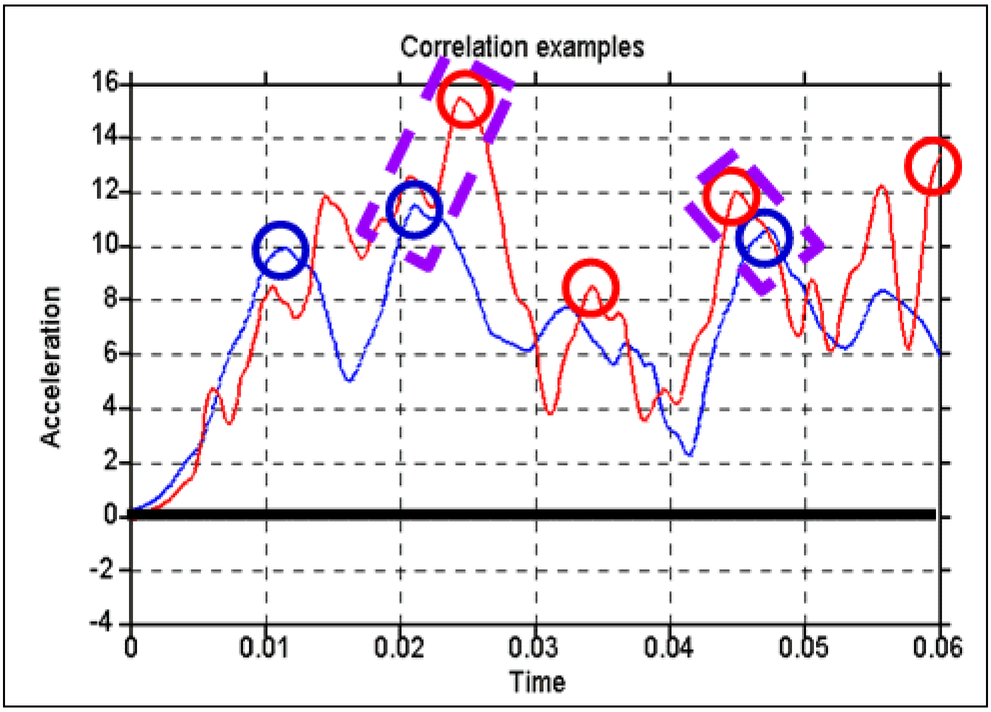
Measure 1 – Peak values

For each identified peak in Curve A, find the maximum value in Curve B within the same time range for which the value in Curve A is within a tolerance of the peak value. Points are lost according to the error in y-values compared to a tolerance limit. Repeat for peaks in curve B against values in Curve A.

This measure allows for the situation where curves are similar but the peaks are more strongly delineated in one of the curves, such that the program does not recognise the other curve as having a peak in that location.

Measure 2 – Peak matching

For each identified peak in Curve A, find the closest identified peak in Curve B. Points are lost according to the largest error (timing or y-value) compared to tolerance limits; points are also lost if there is no corresponding peak in Curve B. Repeat for Curve A peaks matched against those of Curve B.



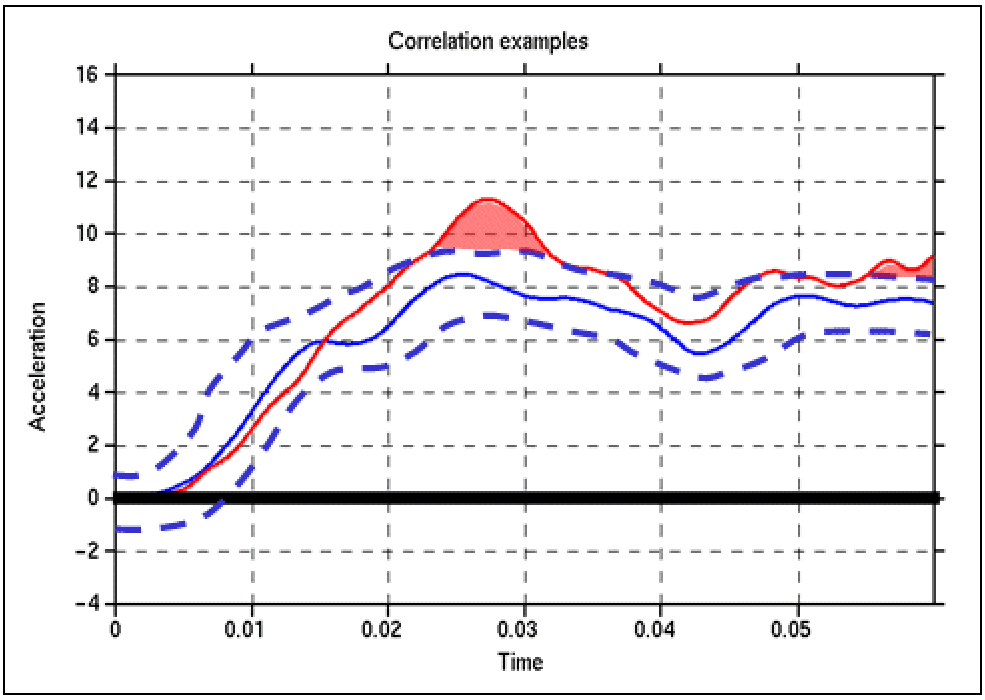
This measure picks up matching of primary and secondary peaks in the curves, which may correspond to physical events.

Measure 3 – Area matching

The integral of each curve is calculated by summing the area of the curve above y=0 and the absolute area of the curve below y=0. Points are lost according to the difference compared to a tolerance limit.

Measure 4 – Curve shape (low frequency excursion)

The curves are filtered. A band is drawn around filtered curve A (using positive and negative offsets in x and y). The area of excursions of filtered Curve B outside the band is calculated. Points are lost according to the excursion area compared to a tolerance limit. The process is repeated for filtered Curve A excursions from a band drawn around filtered curve B.



Measure 5 – Curve shape (full curve)

The same as Measure 4 except that the curves are not filtered and different tolerance limits and band sizes may be used.

Output

T/HIS prints the overall correlation percentage and the marks from each measure to the screen or to a text file. A new curve is created from each input curve showing the identified peaks (used in measures 1 and 2). As the same curve could be used as input to multiple correlations the correlation percentage is stored internally in T/HIS with the 2 output curves NOT the input curves.

The correlation percentage can be accessed from within FAST-TCF scripts by requesting the "correlate" property for either of the 2 ouptut curves.

e.g. `operation correlate strict` `curve_1 curve_2` `tag curve_3 curve_4`

Calculate correlation between "curve_1" and "curve_2". Tag the curves containing the peaks as "curve_3" and "curve_4"

`tab output.txt` `curve_3` `correlate`

Output the curve correlation value from "curve_3" to the file "output.txt"

`taba output.txt` `curve_4` `correlate`

Append the curve correlation value from "curve_4" to the file "output.txt"

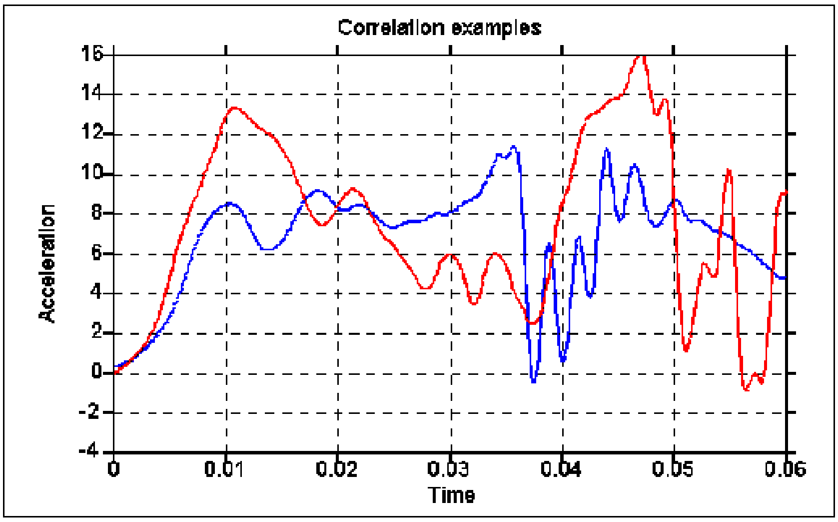
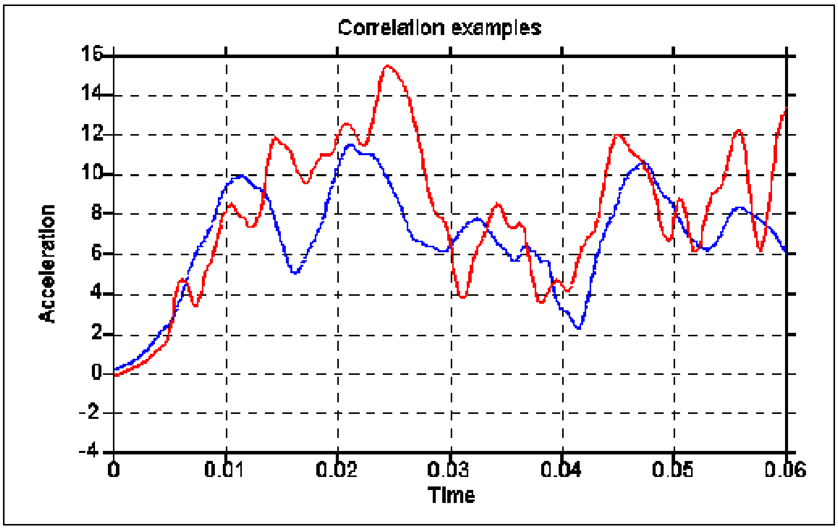
Selection of Parameters

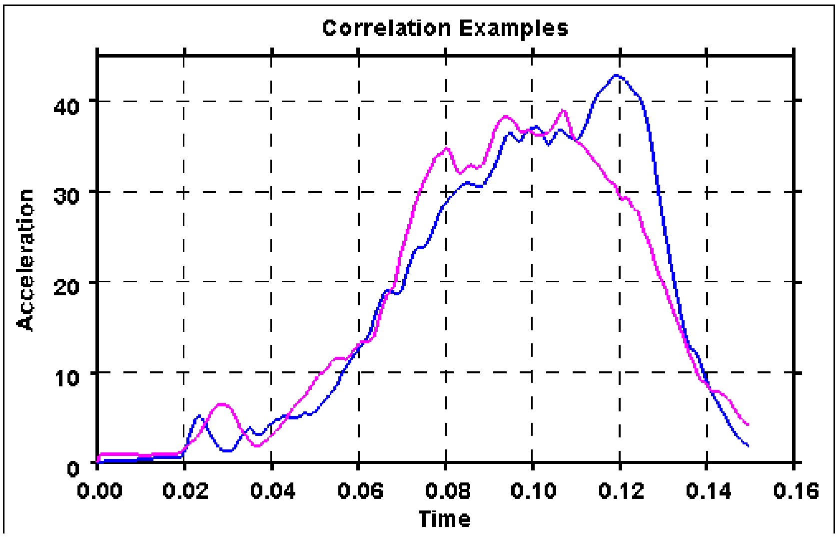
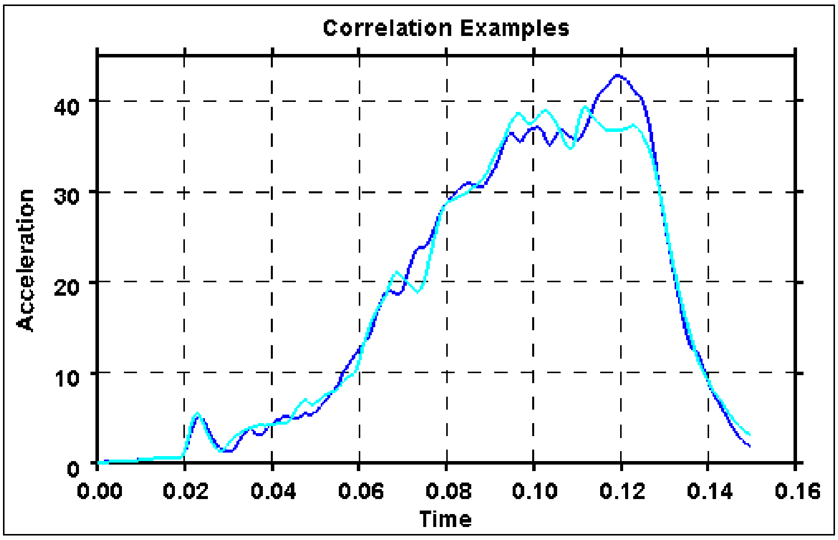
The Correlation algorithm has many tolerance limits and other inputs. Two sets of these parameters have been pre-selected, to offer strict or less strict judgement of correlation (buttons COR1 and COR2 in the Automotive menu). The parameters selected are:

Criterion	Decription	COR1 Value	COR2 Value
Peak matching	Fraction difference in timing that scores zero points for this peak	0.2	0.4
Peak matching and peak values	Fraction difference in value that scores zero points for this peak	0.25	0.5
Area matching	Fraction difference in integral that scores zero points	0.3	0.5
Curve shape (low frequency trend)	Size of tolerance band in X and Y, as fractions of the curve extent in X and Y	0.025	0.05
Curve shape (low frequency trend)	Excursion area fraction scoring zero points	0.1	0.2
Curve shape (full curve)	Size of tolerance band in X and Y, as fractions of the	0.025	0.05

	curve extent in X and Y		
Curve shape (full curve)	Excursion area fraction scoring zero points	0.2	0.4

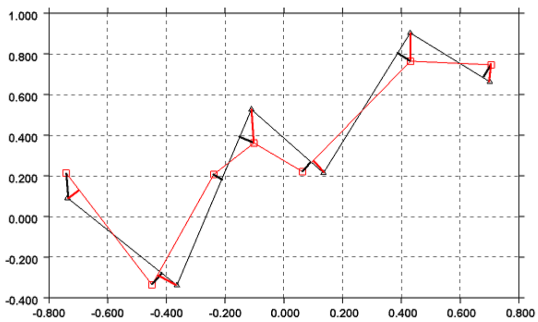
It is expected that, if COR1 rates Curves A and B as better correlated than C and D, then COR2 would also rate the pairs of curves in the same order. The percentage correlation would be greater in each case from COR2 than from COR1. COR1 will provide a greater difference (discrimination) between well-correlated and very well-correlated pairs of curves; while COR2 will provide greater discrimination between averagely-correlated and poorly-correlated pairs of curves. The purpose of offering both versions of the correlation function is to allow the user to select a calibration of the function appropriate to the typical input curves used.

Examples	COR1	COR2
	17%	42%
	27%	62%

<table>
<tr><td>
</td><td>70%</td><td>88%</td></tr>
<tr><td>
</td><td>84%</td><td>92%</td></tr>
</table>

COR3

The Correlation function COR3 provides another measure of the degree to which two curves match based on the distance between the two curves.



This function first normalises the curves using two factors, specified either by the user or defaults calculated by the program (the maximum absolute X and Y values of both graphs).

For each point on the first normalised curve, the shortest distance to the second normalised curve is calculated (the thick black lines on the image above). The root mean square value of all these distances is subtracted from 1 and then multiplied by 100 to get an index between 0 and 100.

The process is repeated along the second curve (the thick red lines show the distances) and the two indices are averaged to get a final index. The higher the index the closer the correlation between the two curves.

Note that the choice of normalising factors is important. Incorrect factors may lead to a correlation index outside the range of 0 to 100.

WIF

The Correlation function WIF provides another measure of the degree to which two curves match. It uses the Weighted Integrated Factor method:

$$crit = 1 - \sqrt{\frac{\sum \max\left(f[n]^2, g[n]^2\right) \cdot \left(1 - \frac{\max\left(0, f[n] \cdot g[n]\right)}{\max\left(f[n]^2, g[n]^2\right)}\right)^2}{\sum \max\left(f[n]^2, g[n]^2\right)}}$$

References

[F1] Thunert (GNS mbH), *CORAplus release 4.04 user's manual*, May 2017

[F2] Gehre (PDB); Gades (Volkswagen AG); Wernicke (BMW Group), *Objective rating of signals using test and simulation responses*, Paper number 09-0407, 2009.

[F3] BSI Standards publication, *Road vehicles - Objective rating metric for non-ambiguous signals*, PD ISO/TS 18571:2014.

[F4] Bastien, C., Diederich, A., Christensen, J., & Ghaleb, S. (2021). *Improving Correlation Accuracy of Crashworthiness Applications by Combining the CORA and MADM Methods.* Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering. https://journals.sagepub.com/doi/10.1177/09544070211069666

[F5] Peres, J, Bastien, C, Christensen, J & Asgharpour, Z 2019, *A Minimum Area Discrepancy Method (MADM) for Force Displacement Response Correlation*, Computer

Methods in Biomechanics and Biomedical Engineering, vol. 22, no. 11, GCMB-2018-045, pp. 981-996. https://doi.org/10.1080/10255842.2019.1610745

13.7. APPENDIX G - The ERROR Calculation

APPENDIX G - The ERROR Calculation

The ERROR function outputs a number of values to indicate the degree of correlation between 2 curves. The function requires two input curves

- A reference curve to compare against (the first curve selected)
- The curve to compare against the reference

Once 2 curves have been selected the a check is carries out to see if the two curves contain the same number of points and if the range of x-axis values the same for the two curves. If any inconsistencies are found then a warning message is generated.

The following values are then calculated

Maximum difference and time of variation

Maximum difference as a %age of the reference value at the same time

Maximum difference as a %age of the peak reference value

Average difference

Average difference as a %age of the peak reference value

$$\text{Area weighted difference} = \left(\frac{\int |y_r - y_c| dx}{\frac{1}{2}\left(\int y_r dx + \int y_c dx \right)} \right)$$

$$\text{where} \qquad y_r = \text{Reference Curve}$$

$$y_c = \text{Data Curve}$$

T/HIS Regression coefficient.

$$R^2 = \left[1 - \frac{\sum (y_c - y_a)^2}{\sum y_a{}^2 - \frac{\sum y_a{}^2}{n}} \right]$$

y_c = Data Curve

y_a = Average of Data and Reference Curve = $\frac{1}{2}(y_c + y_r)$

n = Number of Data Points

This is a value between 0 and 1 where 1 means 100% correlation

13.8. APPENDIX H - The "oa_pref" Preference File

APPENDIX H - The "oa_pref" preference file

This file contains code-specific preferences that can be used to modify the behaviour of T/HIS. It is optional and, where entries (or the whole file) are omitted T/HIS will revert to its default settings.

"oa_pref" naming convention and locations

The file is called "oa_pref. It is looked for in the following places in the order given:

- The optional administration directory defined by the environmental variable (**`$OA_ADMIN`** or **`$OA_ADMIN_xx`** where xx is the release number).
- The site-wide installation directory defined by the environment variable (**`$OA_INSTALL`**)
- The user's home directory: **`$HOME`** (Unix/Linux) or **`%USERPROFILE%`** (Windows)
- The current working directory

See Installation organisation for an explanation of the directory structure.

All four files are read (if they exist) and the last preference read will be the one used, so the file can be customised for a particular job or user at will.
Files do not have to exist in any of these locations, and if none exists the programme defaults will be used.

On Unix and Linux:

> **`$HOME`** on Unix and Linux is usually the home directory specified for each user in the system password file.
> The shell command " **`printenv`** " (or on some systems " **`setenv`** ") will show the value of this variable if set.
> If not set then it is defined as the " **`~`** " directory for the user. The command " **`cd; pwd`** " will show this.

On Windows:

> **`%USERPROFILE%`** on Windows is usually **`C:\Documents and Settings\ <user id> \`**
> Issuing the " **`set`** " command from an MS-DOS prompt will show the value of this and other variables.

Generally speaking you should put

- Organisation-wide options in the version in $OA_ADMIN_xx and/or $OA_INSTALL,
- User-specific options in $HOME / %USERPROFILE%
- Project-specific options in the current working directory.

The file contains preferences for the SHELL (lines commencing shell*), THIS (lines commencing this*), D3PLOT (lines commencing d3plot*), PRIMER (lines commencing primer*)and REPORTER (lines commencing reporter*). All lines take the format <preference name> <preference value>.

The general copy of the preference file should be present in the $OA_ADMIN_xx and/or $OA_INSTALL directory. This should contain the preferences most suitable for all software users on the system.

An individual's specific preferences file can be stored in the individual's home area. This can be used to personally customise the software to the individual's needs.

Whenever one of the programs whose preferences can be stored in the oa_pref file is fired up, the program will take preferences first from the general preference file in the $OA_ADMIN_xx directory (if it exists) then the $OA_INSTALL directory, then from the file in the user's home area, then from the current working directory.

Preferences defined in the general oa_pref file can be modified in the user's personal file but they can't be removed by it.

From version 9.4 onwards preferences can be locked. If a preference is locked it cannot be changed in an oa_pref file in a more junior directory. To lock a preference use the syntax **'this#'** rather than **'this*'** .

13.8.1. The Interactive Preferences Editor

The interactive Preferences Editor

You are free to edit oa_pref files by hand, but there is an interactive "Preferences Editor" that may be called from within T/HIS that makes the job much easier.

It is started by **Options, Edit Preferences** or through the Preferences Button in the Tool menu

The preferences editor reads an XML file that contains all possible preferences and their valid options, and allows you to change them at will. In this example the user is changing the background colour in D3PLOT.

Note that changes made in the Preferences editor will not affect the current session of D3PLOT, they will only take effect the next time it is run.

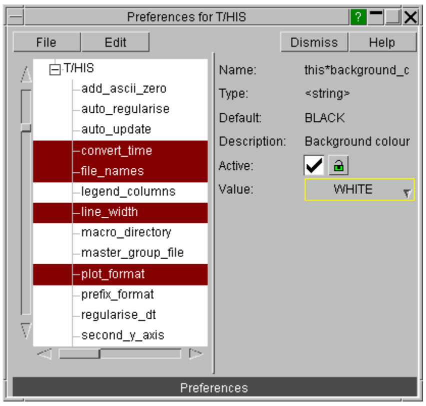
If you have write permission on the oa_pref file in the $OASYS directory you will be asked if you want to update that file, otherwise you will only be given the option of updating your own file in your $HOME / $USERPROFILE directory.

In this example the user is changing the background colour.

The option is "active" (ie present in the oa_pref file) and currently is set to WHITE.

Usage is:

- Select an option in the Tree on the left hand side
- Make it active / inactive
- If active select a value from the popup, or type in a value if necessary



The colour of the highlighting in the left hand side tree is significant:

Green	Means that the option has been read from your $HOME/$USERPROFILE file.
Red	Means that the option has been read from the $OA_INSTALL file.
Magenta	Means that the option had been read from the $OA_ADMIN file.

In either event, regardless of the data source, the updated option will be written to the file chosen when you started the preferences editor.

Because of the order of file reading (see above), and option read from the master $OASYS file, amended, and written to your local $HOME file will take precedence when you next run T/HIS.

13.8.2. Locking Preferences

Locking Preferences

From version 9.4 onwards preferences can be locked. Beside each option in the preference editor is a padlock symbol. If the symbol is green then the option is unlocked, if it is red then it is locked. If a preference option has been locked in a file that the user can not modify then an error message will be generated if the user tries to edit that option.

If a user manually edits the "oa_pref" file to try and set an option that has been locked in another preference file then the option will be ignored in the users preference file.

13.8.3. Format of the oa_pref File

Format of the oa_pref File

Entries are formatted in the following way:

<programme>*<option>: <setting>

For example: this*laser_paper_size: A4

The rules for formatting are:

- The **<programme>*<option>:** string must start at column 1;
- This string must be in lower case, and must not have any spaces in it.
- The **<setting>** must be separated from the string by at least one space.
- Lines starting with a " **#** " are treated as comments and are ignored.

(Users accustomed to setting the attributes of their window manager with the **.Xdefaults** file will recognise this format and syntax.)

"oa_pref" arguments valid for T/HIS

Preference	**Type**	**Description**	**Valid arguments**	**Default**
splash_screen_seen	<real>	Most recent version (as major.minor, eg 17.1) for which a splash screen has been seen		0.0
legend_layout	<string>	Layout of legend	COL_LIST, AUTO, OFF, FLOAT	AUTO
add_ascii_zero	<logical>	Automatically add point at time zero if required	TRUE, FALSE	FALSE
auto_regularise	<logical>	Always regularise curves before filtering	TRUE, FALSE	FALSE
auto_update	<logical>	Automatically replot graph after changing axis/title options	TRUE, FALSE	TRUE
write_checkpoint_files	<logical>	Record checkpoint files for the T/His session.	TRUE, FALSE	FALSE
checkpoint_dir	<string>	Directory for checkpoint files, or "none"		<none>

		to suppress them altogether		
show_checkpoint_files	<logical>	Show checkpoint playback panel upon T/His startup.	TRUE, FALSE	FALSE
curve_property_number_format	<string>	Number format option for curves	AUTO, SCIENTIFIC, GENERAL	SCIENTIFIC
curve_property_dec_places	<integer>	Number of decimal places to display for curves	0 - 9	3
error_handler	<string>	how to handle errors and exceptions	no_action, mini_dump, trap_continue, trace_exit	mini_dump
convert_time	<logical>	Automatically convert from ms->s when filtering	TRUE, FALSE	FALSE
file_names	<string>	Controls default file filters. LSTC = d3thdt*, xtfile*, OASYS/ARUP = *.thf, *.xtf	OASYS, ARUP, LSTC	OASYS
file_timeout	<integer>	Timeout period in seconds before T/HIS automatically closes LSDA(binout) and THF (d3thdt) files	0 - 300	10
iso_curve_labels	<string>	Curve label for ISO	CHANNEL_NAME, CHANNEL_CODE	CHANNEL_CODE
iso_list_labels	<string>	Default list labels in Read ISO panel	FILENAME, CHANNEL_CODE	FILENAME
legend_columns	<string>	Number of columns to display in legend	1, 2, 3	2
line_antialias	<string>	Draw lines using antialiasing	OFF, ON	ON
line_scale_dpi	<string>	Scale lines according to	OFF, ON	ON

		display DPI resolution		
line_width	<real>	Default line width for curves (pixels)	1.0, 2.0, 4.0, 8.0	2.0
datum_file	<string>	File containing DATUM line definitons		<none>
macro_directory	<string>	Specify a directory for T/HIS to look in for MACRO definitions		$OA_INSTALL/this_library/macros
master_group_file	<string>	Filename for default group information		<none>
read_group_files	<string>	Default action when a group file is found in a model directory and T/HIS has already read a group file.	IGNORE, DELETE, OVERWRITE, INCREMENT	IGNORE
pemag_calculation_v12	<logical>	Whether the PEMAG calculation uses the v12 logic.	TRUE, FALSE	FALSE
plot_format	<string>	Default format of plot	COLUMN, DEFAULT, AUTO, OFF, FULL, FLOATING	DEFAULT
prefix_format	<string>	Select the prefix formatting for Legend curve labels.	MODEL, DIRECTORY, ROOTNAME, USER	MODEL
regularise_dt	<real>	Time interval for automatic curve regularisation		0.0001
second_y_axis	<logical>	Display 2nd y axis	TRUE, FALSE	FALSE
show_hic_value	<string>	Display HIC value	ON, OFF	ON
show_3ms_value	<string>	Display 3ms Clip value	ON, OFF	ON
show_thiv_value	<string>	Display THIV value	ON, OFF	OFF

show_phd_value	<string>	Display PHD value	ON, OFF	OFF
injury_text_colour	<string>	Colour used to display injury criteria values (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND
injury_line_colour	<string>	Colour used to display injury criteria lines (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND

hic_time_window	<real>	Length of time window for HIC automotive operation.	1e-10 - 1e37	0.036
hic_scale_factor	<real>	Scale factor for acceleration used in HIC automotive operation.	1e-10 - 1e37	9.81
equation_x_start	<real>	X axis start value for equation curves.	-1e37 - 1e37	0.0
equation_x_interval	<real>	X interval between points for equation curves.	1e-10 - 1e37	0.001
equation_npoints	<real>	Number of points for equation curves. Not used if equation_x_interval is defined.	1 - 1e7	0.001
equation_x_end	<real>	X axis end value for equation curves.	-1e37 - 1e37	1.0
hic_scale_factor	<real>	Scale factor for acceleration used in HIC automotive operation.	1e-10 - 1e37	9.81
auto_blank	<string>	Turn ON/OFF AutoBlank	ON, OFF	ON
auto_blank_mode	<string>	Set the default AutoBlank mode	MODEL, COMPONENT_ID, ENTITY_TYPE, ENTITY_ID, COMPONENT_TYPE, SURFACE, CURVE	MODEL
start_in	<string>	Directory to start T/HIS in		<none>
show_license_warning	<logical>	Display Window containing License System messages	TRUE, FALSE	TRUE

save_window_positions	<logical>	Save position of undocked windows between sessions	TRUE, FALSE	TRUE
vc_method	<string>	Default method for calculating Viscous Criteria	ECER95, IIHS	ECER95
asi_method	<string>	Default method for calculating Acceleration Severity Index	2010, 1998	2010
curve_palette	<string>	Controls how many colours are used by curves, default(6), extended(13), no_grey(27), full(30+any user defined)	DEFAULT, EXTENDED, NO_GREY, FULL	OFF
ftcf_error_count	<integer>	Maximum number of errors before a FAST-TCF script terminates		10
ftcf_write_entity_names	<string>	Write entity names instead of entity IDs into FAST-TCF scripts	TRUE, FALSE	TRUE
ftcf_write_diadem_channel_names	<string>	Write DIAdem channel names instead of channel numbers into FAST-TCF scripts	TRUE, FALSE	TRUE
ftcf_write_user_colours	<string>	Write all user-defined colour definitions into FAST-TCF scripts	TRUE, FALSE	FALSE
ftcf_write_required_models	<string>	Reference only the models required by the FAST-TCF script, rather than all models in the session.	TRUE, FALSE	TRUE

		E.g. capturing a graph that contains only data from model M2 will write the model into the script as model 1, so it can be run in a session containing 1 model.		
null_value	<real>	Value to assign to curves when data doesn't exist		1.0E+18
csv_separator	<string>	CSV file field separator	COMMA, TAB, SPACE	COMMA
edit_output_in_primer	<string>	Edit/Create DATABASE cards related to T/HIS entities in PRIMER	ON, OFF	ON

The following options control the automatic creation of curve groups.

Preference	Type	Description	Valid arguments	Default
group_by_model	<logical>	Automatically create a curve group for each model	TRUE, FALSE	TRUE
group_by_type	<logical>	Automatically create a curve group for each entity type data is read for	TRUE, FALSE	FALSE
group_by_component	<logical>	Automatically create a curve group for each component type data is read for	TRUE, FALSE	FALSE
component_group_name	<string>	Controls how curve groups created for components are named)	COMPONENT, COMPONENT_AND_TYPE	COMPONENT
group_by_file_index	<logical>	Automatically create a curve group based on the index of a curve read from a curve file	TRUE, FALSE	FALSE

The following options control the columns that are dsiplayed by default in the curve table

Preference	Type	Description	Valid arguments	Default
ctable_show_curve_id	<logical>	Display Curve IDs	TRUE, FALSE	TRUE
ctable_show_label	<logical>	Display Curve Labels	TRUE, FALSE	TRUE
ctable_show_model	<logical>	Display Files / Models	TRUE, FALSE	TRUE
ctable_show_type	<logical>	Display Entity Types	TRUE, FALSE	TRUE
ctable_show_entity_id	<logical>	Display Entity Ids	TRUE, FALSE	TRUE
ctable_show_component	<logical>	Display Components	TRUE, FALSE	TRUE
ctable_show_style	<logical>	Display Curve Styles	TRUE, FALSE	TRUE
ctable_show_directory	<logical>	Display Directories	TRUE, FALSE	TRUE
ctable_show_miny	<logical>	Display minimum Y value	TRUE, FALSE	TRUE
ctable_show_maxy	<logical>	Display maximum Y value	TRUE, FALSE	TRUE
ctable_show_minposy	<logical>	Display minimum positive Y value	TRUE, FALSE	FALSE
ctable_show_minx	<logical>	Display minimum X value	TRUE, FALSE	TRUE
ctable_show_maxx	<logical>	Display maximum X value	TRUE, FALSE	TRUE
ctable_show_minposx	<logical>	Display minimum positive Y value	TRUE, FALSE	FALSE
ctable_show_xatminy	<logical>	Display X at minimum Y value	TRUE, FALSE	TRUE
ctable_show_xatmaxy	<logical>	Display X at maximum Y value	TRUE, FALSE	TRUE
ctable_show_xatminposy	<logical>	Display X atminimum positive Y value	TRUE, FALSE	FALSE
ctable_show_average	<logical>	Display average value	TRUE, FALSE	TRUE
ctable_show_rms	<logical>	Display RMS value	TRUE, FALSE	TRUE
ctable_show_points	<logical>	Display number of points	TRUE, FALSE	TRUE
ctable_show_hic	<logical>	Display HIC value	TRUE, FALSE	TRUE
ctable_show_hicd	<logical>	Display HICD value	TRUE, FALSE	TRUE
ctable_show_tms	<logical>	Display TMS value	TRUE, FALSE	TRUE
ctable_show_tti	<logical>	Display TTI value	TRUE, FALSE	TRUE

ctable_show_thiv	<logical>	Display THIV value	TRUE, FALSE	TRUE
ctable_show_phd	<logical>	Display PHD value	TRUE, FALSE	TRUE
ctable_show_corr	<logical>	Display CORR value	TRUE, FALSE	TRUE
ctable_properties_on	<logical>	Display the properties columns	TRUE, FALSE	false
ctable_injuryvals_on	<logical>	Display the injury values columns	TRUE, FALSE	false

The following options control the default location and name of where T/HIS looks for model database files.

Preference	**Type**	**Description**	**Valid arguments**	**Default**
database_dir	<string>	Directory to look in for model database (XML) files		<none>
database_file	<string>	Default model database (XML) file		<none>
database_expand	<integer>	Number of levels to automatically expand in model database tree (-1 ALL)	-1 - 2147483646	0

The following strings and values control display options

Preference	**Type**	**Description**	**Valid arguments**	**Default**
axis_width	<real>	Default line width for axis (pixels)	1.0, 2.0, 4.0, 8.0	2.0
axis_colour	<string>	Axis colour (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY,	FOREGROUND

			MEDIUM_GREY, LIGHT_GREY	
axis_top	<string>	Turn ON/OFF drawing of graph top axis	ON, OFF	ON
axis_right	<string>	Turn ON/OFF drawing of graph right axis	ON, OFF	ON
border_on	<logical>	Display border	TRUE, FALSE	TRUE
border_width	<real>	Default line width for border (pixels)	1.0, 2.0, 4.0, 8.0	1.0
border_colour	<string>	Border colour (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND
grid_on	<logical>	Display grid	TRUE, FALSE	TRUE
grid_width	<real>	Default line width for grid (pixels)	1.0, 2.0, 4.0, 8.0	1.0
grid_colour	<string>	Grid colour (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK,	FOREGROUND

			LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	
symbols_on	<logical>	Display symbols (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	TRUE, FALSE	FALSE
symbol_freq	<integer>	Symbol Frequency	1 - 2147483646	1
lines_on	<logical>	Display lines	TRUE, FALSE	TRUE
fix_styles	<logical>	Fix curve styles to cycle through the default colours/styles regardless of the curve number	TRUE, FALSE	FALSE
legend_bg_colour	<string>	Legends background colour (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	BLACK
legend_bg_trans	<integer>	Legend background transparency	0 - 100	0
show_prefix	<string>	Allows you to toggle the Legend curve label prefix On/Off	AUTO, ON, OFF	AUTO

The following strings and values control formatting of values for graphs

Preference	Type	Description	Valid arguments	Default

x_axis_type	<string>	Linear or Logarithmic X Axis type	LOGARITHMIC, LINEAR	LINEAR
x_grid_spacing_off	<real>	X-Axis Grid Spacing value		0.0
x_grid_spacing_int	<real>	X-Axis Grid Interval value		0.0
x_grid_spacing_auto	<string>	X-Axis Grid Spacing	AUTOMATIC, LOCKED	AUTOMATIC
y_axis_type	<string>	Linear or Logarithmic X Axis type	LOGARITHMIC, LINEAR	LINEAR
y_grid_spacing_off	<real>	Y-Axis Grid Spacing value		0.0
y_grid_spacing_int	<real>	Y-Axis Grid Interval value		0.0
y_grid_spacing_auto	<string>	Y-Axis Grid Spacing	AUTOMATIC, LOCKED	AUTOMATIC
y2_axis_type	<string>	Linear or Logarithmic X Axis type	LOGARITHMIC, LINEAR	LINEAR
y2_align_zero	<logical>	Y2-Axis align with Y=0	TRUE, FALSE	FALSE
add_exponent_to_label	<logical>	Add axis multiplier to label	TRUE, FALSE	TRUE
x_axis_decimal_places	<string>	Number of decimal places displayed for X axis values	0, 1, 2, 3, 4, 5, 6, 7, 8, 9, Default(3)	Default(3)
x_axis_format	<string>	Format used to display X axis values	Automatic, General, Scientific, Default(Automatic)	Default(Automatic)
y_axis_decimal_places	<string>	Number of decimal places displayed for Y axis values	0, 1, 2, 3, 4, 5, 6, 7, 8, 9, Default(3)	Default(3)
y_axis_format	<string>	Format used to display Y axis values	Automatic, General, Scientific, Default(Automatic)	Default(Automatic)
y2_axis_decimal_places	<string>	Number of decimal places displayed for second Y axis values	0, 1, 2, 3, 4, 5, 6, 7, 8, 9, Default(3)	Default(3)
y2_axis_format	<string>	Format used to display second Y axis values	Automatic, General, Scientific, Default(Automatic)	Default(Automatic)
colours				
background_colour	<string>	Background colour (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK,	BLACK

			PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	
foreground_colour	<string>	Foreground colour (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	WHITE
user_colour1	<string>	User defined colour 1 (hex code e.g. 0XA1B2C3)		<none>
user_colour2	<string>	User defined colour 2 (hex code e.g. 0XA1B2C3)		<none>
user_colour3	<string>	User defined colour 3 (hex code e.g. 0XA1B2C3)		<none>
user_colour4	<string>	User defined colour 4 (hex code e.g. 0XA1B2C3)		<none>
user_colour5	<string>	User defined colour 5 (hex code e.g. 0XA1B2C3)		<none>

user_colour6	<string>	User defined colour 6 (hex code e.g. 0XA1B2C3)		<none>
user_colours_file	<string>	Location of the user-defined colours XML file.		<none>
save_colours_on_exit	<logical>	Automatically save the user colours XML file when the program exits.	TRUE, FALSE	TRUE

The following options control the preferred order of data sources for various entities

Preference	**Type**	**Description**	**Valid arguments**	**Default**
use_elout	<logical>	Use ELOUT in preference to ELOUTDET for Shell and ThickShell data components from LSDA file	TRUE, FALSE	FALSE
global	<ordered>	Data source for global data	LSDA, ASCII, THF, none	<none>
part	<ordered>	Data source for part data	LSDA, ASCII, THF, none	<none>
node	<ordered>	Data source for node data	THF, LSDA, ASCII, none	<none>
elements				
solid	<ordered>	Data source for solid data	THF, LSDA, none	<none>
beam	<ordered>	Data source for beam data	THF, LSDA, none	<none>
shell	<ordered>	Data source for shell data	THF, LSDA, none	<none>
tshell	<ordered>	Data source for thick shell data	THF, LSDA, none	<none>
spring	<ordered>	Data source for spring data	LSDA, ASCII, XTF, none	<none>
seatbelt	<ordered>	Data source for seatbelt data	LSDA, ASCII, XTF, none	<none>
retractor	<ordered>	Data source for retractor data	LSDA, ASCII, XTF, none	<none>
slipring	<ordered>	Data source for slipring data	LSDA, ASCII, XTF, none	<none>
wall	<ordered>	Data source for rigid wall data	LSDA, ASCII, XTF, none	<none>

contact	<ordered>	Data source for contact data	LSDA, ASCII, XTF, none	<none>
reaction	<ordered>	Data source for nodal reaction data	LSDA, ASCII, XTF, none	<none>
airbag	<ordered>	Data source for airbag data	LSDA, ASCII, XTF, none	<none>
joint	<ordered>	Data source for joint data	LSDA, ASCII, none	<none>
section	<ordered>	Data source for section data	LSDA, ASCII, none	<none>
subsystem	<ordered>	Data source for subsystems data	LSDA, ASCII, none	<none>
geo_contact	<ordered>	Data source for geometric contact data	LSDA, ASCII, none	<none>
nodal_rb	<ordered>	Data source for nodal rigid body data	LSDA, ASCII, none	<none>
weld	<ordered>	Data source for spotweld data	LSDA, ASCII, none	<none>
spc	<ordered>	Data source for spc data	LSDA, ASCII, none	<none>
boundary	<ordered>	Data source for boundary data	LSDA, ASCII, none	<none>
fsi	<ordered>	Data source for fluid structural interaction data	LSDA, ASCII, none	<none>
sph	<ordered>	Data source for SPH data	LSDA, ASCII, none	<none>
tracer	<ordered>	Data source for TRACER data	LSDA, ASCII, none	<none>
pulley	<ordered>	Data source for PULLEY data	LSDA, ASCII, none	<none>
prtube	<ordered>	Data source for PRTUBE data	LSDA, ASCII, none	<none>
pblast	<ordered>	Data source for Particle Blast data	LSDA, ASCII, none	<none>
bearing	<ordered>	Data source for BEARING data	LSDA, ASCII, none	<none>

The following set of options defines how various files are handled by T/HIS.

Preference	**Type**	**Description**	**Valid arguments**	**Default**
output_directory	<string>	Default output directory for images and other files when multiple models have been read	LATEST_MODEL_READ, FIRST_MODEL_READ, LEGACY	LATEST_MODEL_READ

The following strings and values control axes, title, and legend formatting for graphs

Preference	Type	Description	Valid arguments	Default
title_size	<string>	Font size for title	8, 10, 12, 14, 18, 24, Default	Default
title_font	<string>	Font for title	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
title_colour	<string>	Colour of title (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND
x_label_size	<string>	Font size for X axis label	8, 10, 12, 14, 18, 24, Default	Default
x_label_font	<string>	Font for X axis label	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
x_label_colour	<string>	Colour of X axis label (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME,	FOREGROUND

			SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	
x_axis_size	<string>	Font size for X axis units	8, 10, 12, 14, 18, 24, Default	Default
x_axis_font	<string>	Font for X axis units	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
x_axis_colour	<string>	Colour of X axis units (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND
y_label_size	<string>	Font size for Y axis label	8, 10, 12, 14, 18, 24, Default	Default
y_label_font	<string>	Font for Y axis label	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold,	Default

			Times_Medium, Times_bold, Default	
y_label_colour	<string>	Colour of Y axis label (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND
y_axis_size	<string>	Font size for Y axis units	8, 10, 12, 14, 18, 24, Default	Default
y_axis_font	<string>	Font for Y axis units	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
y_axis_colour	<string>	Colour of Y axis units (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON,	FOREGROUND

			DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	
y2_label_size	<string>	Font size for second Y axis label	8, 10, 12, 14, 18, 24, Default	Default
y2_label_font	<string>	Font for second Y axis label	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
y2_label_colour	<string>	Colour of second Y axis label (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND
y2_axis_size	<string>	Font size for second Y axis units	8, 10, 12, 14, 18, 24, Default	Default
y2_axis_font	<string>	Font for second Y axis units	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
y2_axis_colour	<string>	Colour of second Y axis units (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE,	FOREGROUND

			INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	
legend_size	<string>	Font size for curve legends	8, 10, 12, 14, 18, 24, Default	Default
legend_font	<string>	Font for second curve legends	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
legend_colour	<string>	Colour of curve legends (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	FOREGROUND
legend_display_lines	<string>	Turn User Lines On/Off	ON, OFF	ON

The following strings and values control how T/HIS starts

Preference	**Type**	**Description**	**Valid arguments**	**Default**
auto_hide	<logical>	Hide graph tool bar	TRUE, FALSE	FALSE
graphics_type	<string>	Graphics format to start T/HIS with	OPENGL, TTY, DEFAULT	OPENGL
maximise	<logical>	Maximise window when T/HIS started	TRUE, FALSE	TRUE
image_format	<string>	Default image format	BMP_8_C, BMP_8_UN, PNG_8, GIF_8, BMP_24_UN, PNG_24, JPG_24, PPM_24	PNG_24
intel_hd_use_shaders	<string>	Control usage of hardware shaders on Intel HD graphics cards	AUTO_DETECT, FORCE_OFF, FORCE_ON	AUTO_DETECT
placement	<string>	Location for initial window on multi-screen display	LEFT, RIGHT, BOTTOM, TOP, LEFT_BOTTOM, LEFT_TOP, RIGHT_BOTTOM, RIGHT_TOP	<none>
rhs_number_columns	<integer>	Number of columns of Tools buttons	4 - 50	4
white_background_image	<logical>	Write images with white background	TRUE, FALSE	FALSE
bg_img_on	<string>	Turn the Background Image on or off.	ON, OFF	<none>
bg_img_path	<string>	Valid Background Image file path		<none>
bg_img_scale	<string>	Preset Background Image scaling	WIDTH, HEIGHT, W+H	<none>
bg_img_fact	<real>	Scale factor for Background Image Size		1
bg_img_just	<string>	Background Image Justification	N, NE, E, SE, S, SW, W, NW	<none>
bg_img_pos	<string>	Background Image Positioning	TILED, SINGLE	SINGLE
window_layout	<string>	Multiple window layout type	TILE_WIDE, TILE_TALL, CASCADE, 1X1, 2X2, 3X3, XY	TILE_WIDE
x_layout	<integer>	Number used for 'X x Y' layout, number of cols	1 - 8	1
y_layout	<integer>	Number used for 'X x Y' layout, number of rows	1 - 8	1
page_width	<integer>	Width of the page (pixels)		<none>
page_height	<integer>	Height of the page (pixels)		<none>
graphical_user_interface				

gui_theme	<string>	Graphical User Interface (GUI) theme	LIGHT, DARK, CLASSIC, LEGACY	LIGHT
gui_styling_mode	<string>	Graphical User Interface (GUI) styling and decoration	NOT_USED, TIME_LIMIT, ALWAYS	TIME_LIMIT
gui_styling_tlimit	<integer>	Graphical User Interface (GUI) menu repaint time limit to turn off decorations	0 - 100000	500

The following strings and values control laser plotting setup

Preference	**Type**	**Description**	**Valid arguments**	**Default**
laser_paper_size	<string>	Default paper size	A4, A3, US	A4
laser_orientation	<string>	Default page orientation	Portrait, Landscape	Landscape
laser_top_margin	<real>	Top margin size in mm		10
laser_bottom_margin	<real>	Bottom margin size in mm		30
laser_left_margin	<real>	Left margin size in mm		20
laser_right_margin	<real>	Right margin size in mm		10

The following options affect the appearance and behaviour of the graphical user interface, left handed support, and the mouse

Preference	**Type**	**Description**	**Valid arguments**	**Default**
display_factor	<real>	Factor on display size (0.5 - 2.0, automatic if undefined)	0.5 - 2.0	1.2
display_brightness	<real>	Menu brightness (0.0-1.0)	0.0 - 1.0	1.0
display_saturation	<real>	Menu colour saturation (0.0-1.0)	0.0 - 1.0	1.0
button_gradation	<real>	Button shade gradation (0.0-1.0)	0.0 - 1.0	0.0
dv_sync_windows	<string>	Dyn view method(s) for synchronising windows	ICON, ICON+CAPS, ICON+NUM, ICON+CAPS+NUM	ICON+CAPS
dv_left_shift	<string>	Dyn view action for shift + Left mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE,	ROTATION_XYZ

			ZOOM_DOWN_+VE, UNUSED	
dv_middle_shift	<string>	Dyn view action for shift + Middle mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	TRANSLATION
dv_right_shift	<string>	Dyn view action for shift + Right mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ZOOM_UP_+VE
dv_left_ctrl	<string>	Dyn view action for ctrl + Left mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ROTATION_XYZ
dv_middle_ctrl	<string>	Dyn view action for ctrl + Middle mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	TRANSLATION
dv_right_ctrl	<string>	Dyn view action for ctrl + Right mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ZOOM_UP_+VE
dv_left_both	<string>	Dyn view action for shift+ctrl + Left mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ROTATION_XYZ
dv_middle_both	<string>	Dyn view action for shift+ctrl + Middle mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z,	TRANSLATION

			ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	
dv_right_both	<string>	Dyn view action for shift+ctrl + Right mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ZOOM_UP_+VE
dv_shift_action	<string>	Dynamic viewing mode for shift + mouse button	CURRENT, WIREFRAME, FREE_EDGE, UNUSED	CURRENT
dv_ctrl_action	<string>	Dynamic viewing mode for ctrl + mouse button	CURRENT, WIREFRAME, FREE_EDGE, UNUSED	WIREFRAME
dv_both_action	<string>	Dynamic viewing mode for shift+ctrl + mouse button	CURRENT, WIREFRAME, FREE_EDGE, UNUSED	FREE_EDGE
font_cache	<logical>	Whether to use cached fonts on Linux machines with no core X11 fonts loaded	TRUE, FALSE	TRUE
font_quality	<string>	The quality of font rendering in the graphical user interface	PLAIN, ANTI_ALIAS	ANTI_ALIAS
font_scaling	<string>	Whether text in GUI buttons can be scaled down to fit (TRUE means both width and height)	FALSE, WIDTH, HEIGHT, TRUE	WIDTH
font_silent	<logical>	whether to write explanatory text if wanted fonts are not found	TRUE, FALSE	FALSE
font_size	<string>	Menu font size	TINY, SMALL, DEFAULT, LARGE, HUGE	DEFAULT
font_type	<string>	Menu font typeface and strength	HELVETICA, HELVETICA-BOLD, TIMES, TIMES-BOLD, COURIER, COURIER-BOLD	HELVETICA

unix_prop_font	<string>	GUI proportional font for menu panels on Linux/Unix		Helvetica
unix_mono_font	<string>	GUI monospaced font for listing boxes on Linux/Unix		Courier New
windows_prop_font	<string>	GUI proportional font for menu panels on Windows		Helvetica
windows_mono_font	<string>	GUI monospaced font for listing boxes on Windows		Courier New
left_handed	<string>	Left handed switching of mouse and/or keyboard	NONE, MOUSE, KEYBOARD, ALL	NONE
zoom_factor	<real>	Zoom Factor for mouse wheel (0.01-1.0)	0.01 - 1.0	0.05
czoom_factor	<real>	Factor for right mouse dynamic zoom (0.01-0.2)	0.01 - 0.2	0.05
kzoom_factor	<real>	Factor for +/- keyboard short-cut keys	0.01 - 100.0	2.0
menu_dragging_mode	<string>	Mode used when moving menu panels with the mouse	WIREFRAME, OPAQUE	WIREFRAME
mouse_action_middle_button	<string>	Set the action for the middle mouse key during picking	APPLY, REJECT, DESELECT	REJECT
mouse_action_right_button	<string>	Set the action for the right mouse key during picking	APPLY, REJECT, DESELECT	DESELECT

The following control settings related to quickfind

Preference	Type	Description	Valid arguments	Default
quickfind_unmatched_text_colour	<string>	Text colour for unmatched characters (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME,	BLACK

			SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	
quickfind_matched_text_colour	<string>	Text colour for matched characters (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN, MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	BLUE
quickfind_found_list_length	<integer>	Number of items to display in the found list	1 - 20	10
quickfind_recent_history	<integer>	Number of recently selected items to store	0 - 2147483646	10
quickfind_sequential_bonus	<integer>	Modifier for two successful adjacent matches	1 - 100	50
quickfind_word_start_bonus	<integer>	Modifier for successful match at word beginning	1 - 100	10
quickfind_box_size	<string>	Size and layout of Search box	SMALL, LARGE	SMALL

The following options define how Javascripts are processed by THIS. See the JavaScript interface for further details.

Preference	Type	Description	Valid arguments	Default
modules_directory	<string>	Directory for T/HIS to look for modules in		<none>
script_directory	<string>	Directory in which T/HIS looks for scripts		$OA_INSTALL/this_library/scripts
javascript_memory_size	<integer>	Maximum memory allocated for garbage collection		25
javascript_update_curve_menu	<logical>	Update curve menu contenst while a JavaScript is running	TRUE, FALSE	FALSE

The following options define how T/HIS sessions are processed See T/HIS Session Save and Retrieve for further details.

Preference	Type	Description	Valid arguments	Default
session_auto_save	<string>	Save a session unconditionally on exit	OFF, ON	OFF
session_save_option	<string>	Location for automatically saving sessions	HOME, USER_DEFINED, DESKTOP	HOME
session_save_dir	<string>	User-defined location for session save		<none>
session_embed_cur_csv_files	<string>	Embed the external cur/csv files into the session.	OFF, ON	OFF
session_embed_curve_data	<string>	Embed the curve xy data for all curves into the session.	OFF, ON	OFF
show_session_retrieve_on_start	<string>	A pop-up panel to retrieve a saved T/HIS session file would show up every time T/HIS is launched.	ON, OFF	OFF

Keys can have functions assigned to them:

Preference	Type	Description	Valid arguments	Default

F1_key	<string>	Shortcut for F1		<none>
F2_key	<string>	Shortcut for F2		<none>
F3_key	<string>	Shortcut for F3		<none>
F4_key	<string>	Shortcut for F4		<none>
F5_key	<string>	Shortcut for F5		<none>
F6_key	<string>	Shortcut for F6		<none>
F7_key	<string>	Shortcut for F7		<none>
F8_key	<string>	Shortcut for F8		<none>
F9_key	<string>	Shortcut for F9		<none>
F10_key	<string>	Shortcut for F10		<none>
F11_key	<string>	Shortcut for F11		<none>
F12_key	<string>	Shortcut for F12		<none>
A_key	<string>	Shortcut for A		AUTOSCALE
B_key	<string>	Shortcut for B		BLANK
C_key	<string>	Shortcut for C		CURVE_MENU
D_key	<string>	Shortcut for D		DATUM_MENU
E_key	<string>	Shortcut for E		<none>
F_key	<string>	Shortcut for F		FAST_TCF_MENU
G_key	<string>	Shortcut for G		NEW_WINDOW
H_key	<string>	Shortcut for H		<none>
I_key	<string>	Shortcut for I		<none>
J_key	<string>	Shortcut for J		JAVASCRIPT_MENU
K_key	<string>	Shortcut for K		<none>
L_key	<string>	Shortcut for L		<none>
M_key	<string>	Shortcut for M		<none>
N_key	<string>	Shortcut for N		EDIT_NEXT
O_key	<string>	Shortcut for O		<none>
P_key	<string>	Shortcut for P		PLOT
Q_key	<string>	Shortcut for Q		QUICK_PICK
R_key	<string>	Shortcut for R		REVERSE
S_key	<string>	Shortcut for S		<none>
T_key	<string>	Shortcut for T		TIDY_MENUS
U_key	<string>	Shortcut for U		UNBLANK
V_key	<string>	Shortcut for V		CURVE_GROUP
W_key	<string>	Shortcut for W		<none>
X_key	<string>	Shortcut for X		CURVE_TABLE
Y_key	<string>	Shortcut for Y		Y_AUTOSCALE
Z_key	<string>	Shortcut for Z		ZOOM
a_key	<string>	Shortcut for a		AUTOSCALE
b_key	<string>	Shortcut for b		BLANK
c_key	<string>	Shortcut for c		CURVE_MENU
d_key	<string>	Shortcut for d		DATUM_MENU
e_key	<string>	Shortcut for e		<none>
f_key	<string>	Shortcut for f		FAST_TCF_MENU
g_key	<string>	Shortcut for g		NEW_WINDOW
h_key	<string>	Shortcut for h		<none>

i_key	<string>	Shortcut for i		<none>
j_key	<string>	Shortcut for j		JAVASCRIPT_MENU
k_key	<string>	Shortcut for k		<none>
l_key	<string>	Shortcut for l		<none>
m_key	<string>	Shortcut for m		<none>
n_key	<string>	Shortcut for n		EDIT_NEXT
o_key	<string>	Shortcut for o		<none>
p_key	<string>	Shortcut for p		PLOT
q_key	<string>	Shortcut for q		QUICK_PICK
r_key	<string>	Shortcut for r		REVERSE
s_key	<string>	Shortcut for s		<none>
t_key	<string>	Shortcut for t		TIDY_MENUS
u_key	<string>	Shortcut for u		UNBLANK
v_key	<string>	Shortcut for v		CURVE_GROUP
w_key	<string>	Shortcut for w		<none>
x_key	<string>	Shortcut for x		CURVE_TABLE
y_key	<string>	Shortcut for y		Y_AUTOSCALE
z_key	<string>	Shortcut for z		ZOOM
SPACE_key	<string>	Shortcut for space		PLOT
ZERO_key	<string>	Shortcut for 0		COPY_AXIS
ONE_key	<string>	Shortcut for 1		TILE_TALL
TWO_key	<string>	Shortcut for 2		TILE_WIDE
THREE_key	<string>	Shortcut for 3		CASCADE
FOUR_key	<string>	Shortcut for 4		LAYOUT_1X1
FIVE_key	<string>	Shortcut for 5		LAYOUT_2X2
SIX_key	<string>	Shortcut for 6		LAYOUT_3X3
SEVEN_key	<string>	Shortcut for 7		<none>
EIGHT_key	<string>	Shortcut for 8		<none>
NINE_key	<string>	Shortcut for 9		<none>
EXCLAMATION_key	<string>	Shortcut for !		<none>
DOUBLEQUOTE_key	<string>	Shortcut for "		<none>
HASH_key	<string>	Shortcut for #		<none>
DOLLAR_key	<string>	Shortcut for $		<none>
PERCENT_key	<string>	Shortcut for %		<none>
AMPERSAND_key	<string>	Shortcut for &		<none>
SINGLEQUOTE_key	<string>	Shortcut for '		<none>
LEFTBRACKET_key	<string>	Shortcut for (		<none>
RIGHTBRACKET_key	<string>	Shortcut for)		<none>
ASTERISK_key	<string>	Shortcut for *		<none>
PLUS_key	<string>	Shortcut for +		ZOOM_IN
COMMA_key	<string>	Shortcut for ,		<none>
MINUS_key	<string>	Shortcut for -		ZOOM_OUT
DOT_key	<string>	Shortcut for .		<none>
SLASH_key	<string>	Shortcut for /		SHORTCUT
COLON_key	<string>	Shortcut for :		<none>
SEMICOLON_key	<string>	Shortcut for ;		<none>

LESSTHAN_key	<string>	Shortcut for <		<none>
EQUALS_key	<string>	Shortcut for =		ZOOM_IN
GREATERTHAN_key	<string>	Shortcut for >		<none>
QUESTIONMARK_key	<string>	Shortcut for ?		SHORTCUT
AT_key	<string>	Shortcut for @		<none>
LEFTSQUAREBRACKET_key	<string>	Shortcut for [		<none>
BACKSLASH_key	<string>	Shortcut for \		<none>
RIGHTSQUAREBRACKET_key	<string>	Shortcut for]		<none>
CIRCUMFLEX_key	<string>	Shortcut for ^		<none>
UNDERSCORE_key	<string>	Shortcut for _		ZOOM_OUT
BACKTICK_key	<string>	Shortcut for `		<none>
LEFTCURLYBRACKET_key	<string>	Shortcut for {		<none>
PIPE_key	<string>	Shortcut for |		<none>
RIGHTCURLYBRACKET_key	<string>	Shortcut for }		<none>
TILDE_key	<string>	Shortcut for ~		<none>

The following strings control the T/HIS header and version number at the bottom right of the plot space

Preference	Type	Description	Valid arguments	Default
user_text_line_1	<string>	Text for line 1		<none>
user_text_line_2	<string>	Text for line 2		<none>
user_text_line_3	<string>	Text for line 3		<none>
user_text_line_4	<string>	Text for line 4		<none>
user_text_line_5	<string>	Text for line 5		<none>
user_text_line_6	<string>	Text for line 6		<none>
user_text_size_1	<string>	Size of text on line 1	8, 10, 12, 14, 18, 24, Default	Default
user_text_size_2	<string>	Size of text on line 2	8, 10, 12, 14, 18, 24, Default	Default
user_text_size_3	<string>	Size of text on line 3	8, 10, 12, 14, 18, 24, Default	Default
user_text_size_4	<string>	Size of text on line 4	8, 10, 12, 14, 18, 24, Default	Default
user_text_size_5	<string>	Size of text on line 5	8, 10, 12, 14, 18, 24, Default	Default
user_text_size_6	<string>	Size of text on line 6	8, 10, 12, 14, 18, 24, Default	Default
user_text_font	<string>	Font for user text	Helvetica_Medium, Helvetica_Bold, Courier_Medium, Courier_Bold, Times_Medium, Times_bold, Default	Default
user_text_colour	<string>	Colour for user text (hex code e.g. 0XA1B2C3 or core colour name e.g. OLIVE)	FOREGROUND, WHITE, BLACK, RED, GREEN, BLUE, CYAN,	FOREGROUND

			MAGENTA, YELLOW, ORANGE, TURQUOISE, INDIGO, LIME, SKY, PINK, PALE_YELLOW, GOLD, OLIVE, DARK_MAGENTA, MEDIUM_GREEN, MEDIUM_BLUE, HOT_PINK, LIGHT_PINK, SEA_GREEN, MAROON, DARK_GREEN, PURPLE, NAVY, DARK_GREY, MEDIUM_GREY, LIGHT_GREY	

The following control treatment of unicode

Preference	**Type**	**Description**	**Valid arguments**	**Default**
cjk_unix_font	<string>	Font to use for CJK text on unix machines		-misc-fixed-medium-r-normal-*-12-*-*-*-*-*-*-*
cjk_windows_font	<string>	Font to use for CJK text on windows machines		MS Gothic 10
file_encoding	<string>	Character encoding for script files	Latin-1, BIG5, EUC-CN, EUC-JP, EUC-KR, GB, GBK, ISO-2022-CN, ISO-2022-CN-EXT, ISO-2022-JP, ISO-2022-JP-2, ISO-2022-KR, JOHAB, Shift-JIS, UTF-8, UTF-16BE, UTF-16LE, UTF-16, UTF-32BE, UTF-32LE, UTF-32	Latin-1

The following strings and values control the display of UNIT information in T/HIS

Preference	Type	Description	Valid arguments	Default
model_units	<string>	Sets the default UNIT system for models	U1 m:kg:s (SI), U2 mm:T:s, U3 mm:kg:ms, U4 mm:gm:ms, U5 ft:slug:s, U6 m:T:s	U1 m:kg:s (SI)
display_units	<string>	Sets the default UNIT system used to display results	U1 m:kg:s (SI), U2 mm:T:s, U3 mm:kg:ms, U4 mm:gm:ms, U5 ft:slug:s, U6 m:T:s	U1 m:kg:s (SI)
write_csv_units	<logical>	Write UNIT information to CSV files	TRUE, FALSE	TRUE

The drive mappings allow T/HIS to convert equivalent folder names from Windows to Unix and visa versa. This is currently only in use for the JavaScript function DriveMapFilename.

Preference	Type	Description	Valid arguments	Default
drive_a	<string>	Mapping from Windows drive A: to unix path		<none>
drive_b	<string>	Mapping from Windows drive B: to unix path		<none>
drive_c	<string>	Mapping from Windows drive C: to unix path		<none>
drive_d	<string>	Mapping from Windows drive D: to unix path		<none>
drive_e	<string>	Mapping from Windows drive E: to unix path		<none>
drive_f	<string>	Mapping from Windows drive F: to unix path		<none>
drive_g	<string>	Mapping from Windows drive G: to unix path		<none>
drive_h	<string>	Mapping from Windows drive H: to unix path		<none>
drive_i	<string>	Mapping from Windows drive I: to unix path		<none>
drive_j	<string>	Mapping from Windows drive J: to unix path		<none>
drive_k	<string>	Mapping from Windows drive K: to unix path		<none>

drive_l	<string>	Mapping from Windows drive L: to unix path		<none>
drive_m	<string>	Mapping from Windows drive M: to unix path		<none>
drive_n	<string>	Mapping from Windows drive N: to unix path		<none>
drive_o	<string>	Mapping from Windows drive O: to unix path		<none>
drive_p	<string>	Mapping from Windows drive P: to unix path		<none>
drive_q	<string>	Mapping from Windows drive Q: to unix path		<none>
drive_r	<string>	Mapping from Windows drive R: to unix path		<none>
drive_s	<string>	Mapping from Windows drive S: to unix path		<none>
drive_t	<string>	Mapping from Windows drive T: to unix path		<none>
drive_u	<string>	Mapping from Windows drive U: to unix path		<none>
drive_v	<string>	Mapping from Windows drive V: to unix path		<none>
drive_w	<string>	Mapping from Windows drive W: to unix path		<none>
drive_x	<string>	Mapping from Windows drive X: to unix path		<none>
drive_y	<string>	Mapping from Windows drive Y: to unix path		<none>
drive_z	<string>	Mapping from Windows drive Z: to unix path		<none>

oa_pref" arguments valid for all programs

Preference	Type	Description	Valid arguments	Default
file_names	<string>	Controls input filename syntax. LSTC = d3*, OASYS = job.ptf*	OASYS, LSTC	OASYS
html_application	<string>	Location of HTML browser		<none>
html_application_linux	<string>	Location of HTML browser for linux (use if the same oa_pref file is used for windows and linux)		<none>

html_application_windows	<string>	Location of HTML browser for windows (use if the same oa_pref file is used for windows and linux)		<none>
image_format	<string>	Default image format	BMP_8_C, BMP_8_UN, PNG_8, GIF_8, BMP_24_UN, PNG_24, JPG_24, PPM_24	PNG_24
intel_hd_use_shaders	<string>	Control usage of hardware shaders on Intel HD graphics cards	AUTO_DETECT, FORCE_OFF, FORCE_ON	AUTO_DETECT
locale	<string>	Language and country locale to use (overrides system one)		<none>
manuals_url	<string>	URL of the online manuals		help.oasys-software.com/articles/?readerUiPreview=1#!
maximise	<logical>	Maximise window when Program is started	TRUE, FALSE	TRUE
online_manuals	<logical>	Open the online version of the manuals from Help buttons (TRUE) or open the local (offline) HTML copy (FALSE)	TRUE, FALSE	TRUE
pdf_application	<string>	Location of PDF browser		<none>
pdf_application_linux	<string>	Location of PDF browser for linux (use if the same oa_pref file is used for windows and linux)		<none>
pdf_application_windows	<string>	Location of PDF browser		<none>

		for windows (use if the same oa_pref file is used for windows and linux)		
placement	<string>	Location for initial window on multi-screen display	LEFT, RIGHT, BOTTOM, TOP, LEFT_BOTTOM, LEFT_TOP, RIGHT_BOTTOM, RIGHT_TOP	<none>
start_in	<string>	Directory to start Program in		<none>
temp_file_expiry	<integer>	Age in days after which a temporary filename can be reused, 0 = never	0 - 10000	31
show_license_warning	<logical>	Display Window containing License System messages	TRUE, FALSE	TRUE
post_uses_primer	<logical>	ADMIN/INSTALL pref which allows D3Plot, T/his to take an available Primer license	TRUE, FALSE	TRUE
save_window_positions	<logical>	Save position of undocked windows between sessions	TRUE, FALSE	TRUE

The following control directories

Preference	**Type**	**Description**	**Valid arguments**	**Default**
home_dir	<string>	"home" directory for user		<none>
manuals_dir	<string>	Directory user manuals are installed in		<none>
temp_dir	<string>	temporary directory for user		<none>

write_checkpoint_files	<logical>	Record checkpoint files for the PRIMER, D3PLOT or T/His sessions.	TRUE, FALSE	FALSE
checkpoint_dir	<string>	Directory for checkpoint files. If omitted use cwd.		<none>
show_checkpoint_files	<logical>	Show checkpoint playback panel upon PRIMER, D3PLOT or T/His startup.	TRUE, FALSE	FALSE
graphics				
initial_view_orientation	<string>	Initial view orientation for the graphics window.	+XY, +YZ, +XZ, +ISO, -XY, -YZ, -XZ, -ISO	+XY
graphical_user_interface				
gui_theme	<string>	Graphical User Interface (GUI) theme	LIGHT, DARK, CLASSIC, LEGACY	LIGHT
gui_styling_mode	<string>	Graphical User Interface (GUI) styling and decoration	NOT_USED, TIME_LIMIT, ALWAYS	TIME_LIMIT
gui_styling_tlimit	<integer>	Graphical User Interface (GUI) menu repaint time limit to turn off decorations	0 - 100000	500

The following control laser options

Preference	Type	Description	Valid arguments	Default
laser_paper_size	<string>	Default paper size	US, A4	A4
laser_orientation	<string>	Default page orientation	Portrait, Landscape	Landscape
laser_top_margin	<real>	Top margin size in mm		10
laser_bottom_margin	<real>	Bottom margin size in mm		30
laser_left_margin	<real>	Left margin size in mm		20
laser_right_margin	<real>	Right margin size in mm		10

The following control menu and mouse attributes

Preference	Type	Description	Valid arguments	Default
display_factor	<real>	Factor on display size (0.5 - 2.0, automatic if undefined)	0.5 - 2.0	1.2
display_brightness	<real>	Menu brightness (0.0-1.0)	0.0 - 1.0	1.0

display_saturation	<real>	Menu colour saturation (0.0-1.0)	0.0 - 1.0	1.0
button_gradation	<real>	Button shade gradation (0.0-1.0)	0.0 - 1.0	0.0
dv_sync_windows	<string>	Dyn view method(s) for synchronising windows	ICON, ICON+CAPS, ICON+NUM, ICON+CAPS+NUM	ICON+CAPS
dv_left_shift	<string>	Dyn view action for shift + Left mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ROTATION_XYZ
dv_middle_shift	<string>	Dyn view action for shift + Middle mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	TRANSLATION
dv_right_shift	<string>	Dyn view action for shift + Right mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ZOOM_UP_+VE
dv_left_ctrl	<string>	Dyn view action for ctrl + Left mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ROTATION_XYZ
dv_middle_ctrl	<string>	Dyn view action for ctrl + Middle mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	TRANSLATION
dv_right_ctrl	<string>	Dyn view action for ctrl + Right mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z,	ZOOM_UP_+VE

			ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	
dv_left_both	<string>	Dyn view action for shift+ctrl + Left mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ROTATION_XYZ
dv_middle_both	<string>	Dyn view action for shift+ctrl + Middle mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	TRANSLATION
dv_right_both	<string>	Dyn view action for shift+ctrl + Right mouse	ROTATION_XYZ, ROTATION_XY, ROTATION_Z, ROTATION_SPHERE, TRANSLATION, ZOOM_UP_+VE, ZOOM_DOWN_+VE, UNUSED	ZOOM_UP_+VE
dv_shift_action	<string>	Dynamic viewing mode for shift + mouse button	CURRENT, WIREFRAME, FREE_EDGE, UNUSED	CURRENT
dv_ctrl_action	<string>	Dynamic viewing mode for ctrl + mouse button	CURRENT, WIREFRAME, FREE_EDGE, UNUSED	WIREFRAME
dv_both_action	<string>	Dynamic viewing mode for shift+ctrl + mouse button	CURRENT, WIREFRAME, FREE_EDGE, UNUSED	FREE_EDGE
font_cache	<logical>	Whether to use cached fonts on Linux machines with no core X11 fonts loaded	TRUE, FALSE	TRUE
font_quality	<string>	The quality of font rendering in the graphical user interface	PLAIN, ANTI_ALIAS	ANTI_ALIAS
font_scaling	<string>	Whether text in GUI buttons can	FALSE, WIDTH, HEIGHT, TRUE	WIDTH

		be scaled down to fit (TRUE means both width and height)		
font_silent	<logical>	whether to write explanatory text if wanted fonts are not found	TRUE, FALSE	FALSE
font_size	<string>	Menu font size	TINY, SMALL, DEFAULT, LARGE, HUGE	DEFAULT
font_type	<string>	Menu font typeface and strength	HELVETICA, HELVETICA-BOLD, TIMES, TIMES-BOLD, COURIER, COURIER-BOLD	HELVETICA
unix_prop_font	<string>	GUI proportional font for menu panels on Linux/Unix		Helvetica
unix_mono_font	<string>	GUI monospaced font for listing boxes on Linux/Unix		Courier New
windows_prop_font	<string>	GUI proportional font for menu panels on Windows		Helvetica
windows_mono_font	<string>	GUI monospaced font for listing boxes on Windows		Courier New
left_handed	<string>	Left handed switching of mouse and/or keyboard	NONE, MOUSE, KEYBOARD, ALL	NONE
zoom_factor	<real>	Zoom Factor for mouse wheel (0.01-1.0)	0.01 - 1.0	0.05
czoom_factor	<real>	Factor for right mouse dynamic zoom (0.01-0.2)	0.01 - 0.2	0.05
kzoom_factor	<real>	Factor for +/- keyboard short-cut keys	0.01 - 100.0	2.0
menu_dragging_mode	<string>	Mode used when moving menu panels with the mouse	WIREFRAME, OPAQUE	WIREFRAME
mouse_3d_rotation_factor	<real>	Factor applied to the speed of rotation when using a 3D mouse		1.0

mouse_3d_pan_factor	<real>	Factor applied to the speed of panning when using a 3D mouse		1.0
mouse_3d_zoom_factor	<real>	Factor applied to the speed of zooming when using a 3D mouse		1.0
mouse_action_middle_button	<string>	Set the action for the middle mouse key during picking	APPLY, REJECT, DESELECT	REJECT
mouse_action_right_button	<string>	Set the action for the right mouse key during picking	APPLY, REJECT, DESELECT	DESELECT

The following control treatment of recent files popups

Preference	**Type**	**Description**	**Valid arguments**	**Default**
recent_files_dropdown	<string>	Turn the recent files popup on or off	OFF, ON	ON
recent_files_max_but	<integer>	Maximum number of buttons displayed in a recent files popup	1 - 50	10
recent_files_max_char	<integer>	Maximum number of characters displayed on each recent files button	1 - 512	50

The following control treatment of unicode

Preference	**Type**	**Description**	**Valid arguments**	**Default**
cjk_unix_font	<string>	Font to use for CJK text on unix machines		-misc-fixed-medium-r-normal-*-12-*-*-*-*-*-*-*
cjk_windows_font	<string>	Font to use for CJK text on windows machines		MS Gothic 10
file_encoding	<string>	Character encoding for script files	Latin-1, BIG5, EUC-CN, EUC-JP, EUC-KR, GB, GBK, ISO-2022-CN, ISO-2022-CN-EXT, ISO-2022-JP, ISO-2022-JP-2, ISO-2022-KR, JOHAB, Shift-	Latin-1

			JIS, UTF-8, UTF-16BE, UTF-16LE, UTF-16, UTF-32BE, UTF-32LE, UTF-32	

The drive mappings allow PRIMER to convert equivalent folder names from Windows to Unix and visa versa. This is currently only in use for the JavaScript function DriveMapFilename for D3PLOT and T/HIS.

Preference	**Type**	**Description**	**Valid arguments**	**Default**
drive_a	<string>	Mapping from Windows drive A: to unix path		<none>
drive_b	<string>	Mapping from Windows drive B: to unix path		<none>
drive_c	<string>	Mapping from Windows drive C: to unix path		<none>
drive_d	<string>	Mapping from Windows drive D: to unix path		<none>
drive_e	<string>	Mapping from Windows drive E: to unix path		<none>
drive_f	<string>	Mapping from Windows drive F: to unix path		<none>
drive_g	<string>	Mapping from Windows drive G: to unix path		<none>
drive_h	<string>	Mapping from Windows drive H: to unix path		<none>
drive_i	<string>	Mapping from Windows drive I: to unix path		<none>
drive_j	<string>	Mapping from Windows drive J: to unix path		<none>
drive_k	<string>	Mapping from Windows drive K: to unix path		<none>
drive_l	<string>	Mapping from Windows drive L: to unix path		<none>
drive_m	<string>	Mapping from Windows drive M: to unix path		<none>
drive_n	<string>	Mapping from Windows drive N: to unix path		<none>
drive_o	<string>	Mapping from Windows drive O: to unix path		<none>
drive_p	<string>	Mapping from Windows drive P: to unix path		<none>
drive_q	<string>	Mapping from Windows drive Q: to unix path		<none>
drive_r	<string>	Mapping from Windows drive R: to unix path		<none>

drive_s	<string>	Mapping from Windows drive S: to unix path		<none>
drive_t	<string>	Mapping from Windows drive T: to unix path		<none>
drive_u	<string>	Mapping from Windows drive U: to unix path		<none>
drive_v	<string>	Mapping from Windows drive V: to unix path		<none>
drive_w	<string>	Mapping from Windows drive W: to unix path		<none>
drive_x	<string>	Mapping from Windows drive X: to unix path		<none>
drive_y	<string>	Mapping from Windows drive Y: to unix path		<none>
drive_z	<string>	Mapping from Windows drive Z: to unix path		<none>

The following control workflows functionality

Preference	**Type**	**Description**	**Valid arguments**	**Default**
workflow_definitions_directory	<string>	Location that will be scanned for Workflow definitions		<none>
workflow_only_use_specified_directory	<logical>	Only scan location set by preference oasys*workflow_definitions_directory for Workflow definitions (if it is set)	TRUE, FALSE	FALSE
workflow_user_data_directory_name	<string>	Name of a folder to search in for workflow user data		<none>
workflow_auto_open_post_menu	<logical>	Automatically open the Workflow menu in D3PLOT or T/HIS when reading in a model that has workflow data	TRUE, FALSE	FALSE
workflow_max_upward_folder_search_	<integer	Maximum number of folders to	0 - 100	4

13.9. APPENDIX I - Windows File Associations

13.9.1. WINDOWS (PC's)

WINDOWS (PC's)

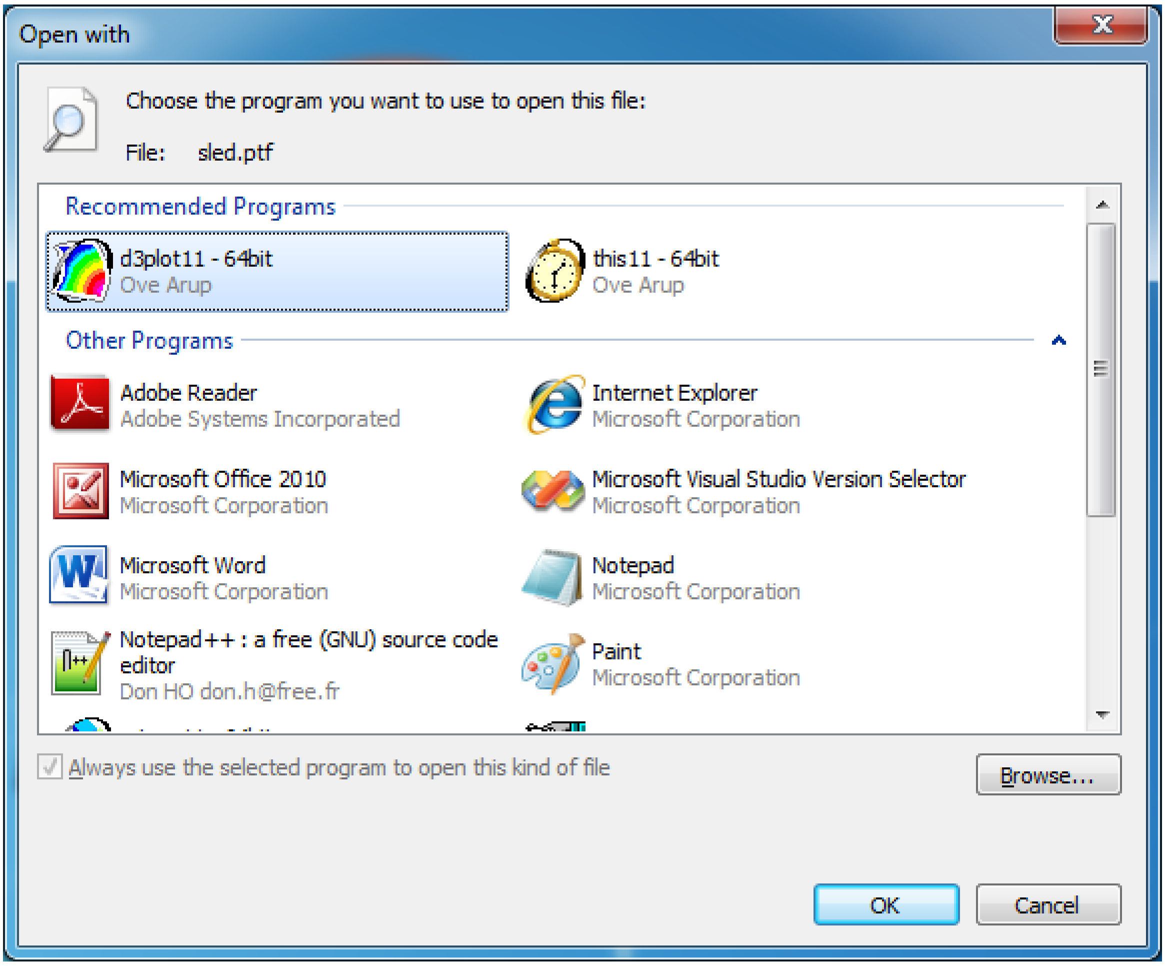
Under Windows on PC it is possible to set up file associations so that double clicking on files with the `.thf` , `.xtf` , `.cur` and `.bdf` extension opens them automatically in T/HIS.

All of these settings are optional: you should be aware that under the Windows operating system associating a filetype (via its extension) with an application is convenient, but can also be restricting and hard to undo.

To make .thf files open in T/HIS by double-clicking on them

If no application is currently associated with .thf files, a "double-click" won't work, and some non-specific, usually "windows", icon will be displayed with the file.

Right click on any `.thf` file, and select **properties** and then press the Change... tab next to Opens with: from the popup menu.



1. This will bring up the **"Open with"** panel.
2. Ensure the **Always use...** box is ticked
3. Use the directory browsing window to find the correct T/HIS executable. You are looking for file **this11.exe** or **this11_x64.exe** .
4. Select the executable and click on **OK** to close the "Open With" window.

T/HIS should now open and read in the selected file and you should now find that:

- All **.thf** files on your system show the T/HIS icon.
- Double-clicking on any such file starts T/HIS and opens that file.

It is not possible to set up the filename "d3thdt" for double-clicking in this way since Windows requires filename extensions when assigning applications to files.)

`To make .xtf`

The procedure is exactly the same as for **`.thf`** files, and must be carried out for each of the file types that you wish to process by double-clicking:

.xtf	LS-DYNA Extra Time History file
.cur	T/HIS Curve file
.bdf	T/HIS Bulk Data file

Note that: File types `.thf` and `.xtf` are opened in this way, but no contents are read in.

File types `.cur` and `.bdf` are opened and their complete contents read in.

13.10. APPENDIX J - Typed Commands

13.10.1. Global Menu

Global Menu	
PL - Plot	**CL** - Clear Screen
ZM - Zoom	**AU** - Auto Scale Plot
CE - Centre	**PT** - Point on Screen
PF - Write Postscript file (use default)	
PC - Write Postscript file (Colour)	
PB - Write Postscript file (Blank/White)	
BL - Blank Curve	**UB** - Unblank Curve
RM - Remove a Curve	**ER** - Erase all curves
GS - Global Status	**CO** - Condense Curves
Y1 - 1st Y axis	**Y2** - Second Y axis
DOU - Double Y axis (ON/ OFF)	
CF - Command file (read)	**SF** - Session file (write)
CS - Close session file	
EX - Exit	
! - Backspace	**/** - Top level menu
Q - Abort operation	
; - End of command string	

13.10.2. List Commands

List Commands

LS - List all files in current directory
LB - List all files "*.bdf" in current directory
LI - List all files ASCII files in current directory
LC - List all files "*.cur" in current directory
LK - List all files "*.key" in current directory

<table>
<tr><td>GM - Global Menu</td><td colspan="2"></td></tr>
<tr><td rowspan="16">MO - Model options</td><td colspan="2">RE <file> - Read Model Files</td></tr>
<tr><td rowspan="15">DA - Read Data from model</td><td>GL <component> - Global data</td></tr>
<tr><td>PA <id> <component> - Part data</td></tr>
<tr><td>NO <id> <component> - Node data</td></tr>
<tr><td>SO <id> <component> - Solid data</td></tr>
<tr><td>BE <id> <component> - Beam data</td></tr>
<tr><td>SH <id> <component> - Shell data</td></tr>
<tr><td>TS <id> <component> - Thick Shell data</td></tr>
<tr><td>WA <id> <component> - Part data</td></tr>
<tr><td>SPR <id> <component> - Spring data</td></tr>
<tr><td>SEA <id> <component> - Seatbelt data</td></tr>
<tr><td>RET <id> <component> - Retractor data</td></tr>
<tr><td>SL <id> <component> - Slipring data</td></tr>
<tr><td>CO <id> <component> - Contact data</td></tr>
<tr><td>REA <id> <component> - Reaction data</td></tr>
<tr><td>AI <id> <component> - Airbag data</td></tr>
</table>

		JO <id> <component> - Joint data
		SEC <id> <component> - Section data
		SU <id> <component> - Subsystem data
		P_G <id> <component> - Part Group data
		G_C <id> <component> - Geometrical Contact data
		RI <id> <component> - Rigid Body data
		SPO <id> <component> - Spotweld data
		SPC <id> <component> - SPC data
		FS <id> <component> - Fluid structural interaction data
		BO <id> <component> - Boundary condition data
		SPH <id> <component> - SPH data
	SE - Select Models	
	DE - Delete Models	
	LI - List Models	
	SU - Set Surface	
RE - Read data	**CU** - Read T/HIS curve file	
	CU_NO - Read T/HIS curve file (ignore any style definitions)	
	BD - Read Bulk data file	
	KW - Read from LS-DYNA KEYWORD input file	
	KY - Input curve from keyboard	
	CSV - Read a CSV file (X,Y,X,Y,X,Y)	
	CSV2 - Read a CSV file (X,Y,|Y,Y,Y,Y)	
	ISO - Read ISO curve data (multiple channels)	
	ISO2 - Read ISO curve data (single channel)	
WR - Write options	**WR** - Write curve file	
	WA - Write all curves to a T/HIS curve file	

	KEY - Write curves to a LS-DYNA Keyword file	
	CSV - Write curves to a CSV file (X,Y,X,Y,X,Y)	
	CSV2 - Write curves to a CSV file (X,Y,|Y,Y,Y,Y)	
	LI - List curve data on screen	
	RE - Report curve data to file	
	SU - Sumary of curve	
	ST - Status	
DE - Defaults	**AU** - Auto Scaling	**ON** - Autoscaling on
		OFF - Autoscaling off
		DX - Define new x limits (minimum,maximum)
		XMN - Define new minimum x limit
		XMX - Define new maximum x limit
		DY - Define new y limit (min,max)
		YMN - Define new minimum y limit
		YMX - Define new maximun y limit
		2DY - Define new second y axis limits (min,max)
		YMN2 - Define new minimum second y limit
		YMX2 - Define new maximum second y limit
		ST - Status
	TI - Title	
	TI_ON - Toggle Title on	
	TI_OFF - Toggle Title off	
	LA - Axes labels (user defined)	**AU** - Use automatic axes labels (both)
		AX - Use automatic x axis labels
		AY - Use automatic y axis labels
		2AY - Use automatic 2nd y axis labels

		DX - Define new x axis plot label
		DY - Define new y axis plot label
		2DY - Define new 2nd y axis plot label
		ST - Status
	AW - Axis line width	
	AX - Axis types	
	AC - Axis Colour	
	GR - Grid lines	**ON** - Turn grid on
		OFF - Turn grid off
		AX - Automatic x-axis grid intervals
		AY - Automatic y-axis grid intervals
		MX - Manual x-axis grid intervals
		MY - Manual y-axis grid intervals
		IX - Define x-axis grid intervals
		IY - Define y-axis grid intervals
		OX - Define x-axis grid offset
		OY - Define y-axis grid offset
		TH - Define grid line thickness
	GW - Grid width	
	UL - User Line	
	LL - Line labels	
	MP - Model Prefix	**ON** - Turn model prefix on
		OFF - Turn model prefix off
		AUTO - Add prefix if more than one model
	PR - Prefix Format	**ID** - Model ID
		DIR - Model directory
		THF - Root of THF filename
		USER - User defined
DE - Defaults (continued)	**PF** - Plot format	
	WX - Window size (x) "pixels"	

	WY - Window size (y) "pixels"
	RV - Reverse Foregorund / Background
	FO - Foreground Colour
	BA - Background Colour
	CU - Curve through points ON/OFF
	SY - Symbols ON/OFF
	BD - Border ON/OFF
	BW - Border width
	BC - Border Colour
	LW - Default line width
	SMN - Show minimum value
	SMX - Show maximum value
	LXMN - Label x value at minimum
	LYMN - Label y value at minimum
	LXMX - Label x value at maximum
	LYMX - Label y value at maximum
	RE - Reset to defaults
	ST - Status
FO - Font	**TI <font> <size> <colour>** - Title
	XL <font> <size> <colour> - X Axis Label
	XU <font> <size> <colour> - X Axis Units
	YL <font> <size> <colour> - Y Axis Label
	YU <font> <size> <colour> - Y Axis Units
	Y2L <font> <size> <colour> - 2nd Y Axis Label
	Y2U <font> <size> <colour> - 2nd Y Axis Units
	LE <font> <size> <colour> - Curve Legend
	ALL <font> <size> <colour> - All labels
ED <curve ID> - Edit option	**F** - move Forward next 16 lines
	B - move Back 16 lines
	T - move to Top of curve
	E - move to End of curve
	n(umber) - move to line n
	C n - Change line n
	I n - Insert before line n
	A n - Append after line n

	D n1 n2 - Delete from line n1 to n2
	L - change Line label
	R - Reset edited curve back to original
	W or S - write curve
	PE - Plot Edited curve
	PA - Plot Edited And original curve
	PL - PLot stored T/HIS curves
	Q - Quit the editor
OP - Operate	**ADX/Y** - Add
	MUX/Y - Multiply
	SUX/Y - Subtract
	DIX/Y - Divide
	CAT - Concatenate 2 curves
	MAP - Map one curve onto another
	COM - Combine curves
	ERR - Error functions
	INT - Integrate
	DIF - Differentiate
	SMO - Smooth
	LSQ - Least squares fit
	SQR - Square root
	NOR - Normalise
	REC - Reciprocal
	ABS - Absolute values
	TRA - Translate
	REV - Reverse
	CLP - Clip
	ZERO - Translate the curve to (0,0)
	ORDER - Reverse the order of the curve points
	VEC - Vector magnitude
	VEC2 - Vector Magnitude (2D)
	SUM - Sum of 'n' curves
	ENV - Envelope of 'n' curves
	MIN - Minimum of 'n' curves
	MAX - Maximum of 'n' curves

	AVE - Average of 'n' curves
	R-AV - Rolling Average of 'n' curves
	STR - Convert stress/strain curve
AM - Automotive options	**C60** - Class 60 filter
	C180 - Class 180 filter
	C600 - Class 600 filter
	C1000 - Class 100 filter
	BUT - Butterworth filter
	FIR - FIR filter
	HIC - HIC value
	HICD - HIC(d) value
	CLI - 3ms Clip value
	EXC - Exceedence Plot
	VC - Viscous Criteria (ECER95)
	VC2 - Viscous Criteria (IIHS)
	ASI - Acceleration Severity Index (BS EN 1317-1:1998)
	ASI2 - Acceleration Severity Index (BS EN 1317-1:2010)
	THIV - Theoretical Head Impact Velocity
	NIJ - Neck Injury
	TTI - Thoracic Trauma Index
	NOR - Normalise
	REG - Regularise
	VEC - Vector Magnitude
	VEC2 - Vector Magnitude (2D)
	ACU - Airbag Control Unit
MA - Maths operations	**SQRT** - Squre Root
	LOG - Natural Log
	EXP - e to power of
	LOG10 - Log to base 10
	****** - To raise to power
	SIN - Sine
	COS - Cosine
	TAN - Tangent
	ASIN - Arc sine
	ACOS - Arc cosine

	ATAN - Arc tangent
SE - Seismic options	**DV** - Displacement to velocity spectra
	DA - Displacement to acceleration spectra
	VD - Velocity to displacement spectra
	VA - Velocity to acceleration spectra
	AD - Acceleration to displacement spectra
	AV - Acceleration to velocity spectra
	DS - Produce a design spectrum from a response spectrum
	RS - Produce response spectra from input accelerations
	FFT - Fast fourier transformation
UT - Utility functions	**CL** - Colour laser output
	GL - Greyscale laser output
	LW - Line width
	SA - Solid axes (x=0 & y=0 axes solid)
ST - Line styles	**RE** - Read in style file
	WR - Write out style file
	DE - Reset styles to default settings
	SET - Set a T/HIS line style
	FIX - Turn fix line styles on/off
HE - Help	
CU - Curve editing options	**LA** - Set a new curve label
	TI - Set a new curve title
	XL - Set a new curve x-axis label
	YL - Set a new curve y-axis label
	TA - Set a new curve tag
PGR or **GRO** - Group options	**READ** - Read a PART group file
	LIST - List all PART groups
	DELETE - Delete all PART groups
	CREATE - Create a new PART group
CGR - Group options	**CREATE** - Create a curve group
	LIST - List all curve groups
	ADD - Add to an existing curve group
	REMOVE - Remove from an existing curve group
IM - Image output options	**JPEG <file>** - Capture a JPEG image
	BMP_U <file> - Capture an uncompressed Bitmap image

	BMP_C <file> - Capture a compressed Bitmap image
	PPM <file> - Capture a portable pixmap file
PREF - Define T/HIS user preferences	**REG** - Set time interval for automatic curve resularising
	CONV - Set/unset automatic conversion from ms to s when filtering
	FILE - Turn on/off output of injury criteria values and error calculations to ASCII files
	SHOW - Turn on/off display of HIC/ 3ms clip values
	ZERO - Turn on/off automatic creation of (0,0) point when reading data from ASCII files
GRAPH - Graphics window commands for linked T/HIS	**TI_ON** - Turns the Timeline on
	TI_OFF - Turns the Timeline off
	EXIT - Closes the T/HIS link

14. Installation Organisation

Installation organisation

The Oasys Suite 21.0 installation can be customised to try and avoid a number of issues that often occur in large organisations with many users.

- Large organisations generally imply large networks, and it is often the case that the performance of these networks can be intermittent or poor, therefore it is common practice to perform an installation of the software on the local disk of each machine, rather then having a single installation on a remote disk.

  This avoids the pauses and glitches that can occur when running executable files over a network, but it also means that all the configuration files in, or depending upon, the top level "Admin" directory have to be copied to all machines and, more to the point, any changes or additions to such files also have to be copied to all machines.

- In larger organisations the "one person per computer" philosophy may not apply, with the consequence that users will tend to have a floating home area on a network drive and may not use the same machine every day.

  This is not usually a problem on Linux where the "home" directory is tied to the login name not the machine. However on Windows platforms it means that %USERPROFILE%, which is typically on the local C drive of a machine, is not a good place to consider as "home" since it will be tied to a given computer, therefore a user who saves a file in their home directory on machine A may not be able to access it from machine B.

- In a similar vein placing large temporary files on the /tmp partition (Linux) or the C: drive (Windows) may result in local disks becoming too full, or quotas exceeded.

This section gives only a brief summary of the installation organisation, and you should refer to the separate Installation Guide if you want to find out more about the details of installation, licensing, and other related issues.

14.1. Version 21 Installation Structure

Oasys Suite 21.0 Installation structure

In Oasys Suite 21.0 the option is provided to separate a top-level 'administration' directory from the 'installation' one where the executables are located.

For large installations on many machines this allows central configuration and administration files to exist in one place only, but executables to be installed locally on users' machines to give better performance. Oasys Suite 21.0 also allows the following items to be configured

- The location for user manuals and other documentation.
- The definition of a user's home directory.
- The definition of the temporary directory for scratch files.

In addition parsing of the 'oa_pref' (preferences) file will now handle environment variables, so that a generic preference can be configured to give a user-specific result, and preferences may be 'locked' so that those set at the administration level cannot be changed by users.

These changes are entirely optional, and users performing a simple installation on a single machine do not need to make any changes to their existing installation practice.

Directory	Status	Directory Content and purpose	oa_pref file option
OA_ADMIN_xx	*Optional*	Top level configuration files. (**xx** =21 for Oasys Suite 21.0, thus **OA_ADMIN_21**) Admin level oa_pref file Other configuration files Timeout configuration file	
OA_ADMIN	*Optional*	Same as **OA_ADMIN_21**, provided for backwards compatibility with earlier releases. It is recommended that plain **OA_ADMIN**, without the **_xx** version suffix, is not used since otherwise there is no easy way of distinguishing between parallel installations of different releases of the Oasys Ltd software in an	

		installation. *If **OA_ADMIN_21** is not defined then this non-release specific version is checked.*	
OA_INSTALL_xx	*Optional*	(**xx** =21 for release 21.0, thus **OA_ADMIN_21** All executables Installation level oa_pref file	**oasys*install_dir:** ***<pathname>***
OA_INSTALL	*Optional*	Same as **OA_INSTALL_21**. If no "**OA_ADMIN_xx**" directory is used and all software is simply placed in this "install" directory, which would be typical of a single-user installation, then it is recommended that the **_xx** version suffix is used in order to keep parallel installations of different releases of the Oasts Ltd software separate on the machine. *If **OA_INSTALL_21** is not defined then this non-release specific version is checked*	**oasys*install_dir:** ***<pathname>***
OA_MANUALS	*Optional*	Specific directory for user manuals. If not defined then will search in: **OA_ADMIN_xx/manuals** (xx = major version number) **OA_INSTALL/manuals**	**oasys*manuals_dir:** ***<pathname>***
OA_HOME	*Optional*	Specific "home" directory for user when using Oasys Ltd software. If not defined will use: **$HOME** (Linux) **%USERPROFILE%** (Windows)	**oasys*home_dir:** ***<pathname>***
OA_TEMP	*Optional*	Specific "temporary" directory for user when using Oasys Ltd software. If not defined will use: **P_tmpdir** (Linux, typically /tmp) **%TEMP%** (Windows, typically C:\temp)	**oasys*temp_dir:** ***<pathname>***

It will be clear from the table above that no Environment variables have to be set, and that all defaults will revert to pre-Oasys Suite 9.4 behaviour. In other words users wishing to keep the status quo will find behaviour and layout unchanged if they do nothing.

OA_INSTALL_XX

Previously the software used the **OA_INSTALL** (renamed from **OASYS**) environment variable to locate the directory the software was installed in.

- On Windows this is no longer required as the software can work out its own installation directory. As this environment variable is no longer required it is recommended that it is removed from machines it is currently set on as in some cases where more than one version has been installed in different directories it can cause problems.
- On LINUX systems the "oasys_21" script that starts the SHELL automatically sets this Environment Variable and passes it to any application started from the SHELL. If you run applications directly from the command line and bypass the SHELL then you should set **OA_INSTALL_XX** so that the software can locate manuals and other required files.

OA_ADMIN_XX

Users wishing to separate configuration and installation directories will be able to do so by making use of the new top level **OA_ADMIN_xx** directory.

14.1.1. Installation Examples

Installation Examples

The following diagrams illustrate how the installation might be organised in various different scenarios..

a) Single user installation on one machine

There is no need to worry about separating administration and installation directories, and the default installation of all files in and below the single installation directory will suffice.

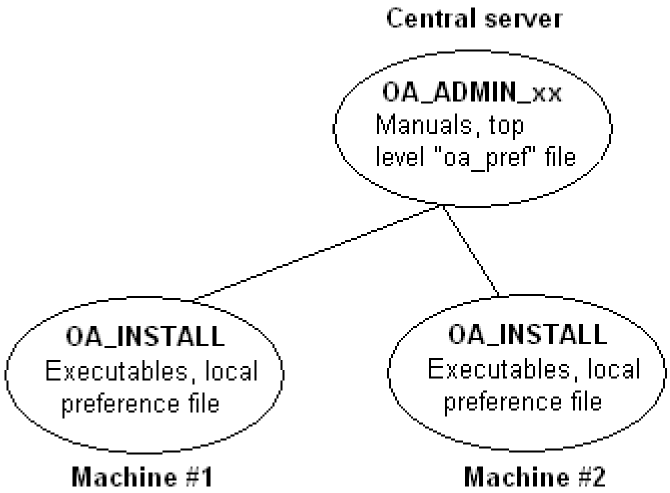
It is suggested that the **`_xx`** version suffix of **`OA_INSTALL_xx`** is used in order to keep parallel installations of different releases of the Oassys Ltd software separate on the machine.

b) A few machines on a small network, each user has their own machine

The top level administration directory can be installed on a network server, possibly also locating the manuals centrally.

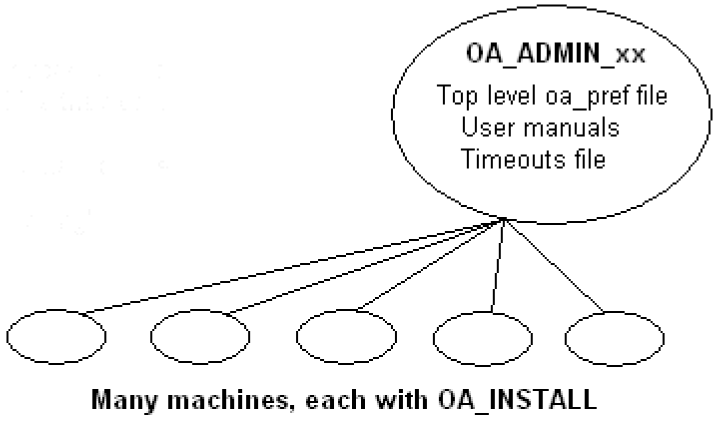
Each user's machine has its own 'installation' directory to give good performance, but there is no need to manage home or temporary directories centrally since each user 'owns' their machine.

If network performance is good an alternative would be to install executables on the central server, meaning that local OA_INSTALL directories are not required.



c) Large corporate network

There is no need to worry about separating administration and installation directories, and the default installation of all files in and below the single installation directory will suffice.



14.1.2. Dynamic Configuration Using the Top Level oa_pref File

Dynamic configuration using the top level oa_pref file.

A further improvement is that all environment variables below **`OA_ADMIN_xx`** may either be set explicitly, or dynamically using the options in the oa_pref file at the top **`OA_ADMIN_xx`** level. This permits parallel installations of different versions of the software to co-exist, with only the top level administration directory names being distinct. For example:

Oasys Suite 21.0	**Oasys Suite 21.1**
Top level directory **`OA_ADMIN_21`**	Top level directory **`OA_ADMIN_211`**
oa_pref file in **`OA_ADMIN_21`** contains: **`oasys*install_dir:`** *<pathname for **21.0** installation>* **`oasys*manuals_dir:`** *<pathname for **21.0** manuals>* **`oasys*home_dir:`** *<pathname for home directory>* **`oasys*temp_dir:`** *<pathname for temporary files>*	oa_pref file in **`OA_ADMIN_211`** contains: **`oasys*install_dir:`** *<pathname for **21.1** installation>* **`oasys*manuals_dir:`** *<pathname for **21.1** manuals>* } would almost certainly be unchanged between major } versions, although they could be different if desired
Pathnames in the oa_pref file may contain environment variables which will be resolved before being applied.	

14.1.3. The Hierarchy of oa_pref File Reading

The hierarchy of oa_pref file reading

It will be clear from the above that in a large installation the "oa_pref" files have a significant role. Each piece of software reads them in the following order:

`OA_ADMIN_xx`	Top level configuration
`OA_INSTALL_xx`	Installation level
`OA_HOME`	User's personal "home" file
Current working directory	File specific to the current directory (rarely used)

The rules for reading these files are:

- If a given directory does not exist, or no file is found in that directory, then no action is taken. This is not an error.
- A more recently read definition supersedes one read earlier, therefore "local" definitions can supersede "global" ones (unless it was locked).
- If two of more of the directories in the table above are the same then that file is only read once from the first instance.

14.1.4. Locking Preference Options

Locking Preference Options

From Oasys Suite 9.4 onwards, preference options can be locked. If a preference option is locked in a file then that preference option will be ignored in any of the subsequent preference files that are read.

Therefore by locking a preference in a top-level file in the hierarchy above, eg in **`OA_ADMIN_xx`**, and then protecting that file to be read-only, an administrator can set preferences that cannot be altered by users since any definitions of that preference in their private oa_pref files will be ignored.

Preferences are locked by using a hash (#) rather than an asterisk (*) between the code name and the preference string. For example:

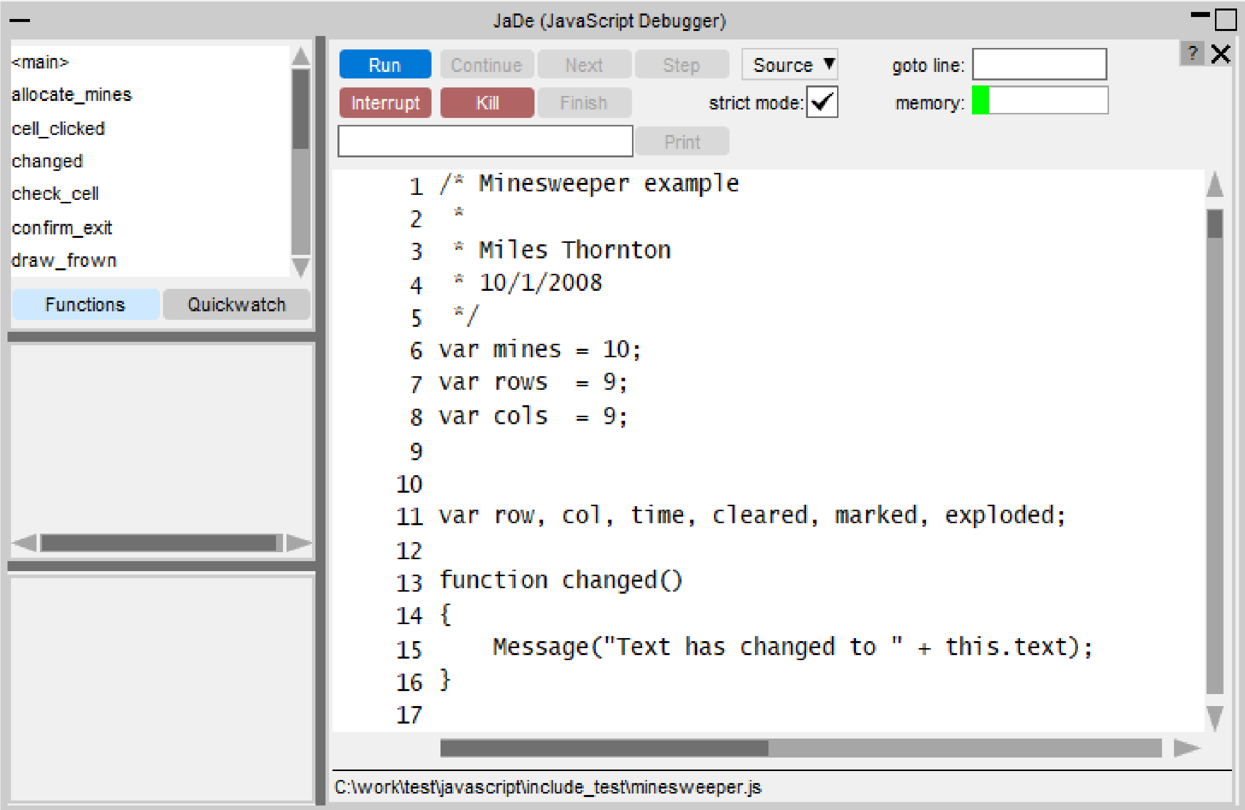
`*maximise: true`	Normal case using "*", means an unlocked preference
`#maximise: true`	Locked case using "#"

These changes may be made either by editing the file manually, or by using the preferences editor.

15. JaDe: The JavaScript debugger

JaDe: The JavaScript debugger

JaDe is included in D3PLOT, PRIMER and T/HIS to help debug and develop JavaScripts. It is started by selecting a script and pressing the **Debug** button in the JavaScript menu in any of the programs. The initial screen is shown below.



It is fairly basic but hopefully has enough functionality for people to be able to find and fix problems in scripts.

15.1. Viewing the Script Files and Functions

Viewing the script files and functions

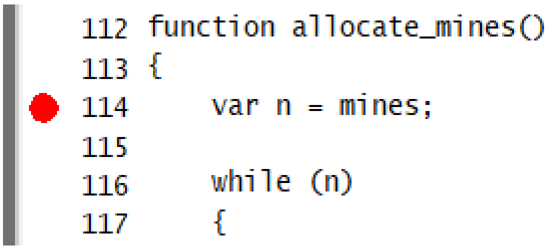
The main part of the window shows the script file. If your script is broken up into separate file (by using Use) then you can get a list of the different files and view them by using the **Source** popup. To go to a particular line in the file use the **goto line** textbox.

A list of the functions in the script is shown in the **Functions** menu on the top left. If you want to look at a particular function then click on the function name and the main text window will jump to the correct file and line.

15.2. Adding/Removing Breakpoints

Adding/removing breakpoints

A breakpoint is a line in the script where execution will pause in JaDe. To add a breakpoint either left click on the line you want the breakpoint on or right click on the line and select **Create breakpoint** from the popup. A red circle is then drawn on the line to show that there is an active breakpoint.

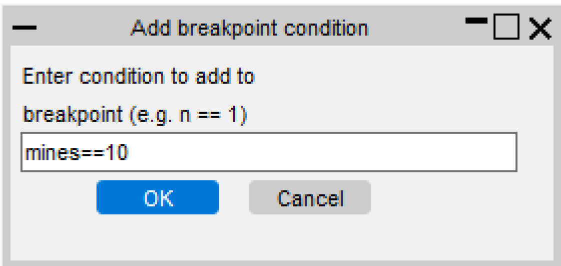


Additionally the breakpoint will also be added to the list in the breakpoint window (bottom left of JaDe). You can click on this at any time and the main text window will jump to the correct file and line.
Active breakpoints are shown with a red circle. Breakpoints can be activated/deactivated by clicking on the line again. Unactive breakpoints are shown as a grey circle instead of a red one. They are also shown in grey text in the breakpoint window .

To delete a breakpoint right click on the line and select **Delete breakpoint**. The breakpoint will be deleted.

Conditional breakpoints

Sometimes it is useful to only stop at a breakpoint if a certain condition is met. For example in the above example we may only want to stop at line 114 if `mines` is 10. You can do this by right clicking on the the breakpoint and selecting **Add condition**.



A window is mapped allowing you type in the condition you want to try to meet. The condition should be a JavaScript expression which evaluates to true if you want the breakpoint to stop execution, or false if you want the breakpoint to be skipped. In this example the condition is `n == 10`.

If a breakpoint has a condition associated with it a C is drawn on the circle and in the breakpoint window. The condition can be edited again or removed by right clicking on the breakpoint and selecting either **Edit condition** or **Remove condition** from the popup.

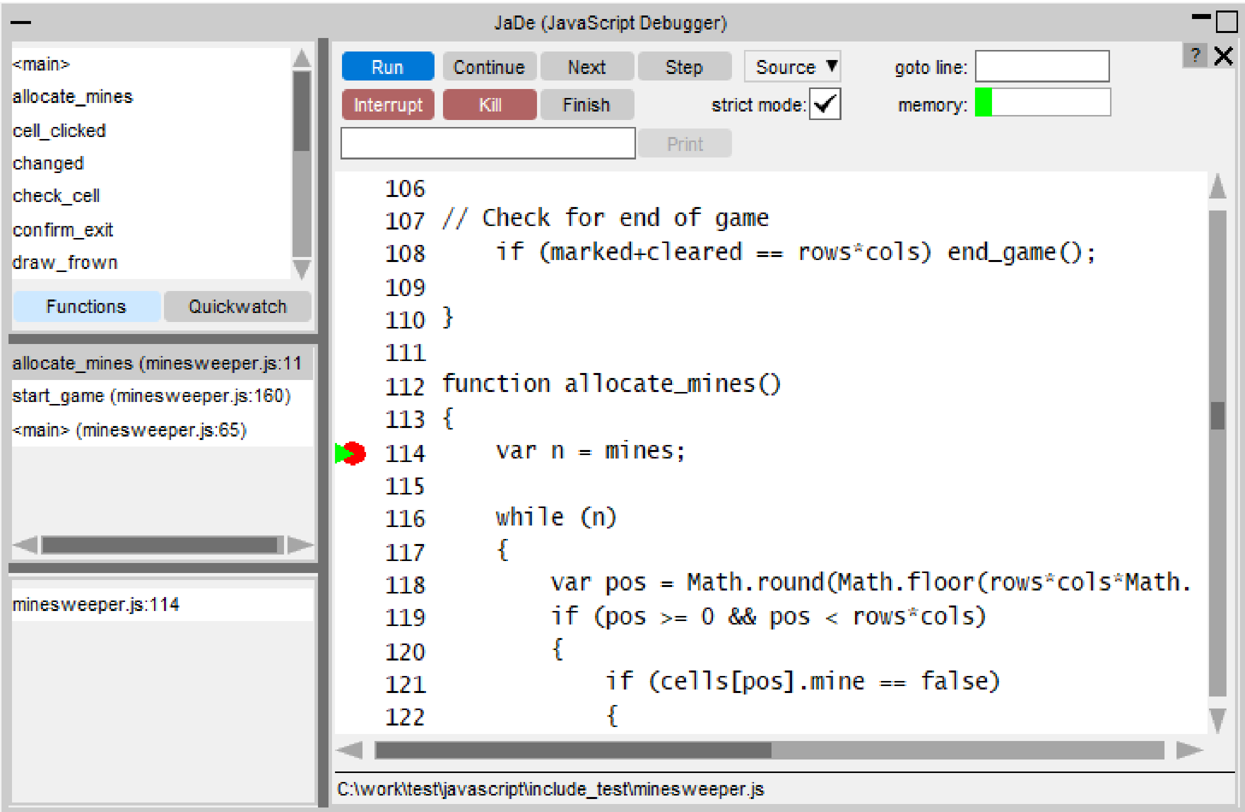
15.3. Running the Script

Running the script

Running the script is controlled by the buttons at the top of the debugger window. By default the script will be run in the debugger in 'strict mode'. This tries to pick up things which you might not have intended by running the script in a stricter environment doing more checking. You can toggle this on/off by using the **strict mode** checkbox.

Starting and stopping

To start the script press the **Run** button. Execution of the script will start. If you have not defined any breakpoints then the script will run until it finishes (unless there are some script errors or exceptions). If there is a breakpoint then the debugger will stop execution of the script when it reaches it. If the script is running and you want to pause execution of the script at any time you can press **Interrupt**.



The line that the debugger has paused the script on is shown by a green triangle. In the above example it is paused at line 114. The middle panel on the left shows the call stack. See the call stack section for more details.

Stepping and continuing

Once the script is paused in the debugger you can step through the source code by using the **Continue**, **Next**, **Step** and **Finish** buttons.

Continue will resume execution of the script again.
Next continues to the next line in the current function. i.e. it will step *over* a function call.
Step continues execution to the next source line (which may be in a different function. i.e. it will step *into* a function call).
Finish will finish executing the current function and stop at the next line in the calling function (the function above this in the call stack).

Alternatively, if you want to continue until a particular line you can right click on the line you want to continue until and select **Continue to here** from the popup.

15.4. Printing the Value of a Variable

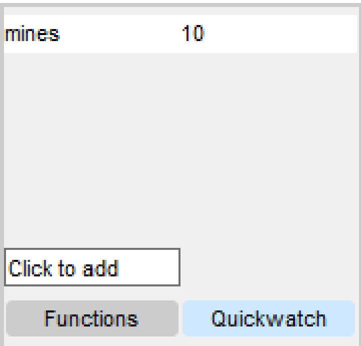
Printing the value of a variable

If you want to see the value of a variable you can type the name of the variable you want to see in the textbox at the top of the debugger and press **Print**. JaDe will evaluate the variable and output the result in the statusbar at the bottom of the debugger.

Using Quickwatch

If you want to look at the values for lots of variables it is annoying to have to type the variable name in and press **Print** for each one. A better way is to use **Quickwatch** at the top left of JaDe

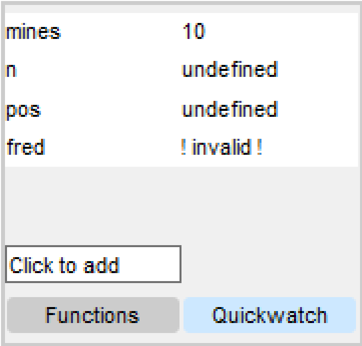
Type the name of the variable that you want to watch in the **Click to add** textbox. A line will be added for the variable showing its name and value. e.g. in the following image the variable `mines` is being displayed and its current value is `10`. If the value is very long hover over the value to get the whole string.



You can add any number of variables to watch. To remove one right click on the variable and select **Remove quickwatch** from the popup.

If a variable exists and has been assigned to then the value is displayed. e.g. `mines` in the following example.
If the variable exists but it has not yet had a value assigned its value is the `undefined` value. e.g. `pos` in the following example.

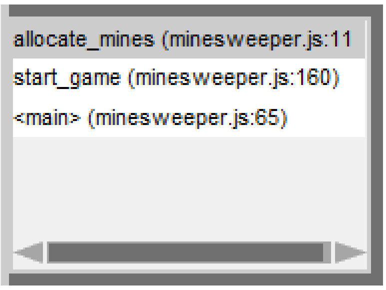
If the variable does not exist the value is shown as `! invalid !`. e.g. `fred` in the following example.



15.5. The Call Stack

The call stack

The call stack shows which functions have been called in the script to get to the current point. It is the middle left window in JaDe.



The top line shows the function that the script is currently paused at. The other lines show the calling functions in order. The above example can be read as:

1. The script starts
2. On line 65 in script file minesweeper.js in the 'main' program the function `start_game` is called.
3. On line 160 in script file minesweeper.js in function `start_game` the function `allocate_mines` is called
4. On line 114 in script file minesweeper.js in function `allocate_mines` the script is paused.

This information is sometimes very useful in more complicated scripts to find out the order things are done in.

The function that the user is currently looking at is highlighted in blue. You can move up or down the call stack by clicking on a line. The main text window will jump to the correct file and line. The line will be shown with a blue triangle instead of a green triangle.

15.6. Exceptions

Exceptions

Sometimes when developing a script you get errors that you need to try to investigate and fix. e.g. an object is null when it should be defined or you try to call a method that does not exist for an object. In these cases an exception is thrown by JavaScript and the script would terminate is run normally. JaDe will trap the exception and stop at the line where the exception occured. e.g. If for example you has the following code:

Copy Code
JavaScript

```javascript
var w = new Window('Example', 0.5, 1.0, 0.5, 1.0);

w.BadMethod();

w.Show()
```

There is no method called `BadMethod` for a `Window`. JaDe will stop at this point and allow you to look at the script.

15.7. Memory Usage

Memory usage

When a script creates arrays, objects or strings it has to allocate some memory to be able to do so (for example an array storing 1,000,000 items will use considerably more memory than an array to store 100 items). To manage this memory JavaScript uses a process called 'garbage collection'. When the array, object or string goes out of scope (can no longer be reached by the script) it can be garbage collected and the memory freed. For the JavaScript engine to be able to do this it must keep track of what memory has been allocated. It does this by keeping a list of the live memory. This list also uses a small amount of memory and this memory is the garbage collection memory. The maximum size for the garbage collection memory is set when running a script.

JaDe allows you do see how much garbage collection memory has been used with a usage bar.

If you hover over the usage bar you can see exactly how much garbage collection memory is being used. As the JavaScript engine allocates memory for objects, arrays etc this will increase. When the engine performs garbage collection to free memory the usage will go down. Note that the engine will normally only perform garbage collection when it thinks it is necessary so if you run a script multiple times in JaDe the memory could continue to increase until the engine decides to do garbage collection, then the memory will reduce.

Note also that JaDe also requires some garbage collection memory to function so the bar also includes some memory for JaDe.

16. Licences Used in Software

Licences used in software

The Oasys LS-DYNA Environment Ltd software uses several third party libraries and executables. The licences for them are given below

16.1. Apple Public Source

Apple Public Source

```
Copyright (c) 1999 Apple Computer, Inc. All rights reserved.

The contents of this file constitute Original Code as defined in and
are subject to the Apple Public Source License Version 1.1 (the
"License"). You may not use this file except in compliance with the
License. Please obtain a copy of the License at
http://www.apple.com/publicsource and read it before using this
file.

This Original Code and all software distributed under the License
are
distributed on an "AS IS" basis, WITHOUT WARRANTY OF ANY KIND,
EITHER
EXPRESS OR IMPLIED, AND APPLE HEREBY DISCLAIMS ALL SUCH WARRANTIES,
INCLUDING WITHOUT LIMITATION, ANY WARRANTIES OF MERCHANTABILITY,
FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT. Please see the
License for the specific language governing rights and limitations
under the License.

Copyright (c) 1992 NeXT Computer, Inc.  All rights reserved.
```

Note: the URL http://www.apple.com/publicsource cited above no longer exists, see instead https://spdx.org/licenses/APSL-1.1.html

T/HIS

16.2. Draco

Draco

google/draco is licensed under the Apache License:

```
Licensed under the Apache License, Version 2.0 (the "License");
you may not use this file except in compliance with the License.
You may obtain a copy of the License at

    http://www.apache.org/licenses/LICENSE-2.0

Unless required by applicable law or agreed to in writing, software
distributed under the License is distributed on an "AS IS" BASIS,
WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or
implied.
See the License for the specific language governing permissions and
limitations under the License.
```

T/HIS

16.3. Expat

Expat

```
Copyright (c) 1998, 1999, 2000 Thai Open Source Software Center Ltd
                               and Clark Cooper
Copyright (c) 2001, 2002, 2003, 2004, 2005, 2006 Expat maintainers.

Permission is hereby granted, free of charge, to any person
obtaining
a copy of this software and associated documentation files (the
"Software"), to deal in the Software without restriction, including
without limitation the rights to use, copy, modify, merge, publish,
distribute, sublicense, and/or sell copies of the Software, and to
permit persons to whom the Software is furnished to do so, subject
to
the following conditions:

The above copyright notice and this permission notice shall be
included
in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND,
EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF
MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND
NONINFRINGEMENT.
IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY
CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT,
TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE
SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE
```

16.4. FreeType

FreeType


```
                    The FreeType Project LICENSE
                    ----------------------------

                            2006-Jan-27

                    Copyright 1996-2002, 2006 by
          David Turner, Robert Wilhelm, and Werner Lemberg



Introduction
============

  The FreeType  Project is distributed in  several archive packages;
  some of them may contain, in addition to the FreeType font engine,
  various tools and  contributions which rely on, or  relate to, the
  FreeType Project.

  This  license applies  to all  files found  in such  packages, and
  which do not  fall under their own explicit  license.  The license
  affects  thus  the  FreeType   font  engine,  the  test  programs,
  documentation and makefiles, at the very least.

  This  license   was  inspired  by  the  BSD,   Artistic,  and  IJG
  (Independent JPEG  Group) licenses, which  all encourage inclusion
  and  use of  free  software in  commercial  and freeware  products
  alike.  As a consequence, its main points are that:

    o We don't promise that this software works. However, we will be
      interested in any kind of bug reports. (`as is' distribution)

    o You can  use this software for whatever you  want, in parts or
      full form, without having to pay us. (`royalty-free' usage)

    o You may not pretend that  you wrote this software.  If you use
      it, or  only parts of it,  in a program,  you must acknowledge
      somewhere  in  your  documentation  that  you  have  used  the
      FreeType code. (`credits')

  We  specifically  permit  and  encourage  the  inclusion  of  this
  software, with  or without modifications,  in commercial products.
  We  disclaim  all warranties  covering  The  FreeType Project  and
  assume no liability related to The FreeType Project.
```

Finally, many people asked us for a preferred form for a credit/disclaimer to use in compliance with this license. We thus encourage you to use the following text:

```
"""
 Portions of this software are copyright  <year> The FreeType
 Project (www.freetype.org).  All rights reserved.
"""
```

Please replace <year> with the value from the FreeType version you actually use.

Legal Terms
===========

0. Definitions

Throughout this license, the terms `package', `FreeType Project', and `FreeType archive' refer to the set of files originally distributed by the authors (David Turner, Robert Wilhelm, and Werner Lemberg) as the `FreeType Project', be they named as alpha, beta or final release.

`You' refers to the licensee, or person using the project, where `using' is a generic term including compiling the project's source code as well as linking it to form a `program' or `executable'. This program is referred to as `a program using the FreeType engine'.

This license applies to all files distributed in the original FreeType Project, including all source code, binaries and documentation, unless otherwise stated in the file in its original, unmodified form as distributed in the original archive. If you are unsure whether or not a particular file is covered by this license, you must contact us to verify this.

The FreeType Project is copyright (C) 1996-2000 by David Turner, Robert Wilhelm, and Werner Lemberg. All rights reserved except as specified below.

1. No Warranty

THE FREETYPE PROJECT IS PROVIDED `AS IS' WITHOUT WARRANTY OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. IN NO EVENT WILL ANY OF THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY DAMAGES CAUSED BY THE USE OR THE INABILITY TO USE, OF THE FREETYPE PROJECT.

2. Redistribution

This license grants a worldwide, royalty-free, perpetual and irrevocable right and license to use, execute, perform, compile, display, copy, create derivative works of, distribute and sublicense the FreeType Project (in both source and object code forms) and derivative works thereof for any purpose; and to authorize others to exercise some or all of the rights granted herein, subject to the following conditions:

- o Redistribution of source code must retain this license file (`FTL.TXT') unaltered; any additions, deletions or changes to the original files must be clearly indicated in accompanying documentation. The copyright notices of the unaltered, original files must be preserved in all copies of source files.

- o Redistribution in binary form must provide a disclaimer that states that the software is based in part of the work of the FreeType Team, in the distribution documentation. We also encourage you to put an URL to the FreeType web page in your documentation, though this isn't mandatory.

These conditions apply to any software derived from or based on the FreeType Project, not just the unmodified files. If you use our work, you must acknowledge us. However, no fee need be paid to us.

3. Advertising

Neither the FreeType authors and contributors nor you shall use the name of the other for commercial, advertising, or promotional purposes without specific prior written permission.

We suggest, but do not require, that you use one or more of the following phrases to refer to this software in your documentation or advertising materials: `FreeType Project', `FreeType Engine', `FreeType library', or `FreeType Distribution'.

As you have not signed this license, you are not required to accept it. However, as the FreeType Project is copyrighted material, only this license, or another one contracted with the authors, grants you the right to use, distribute, and modify it. Therefore, by using, distributing, or modifying the FreeType Project, you indicate that you understand and accept all the terms of this license.

4. Contacts

There are two mailing lists related to FreeType:

- o freetype@nongnu.org

  Discusses general use and applications of FreeType, as well as future and wanted additions to the library and distribution.

```
      If  you are looking  for support,  start in  this list  if you
      haven't found anything to help you in the documentation.

    o freetype-devel@nongnu.org

      Discusses bugs,  as well  as engine internals,  design issues,
      specific licenses, porting, etc.

  Our home page can be found at

    http://www.freetype.org


--- end of FTL.TXT ---
```

T/HIS

16.5. FFmpeg

FFmpeg

```
FFmpeg is licensed under the LGPL v2.1+. The exception to this is
the x264
library used by FFmpeg, for which Arup have obtained a commercial
license (see
here).

# License

Most files in FFmpeg are under the GNU Lesser General Public License
version 2.1
or later (LGPL v2.1+). Read the file `COPYING.LGPLv2.1` for details.
Some other
files have MIT/X11/BSD-style licenses. In combination the LGPL v2.1+
applies to
FFmpeg.

Some optional parts of FFmpeg are licensed under the GNU General
Public License
version 2 or later (GPL v2+). See the file `COPYING.GPLv2` for
details. None of
these parts are used by default, you have to explicitly pass `--
enable-gpl` to
configure to activate them. In this case, FFmpeg's license changes
to GPL v2+.

Specifically, the GPL parts of FFmpeg are:

- libpostproc
- optional x86 optimization in the files
    - `libavcodec/x86/flac_dsp_gpl.asm`
    - `libavcodec/x86/idct_mmx.c`
    - `libavfilter/x86/vf_removegrain.asm`
- the following building and testing tools
    - `compat/solaris/make_sunver.pl`
    - `doc/t2h.pm`
    - `doc/texi2pod.pl`
    - `libswresample/tests/swresample.c`
    - `tests/checkasm/*`
    - `tests/tiny_ssim.c`
- the following filters in libavfilter:
    - `signature_lookup.c`
    - `vf_blackframe.c`
    - `vf_boxblur.c`
    - `vf_colormatrix.c`
    - `vf_cover_rect.c`
    - `vf_cropdetect.c`
    - `vf_delogo.c`
    - `vf_eq.c`
    - `vf_find_rect.c`
```

  - `vf_fspp.c`
  - `vf_histeq.c`
  - `vf_hqdn3d.c`
  - `vf_kerndeint.c`
  - `vf_lensfun.c` (GPL version 3 or later)
  - `vf_mcdeint.c`
  - `vf_mpdecimate.c`
  - `vf_nnedi.c`
  - `vf_owdenoise.c`
  - `vf_perspective.c`
  - `vf_phase.c`
  - `vf_pp.c`
  - `vf_pp7.c`
  - `vf_pullup.c`
  - `vf_repeatfields.c`
  - `vf_sab.c`
  - `vf_signature.c`
  - `vf_smartblur.c`
  - `vf_spp.c`
  - `vf_stereo3d.c`
  - `vf_super2xsai.c`
  - `vf_tinterlace.c`
  - `vf_uspp.c`
  - `vf_vaguedenoiser.c`
  - `vsrc_mptestsrc.c`

Should you, for whatever reason, prefer to use version 3 of the (L)GPL, then
the configure parameter `--enable-version3` will activate this licensing option
for you. Read the file `COPYING.LGPLv3` or, if you have enabled GPL parts,
`COPYING.GPLv3` to learn the exact legal terms that apply in this case.

There are a handful of files under other licensing terms, namely:

* The files `libavcodec/jfdctfst.c`, `libavcodec/jfdctint_template.c` and
  `libavcodec/jrevdct.c` are taken from libjpeg, see the top of the files for
  licensing details. Specifically note that you must credit the IJG in the
  documentation accompanying your program if you only distribute executables.
  You must also indicate any changes including additions and deletions to
  those three files in the documentation.
* `tests/reference.pnm` is under the expat license.

External libraries

FFmpeg can be combined with a number of external libraries, which sometimes affect the licensing of binaries resulting from the combination.

Compatible libraries

The following libraries are under GPL version 2:
- avisynth
- frei0r
- libcdio
- libdavs2
- librubberband
- libvidstab
- libx264
- libx265
- libxavs
- libxavs2
- libxvid

When combining them with FFmpeg, FFmpeg needs to be licensed as GPL as well by
passing `--enable-gpl` to configure.

The following libraries are under LGPL version 3:
- gmp
- libaribb24
- liblensfun

When combining them with FFmpeg, use the configure option `--enable-version3` to
upgrade FFmpeg to the LGPL v3.

The VMAF, mbedTLS, RK MPI, OpenCORE and VisualOn libraries are under the Apache License
2.0. That license is incompatible with the LGPL v2.1 and the GPL v2, but not with
version 3 of those licenses. So to combine these libraries with FFmpeg, the
license version needs to be upgraded by passing `--enable-version3` to configure.

The smbclient library is under the GPL v3, to combine it with FFmpeg,
the options `--enable-gpl` and `--enable-version3` have to be passed to
configure to upgrade FFmpeg to the GPL v3.

Incompatible libraries

There are certain libraries you can combine with FFmpeg whose licenses are not
compatible with the GPL and/or the LGPL. If you wish to enable these
libraries, even in circumstances that their license may be incompatible, pass
`--enable-nonfree` to configure. This will cause the resulting binary to be
unredistributable.

The Fraunhofer FDK AAC and OpenSSL libraries are under licenses which are
incompatible with the GPLv2 and v3. To the best of our knowledge, they are
compatible with the LGPL.

16.6. HDF5

HDF5

```
Copyright Notice and License Terms for
HDF5 (Hierarchical Data Format 5) Software Library and Utilities
--------------------------------------------------------------------
---------

HDF5 (Hierarchical Data Format 5) Software Library and Utilities
Copyright 2006 by The HDF Group.

NCSA HDF5 (Hierarchical Data Format 5) Software Library and
Utilities
Copyright 1998-2006 by The Board of Trustees of the University of
Illinois.

All rights reserved.

Redistribution and use in source and binary forms, with or without
modification, are permitted for any purpose (including commercial
purposes)
provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright
notice,
   this list of conditions, and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright
notice,
   this list of conditions, and the following disclaimer in the
documentation
   and/or materials provided with the distribution.

3. Neither the name of The HDF Group, the name of the University,
nor the
   name of any Contributor may be used to endorse or promote
products derived
   from this software without specific prior written permission from
   The HDF Group, the University, or the Contributor, respectively.

DISCLAIMER:
THIS SOFTWARE IS PROVIDED BY THE HDF GROUP AND THE CONTRIBUTORS
"AS IS" WITH NO WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED.
IN NO
EVENT SHALL THE HDF GROUP OR THE CONTRIBUTORS BE LIABLE FOR ANY
DAMAGES
SUFFERED BY THE USERS ARISING OUT OF THE USE OF THIS SOFTWARE, EVEN
IF
ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

You are under no obligation whatsoever to provide any bug fixes,
patches, or
```

upgrades to the features, functionality or performance of the source code
("Enhancements") to anyone; however, if you choose to make your Enhancements
available either publicly, or directly to The HDF Group, without imposing a
separate written license agreement for such Enhancements, then you hereby
grant the following license: a non-exclusive, royalty-free perpetual license
to install, use, modify, prepare derivative works, incorporate into other
computer software, distribute, and sublicense such enhancements or derivative
works thereof, in binary and source code form.

Limited portions of HDF5 were developed by Lawrence Berkeley National
Laboratory (LBNL). LBNL's Copyright Notice and Licensing Terms can be
found here: COPYING_LBNL_HDF5 file in this directory or at
http://support.hdfgroup.org/ftp/HDF5/releases/COPYING_LBNL_HDF5.

Contributors: National Center for Supercomputing Applications (NCSA) at
the University of Illinois, Fortner Software, Unidata Program Center
(netCDF), The Independent JPEG Group (JPEG), Jean-loup Gailly and Mark Adler
(gzip), and Digital Equipment Corporation (DEC).

Portions of HDF5 were developed with support from the Lawrence Berkeley
National Laboratory (LBNL) and the United States Department of Energy
under Prime Contract No. DE-AC02-05CH11231.

Portions of HDF5 were developed with support from the University of
California, Lawrence Livermore National Laboratory (UC LLNL).

The following statement applies to those portions of the product and must
be retained in any redistribution of source code, binaries, documentation,
and/or accompanying materials:

```
   This work was partially produced at the University of California,
   Lawrence Livermore National Laboratory (UC LLNL) under contract
   no. W-7405-ENG-48 (Contract 48) between the U.S. Department of
Energy
   (DOE) and The Regents of the University of California
(University)
   for the operation of UC LLNL.

   DISCLAIMER:
   THIS WORK WAS PREPARED AS AN ACCOUNT OF WORK SPONSORED BY AN
AGENCY OF
   THE UNITED STATES GOVERNMENT. NEITHER THE UNITED STATES
GOVERNMENT NOR
   THE UNIVERSITY OF CALIFORNIA NOR ANY OF THEIR EMPLOYEES, MAKES
ANY
   WARRANTY, EXPRESS OR IMPLIED, OR ASSUMES ANY LIABILITY OR
RESPONSIBILITY
   FOR THE ACCURACY, COMPLETENESS, OR USEFULNESS OF ANY INFORMATION,
   APPARATUS, PRODUCT, OR PROCESS DISCLOSED, OR REPRESENTS THAT ITS
USE
   WOULD NOT INFRINGE PRIVATELY- OWNED RIGHTS. REFERENCE HEREIN TO
ANY
   SPECIFIC COMMERCIAL PRODUCTS, PROCESS, OR SERVICE BY TRADE NAME,
   TRADEMARK, MANUFACTURER, OR OTHERWISE, DOES NOT NECESSARILY
CONSTITUTE
   OR IMPLY ITS ENDORSEMENT, RECOMMENDATION, OR FAVORING BY THE
UNITED
   STATES GOVERNMENT OR THE UNIVERSITY OF CALIFORNIA. THE VIEWS AND
   OPINIONS OF AUTHORS EXPRESSED HEREIN DO NOT NECESSARILY STATE OR
REFLECT
   THOSE OF THE UNITED STATES GOVERNMENT OR THE UNIVERSITY OF
CALIFORNIA,
   AND SHALL NOT BE USED FOR ADVERTISING OR PRODUCT ENDORSEMENT
PURPOSES.

---------------------------------------------------------------------
---------
```

T/HIS

16.7. Jpeg

Jpeg

```
The authors make NO WARRANTY or representation, either express or
implied,
with respect to this software, its quality, accuracy,
merchantability, or
fitness for a particular purpose.  This software is provided "AS
IS", and you,
its user, assume the entire risk as to its quality and accuracy.

This software is copyright (C) 1991-2012, Thomas G. Lane, Guido
Vollbeding.
All Rights Reserved except as specified below.

Permission is hereby granted to use, copy, modify, and distribute
this
software (or portions thereof) for any purpose, without fee, subject
to these
conditions:
(1) If any part of the source code for this software is distributed,
then this
README file must be included, with this copyright and no-warranty
notice
unaltered; and any additions, deletions, or changes to the original
files
must be clearly indicated in accompanying documentation.
(2) If only executable code is distributed, then the accompanying
documentation must state that "this software is based in part on the
work of
the Independent JPEG Group".
(3) Permission for use of this software is granted only if the user
accepts
full responsibility for any undesirable consequences; the authors
accept
NO LIABILITY for damages of any kind.

These conditions apply to any software derived from or based on the
IJG code,
not just to the unmodified library.  If you use our work, you ought
to
acknowledge us.

Permission is NOT granted for the use of any IJG author's name or
company name
in advertising or publicity relating to this software or products
derived from
it.  This software may be referred to only as "the Independent JPEG
Group's
software".
```

```
We specifically permit and encourage the use of this software as the
basis of
commercial products, provided that all warranty or liability claims
are
assumed by the product vendor.
```

16.8. Libcurl

Libcurl

COPYRIGHT AND PERMISSION NOTICE

Copyright (c) 1996 - 2012, Daniel Stenberg, <daniel@haxx.se>.

All rights reserved.

Permission to use, copy, modify, and distribute this software for any purpose
with or without fee is hereby granted, provided that the above copyright
notice and this permission notice appear in all copies.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR
IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY,
FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT OF THIRD PARTY RIGHTS. IN
NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM,
DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR
OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE
OR OTHER DEALINGS IN THE SOFTWARE.

Except as contained in this notice, the name of a copyright holder shall not
be used in advertising or otherwise to promote the sale, use or other dealings
in this Software without prior written authorization of the copyright holder.

16.9. Libgif

Libgif

```
The GIFLIB distribution is Copyright (c) 1997  Eric S. Raymond

Permission is hereby granted, free of charge, to any person
obtaining a copy
of this software and associated documentation files (the
"Software"), to deal
in the Software without restriction, including without limitation
the rights
to use, copy, modify, merge, publish, distribute, sublicense, and/or
sell
copies of the Software, and to permit persons to whom the Software
is
furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be
included in
all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND,
EXPRESS OR
IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF
MERCHANTABILITY,
FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT.  IN NO EVENT
SHALL THE
AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR
OTHER
LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE,
ARISING FROM,
OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER
DEALINGS IN
THE SOFTWARE.
```

16.10. Libpng

Libpng

COPYRIGHT NOTICE, DISCLAIMER, and LICENSE
===

PNG Reference Library License version 2
----

* Copyright (c) 1995-2019 The PNG Reference Library Authors.
* Copyright (c) 2018-2019 Cosmin Truta.
* Copyright (c) 2000-2002, 2004, 2006-2018 Glenn Randers-Pehrson.
* Copyright (c) 1996-1997 Andreas Dilger.
* Copyright (c) 1995-1996 Guy Eric Schalnat, Group 42, Inc.

The software is supplied "as is", without warranty of any kind, express or implied, including, without limitation, the warranties of merchantability, fitness for a particular purpose, title, and non-infringement. In no event shall the Copyright owners, or anyone distributing the software, be liable for any damages or other liability, whether in contract, tort or otherwise, arising from, out of, or in connection with the software, or the use or other dealings in the software, even if advised of the possibility of such damage.

Permission is hereby granted to use, copy, modify, and distribute this software, or portions hereof, for any purpose, without fee, subject to the following restrictions:

1. The origin of this software must not be misrepresented; you must not claim that you wrote the original software. If you use this software in a product, an acknowledgment in the product documentation would be appreciated, but is not required.

2. Altered source versions must be plainly marked as such, and must not be misrepresented as being the original software.

3. This Copyright notice may not be removed or altered from any source or altered source distribution.

PNG Reference Library License version 1 (for libpng 0.5 through 1.6.35)
----

libpng versions 1.0.7, July 1, 2000, through 1.6.35, July 15, 2018 are
Copyright (c) 2000-2002, 2004, 2006-2018 Glenn Randers-Pehrson, are derived from libpng-1.0.6, and are distributed according to the same disclaimer and license as libpng-1.0.6 with the following individuals

added to the list of Contributing Authors:

    Simon-Pierre Cadieux
    Eric S. Raymond
    Mans Rullgard
    Cosmin Truta
    Gilles Vollant
    James Yu
    Mandar Sahastrabuddhe
    Google Inc.
    Vadim Barkov

and with the following additions to the disclaimer:

    There is no warranty against interference with your enjoyment of
    the library or against infringement. There is no warranty that
our
    efforts or the library will fulfill any of your particular
purposes
    or needs. This library is provided with all faults, and the
entire
    risk of satisfactory quality, performance, accuracy, and effort
is
    with the user.

Some files in the "contrib" directory and some configure-generated
files that are distributed with libpng have other copyright owners,
and
are released under other open source licenses.

libpng versions 0.97, January 1998, through 1.0.6, March 20, 2000,
are
Copyright (c) 1998-2000 Glenn Randers-Pehrson, are derived from
libpng-0.96, and are distributed according to the same disclaimer
and
license as libpng-0.96, with the following individuals added to the
list of Contributing Authors:

    Tom Lane
    Glenn Randers-Pehrson
    Willem van Schaik

libpng versions 0.89, June 1996, through 0.96, May 1997, are
Copyright (c) 1996-1997 Andreas Dilger, are derived from libpng-
0.88,
and are distributed according to the same disclaimer and license as
libpng-0.88, with the following individuals added to the list of
Contributing Authors:

    John Bowler
    Kevin Bracey
    Sam Bushell
    Magnus Holmgren
    Greg Roelofs
    Tom Tanner

Some files in the "scripts" directory have other copyright owners, but are released under this license.

libpng versions 0.5, May 1995, through 0.88, January 1996, are Copyright (c) 1995-1996 Guy Eric Schalnat, Group 42, Inc.

For the purposes of this copyright and license, "Contributing Authors" is defined as the following set of individuals:

    Andreas Dilger
    Dave Martindale
    Guy Eric Schalnat
    Paul Schmidt
    Tim Wegner

The PNG Reference Library is supplied "AS IS". The Contributing Authors and Group 42, Inc. disclaim all warranties, expressed or implied, including, without limitation, the warranties of merchantability and of fitness for any purpose. The Contributing Authors and Group 42, Inc. assume no liability for direct, indirect, incidental, special, exemplary, or consequential damages, which may result from the use of the PNG Reference Library, even if advised of the possibility of such damage.

Permission is hereby granted to use, copy, modify, and distribute this source code, or portions hereof, for any purpose, without fee, subject to the following restrictions:

1. The origin of this source code must not be misrepresented.

2. Altered versions must be plainly marked as such and must not be misrepresented as being the original source.

3. This Copyright notice may not be removed or altered from any source or altered source distribution.

The Contributing Authors and Group 42, Inc. specifically permit, without fee, and encourage the use of this source code as a component to supporting the PNG file format in commercial products. If you use this source code in a product, acknowledgment is not required but would be appreciated.

T/HIS

16.11. Libxlsxwriter

Libxlsxwriter

```
Libxlsxwriter is released under a FreeBSD license:

    Copyright 2014-2016, John McNamara
    All rights reserved.

    Redistribution and use in source and binary forms, with or
without
    modification, are permitted provided that the following
conditions are
    met:

    1. Redistributions of source code must retain the above
copyright notice,
       this list of conditions and the following disclaimer.
    2. Redistributions in binary form must reproduce the above
copyright
       notice, this list of conditions and the following disclaimer
in the
       documentation and/or other materials provided with the
distribution.

    THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND
CONTRIBUTORS "AS
    IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT
LIMITED TO,
    THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A
PARTICULAR
    PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR
    CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL,
SPECIAL,
    EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED
TO,
    PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA,
OR
    PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY
THEORY OF
    LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT
(INCLUDING
    NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF
THIS
    SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

    The views and conclusions contained in the software and
documentation are
    those of the authors and should not be interpreted as
representing
    official policies, either expressed or implied, of the FreeBSD
Project.
```

Libxlsxwriter includes `queue.h` from FreeBSD and the `minizip` component of
`zlib` which have the following licenses:

Queue.h from FreeBSD:

Zlib has the following License/Copyright:

```
    (C) 1995-2013 Jean-loup Gailly and Mark Adler

    This software is provided 'as-is', without any express or
implied
    warranty.  In no event will the authors be held liable for any
damages
    arising from the use of this software.

    Permission is granted to anyone to use this software for any
purpose,
    including commercial applications, and to alter it and
redistribute it
    freely, subject to the following restrictions:

    1. The origin of this software must not be misrepresented; you
must not
       claim that you wrote the original software. If you use this
software
       in a product, an acknowledgment in the product documentation
would be
       appreciated but is not required.
    2. Altered source versions must be plainly marked as such, and
must not be
       misrepresented as being the original software.
    3. This notice may not be removed or altered from any source
distribution.

    Jean-loup Gailly        Mark Adler
    jloup@gzip.org          madler@alumni.caltech.edu
```

16.12. libzip

libzip

libzip is released under a 3-clause BSD license:

T/HIS

16.13. MPEG-LA

MPEG-LA

16.14. Openssl

Openssl

```
LICENSE ISSUES
  ==============

  The OpenSSL toolkit stays under a double license, i.e. both the
conditions of
  the OpenSSL License and the original SSLeay license apply to the
toolkit.
  See below for the actual license texts.

  OpenSSL License
  ---------------

/*
====================================================================
 * Copyright (c) 1998-2017 The OpenSSL Project.  All rights
reserved.
 *
 * Redistribution and use in source and binary forms, with or
without
 * modification, are permitted provided that the following
conditions
 * are met:
 *
 * 1. Redistributions of source code must retain the above copyright
 *    notice, this list of conditions and the following disclaimer.
 *
 * 2. Redistributions in binary form must reproduce the above
copyright
 *    notice, this list of conditions and the following disclaimer
in
 *    the documentation and/or other materials provided with the
 *    distribution.
 *
 * 3. All advertising materials mentioning features or use of this
 *    software must display the following acknowledgment:
 *    "This product includes software developed by the OpenSSL
Project
 *    for use in the OpenSSL Toolkit. (http://www.openssl.org/)"
 *
 * 4. The names "OpenSSL Toolkit" and "OpenSSL Project" must not be
used to
 *    endorse or promote products derived from this software without
 *    prior written permission. For written permission, please
contact
 *    openssl-core@openssl.org.
 *
 * 5. Products derived from this software may not be called
"OpenSSL"
 *    nor may "OpenSSL" appear in their names without prior written
```

```
 * HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN
CONTRACT, STRICT
 * LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN
ANY WAY
 * OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE
POSSIBILITY OF
 * SUCH DAMAGE.
 *
 * The licence and distribution terms for any publically available
version or
 * derivative of this code cannot be changed.  i.e. this code cannot
simply be
 * copied and put under another distribution licence
 * [including the GNU Public Licence.]
 */
```

16.15. PCRE2

PCRE2

```
PCRE2 LICENCE
-------------

PCRE2 is a library of functions to support regular expressions whose
syntax
and semantics are as close as possible to those of the Perl 5
language.

Releases 10.00 and above of PCRE2 are distributed under the terms of
the "BSD"
licence, as specified below, with one exemption for certain binary
redistributions. The documentation for PCRE2, supplied in the "doc"
directory,
is distributed under the same terms as the software itself. The data
in the
testdata directory is not copyrighted and is in the public domain.

The basic library functions are written in C and are freestanding.
Also
included in the distribution is a just-in-time compiler that can be
used to
optimize pattern matching. This is an optional feature that can be
omitted when
the library is built.


THE BASIC LIBRARY FUNCTIONS
---------------------------

Written by:       Philip Hazel
Email local part: ph10
Email domain:     cam.ac.uk

University of Cambridge Computing Service,
Cambridge, England.

Copyright (c) 1997-2018 University of Cambridge
All rights reserved.


PCRE2 JUST-IN-TIME COMPILATION SUPPORT
--------------------------------------

Written by:       Zoltan Herczeg
Email local part: hzmester
Email domain:     freemail.hu

Copyright(c) 2010-2018 Zoltan Herczeg
All rights reserved.
```

STACK-LESS JUST-IN-TIME COMPILER

Written by: Zoltan Herczeg
Email local part: hzmester
Email domain: freemail.hu

THE "BSD" LICENCE

Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:

* Redistributions of source code must retain the above copyright notices, this list of conditions and the following disclaimer.

* Redistributions in binary form must reproduce the above copyright notices, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.

* Neither the name of the University of Cambridge nor the names of any contributors may be used to endorse or promote products derived from this software without specific prior written permission.

THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE

POSSIBILITY OF SUCH DAMAGE.

EXEMPTION FOR BINARY LIBRARY-LIKE PACKAGES

The second condition in the BSD licence (covering binary redistributions) does
not apply all the way down a chain of software. If binary package A includes
PCRE2, it must respect the condition, but if package B is software that
includes package A, the condition is not imposed on package B unless it uses
PCRE2 independently.

End

16.16. PDFHummus

PDFHummus

Is licensed under the Apache License:

16.17. POV-Ray

POV-Ray

Is licensed under the GNU AFFERO GENERAL PUBLIC LICENSE Version 3, 19 November 2007 which may be found here http://www.povray.org/povlegal.html

Oasys Ltd use the POV-Ray executable in unmodified form as a separate, stand-alone entity. We have not modified the source code or the executable in any way.

We convey the executable as part of our installation package, and in accordance with the licence:

- Users who install POV-Ray must accept the licence terms cited above.
- We provide a download of the POV-Ray executable and source code on our website http://www.oasys-software.com/dyna/en/

T/HIS

16.18. Schemasafe

16.19. SmoothSort

SmoothSort

Oasys Ltd acknowledge Wikibooks as the source of this algorithm, which is used in unmodified form.

T/HIS

16.20. Spidermonkey

Spidermonkey

```
Mozilla Public License Version 2.0
==================================

1. Definitions
--------------

1.1. "Contributor"
    means each individual or legal entity that creates, contributes
to
    the creation of, or owns Covered Software.

1.2. "Contributor Version"
    means the combination of the Contributions of others (if any)
used
    by a Contributor and that particular Contributor's Contribution.

1.3. "Contribution"
    means Covered Software of a particular Contributor.

1.4. "Covered Software"
    means Source Code Form to which the initial Contributor has
attached
    the notice in Exhibit A, the Executable Form of such Source Code
    Form, and Modifications of such Source Code Form, in each case
    including portions thereof.

1.5. "Incompatible With Secondary Licenses"
    means

    (a) that the initial Contributor has attached the notice
described
        in Exhibit B to the Covered Software; or

    (b) that the Covered Software was made available under the terms
of
        version 1.1 or earlier of the License, but not also under
the
        terms of a Secondary License.

1.6. "Executable Form"
    means any form of the work other than Source Code Form.

1.7. "Larger Work"
    means a work that combines Covered Software with other material,
in
    a separate file or files, that is not Covered Software.

1.8. "License"
    means this document.
```

```
1.9. "Licensable"
    means having the right to grant, to the maximum extent possible,
    whether at the time of the initial grant or subsequently, any
and
    all of the rights conveyed by this License.

1.10. "Modifications"
    means any of the following:

    (a) any file in Source Code Form that results from an addition
to,
        deletion from, or modification of the contents of Covered
        Software; or

    (b) any new file in Source Code Form that contains any Covered
        Software.

1.11. "Patent Claims" of a Contributor
    means any patent claim(s), including without limitation, method,
    process, and apparatus claims, in any patent Licensable by such
    Contributor that would be infringed, but for the grant of the
    License, by the making, using, selling, offering for sale,
having
    made, import, or transfer of either its Contributions or its
    Contributor Version.

1.12. "Secondary License"
    means either the GNU General Public License, Version 2.0, the
GNU
    Lesser General Public License, Version 2.1, the GNU Affero
General
    Public License, Version 3.0, or any later versions of those
    licenses.

1.13. "Source Code Form"
    means the form of the work preferred for making modifications.

1.14. "You" (or "Your")
    means an individual or a legal entity exercising rights under
this
    License. For legal entities, "You" includes any entity that
    controls, is controlled by, or is under common control with You.
For
    purposes of this definition, "control" means (a) the power,
direct
    or indirect, to cause the direction or management of such
entity,
    whether by contract or otherwise, or (b) ownership of more than
    fifty percent (50%) of the outstanding shares or beneficial
    ownership of such entity.

2. License Grants and Conditions
--------------------------------
```

2.1. Grants

Each Contributor hereby grants You a world-wide, royalty-free,
non-exclusive license:

(a) under intellectual property rights (other than patent or
trademark)
    Licensable by such Contributor to use, reproduce, make
available,
    modify, display, perform, distribute, and otherwise exploit its
    Contributions, either on an unmodified basis, with
Modifications, or
    as part of a Larger Work; and

(b) under Patent Claims of such Contributor to make, use, sell,
offer
    for sale, have made, import, and otherwise transfer either its
    Contributions or its Contributor Version.

2.2. Effective Date

The licenses granted in Section 2.1 with respect to any Contribution
become effective for each Contribution on the date the Contributor
first
distributes such Contribution.

2.3. Limitations on Grant Scope

The licenses granted in this Section 2 are the only rights granted
under
this License. No additional rights or licenses will be implied from
the
distribution or licensing of Covered Software under this License.
Notwithstanding Section 2.1(b) above, no patent license is granted
by a
Contributor:

(a) for any code that a Contributor has removed from Covered
Software;
    or

(b) for infringements caused by: (i) Your and any other third
party's
    modifications of Covered Software, or (ii) the combination of
its
    Contributions with other software (except as part of its
Contributor
    Version); or

(c) under Patent Claims infringed by Covered Software in the absence
of
    its Contributions.

This License does not grant any rights in the trademarks, service
marks,

or logos of any Contributor (except as may be necessary to comply with
the notice requirements in Section 3.4).

2.4. Subsequent Licenses

No Contributor makes additional grants as a result of Your choice to
distribute the Covered Software under a subsequent version of this
License (see Section 10.2) or under the terms of a Secondary License (if
permitted under the terms of Section 3.3).

2.5. Representation

Each Contributor represents that the Contributor believes its
Contributions are its original creation(s) or it has sufficient rights
to grant the rights to its Contributions conveyed by this License.

2.6. Fair Use

This License is not intended to limit any rights You have under
applicable copyright doctrines of fair use, fair dealing, or other
equivalents.

2.7. Conditions

Sections 3.1, 3.2, 3.3, and 3.4 are conditions of the licenses granted
in Section 2.1.

3. Responsibilities

3.1. Distribution of Source Form

All distribution of Covered Software in Source Code Form, including any
Modifications that You create or to which You contribute, must be under
the terms of this License. You must inform recipients that the Source
Code Form of the Covered Software is governed by the terms of this
License, and how they can obtain a copy of this License. You may not
attempt to alter or restrict the recipients' rights in the Source Code
Form.

3.2. Distribution of Executable Form

If You distribute Covered Software in Executable Form then:

(a) such Covered Software must also be made available in Source Code
Form, as described in Section 3.1, and You must inform recipients of

the Executable Form how they can obtain a copy of such Source Code
Form by reasonable means in a timely manner, at a charge no more
than the cost of distribution to the recipient; and

(b) You may distribute such Executable Form under the terms of this
License, or sublicense it under different terms, provided that the
license for the Executable Form does not attempt to limit or alter
the recipients' rights in the Source Code Form under this License.

3.3. Distribution of a Larger Work

You may create and distribute a Larger Work under terms of Your choice,
provided that You also comply with the requirements of this License for
the Covered Software. If the Larger Work is a combination of Covered
Software with a work governed by one or more Secondary Licenses, and the
Covered Software is not Incompatible With Secondary Licenses, this
License permits You to additionally distribute such Covered Software
under the terms of such Secondary License(s), so that the recipient of
the Larger Work may, at their option, further distribute the Covered
Software under the terms of either this License or such Secondary
License(s).

3.4. Notices

You may not remove or alter the substance of any license notices
(including copyright notices, patent notices, disclaimers of warranty,
or limitations of liability) contained within the Source Code Form of
the Covered Software, except that You may alter any license notices to
the extent required to remedy known factual inaccuracies.

3.5. Application of Additional Terms

You may choose to offer, and to charge a fee for, warranty, support,
indemnity or liability obligations to one or more recipients of Covered
Software. However, You may do so only on Your own behalf, and not on
behalf of any Contributor. You must make it absolutely clear that any
such warranty, support, indemnity, or liability obligation is offered by
You alone, and You hereby agree to indemnify every Contributor for any
liability incurred by such Contributor as a result of warranty, support,

indemnity or liability terms You offer. You may include additional
disclaimers of warranty and limitations of liability specific to any
jurisdiction.

4. Inability to Comply Due to Statute or Regulation

If it is impossible for You to comply with any of the terms of this
License with respect to some or all of the Covered Software due to
statute, judicial order, or regulation then You must: (a) comply
with
the terms of this License to the maximum extent possible; and (b)
describe the limitations and the code they affect. Such description
must
be placed in a text file included with all distributions of the
Covered
Software under this License. Except to the extent prohibited by
statute
or regulation, such description must be sufficiently detailed for a
recipient of ordinary skill to be able to understand it.

5. Termination

5.1. The rights granted under this License will terminate
automatically
if You fail to comply with any of its terms. However, if You become
compliant, then the rights granted under this License from a
particular
Contributor are reinstated (a) provisionally, unless and until such
Contributor explicitly and finally terminates Your grants, and (b)
on an
ongoing basis, if such Contributor fails to notify You of the
non-compliance by some reasonable means prior to 60 days after You
have
come back into compliance. Moreover, Your grants from a particular
Contributor are reinstated on an ongoing basis if such Contributor
notifies You of the non-compliance by some reasonable means, this is
the
first time You have received notice of non-compliance with this
License
from such Contributor, and You become compliant prior to 30 days
after
Your receipt of the notice.

5.2. If You initiate litigation against any entity by asserting a
patent
infringement claim (excluding declaratory judgment actions,
counter-claims, and cross-claims) alleging that a Contributor
Version
directly or indirectly infringes any patent, then the rights granted
to
You by any and all Contributors for the Covered Software under
Section
2.1 of this License shall terminate.

5.3. In the event of termination under Sections 5.1 or 5.2 above,
all
end user license agreements (excluding distributors and resellers)
which
have been validly granted by You or Your distributors under this
License
prior to termination shall survive termination.

```
*********************************************************************
****
*
*
*  6. Disclaimer of Warranty
*
*  -------------------------
*
*
*
*  Covered Software is provided under this License on an "as is"
*
*  basis, without warranty of any kind, either expressed, implied,
or  *
*  statutory, including, without limitation, warranties that the
*
*  Covered Software is free of defects, merchantable, fit for a
*
*  particular purpose or non-infringing. The entire risk as to the
*
*  quality and performance of the Covered Software is with You.
*
*  Should any Covered Software prove defective in any respect, You
*
*  (not any Contributor) assume the cost of any necessary servicing,
*
*  repair, or correction. This disclaimer of warranty constitutes an
*
*  essential part of this License. No use of any Covered Software is
*
*  authorized under this License except under this disclaimer.
*
*
*
*********************************************************************
****

*********************************************************************
****
*
*
*  7. Limitation of Liability
*
*  --------------------------
*
```

```
*
*
*  Under no circumstances and under no legal theory, whether tort
*
*  (including negligence), contract, or otherwise, shall any
*
*  Contributor, or anyone who distributes Covered Software as
*
*  permitted above, be liable to You for any direct, indirect,
*
*  special, incidental, or consequential damages of any character
*
*  including, without limitation, damages for lost profits, loss of
*
*  goodwill, work stoppage, computer failure or malfunction, or any
*
*  and all other commercial damages or losses, even if such party
*
*  shall have been informed of the possibility of such damages. This
*
*  limitation of liability shall not apply to liability for death or
*
*  personal injury resulting from such party's negligence to the
*
*  extent applicable law prohibits such limitation. Some
*
*  jurisdictions do not allow the exclusion or limitation of
*
*  incidental or consequential damages, so this exclusion and
*
*  limitation may not apply to You.
*
*
*
********************************************************************
****

8. Litigation
-------------

Any litigation relating to this License may be brought only in the
courts of a jurisdiction where the defendant maintains its principal
place of business and such litigation shall be governed by laws of
that
jurisdiction, without reference to its conflict-of-law provisions.
Nothing in this Section shall prevent a party's ability to bring
cross-claims or counter-claims.

9. Miscellaneous
----------------

This License represents the complete agreement concerning the
subject
matter hereof. If any provision of this License is held to be
unenforceable, such provision shall be reformed only to the extent
```

necessary to make it enforceable. Any law or regulation which provides
that the language of a contract shall be construed against the drafter
shall not be used to construe this License against a Contributor.

10. Versions of the License

10.1. New Versions

Mozilla Foundation is the license steward. Except as provided in Section
10.3, no one other than the license steward has the right to modify or
publish new versions of this License. Each version will be given a
distinguishing version number.

10.2. Effect of New Versions

You may distribute the Covered Software under the terms of the version
of the License under which You originally received the Covered Software,
or under the terms of any subsequent version published by the license
steward.

10.3. Modified Versions

If you create software not governed by this License, and you want to
create a new license for such software, you may create and use a
modified version of this License if you rename the license and remove
any references to the name of the license steward (except to note that
such modified license differs from this License).

10.4. Distributing Source Code Form that is Incompatible With Secondary
Licenses

If You choose to distribute Source Code Form that is Incompatible With
Secondary Licenses under the terms of this version of the License, the
notice described in Exhibit B of this License must be attached.

Exhibit A - Source Code Form License Notice

This Source Code Form is subject to the terms of the Mozilla Public
License, v. 2.0. If a copy of the MPL was not distributed with this

```
  file, You can obtain one at http://mozilla.org/MPL/2.0/.

If it is not possible or desirable to put the notice in a particular
file, then You may include the notice in a location (such as a
LICENSE
file in a relevant directory) where a recipient would be likely to
look
for such a notice.

You may add additional accurate notices of copyright ownership.

Exhibit B - "Incompatible With Secondary Licenses" Notice
---------------------------------------------------------

  This Source Code Form is "Incompatible With Secondary Licenses",
as
  defined by the Mozilla Public License, v. 2.0.
```

T/HIS

16.21. TOML Parser for C

TOML Parser for C

16.22. Treeview

Treeview

This application includes the TreeView script.

You are not authorized to download and/or use the TreeView source code from this
application for your own purposes. For your own FREE copy of the TreeView script,
please visit the http://www.treeview.net Web site.

THIS SOFTWARE IS PROVIDED "AS IS" AND WITHOUT ANY EXPRESS OR IMPLIED WARRANTIES,
INCLUDING, WITHOUT LIMITATION, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS
FOR A PARTICULAR PURPOSE.

If Customer is using the free version of SOFTWARE, Customer must ensure that the
"JavaScript Tree Menu" link at the top of the TreeView is visible and readable in their
Web page or application.

Customer may not harm the GUBUSOFT intellectual property rights using any media or via
any electronic or other method now known or later discovered.

Customer may not use the GubuSoft name, the name of the TreeView author, or the names
of any source code contributors to endorse or promote products derived from this
SOFTWARE without specific prior written permission.

Customer may not utilize the SOFTWARE in a manner which is disparaging to GUBUSOFT.

16.23. Turf

Turf

```
The MIT License (MIT)

Copyright (c) 2019 Morgan Herlocker

Permission is hereby granted, free of charge, to any person
obtaining a copy of
this software and associated documentation files (the "Software"),
to deal in
the Software without restriction, including without limitation the
rights to
use, copy, modify, merge, publish, distribute, sublicense, and/or
sell copies of
the Software, and to permit persons to whom the Software is
furnished to do so,
subject to the following conditions:

The above copyright notice and this permission notice shall be
included in all
copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND,
EXPRESS OR
IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF
MERCHANTABILITY, FITNESS
FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE
AUTHORS OR
COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER
LIABILITY, WHETHER
IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR
IN
CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE
SOFTWARE.
```

T/HIS

16.24. Win-iconv

Win-iconv

```
win_iconv is a iconv implementation using Win32 API to convert.

win_iconv is placed in the public domain.

Yukihiro Nakadaira <yukihiro.nakadaira@gmail.com>
```

16.25. x264

x264

The x264 software library is used under commercial license from x264, LLC

16.26. Zlib

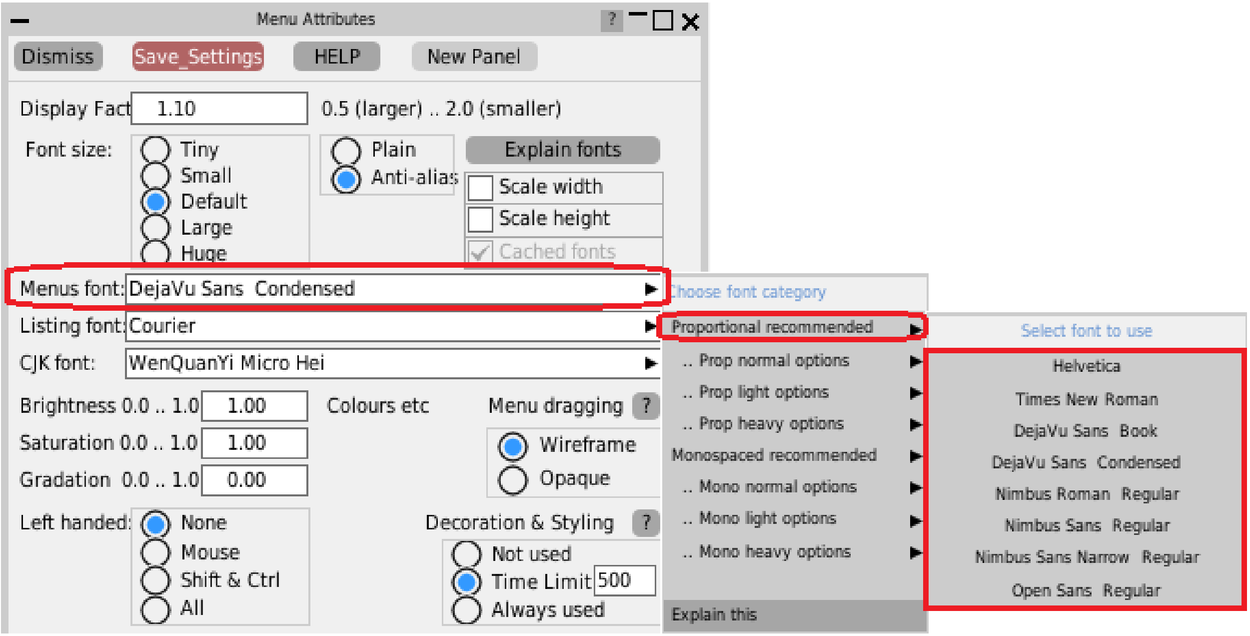
Zlib

17. Fonts on Linux

Fonts on Linux

Prior to Oasys Suite 17.0 the Oasys Ltd. LS-DYNA environment software used "legacy" X11 fixed fonts on Linux, from Oasys Suite 17.0 onwards, the software uses Freetype fonts, which give improved appearance and a wider range of typefaces.

The recommended proportional font for menu panels is "DejaVu Sans Condensed" which is widely available on Linux, but you can change this using **Options, Menu attributes** where a different font can be selected from those available on your system. For example on the author's CentOS 7 system the choice of fonts is:



17.1. The Range of Fonts Available

The range of fonts available

The range of fonts you see on your system will depend on the version of Linux you are using and what fonts you have installed; the image above was captured from a CentOS 7 machine.

The Oasys software interrogates the font server to extract all available fonts, then sorts them for presentation purposes by spacing (proportional or monospaced) and weight (normal, light, bold). The "recommended" fonts, as shown in the right hand popup menu above, are simply those which have been found by trial and error to give the best appearance. However this is a very subjective matter, and you may prefer something different: choose something that you like then use **Save Settings** to save it. If you change your mind later you can always come back to this panel to select something else.

Helvetica is provided as an option for backwards compatibility with the older user interface; it is not natively available on Linux so a different font is substituted, which tends not to look very good in Freetype.

Monospaced font selection problems

We have observed that while proportional font selection works correctly on Linux, the selection of monospaced fonts seems to have some bugs:

- The default "courier" font works, but tends to produce a font that is too small in some situations and probably is not exactly courier, although it looks very similar.
- The "recommended" monospaced font on some systems comes out as "Courier 10 Pt Regular", which is a genuine courier font, however if you select that it will produce something completely different. Experiment shows that if you ask for "Courier 10 Pt" then you get what you expect, but appending "Regular" breaks the font selection somehow

This appears to be a "fontconfig" problem: the system's font server simply gets it wrong. This can be demonstrated by the command

`fc-match` "*font of your choice*"

for example **`fc-match "courier"`** on a RHEL 7 machine produces the result **`"Nimbus Mono PS" "Regular"`**

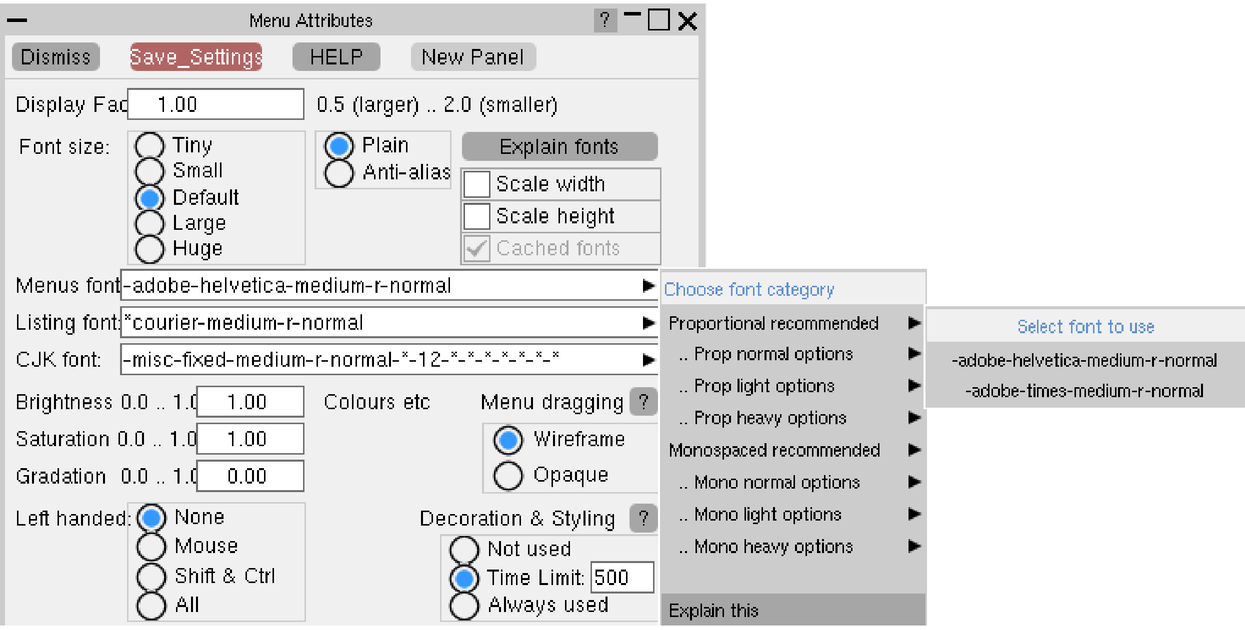
If you are happy with the monospaced font used for help texts and the like you don't need to take any action, however if you want to change it you may need to experiment a bit to find something that looks good on your system by typing different variations of names into the "Listing font: [.....]" text entry box. You can use the "fc-match" command in conjunction with this to see what the font server will map your request onto. Once you have found something satisfactory use **Save Settings** to save it in your oa_pref file and it will be remembered for future use.

17.2. Plain Versus Anti-aliased Fonts

Plain versus Anti-aliased fonts

On some monitors, especially relatively low resolution ones, the anti-aliasing of fonts can result in quite fuzzy text. The quality of this will depend on the version of Freetype installed, and more recent Linuxes will tend to look better since they are more likely to use sub-pixel sampling.

Some users may prefer the cruder but sharper appearance of the original "core X11" legacy fonts, and these can be used by changing to **Plain** so long as you actually have these fonts loaded on your machine. On the CentOS 7 machine being used to create this manual page the equivalent "plain" font image of the above is:



If you try this on your machine and it doesn't work then it means that you need to load the legacy font package(s), see below.

Loading legacy Core X11 fonts

You don't need to load these, it is only necessary if you want the old-style "plain" appearance described in the section above.

You will need root privileges to install these, so unless you are familiar with working as root and using commands such as "rpm", "yum" or "yast" please seek help from your IT department, or alternatively contact Oasys Ltd for help.

The best fonts to install are the 75 dots per inch (dpi) ones, which can be obtained online for a range of common Linux operating systems from https://pkgs.org/download/xorg-x11-fonts-75dpi

If that fails you may already have the relevant packages in your installation files, you should look for (in order)

RedHat/CentOS

```
xorg-x11-fonts-75dpi
xorg-x11-fonts-ISO8859-1-75dpi
xorg-x11-fonts-Type1
xorg-x11-fonts-misc
xorg-x11-fonts-100dpi
xorg-x11-fonts-ISO8859-1-100dpi
```

You don't have to install all of these.

The 75dpi and 100dpi font packages are the same typefaces at different resolutions. You should choose the one which gives the best looking results on your display, but in the author's experience the 75dpi one looks fine but the 100dpi one looks as if a spider was let loose with a leaky pen! Always try the 75dpi one first.

To manage fonts on RHEL/CentOS do the following:

- Log in as root
- To see the X11 fonts currently installed type "**`yum list installed | grep xorg | grep font`**"
- To see X11 fonts available but not installed "**`yum list available | grep xorg | grep font`**"
- To install something "**`yum install`** *package*", for example "**`yum install xorg-x11-fonts-75dpi`**"

You can list the range of "yum" commands available with "man yum".

SUSE

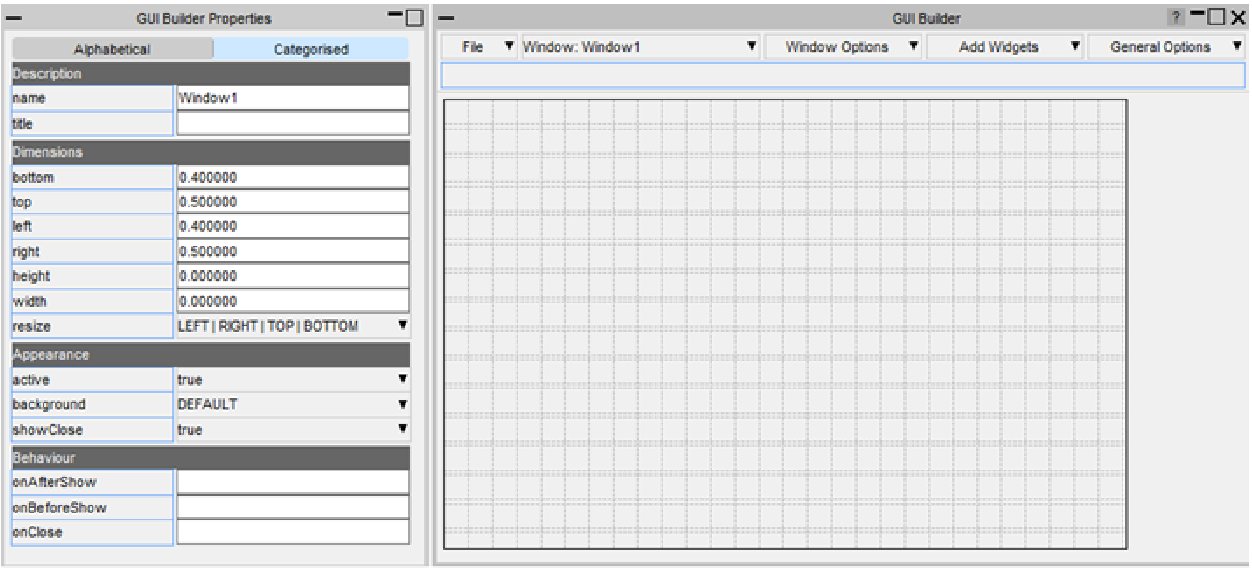
```
xorg-x11-fonts-core
xorg-x11-fonts
```

18. The JavaScript GUI Builder

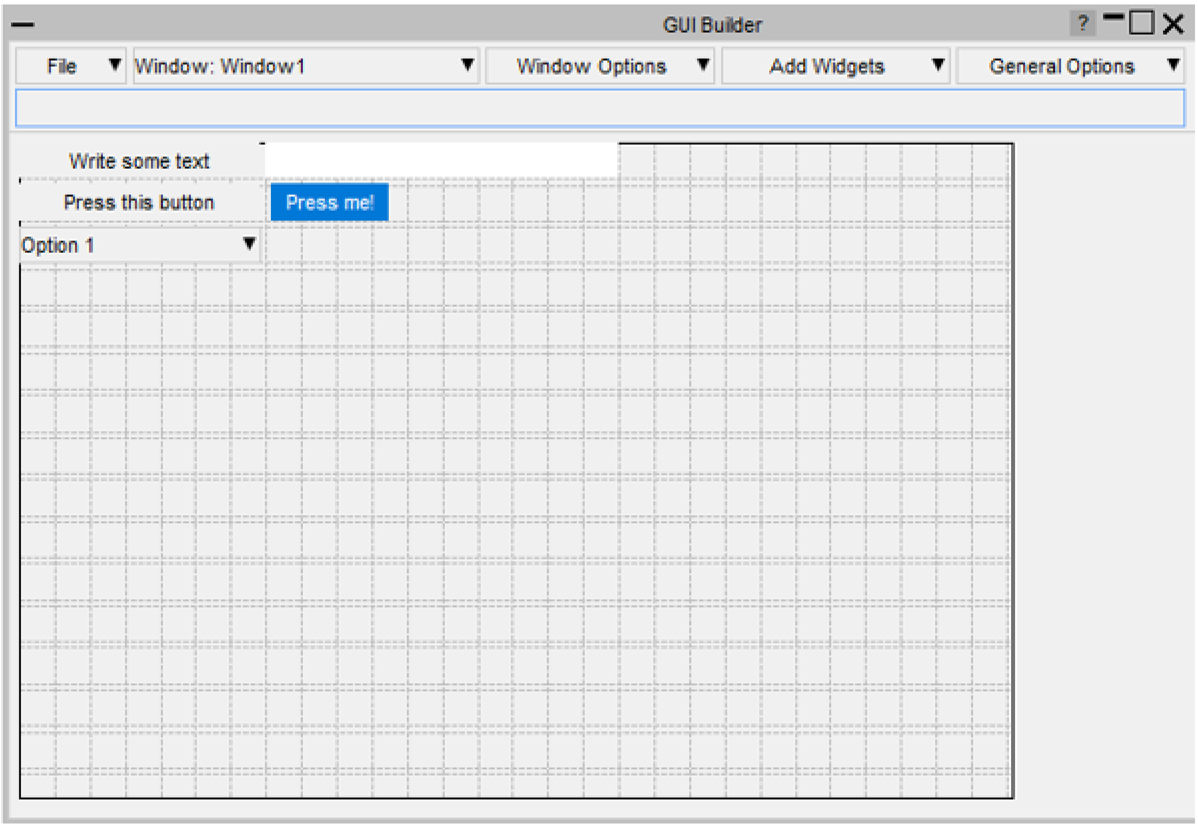
The JavaScript GUI Builder

The JavaScript GUI Builder is an interactive GUI Builder, available in D3PLOT, PRIMER and T/HIS, making it easier to create JavaScript GUIs, removing the need to write code to create windows and widgets.

It can be started by pressing the **GUI Builder** button in the JavaScript menu in any of the programs.



You can then design and save your GUI to a file:

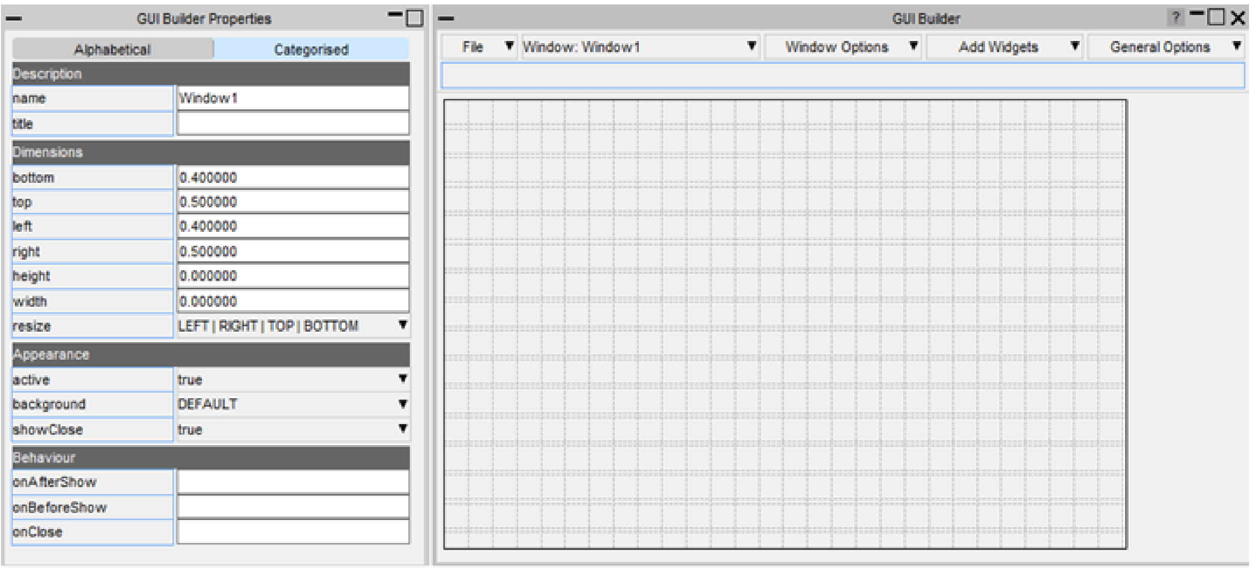


Then read the file in your script to automatically generate the window and widgets:

18.1. How to Build a GUI

How to build a GUI

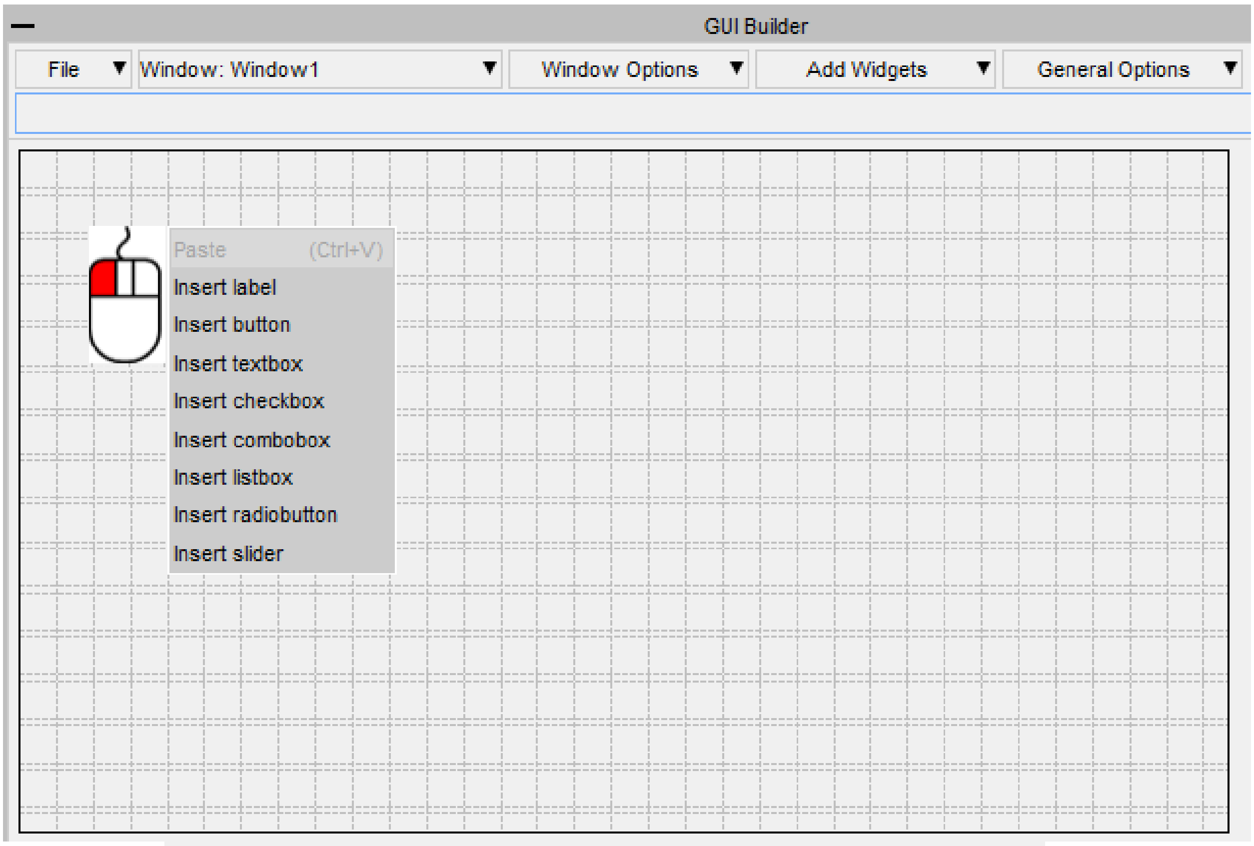
The builder is split into two windows. The properties window for setting the properties of the widgets and windows and a design window for adding, positioning and resizing widgets.

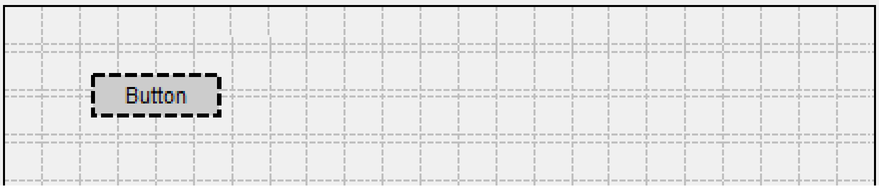


18.1.1. Add a Widget

Add a widget

Widgets can be added by right-clicking on the design window and selecting the widget type to add. The widget will be added with default properties and highlighted with dashed lines to indicate that it's the current widget.

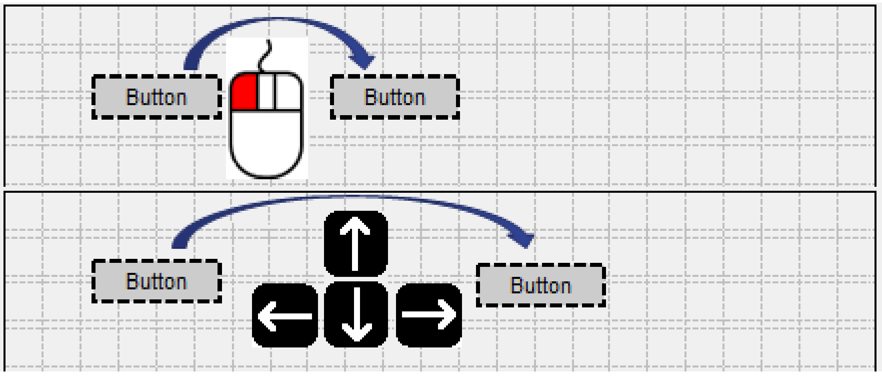




18.1.2. Move a Widget

Move a widget

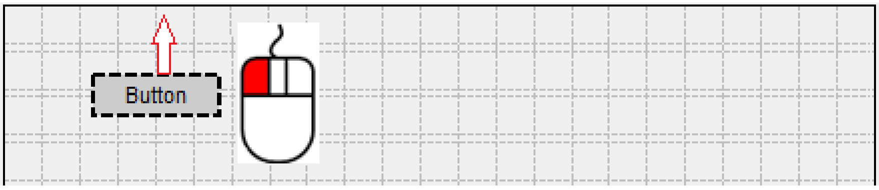
Widgets can be moved by left-clicking on them and dragging, or by using arrow keys.

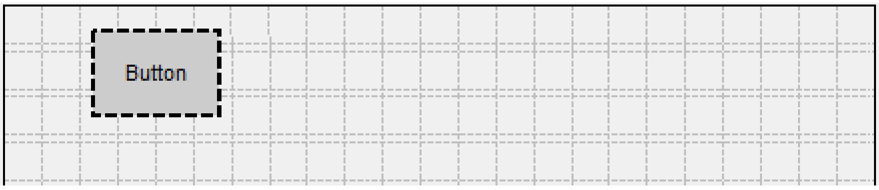


18.1.3. Resize a Widget

Resize a widget

Widgets can be resized by left-clicking on their border and dragging.

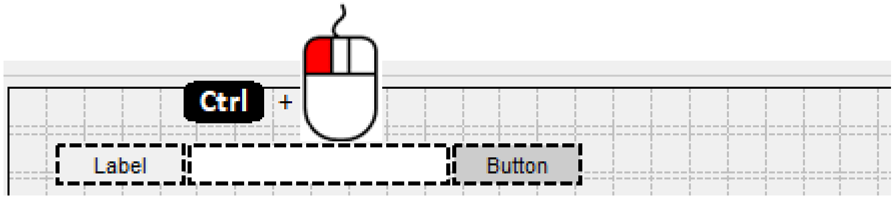




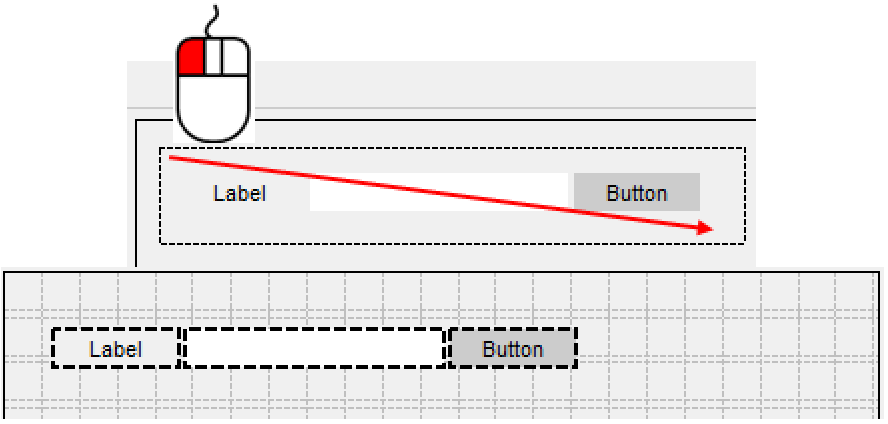
18.1.4. Selecting Widgets

Selecting widgets

Multiple widgets can be selected by holding the Ctrl or Shift keys and left-clicking.



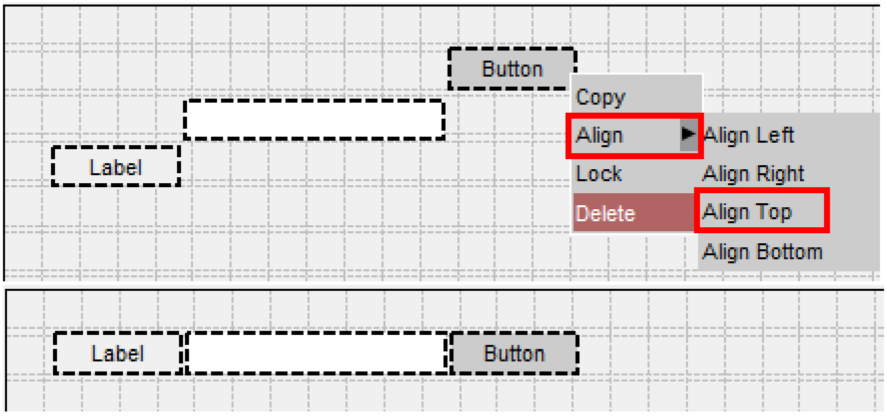
Alternatively a box can be dragged around the widgets you want to select.



18.1.5. Aligning Widgets

Aligning widgets

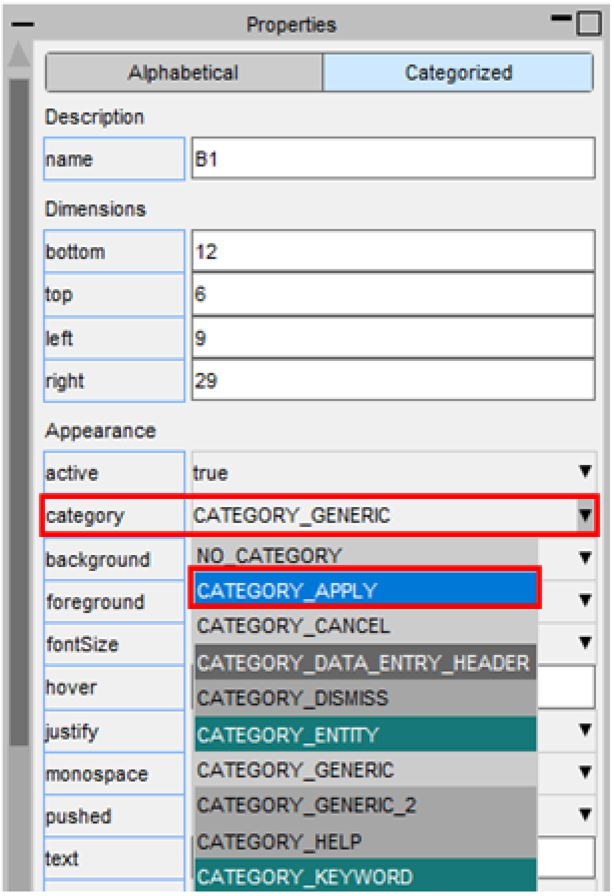
When multiple widgets are selected the borders can be aligned by right-clicking on the widget you want to align the other widgets to, and then selecting how you want them to be aligned.



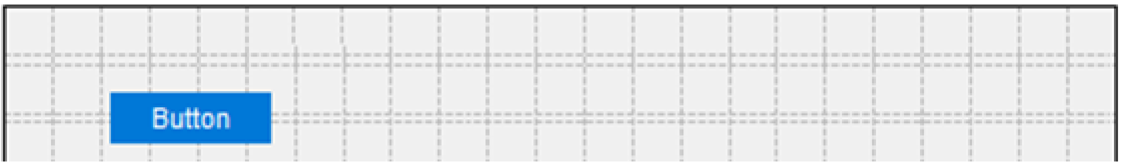
18.1.6. Setting the Properties of Widgets

Setting the properties of widgets

The properties of a widget can be modified in the properties window, e.g. change the category to CATEGORY_APPLY.



The appearance of the widget will update in the design window. If multiple widgets are selected the property will be applied to all the selected widgets.

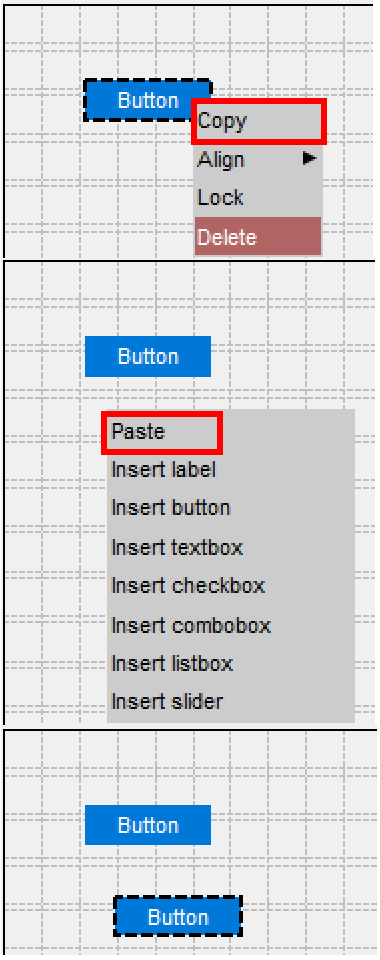


18.1.7. Copying and Pasting Widgets

Copying and pasting widgets

You can copy and paste widgets by right-clicking on them and selecting **Copy** and then right-clicking on the window and selecting **Paste**. The new widget will have all the same properties as the copied widget.

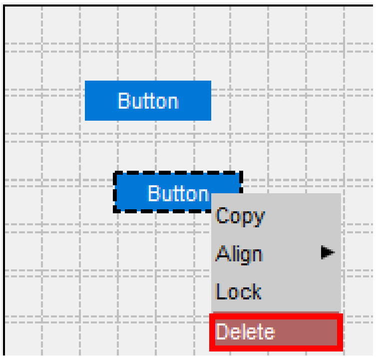
Alternatively you can use the shortcuts Ctrl-C and Ctrl-V.



18.1.8. Deleting Widgets

Deleting widgets

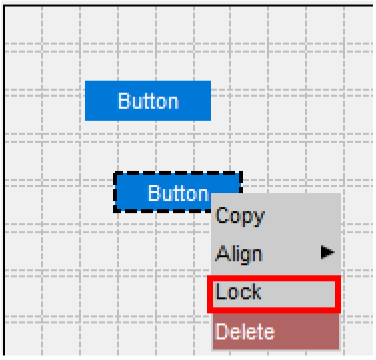
To delete a widget, right-click on it and select **Delete**. Alternatively you can press the Delete shortcut key.



18.1.9. Lock the Position of Widgets

Lock the position of widgets

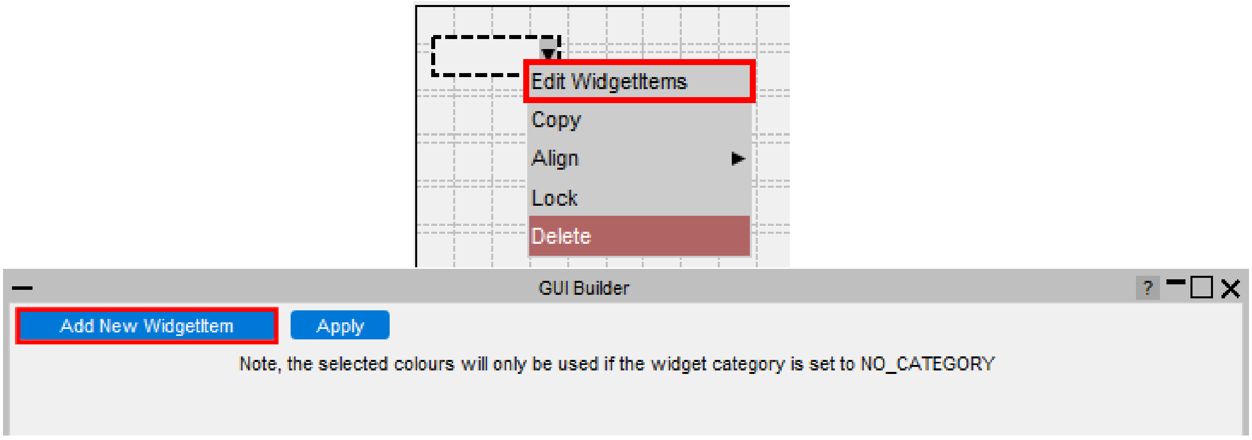
To lock the position of a widget so it can't be repositioned or resized, right-click on it and select **Lock**. To unlock it again, right-click on it and select **Unlock**



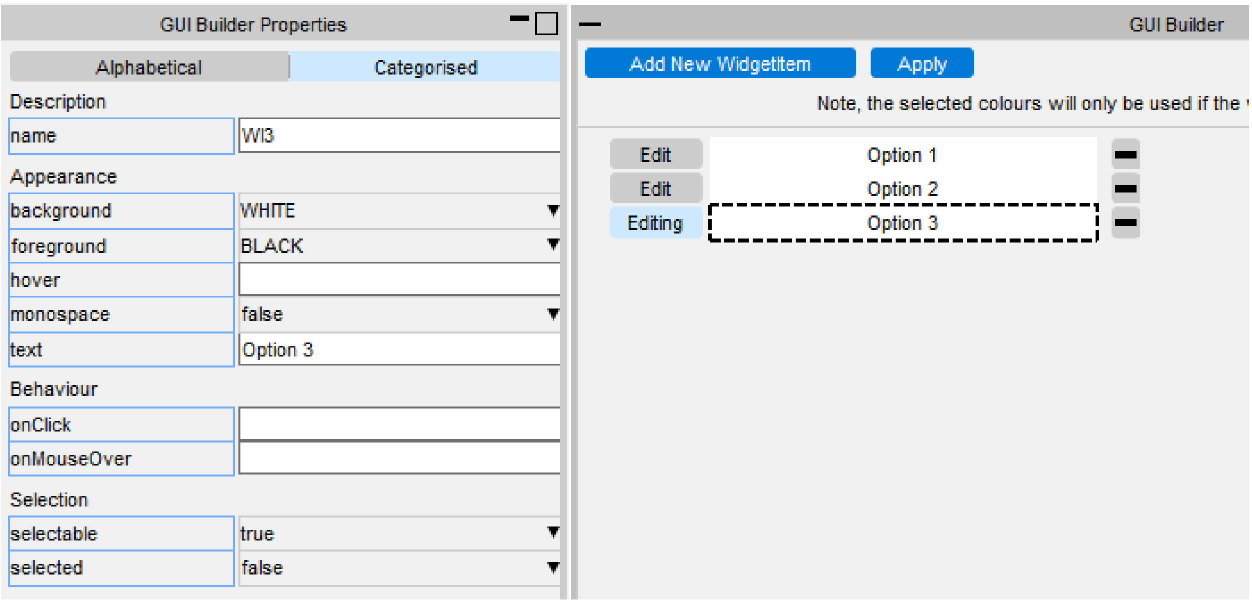
18.1.10. Adding Widgetitems to Comboboxes and Listboxes

Adding widgetitems to comboboxes, listboxes and radiobuttons

To add WidgetItems to a Combobox, Listbox or Radiobutton, right-click on it and select **Edit WidgetItems**. This will update the design window where you can add WidgetItems by pressing the **Add New WidgetItem** button.



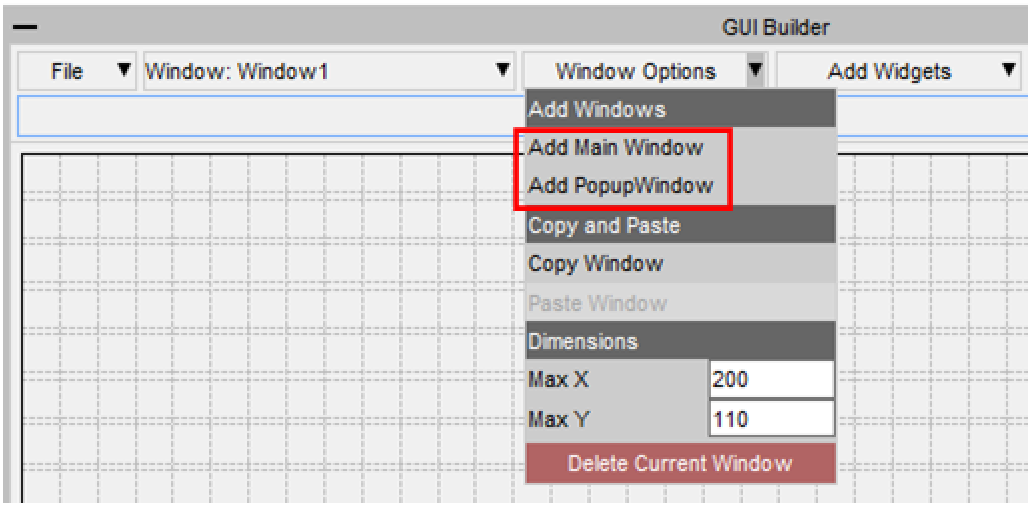
The appearance of the current WidgetItem can be modified in the same way as Widgets by clicking on the WidgetItem and updating its properties. To delete a WidgetItem, click on the **-** on the right hand side. Once you have finished, press **Apply** to return to the normal design window.



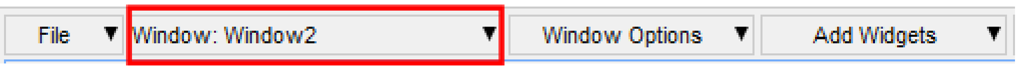
18.1.11. Adding Windows

Adding windows

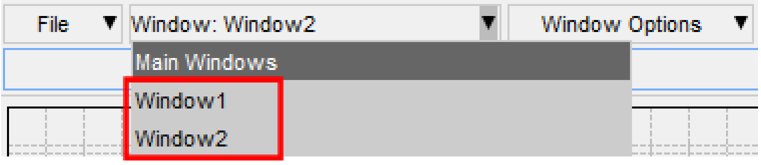
Additional windows can be created by clicking on the Window Options dropdown menu. You can add either a Main Window or PopupWindow.



The name of the current window is displayed in the Window selection dropdown menu.



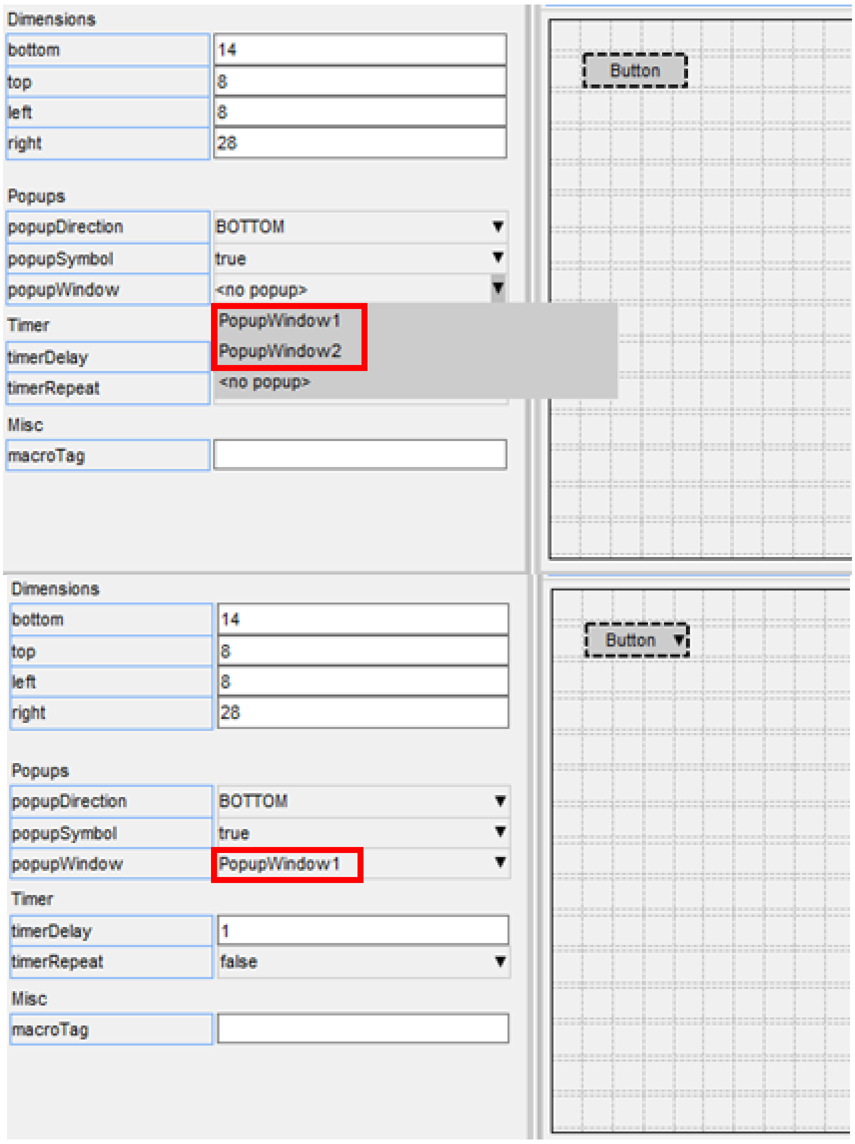
To change to a different window, select it from the dropdown menu.



18.1.12. PopupWindows

PopupWindows

PopupWindows can be linked to widgets by setting the popupWindow property.

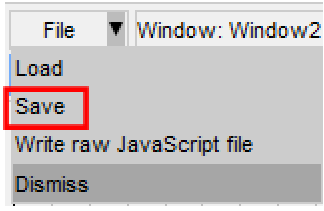


To remove a PopupWindow linked to a widget, set the popupWindow to <no popup>.

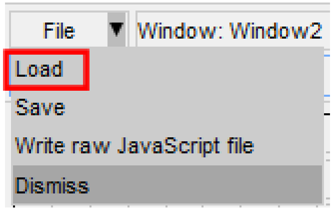
18.1.13. Saving and Loading a GUI

Saving and loading a GUI

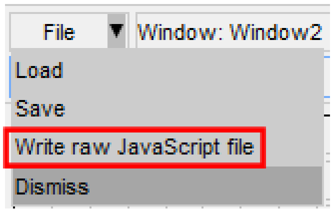
The GUI can be saved to file by pressing the **Save** button and then selecting a file. The saved file is a JavaScript file containing the window and widget definitions in a JSON string, and a call to Window.BuildGUIFromString() which builds the GUI when the script is run. Further details are given in the next section.



It can be reloaded by pressing the **Load** button and selecting the file to load.



The GUI can also be saved as a raw JavaScript file, with the calls to create and position the windows and widgets, explicitly defined, rather than using Window.BuildGUIFromString(). This cannot be loaded back into the GUI Builder, however it may be useful for creating GUIs to run in versions prior to v18 that don't have the Window.BuildGUIFromString() function.



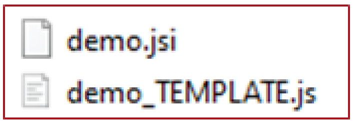
18.2. How to Use the GUI in a Script

How to use the GUI in a script

The GUI is saved to a JavaScript file, containing the GUI definition in a JSON string and a call to Window.BuildGUIFromString(). It is saved with the extension **`.jsi`** to indicate that it should be included from another file. You should not need to edit this file.

When saving the GUI a *.js file is also written to demonstrate how to include the *.jsi file and display the GUI. This can be used as a template to follow and modify.

It is written to the same folder as the *.jsi file and named **<jsi_filename>_TEMPLATE.js**, e.g. if the *.jsi file is called **`demo.jsi`**, the *.js file will be saved as **`demo_TEMPLATE.js`**



The following sections explain how you can reference the Windows, Widgets and WidgetItem objects within your script.

18.2.1. Read the GUI Into a Script

Read the GUI Into a Script

To read the GUI in a script you need to include the *.jsi file with the Use() function.

This will create a global variable (`gui` by default) containing all the GUI objects. The name of the variable can be changed in the GUI builder menu under General Options.

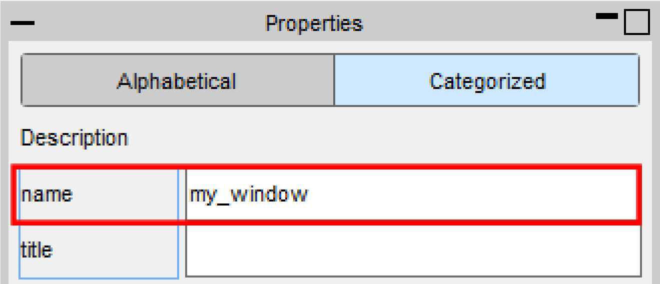
For example, to include the GUI saved in C:\my_gui.jsi:

Use("C:\\my_gui.jsi");

18.2.2. Accessing the Window Objects

Accessing the Window objects

The GUI Window objects are stored as properties on the global GUI object. The name of the property is whatever was defined in the properties window in the GUI builder.



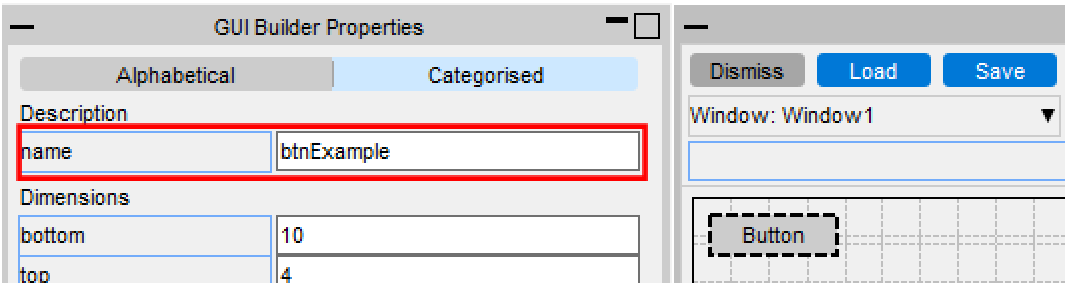
To display the Window called **my_window** use the Show() method:

if (gui) gui.my_window.Show();

18.2.3. Accessing the Widget Objects

Accessing the Widget objects

Similarly, each Widget object is a property of the Window object. The name of the Widget property is whatever was defined in the properties window in the GUI builder.



For example if the window is called **my_window** and the widget is called **btnExample**, the Widget object can be accessed and modified with.

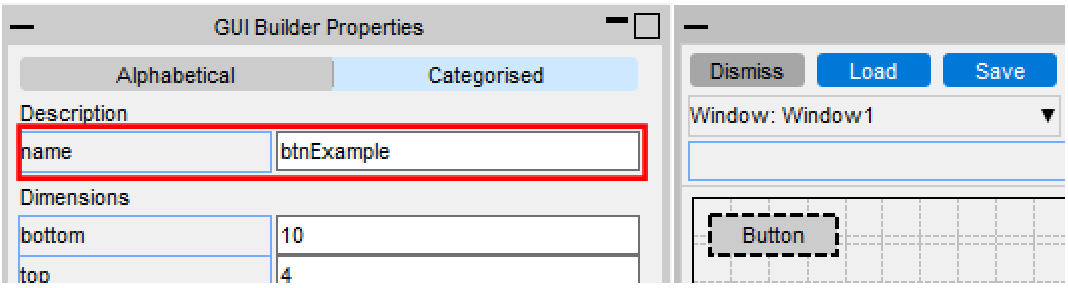
var btn = gui.my_window.btnExample;

btn.text = Test;

18.2.4. Accessing the Widgetitem Objects

Accessing the WidgetItem objects

WidgetItem objects are a property of the Widget.



For, example if the Window is called **my_window**, the Widget the WidgetItem is on is called **cbxExample** and the widget item is called **wi1**, it can be accessed and modified with.

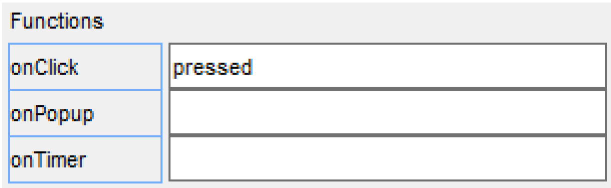
var wi = gui.my_window.cbxExample.wi1;

18.2.5. Defining Callback Functions

Defining callback functions

Callback functions (onClick, onChange, etc.) can be assigned to the window and widgets in the properties window, by adding the name of a function to call.

For example to set the onClick property of a widget so it calls a function called **pressed**:



This function then needs to be defined in your script:

Use("C:\\test.jsi");

if (gui) gui.my_window.Show();

function pressed()

{

Message("You clicked me!");

}

19. Workflow Tools

Workflow Tools

The Oasys Suite contains powerful tools and capabilities that can be used to interrogate and debug your analysis results. However...

1. The tools are not always customised for your specific loadcases or tasks
2. You may need to manually perform a number of steps to process your results, which can be time-consuming and prone to error
3. The JavaScript API can be used to create tools to automate your post-processing workflow, but this requires time, resource and knowledge, which is not always available

To address these issues, the Workflows feature provides tools customised for specific loadcases and tasks, built upon the existing capabilities in the Oasys Suite, to make it easier to interrogate and post-process results.

In addition to the tools provided (described in the following topics) you can create your own bespoke tools. Please contact us if you have an idea for a tool and would like some help creating it.

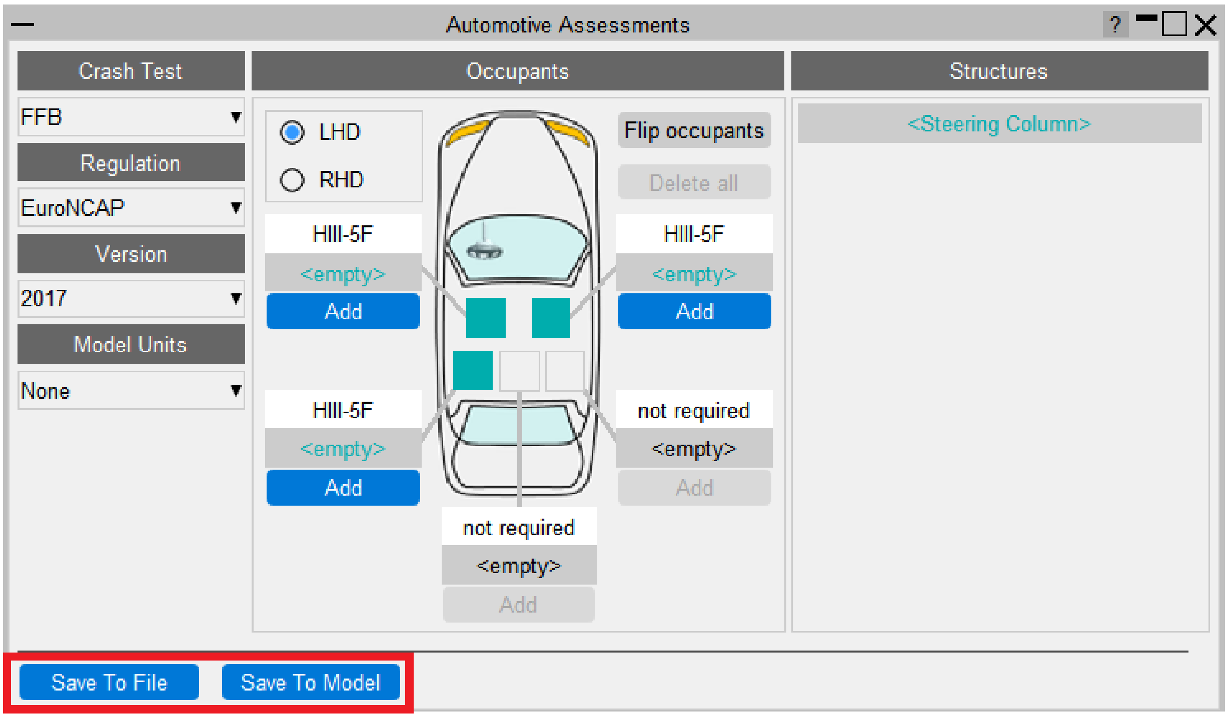
19.1. Workflow User Data

Saving user data

Workflows enable you to tag your models in PRIMER with user data which can be utilized in D3PLOT, T/HIS, and REPORTER to provide relevant post-processing analysis tools.

The user data can be saved either in a JSON file or as post *END data in the model keyword file.

In PRIMER each workflow tool will have buttons on their main window for saving either to a file or to a model, e.g. for the Automotive Assessments workflow:



User data saved to JSON files

If the data is saved to a JSON file it needs to be saved in the same directory as the model or in a parent directory for it to be found when loading model results in D3PLOT, T/HIS and REPORTER.

The name of the file can be anything you want, although it must have the ***.json*** extension.

Saving JSON files in parent directories means that you only have to setup the user data once and it can be used by multiple models.

For example in the below folder structure the ***parent_user_data.json*** file in dir_1 will be used for the results in dir_3 and dir_4 and the ***user_data.json*** file in dir_2 will be used for the model in dir_2:

- dir_1
    |- ***parent_user_data.json***

```
|
|- dir_2
|       |- model1.key
|       |- d3thdt
|       |- d3plot
|       |- user_data.json
|
|- dir_3
|       |- model2.key
|       |- d3thdt
|       |- d3plot
|
|- dir_4
|       |- model3.key
|       |- d3thdt
|       |- d3plot
```

Maximum number of directories to search up

The maximum number of directories up from a model/results directory that will be searched is set to 4 by default, but it can be changed by setting the preference ***oasys*workflow_max_upward_folder_search_depth***.

For example, in the following folder structure the ***grandparent_user_data.json*** file is 2 directories up from the model in dir_3 and will be found when reading the results into D3PLOT, T/HIS and REPORTER.

```
- dir_1
    |- grandparent_user_data.json
    |
    |- dir_2
    |       |- dir_3
    |            |- model1.key
    |            |- d3thdt
    |            |- d3plot
```

Search in workflow_user_data directory

The search for user data JSON files will also look in a folder named ***workflow_user_data*** in the model folder and its parent folders.

For example in the below folder structure, the ***parent_user_data.json*** file in dir_1/workflow_user_data will be used for the models in dir_3 and dir_4, and the ***user_data.json*** file in dir_2 will be used for the model in dir_2:

```
- dir_1
    |- workflow_user_data
    |       |- parent_user_data.json
    |
    |- dir_2
```

```
|       |- model1.key
|       |- user_data.json
|
|- dir_3
|       |- model2.key
|
|- dir_4
|       |- model3.key
```

The name of the directory to search can be changed by setting the preference **oasys*workflow_user_data_directory_name**

Writing user data for multiple workflows

When writing user data for multiple workflows you have two options.

- Write the user data for each workflow to a separate JSON file

```
- dir_1
    |- model1.key
    |- d3thdt
    |- d3plot
    |- user_data_workflow_1.json
    |- user_data_workflow_2.json
    |- user_data_workflow_3.json
```

- Write the user data for each workflow to a single JSON file

```
- dir_1
    |- model1.key
    |- d3thdt
    |- d3plot
    |- user_data.json    <- Contains user data for workflow 1, 2 and 3
```

The option you chose will depend on how you want to organise your files, but in terms of how the data is read in D3PLOT, T/HIS and REPORTER there is no difference.

To write multiple workflows you need to select an existing user data JSON file when saving the file.

- If user data for the workflow already exists in the file it will overwrite that data, but preserve the user data for any other workflows that already exist in the file.
- If user data for the workflow doesn't exist in the file it will append it to the existing user data for any other workflows in the file

User data saved in keyword files

If the data is saved to a model it is stored as post *END data in the master keyword file, e.g.

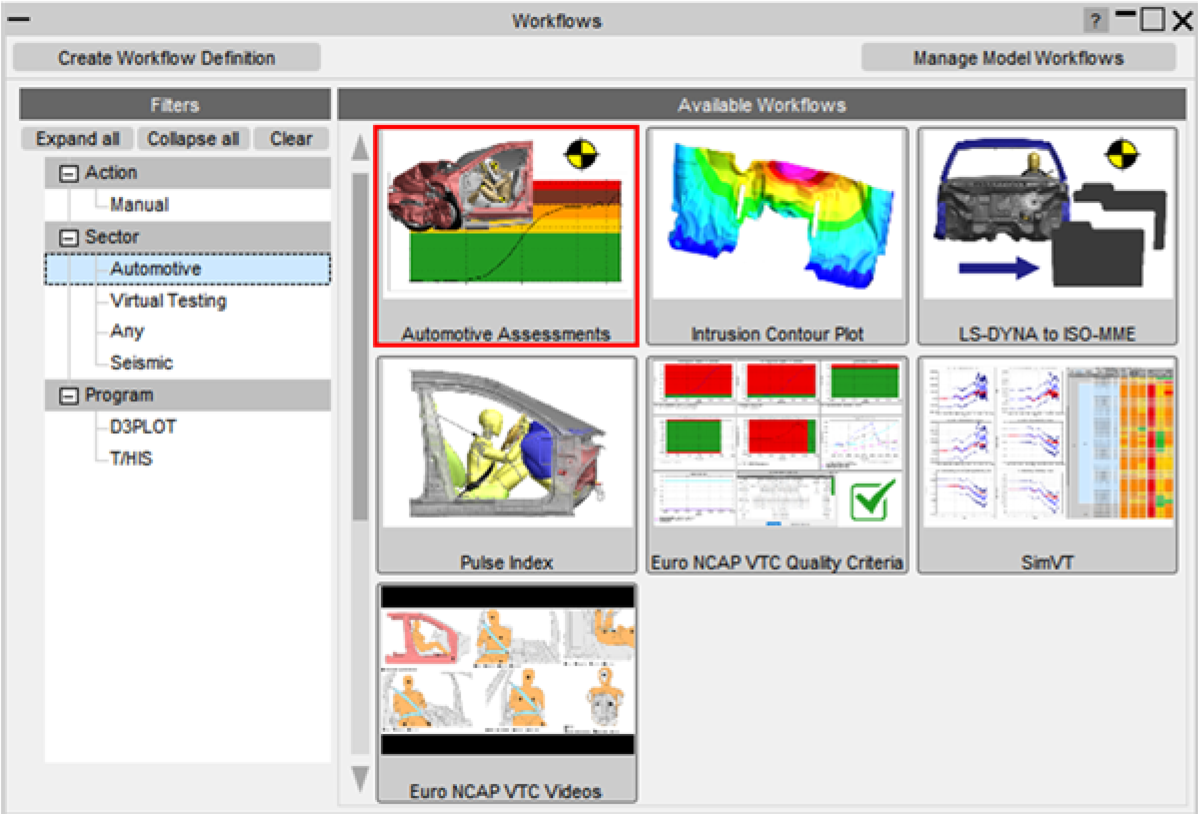
```
$
*END
*PRIMER_USER_DATA
WORKFLOW_START
{"workflows":[{"program":"PRIMER","major_version":21,"minor_version":0,"bu
ild_ +
number":34854,"workflow_definition":{"filename":"$OA_WORKFLOW\\automotive_
asse +
ssments.json"},"data":{"user_data_version":"21.0","regulations":["EuroNCAP
"]," +
crash_test":"FFB","version":"2017","drive_side":"LHD","occupants":[],"stru
ctur +
es":[],"b_pillar":null,"head_excursion":null,"head_intrusion":null},"model
_uni +
t_system":"U2"}]}
WORKFLOW_END
```

When you press the **Save To Model** button in the workflow window it is important to note that this adds the data to the model, but it doesn't automatically write the model to disk. You need to manually use **Model->Write** to save the data.

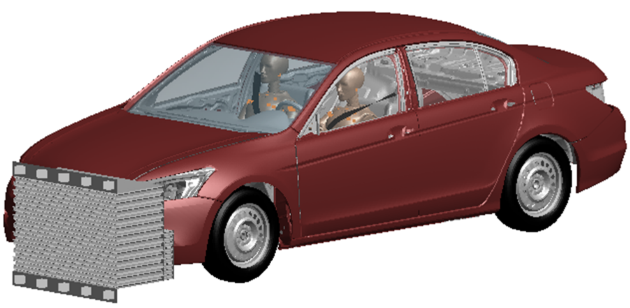
19.2. Automotive Assessments

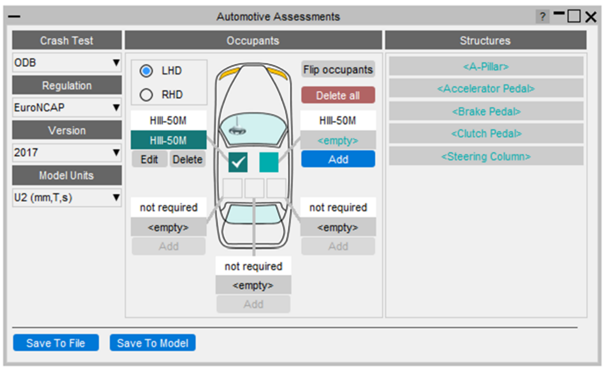
Automotive Assessments

The Automotive Assessments workflow tool is used to post-process analyses according to various crash test regulations.

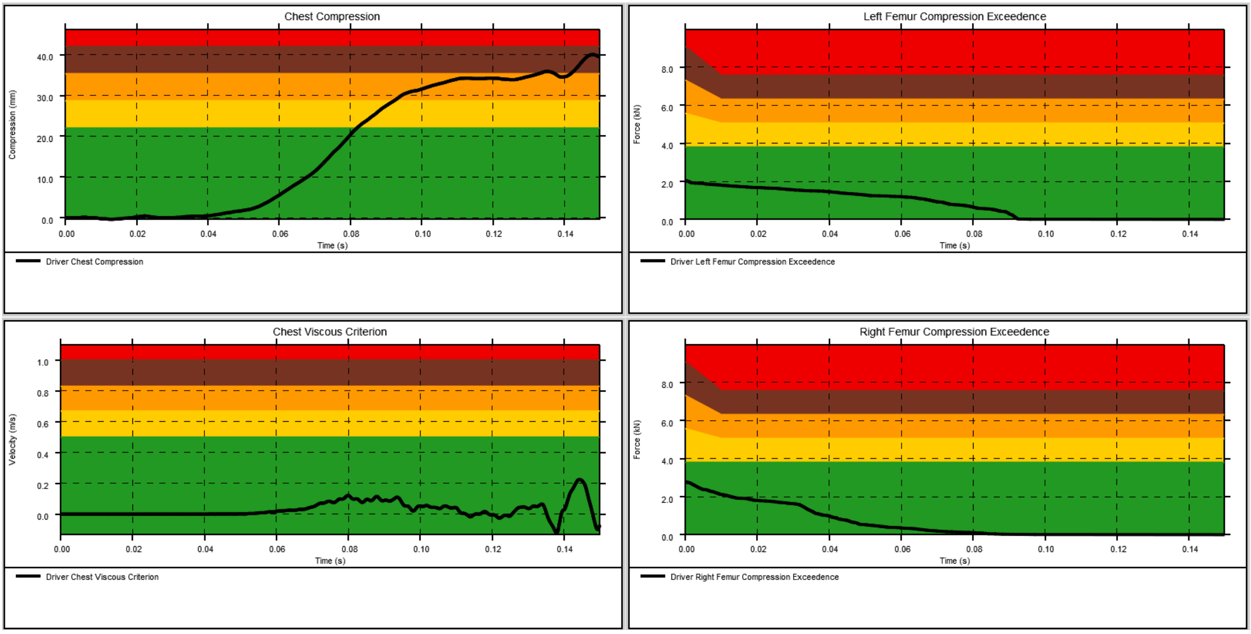


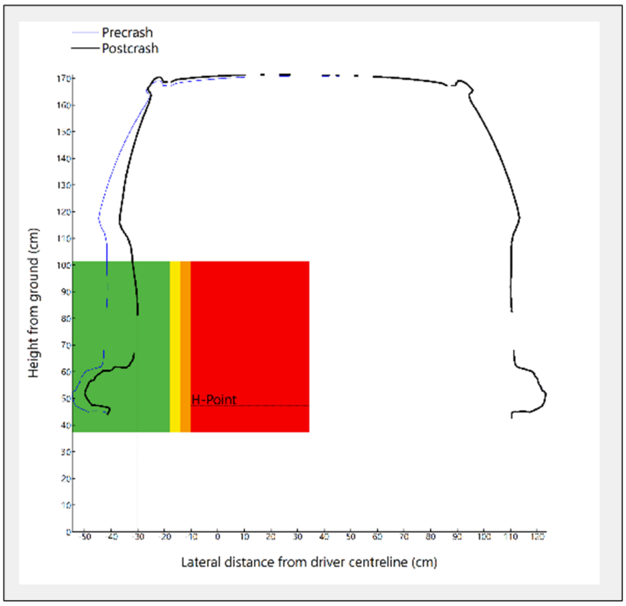
In PRIMER you select the crash test type and the occupants and structures to be assessed.





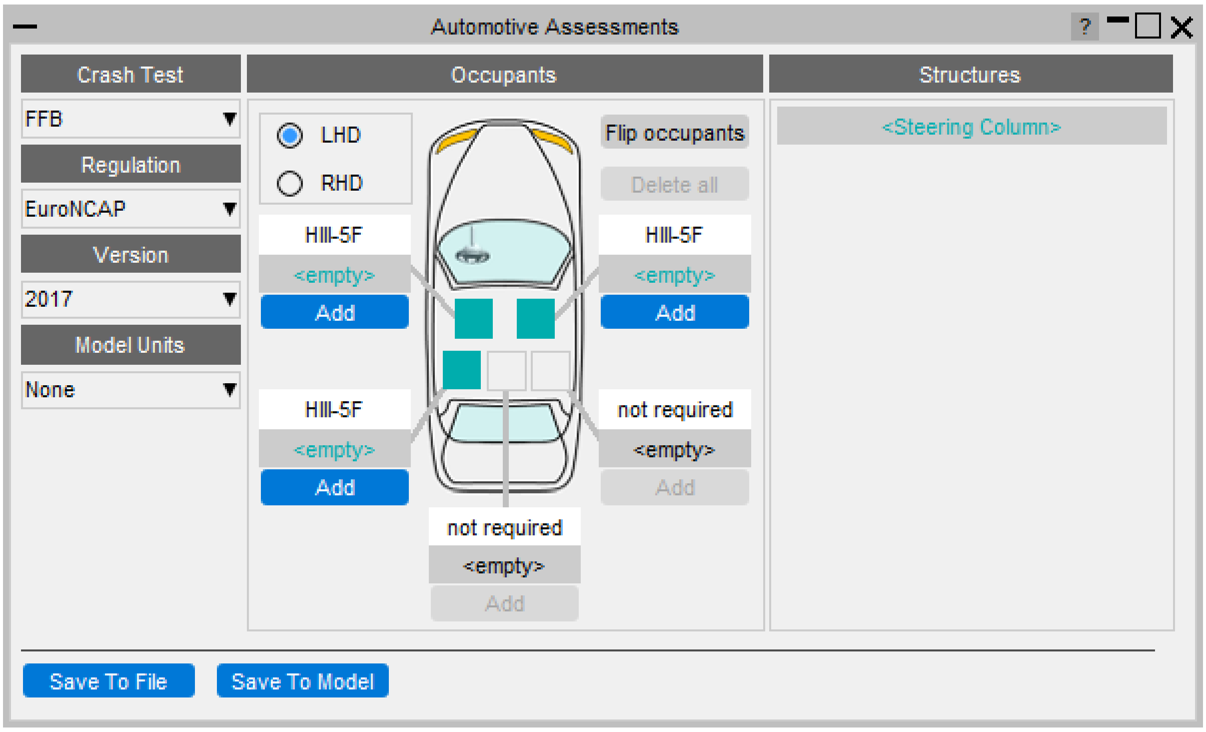
In D3PLOT, T/HIS and REPORTER this data is used to carry out assessments according to the crash test type and regulation.



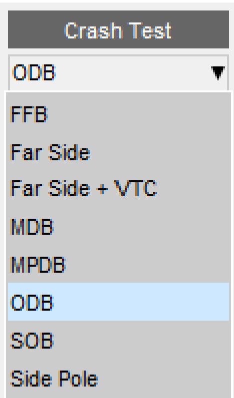


19.2.1. Automotive Assessments PRIMER

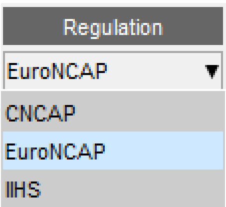
When the tool is launched in PRIMER you are presented with this window. This is where you specify the crash test type and the occupants and structures you want to assess:



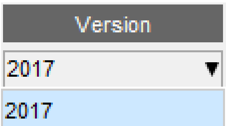
You first specify the crash test type of your model from the dropdown menu:



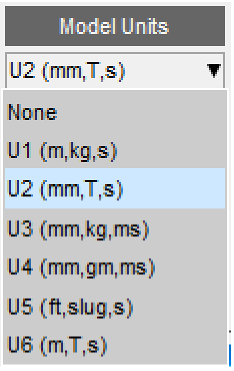
This will update the Regulation dropdown menu with regulations that are supported by the tool for the selected crash test type. Select the regulation you want to assess your model with:



The Version dropdown menu will get updated with versions that are supported by the tool for the selected crash test type and regulation. Select the version you want to assess your model with:

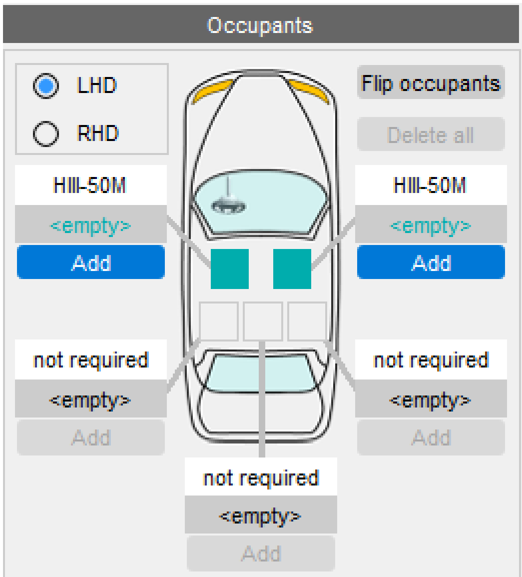


Then select the unit system of your model:

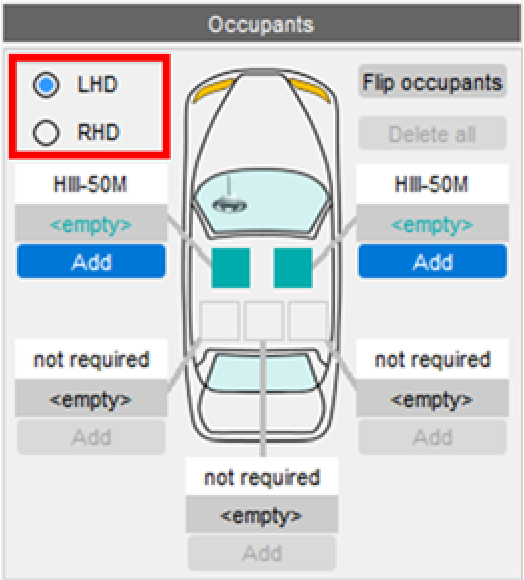


Occupants

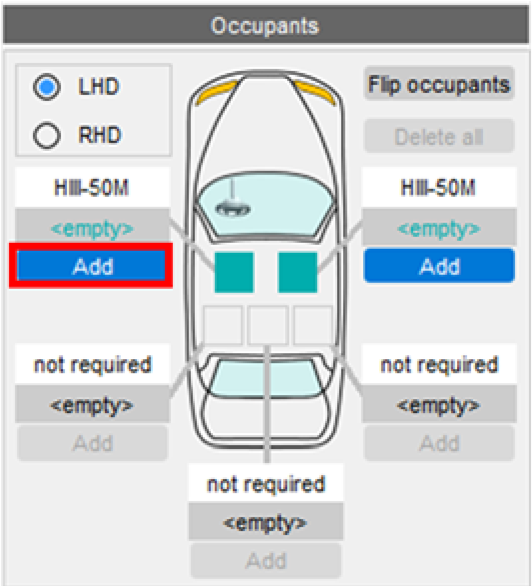
The Occupants section is used to select what occupants are in the vehicle, their positions in the vehicle and the IDs of the entities from which data can be read from. This section updates automatically to show the required occupants for the selected crash test type and regulation. In the image below it shows that two HIII-50M occupants are expected in the driver and front passenger positions.



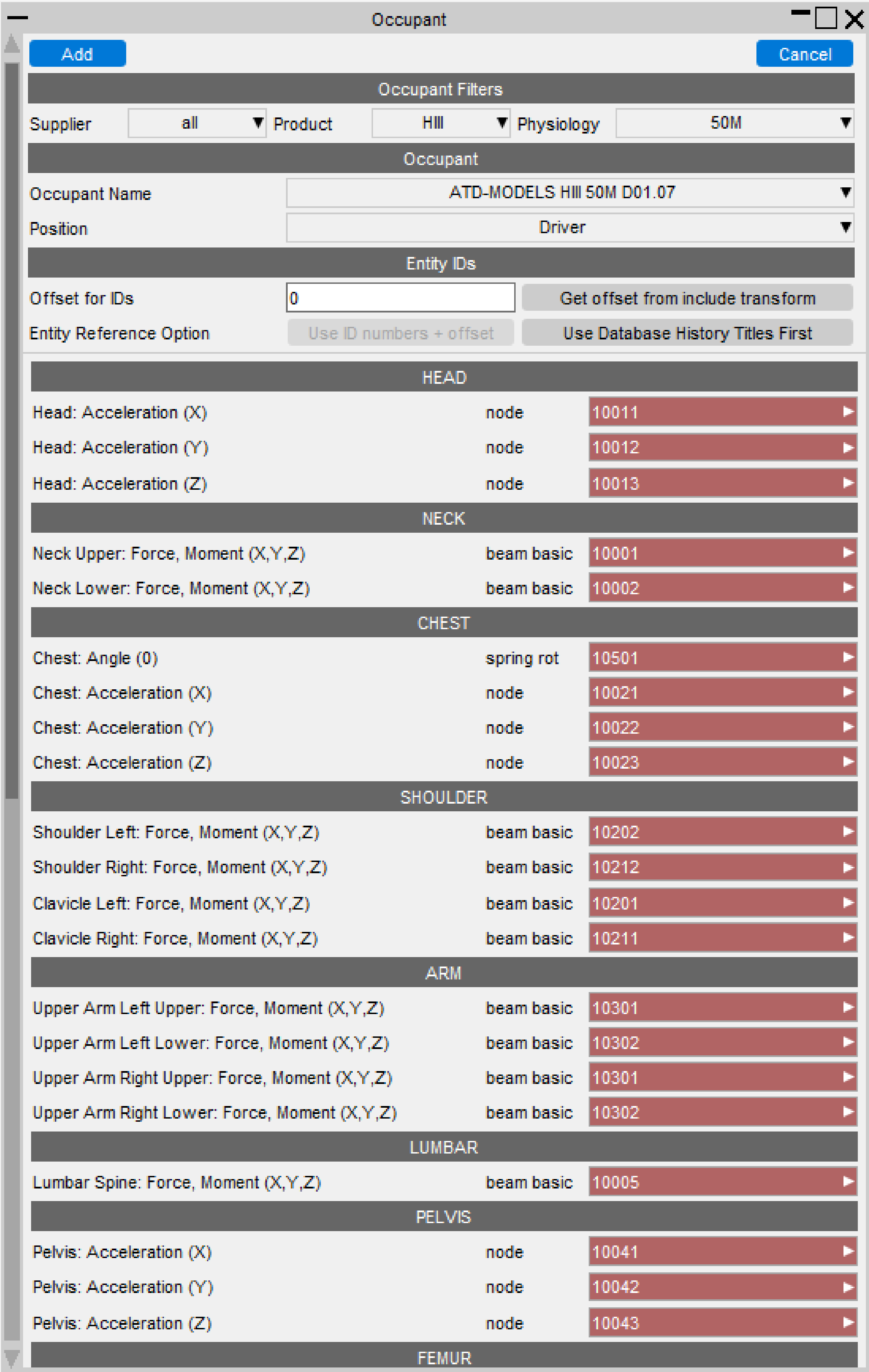
The first thing to do is select whether the vehicle is left hand or right hand drive (LHD or RHD). The vehicle image will update to show the steering wheel on the correct side and the occupant positions will also update:



For each occupant click on the Add button. Note that if your model does not have an occupant (or you don't want to carry out an assessment on it) you can leave it empty. T/HIS and D3PLOT will only attempt to process results for occupants that have been added.



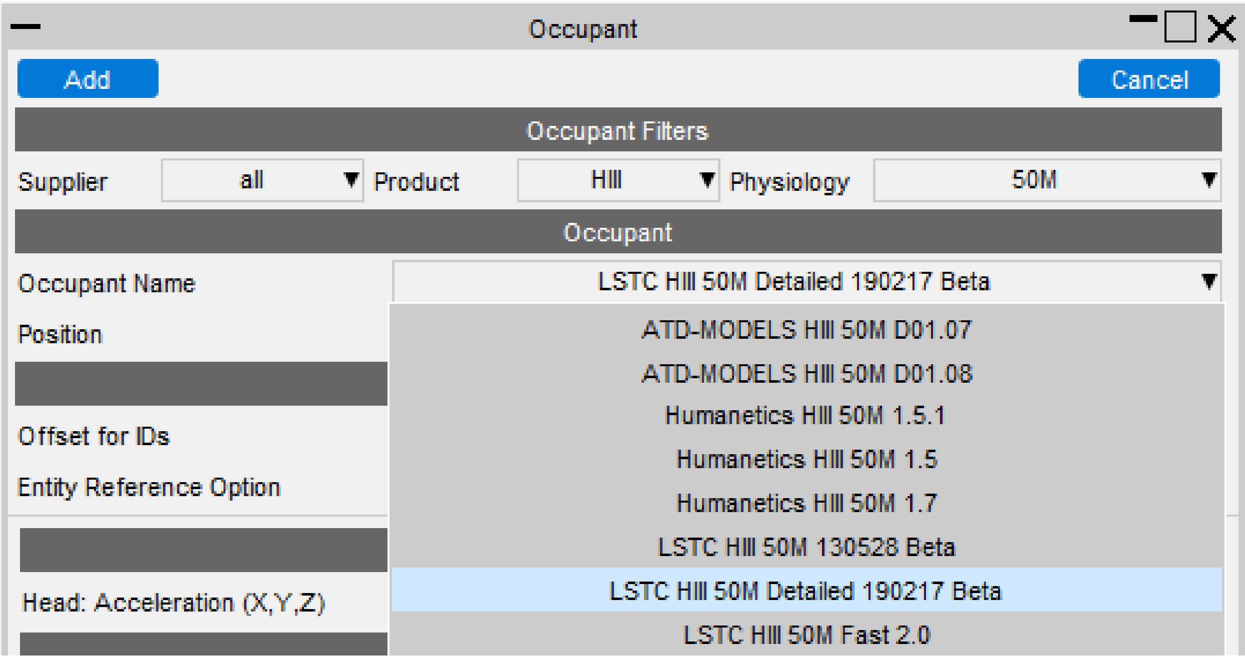
This will open a window where you can set the occupant type and select the IDs of entities:


Occupant
Add
Cancel
Occupant Filters
Supplier
all
Product
HIII
Physiology
50M
Occupant
Occupant Name
ATD-MODELS HIII 50M D01.07
Position
Driver
Entity IDs
Offset for IDs
0
Get offset from include transform
Entity Reference Option
Use ID numbers + offset
Use Database History Titles First
HEAD
Head: Acceleration (X) node 10011
Head: Acceleration (Y) node 10012
Head: Acceleration (Z) node 10013
NECK
Neck Upper: Force, Moment (X,Y,Z) beam basic 10001
Neck Lower: Force, Moment (X,Y,Z) beam basic 10002
CHEST
Chest: Angle (0) spring rot 10501
Chest: Acceleration (X) node 10021
Chest: Acceleration (Y) node 10022
Chest: Acceleration (Z) node 10023
SHOULDER
Shoulder Left: Force, Moment (X,Y,Z) beam basic 10202
Shoulder Right: Force, Moment (X,Y,Z) beam basic 10212
Clavicle Left: Force, Moment (X,Y,Z) beam basic 10201
Clavicle Right: Force, Moment (X,Y,Z) beam basic 10211
ARM
Upper Arm Left Upper: Force, Moment (X,Y,Z) beam basic 10301
Upper Arm Left Lower: Force, Moment (X,Y,Z) beam basic 10302
Upper Arm Right Upper: Force, Moment (X,Y,Z) beam basic 10301
Upper Arm Right Lower: Force, Moment (X,Y,Z) beam basic 10302
LUMBAR
Lumbar Spine: Force, Moment (X,Y,Z) beam basic 10005
PELVIS
Pelvis: Acceleration (X) node 10041
Pelvis: Acceleration (Y) node 10042
Pelvis: Acceleration (Z) node 10043
FEMUR

The occupant type can be selected from the Occupant Name dropdown menu.

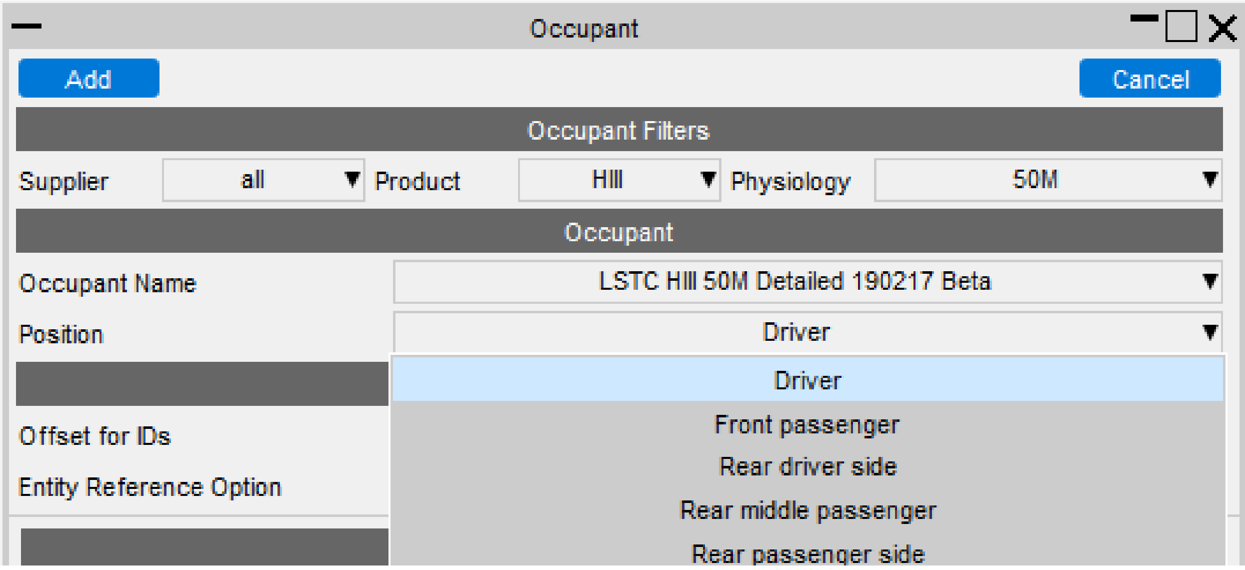
The options shown in this dropdown are filtered by the values in the Supplier, Product and Physiology dropdown menus. When the window is first opened these are automatically set so only occupant types that are relevant for the selected crash test type, regulation and occupant position are shown.

In this example the selected occupant is expected to be a HIII-50M occupant so the Product filter is set to HIII and the Physiology filter is set to 50M.



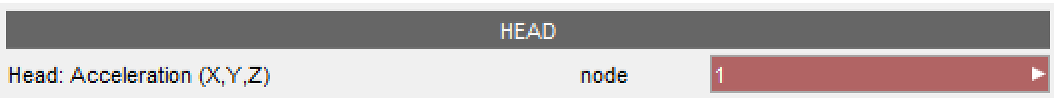
If for some reason you want to select an occupant of a different type to the one expected, you can change the values of the filters to list other occupant types.

The position of the occupant in the vehicle will be set automatically, but you can change this with the Position dropdown menu if required:

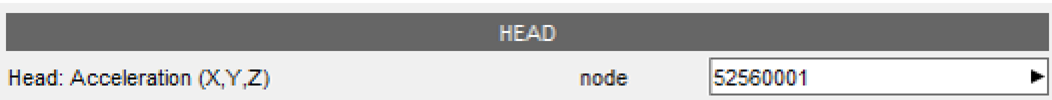


Entity IDs can be specified either by their numerical labels or DATABASE_HISTORY titles (for entities that have them defined) or DATABASE_CROSS_SECTION titles for X-Sections.

The tool knows what the default numerical labels are for each entity in each occupant type and will automatically fill the textboxes in with those values. If they do not exist in the model, for example if the model has been renumbered, the textboxes are coloured red, e.g.

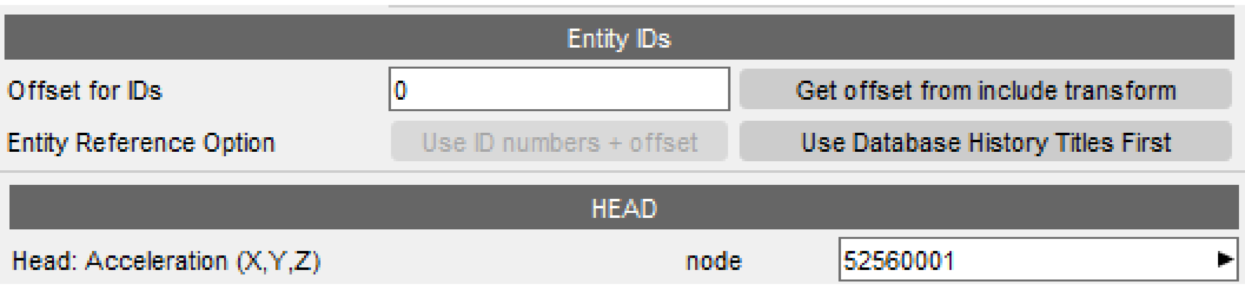


If they do exist the textboxes will change colour to indicate that (the colour will depend on the UI Theme), e.g.

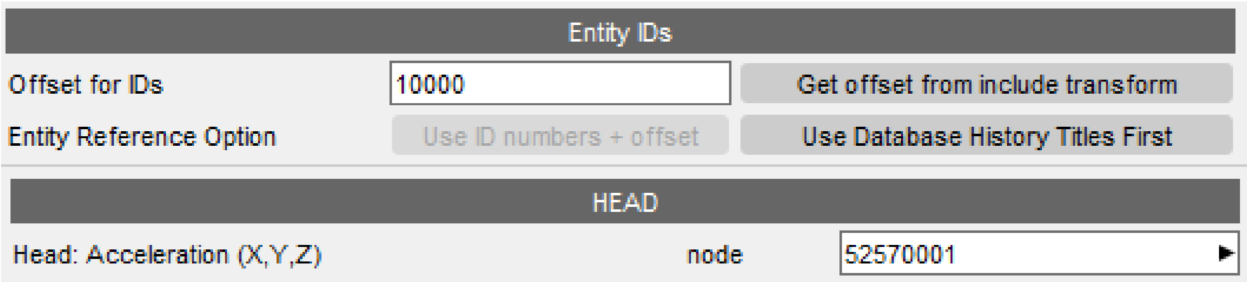


If the occupant has been renumbered so the labels are offset from the default ones, the Offset for IDs option can be used to apply the offset. This is useful when you have two or more occupants of the same type in the model as they both can't have the same entity labels.

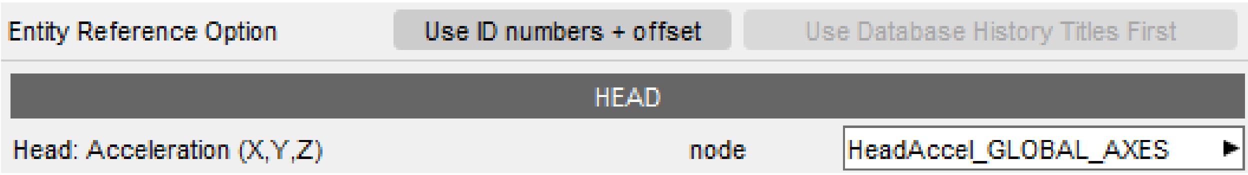
Offset = 0:



Offset = 10000:

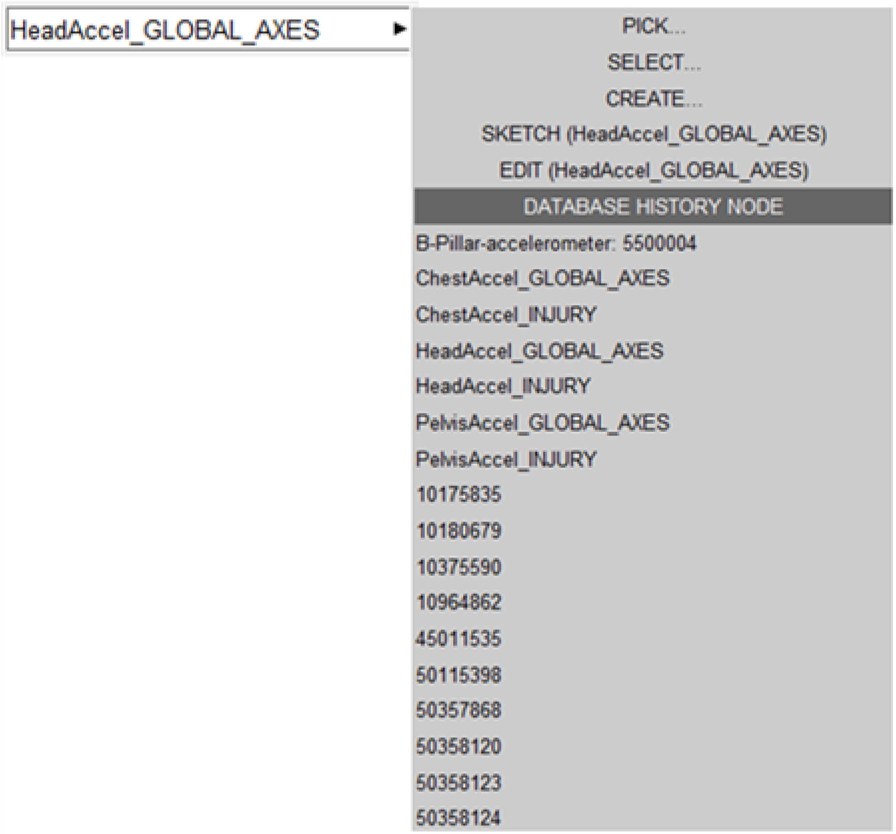


Alternatively, the entity IDs can be specified using DATABASE_HISTORY and DATABASE_CROSS_SECTION titles instead of their numerical labels. To automatically switch to use titles where they exist you can click on the Use Database History Titles button:

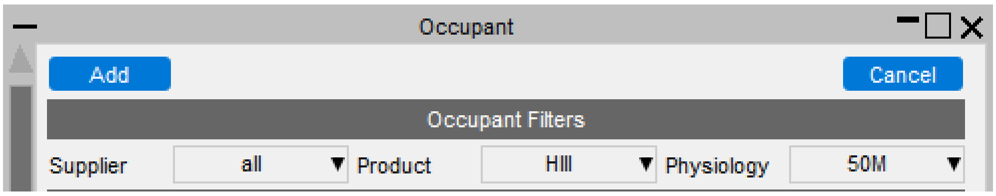


Note that a mix of defining some entities using numerical labels and others with titles is perfectly valid, they don't all have to be defined the same way.

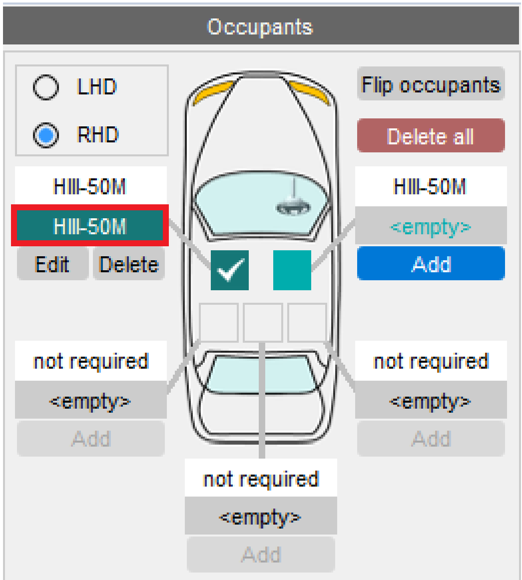
You can manually select entities by right clicking on a textbox. This opens a popup window which allows you to Pick or Select the entity interactively on the screen or select it from a list of DATABASE_HISTORY / DATABASE_CROSS_SECTION entities (ones with titles are listed first and ones without at the bottom).



Once the entities have been defined you can press Add at the top of the window to add the occupant definition.



This will close the window and the Occupants section will update to show that an occupant has been defined in the selected position. If all the entity IDs are defined and valid the occupant will be shown like this:



If there are any undefined or invalid entity IDs it will look like this, i.e. enclosed in < >'s:

If the occupant is a different type to the one expected it will look like this:

T/HIS and D3PLOT will cope with any undefined or invalid, but obviously won't be able to carry out any assessments that require them.

If you want to edit or delete the occupant, you can click on the Edit or Delete buttons.

Supported Dummies

Below is a table listing all the dummies supported in Automotive Assessment workflows, along with the corresponding dummy supplier manuals filename that were referenced in creating the respective occupant JSON files.

Supplier	**Product**	**Physiology**	**Version**	**JSON**	**Manual**
PDB	WS ID	50M	v4.0 LHD	PDB WSID 50M v4.0 LHD	wsid50_pdb_v4.0_manual_v0.0.pdf

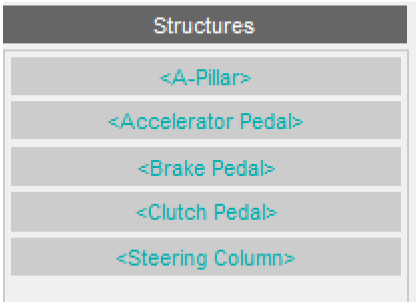
PDB	WS ID	50M	v4.0 RHD	PDB WSID 50M v4.0 RHD	wsid50_pdb_v4.0_manual_v0.0.pdf
PDB	WS ID	50M	v6.0 LHD	PDB WSID 50M v6.0 LHD	wsid50_pdb_v6.0_manual_v0.0.pdf
PDB	WS ID	50M	v6.0 RHD	PDB WSID 50M v6.0 RHD	wsid50_pdb_v6.0_manual_v0.0.pdf
PDB	WS ID	50M	v7.6 LHD	PDB WSID 50M v7.6 LHD	wsid50_pdb_v7.6.1_manual_v0.pdf
PDB	WS ID	50M	v7.6 RHD	PDB WSID 50M v7.6 RHD	wsid50_pdb_v7.6.1_manual_v0.pdf
PDB	WS ID	50M	v8.0 LHD	PDB WSID 50M v8.0 LHD	wsid50_pdb_v8.0_manual.pdf
PDB	WS ID	50M	v8.0 RHD	PDB WSID 50M v8.0 RHD	wsid50_pdb_v8.0_manual.pdf
PDB	WS ID	50M	v8.1 LHD	PDB WSID 50M v8.1 LHD	wsid50_pdb_v8.1_manual.pdf
PDB	WS ID	50M	v8.1 RHD	PDB WSID 50M v8.1 RHD	wsid50_pdb_v8.1_manual.pdf
DYNA MORE	ES-2re	50M	v6.0	DYNA MORE ES-2re	es2_v_6.0_users_manual_v0.0.pdf

				50M v6.0	
LSTC	SID2-SBDL	5F	v.0.150.beta	LSTC SID2-SBDL 5F v.0.150.beta	Documentation_for_LSTC_SID-IIs-D_Version_0.150.beta.pdf
LSTC	HIII	50M	Detailed 190217 Beta	LSTC HIII 50M Detailed 190217 Beta	LSTC.H3_50TH.130528_BETA.pdf
LSTC	HIII	50M	Fast 2.0	LSTC HIII 50M Fast 2.0	LSTC.H3_50TH_FAST.111130_V2.0_Documentation.pdf
LSTC	HIII	5F	Fast 2.0	LSTC HIII 5F Fast 2.0	LSTC.H3_5TH_FAST.111130_V2.0_Documentation.pdf
LSTC	HIII	5F	v2	LSTC HIII 5F v2	LSTC.H3_05TH_DETAILED.160920_V2.0.pdf
LSTC	HIII	5F	v2.1	LSTC HIII 5F v2.1	LSTC.H3_05TH_DETAILED.160920_V2.0.pdf
HUMANETICS	SID2-SBDL	5F	v.4.3.2	HUMANETICS SID2-SBDL 5F v.4.3.2	Humanetics_SID2s_SBLD_V4.3.2_LS-DYNA_UserManual_TechnicalReport.pdf
HUMANETICS	SID2-SBDL	5F	v.4.3.5	HUMANETICS SID2-SBDL 5F v.4.3.5	Humanetics_SID2s_SBLD_V4.3.5_LS-DYNA_UserManual_TechnicalReport.pdf
HUMANETICS	THOR	50M	v1.9	HUMANETICS THOR 50M v1.9	HUMANETICS_THOR_50M_USNCAP_V1.9_LS_DYNA_TECHNICAL_REPORT_USERS_MANUAL.pdf

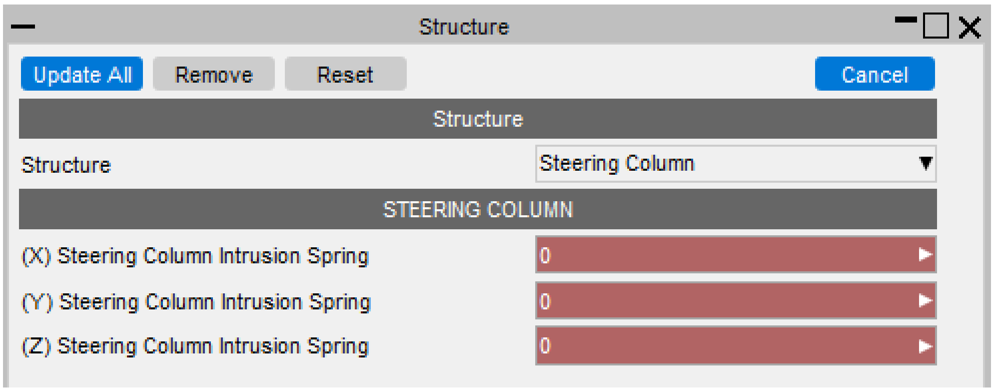
HUMANETICS	THOR	50M	v1.8	HUMANETICS THOR 50M v1.8	HUMANETICS_THOR_50M_EuroNCAP_V1.8_LS_DYNA_TECHNICAL_REPORT_USERS_MANUAL.pdf
HUMANETICS	HIII	50M	v.1.5	HUMANETICS HIII 50M v.1.5	HUMANETICS_HIII_50M_V1.5.1_HARMONIZED_LS_DYNA_TECHNICAL_REPORT_USER_MANUAL.pdf
HUMANETICS	HIII	50M	v.1.5.1	HUMANETICS HIII 50M v.1.5.1	HUMANETICS_HIII_50M_V1.5.1_HARMONIZED_LS_DYNA_TECHNICAL_REPORT_USER_MANUAL.pdf
HUMANETICS	HIII	50M	v1.7	HUMANETICS HIII 50M v1.7	HUMANETICS_HIII_50M_V1.7_HARMONIZED_LS_DYNA_TECHNICAL_REPORT_USER_MANUAL.pdf
HUMANETICS	HIII	5F	v.2.02	HUMANETICS HIII 5F v.2.02	HUMANETICS_HIII_5F_V2.0_HARMONIZED_LS_DYNA_TECHNICAL_REPORT_USER_MANUAL.pdf
ATD_MODELS	HIII	50M	D01.07	ATD_MODELS HIII 50M D01.07	atd-h350-d01.07_91_user_manual_v01.11_en.pdf
ATD_MODELS	HIII	50M	D01.08	ATD_MODELS HIII 50M D01.08	atd-h350-d01.08_91_user_manual_v01.11_en.pdf

Structures

The Structures section is used to select the IDs of the entities from which data can be read from structures in the vehicle. This section updates automatically to show the required structures for the selected crash test type and regulation. In the image below it shows that an A-Pillar, Accelerator Pedal, Brake Pedal, Clutch Pedal and Steering Column structures are to be assessed.

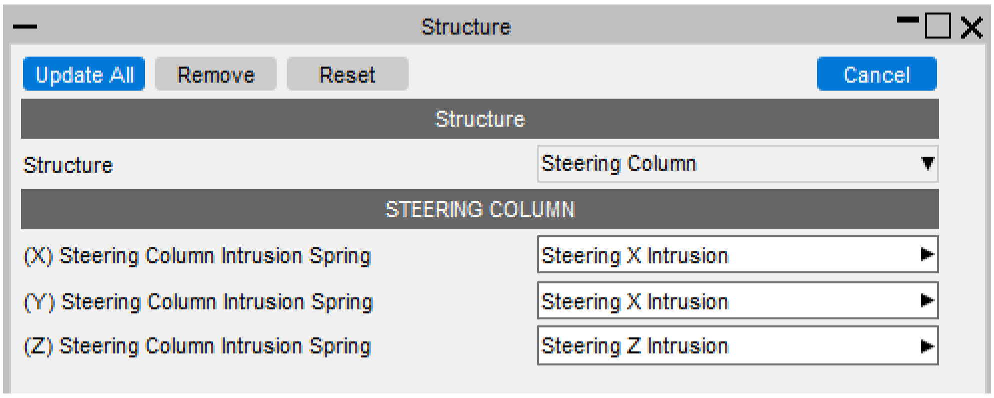


Click on one of the structures to open a window where you can select the entity IDs.

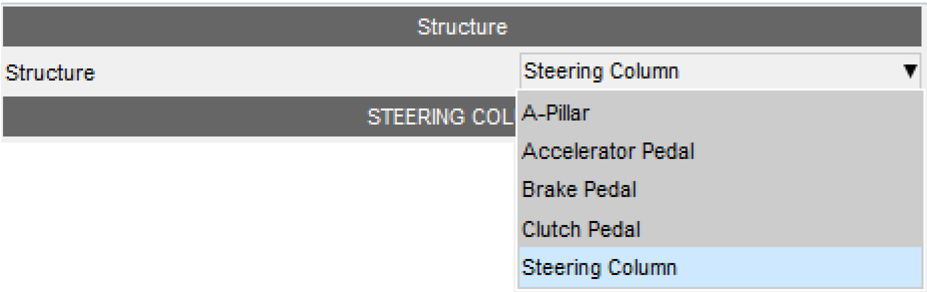


This works in the same way as the occupants window where IDs can be specified either by their numerical labels or DATABASE_HISTORY titles (for entities that have them defined).

If they do not exist in the model, the textboxes are coloured red. If they do exist they change colour (the colour will depend on the UI Theme), e.g.

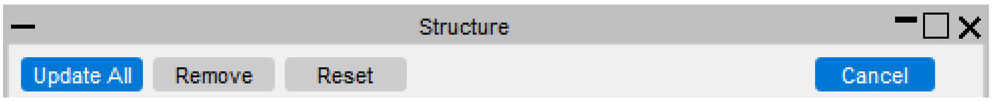


To select entity IDs for other structures you can use the Structure dropdown menu:



Note that if your model does not have a structure (or you don't want to carry out an assessment on it) you can leave it empty. T/HIS and D3PLOT will only attempt to process results for structures that have been added.

Once you have selected all the entity IDs click on Update All to save them and close the window.

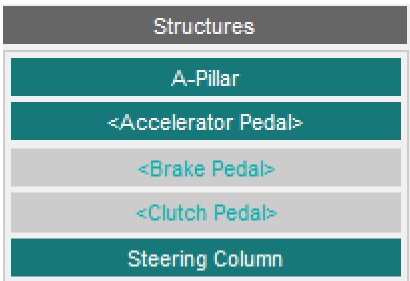


The Remove button sets all the entity IDs of the current structure to 0, effectively removing it from the assessment.

The Reset button sets the entity IDs back to what they were before any edits were made.

The Cancel button closes the window, without saving the selected entity IDs.

After you have selected entity IDs the structures section will update with different colours to show what is defined and what is not, e.g



Structures that are coloured like this mean they have all the required entity IDs defined and the exist in the model:

Structures that are coloured like this and enclosed in < >'s mean some of the required entity IDs are defined and exist in the model, but there are others that are either undefined or don't exist in the model:

Structures coloured like this and enclosed in < >'s mean none of the required entity IDs are defined or exist in the model:

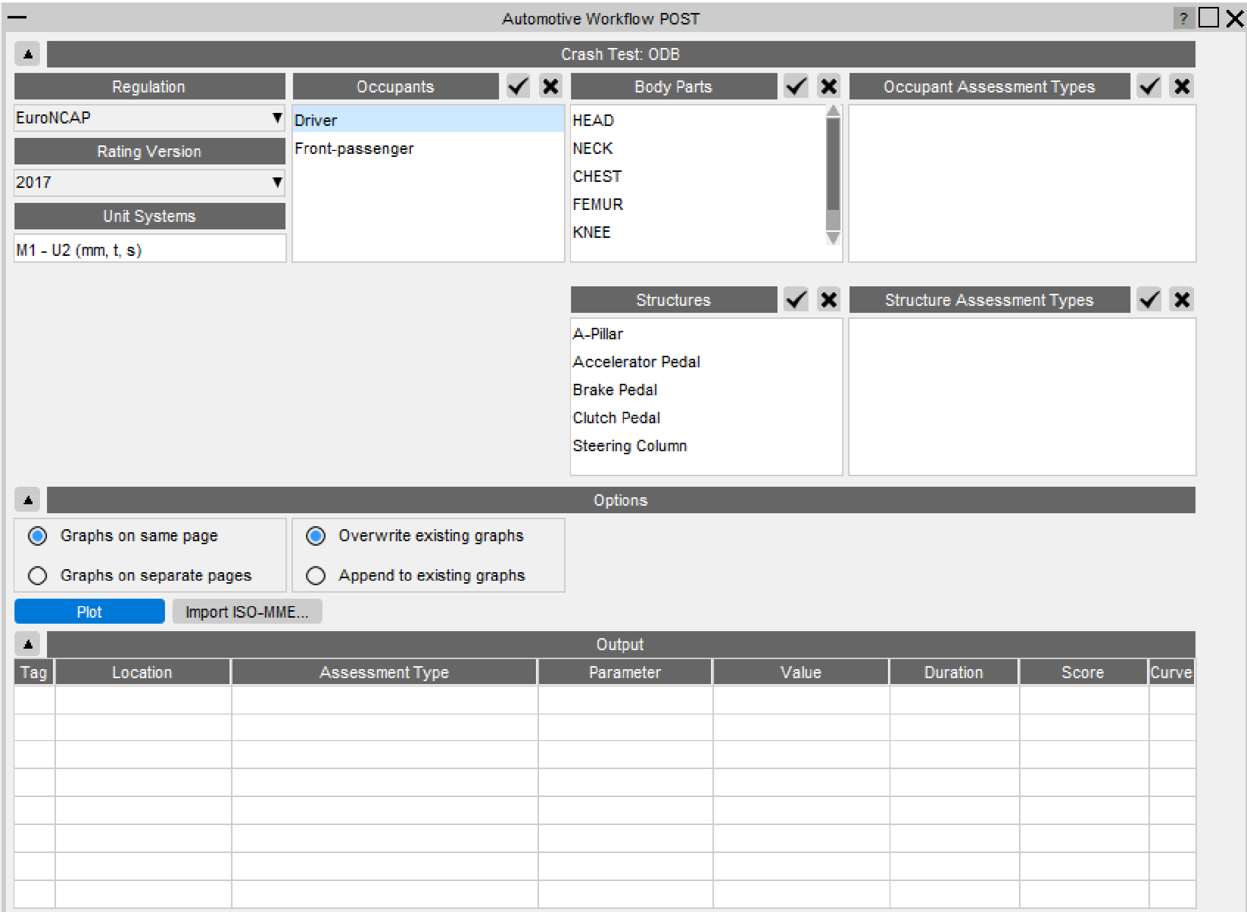
Saving

Users can choose to save the data to a .JSON file or directly to the model. The user data from the file will then be picked up when the workflow is selected in T/HIS or D3PLOT. Note if you choose to save the data to the model then you will need to save the model in PRIMER to ensure that the data is written to the master keyword file.

19.2.2. Automotive Assessments T-HIS

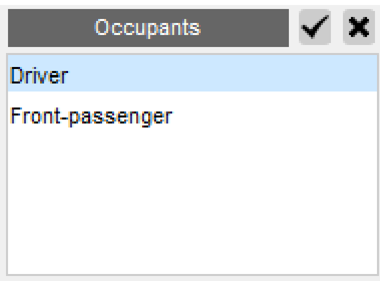
When the tool is launched in T/HIS you are presented with this window. This is where you select what assessments you want to carry out.

The dropdown menus on the left hand side show the regulation being used to carry out the assessments and the version. The model unit system is also shown.

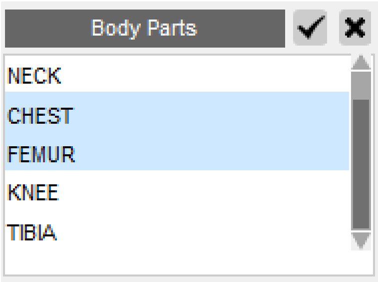


To select what assessments to carry out, you first need to select which occupant(s) you want to assess.

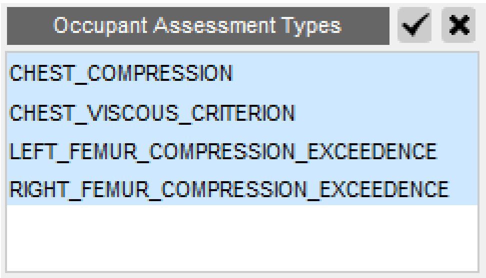
To select a single occupant left click on the one you want to assess. Use shift and left-click or ctrl and left-click to select multiple occupants. If you want to select all the occupants you can press the tick button and to deselect them all press the cross.



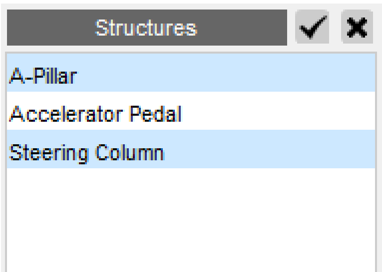
You can then select the body part(s) you want to assess.



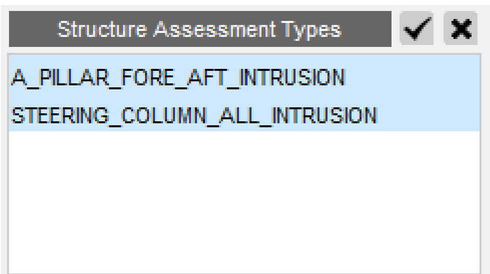
This will populate the Occupant Assessment Types list with the assessments that can be carried out for the selected body parts and occupants. By default they will all be selected, but you can chose to select only a subset of the list if you don't want to do them all.



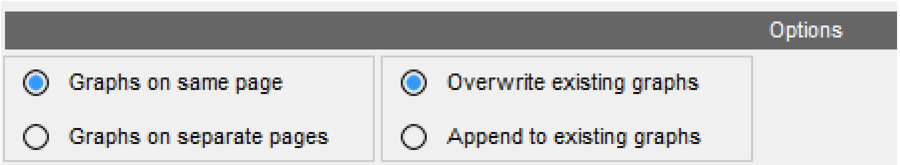
You can also select which structure(s) which you want to assess



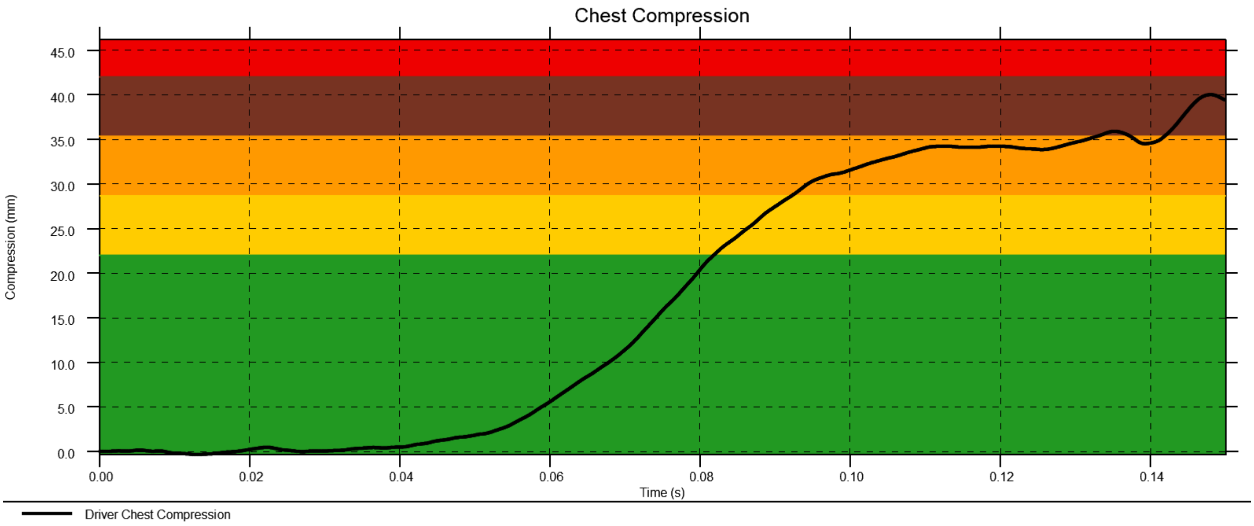
This will populate the Structure Assessment Types list with the assessments that can be carried out for the selected structures.



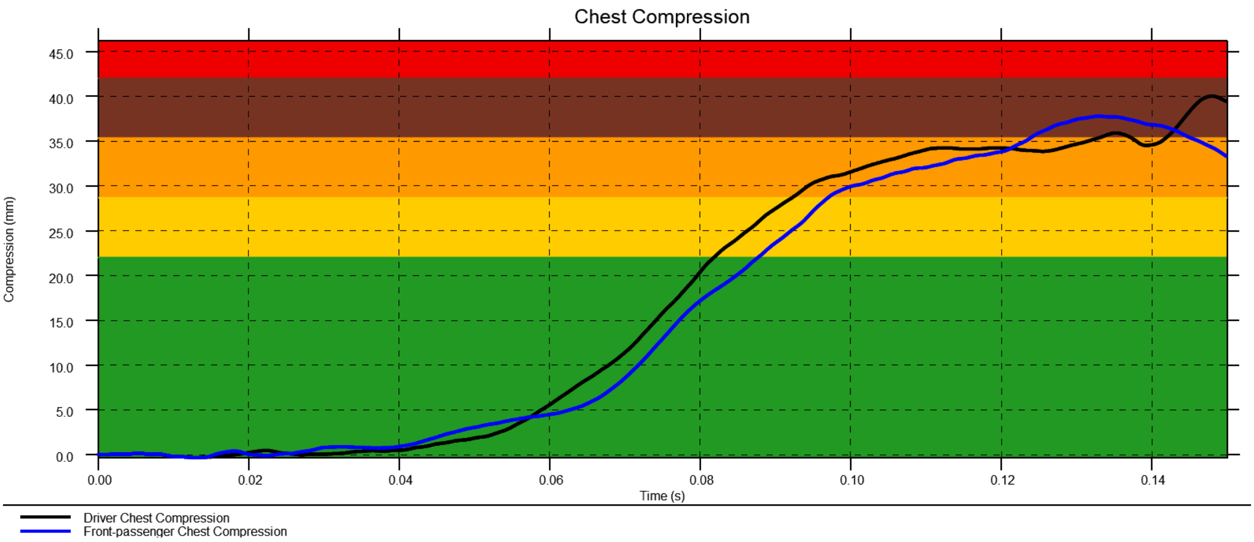
You can then chose how the graphs for each assessment should be displayed. By default they will all be put on one page and overwrite any existing graphs, but you can also chose to put each one on a separate page and append them to existing graphs.



Once you are happy with your choices, press the Plot button to carry out the assessments. T/HIS will extract the data required for each assessment, process it according to the rules set out in the regulation and plot the results on a graph with datums showing allowable limits (where they are defined by the regulation), e.g. the CHEST_COMPRESSION assessment for the driver:



If you have selected multiple occupants the curves for each occupant will be plotted on the same graph if the datum values are the same. If the datum values are different they will be plotted on separate graphs.



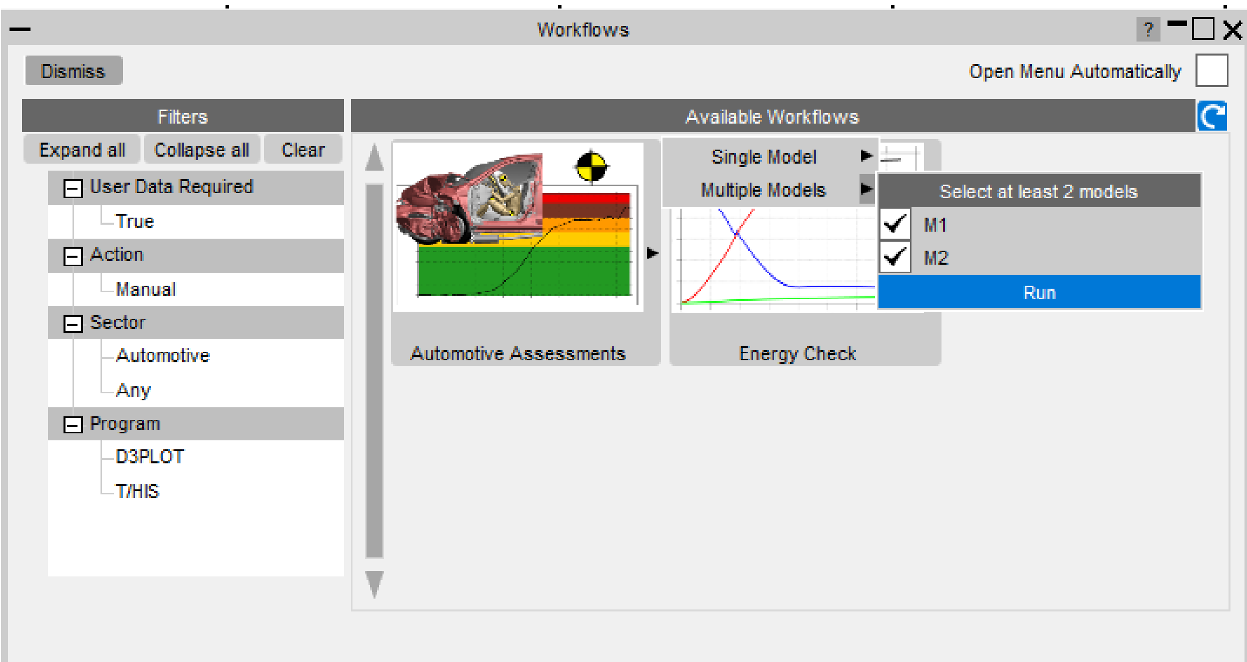
The output box at the bottom of the window lists the values and scores from the assessments carried out. Clicking on the '->' for each assessment will highlight the curve used for the assessment (and select the page if it's not on the current page) to make it easy to locate:

Output							
Tag	Location	Assessment Type	Parameter	Value	Duration	Score	Curve
M1	Driver	CHEST_COMPRESSION	Max	40.0235 mm		0.3953	->
M1	Front passenger	CHEST_COMPRESSION	Max	37.7902 mm		0.8420	->

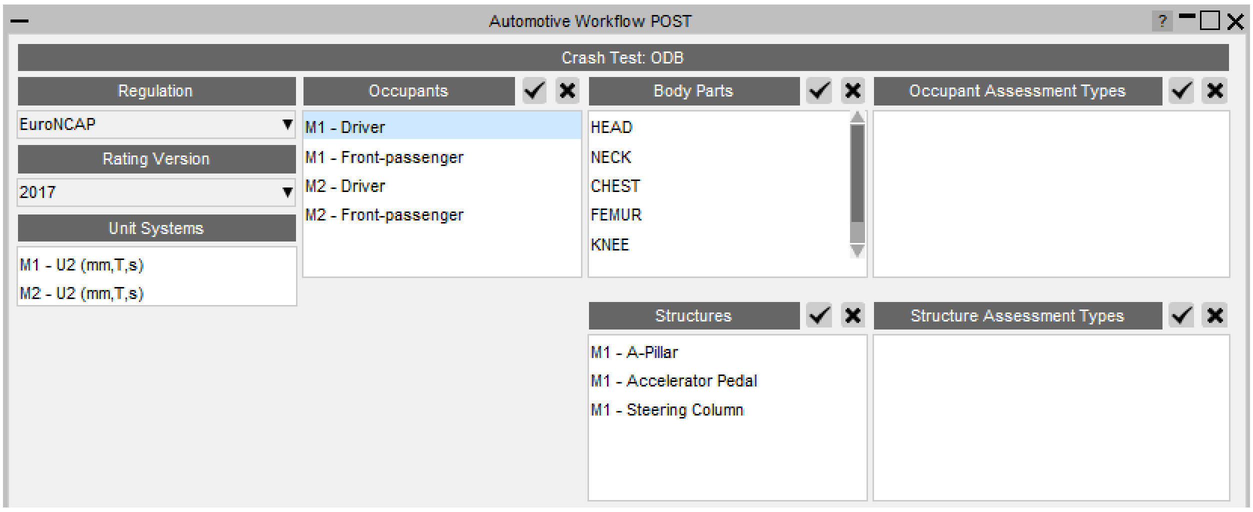
Multiple Models

It is also possible to plot results from multiple models on the same graphs. This is useful when you want to compare results between different runs.

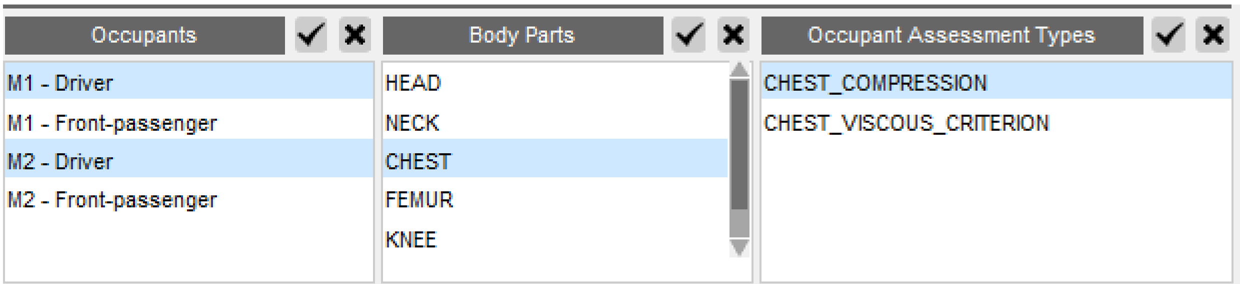
First you'll need to load the results from the models you want to compare into T/HIS and then on the workflow menu, select Multiple Models, pick the models you want to compare and press Run. Note that the models need to be of the same crash test type and regulation. If they're not the tool will refuse to run.

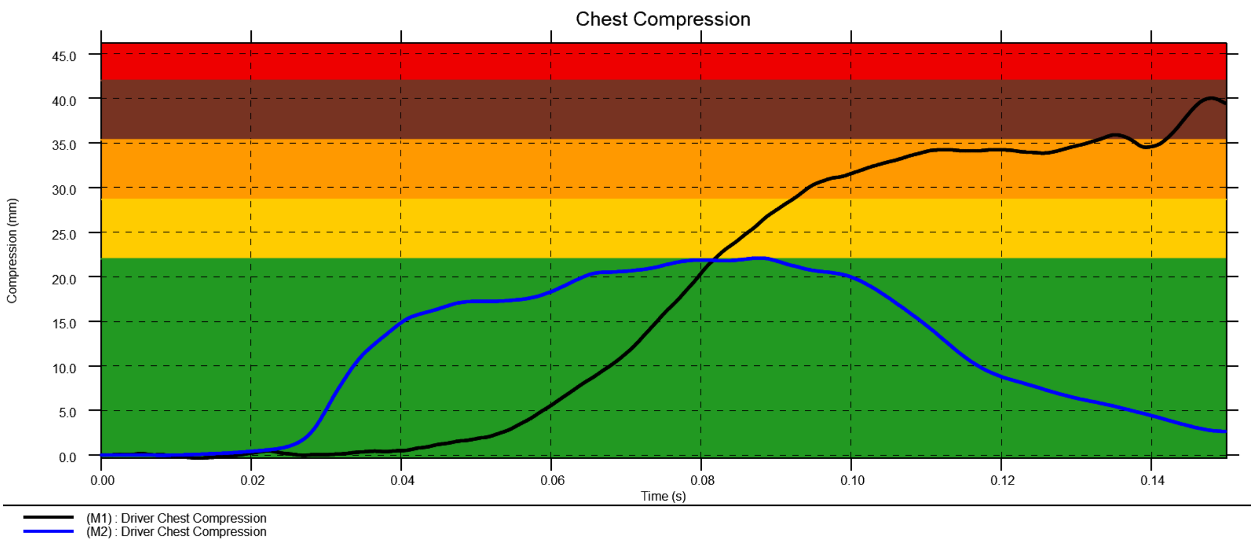


The window will then be populated with the occupants and structures from all the selected models, pre-pending them with the model number (M1, M2, etc)



If you wanted to compare the results for a CHEST_COMPRESSION assessment of the driver you would select the occupants in both models, select the chest body part and the CHEST_COMPRESSION assessment type.

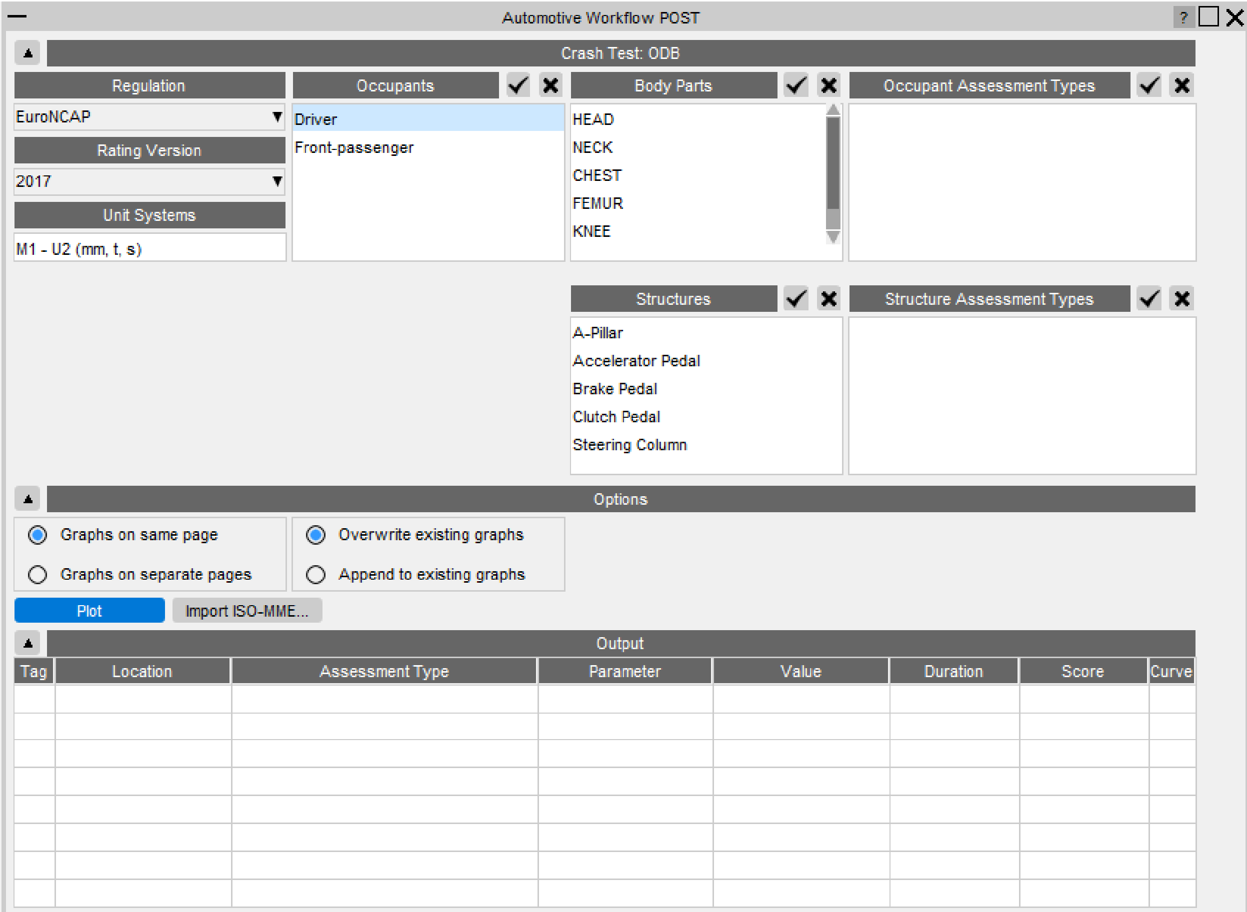




19.2.3. Automotive Assessments D3PLOT

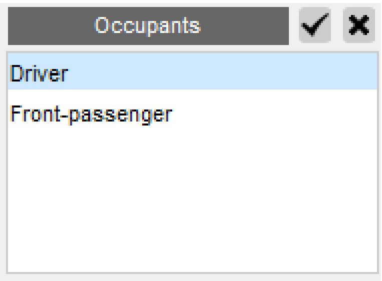
When the tool is launched in T/HIS you are presented with this window. This is where you select what assessments you want to carry out.

The dropdown menus on the left hand side show the regulation being used to carry out the assessments and the version. The model unit system is also shown.

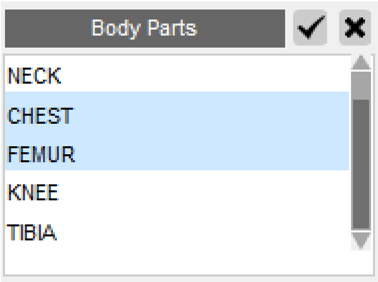


To select what assessments to carry out, you first need to select which occupant(s) you want to assess.

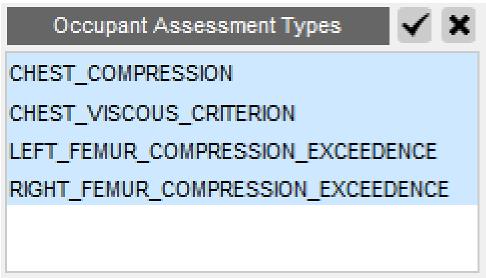
To select a single occupant left click on the one you want to assess. Use shift and left-click or ctrl and left-click to select multiple occupants. If you want to select all the occupants you can press the tick button and to deselect them all press the cross.



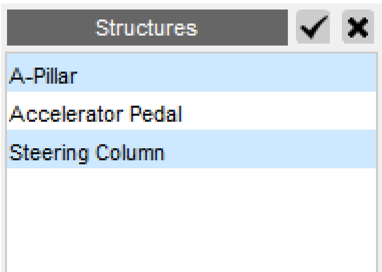
You can then select the body part(s) you want to assess.



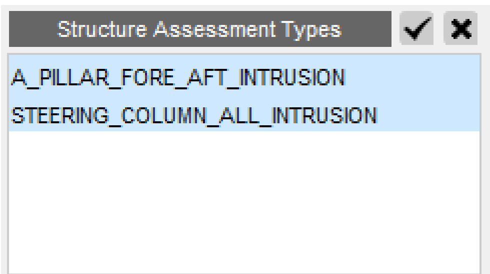
This will populate the Occupant Assessment Types list with the assessments that can be carried out for the selected body parts and occupants. By default they will all be selected, but you can chose to select only a subset of the list if you don't want to do them all.



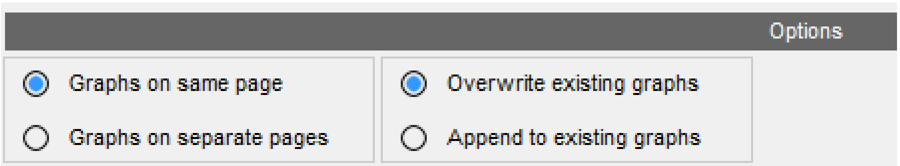
You can also select which structure(s) which you want to assess



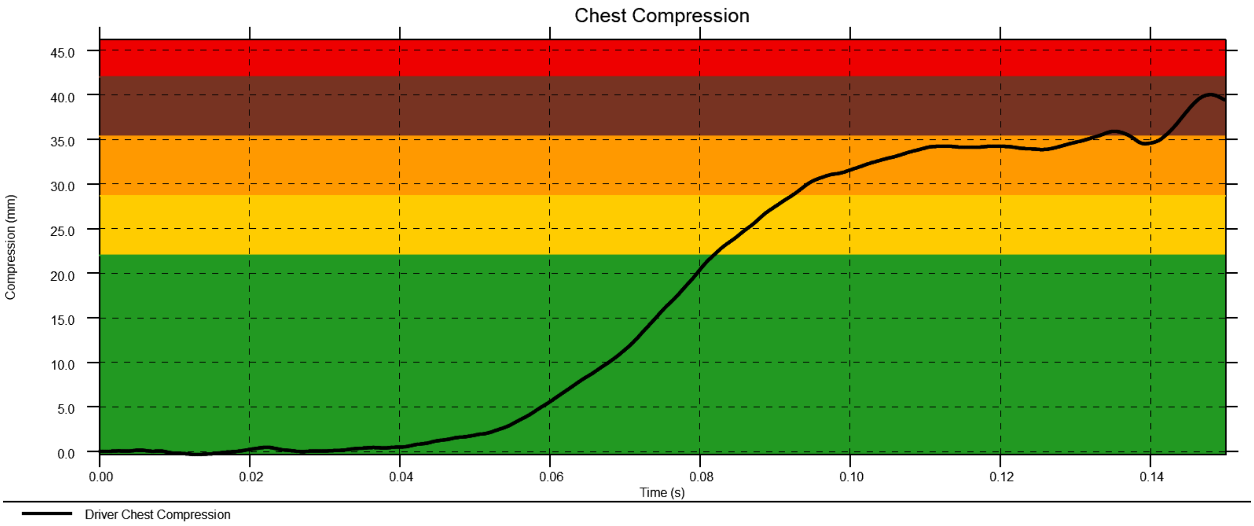
This will populate the Structure Assessment Types list with the assessments that can be carried out for the selected structures.



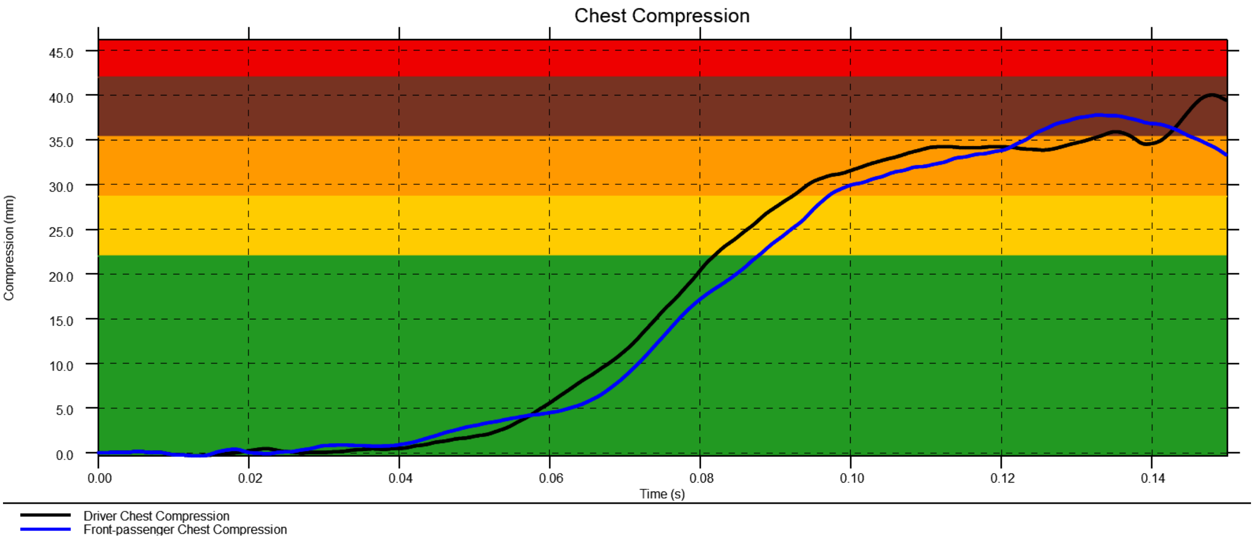
You can then chose how the graphs for each assessment should be displayed. By default they will all be put on one page and overwrite any existing graphs, but you can also chose to put each one on a separate page and append them to existing graphs.



Once you are happy with your choices, press the Plot button to carry out the assessments. T/HIS will extract the data required for each assessment, process it according to the rules set out in the regulation and plot the results on a graph with datums showing allowable limits (where they are defined by the regulation), e.g. the CHEST_COMPRESSION assessment for the driver:



If you have selected multiple occupants the curves for each occupant will be plotted on the same graph if the datum values are the same. If the datum values are different they will be plotted on separate graphs.



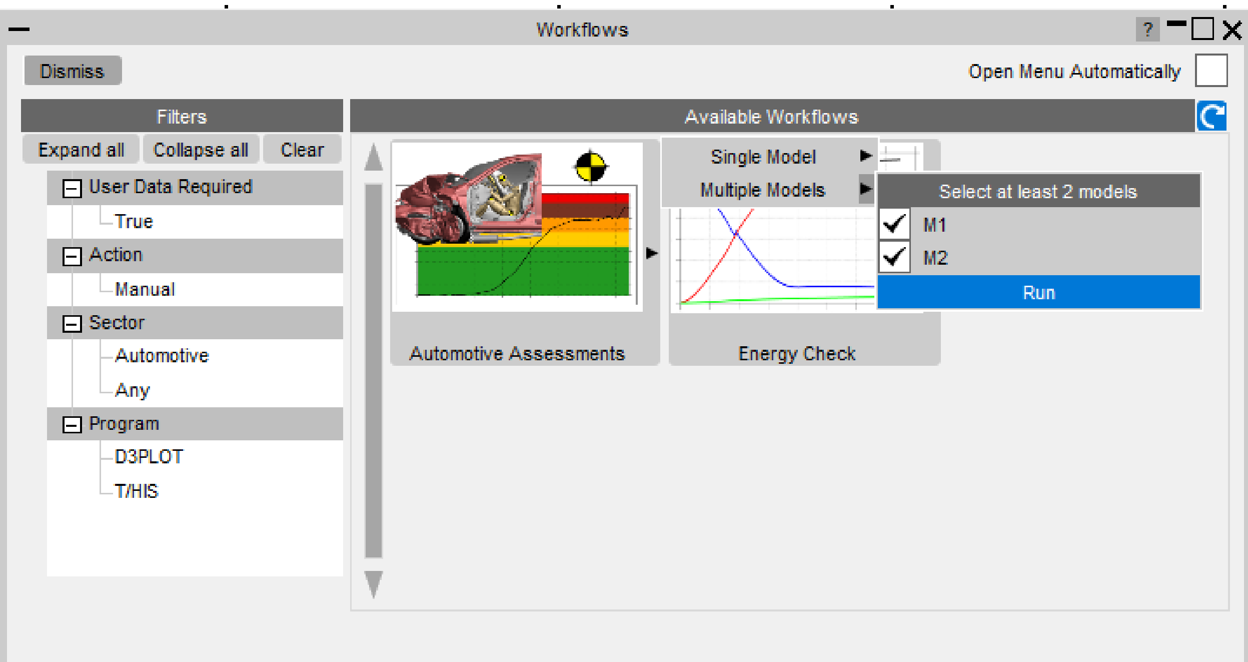
The output box at the bottom of the window lists the values and scores from the assessments carried out. Clicking on the '->' for each assessment will highlight the curve used for the assessment (and select the page if it's not on the current page) to make it easy to locate:

Output							
Tag	Location	Assessment Type	Parameter	Value	Duration	Score	Curve
M1	Driver	CHEST_COMPRESSION	Max	40.0235 mm		0.3953	->
M1	Front passenger	CHEST_COMPRESSION	Max	37.7902 mm		0.8420	->

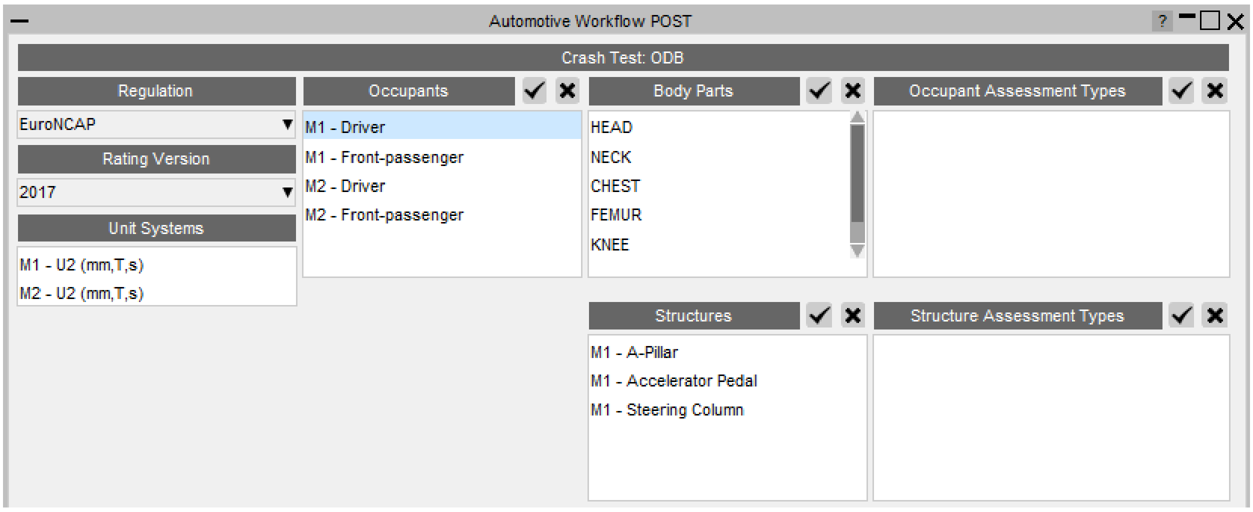
Multiple Models

It is also possible to plot results from multiple models on the same graphs. This is useful when you want to compare results between different runs.

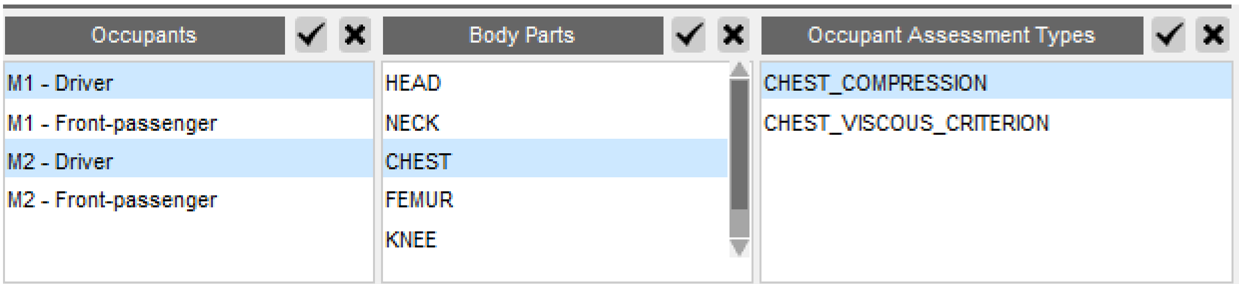
First you'll need to load the results from the models you want to compare into T/HIS and then on the workflow menu, select Multiple Models, pick the models you want to compare and press Run. Note that the models need to be of the same crash test type and regulation. If they're not the tool will refuse to run.

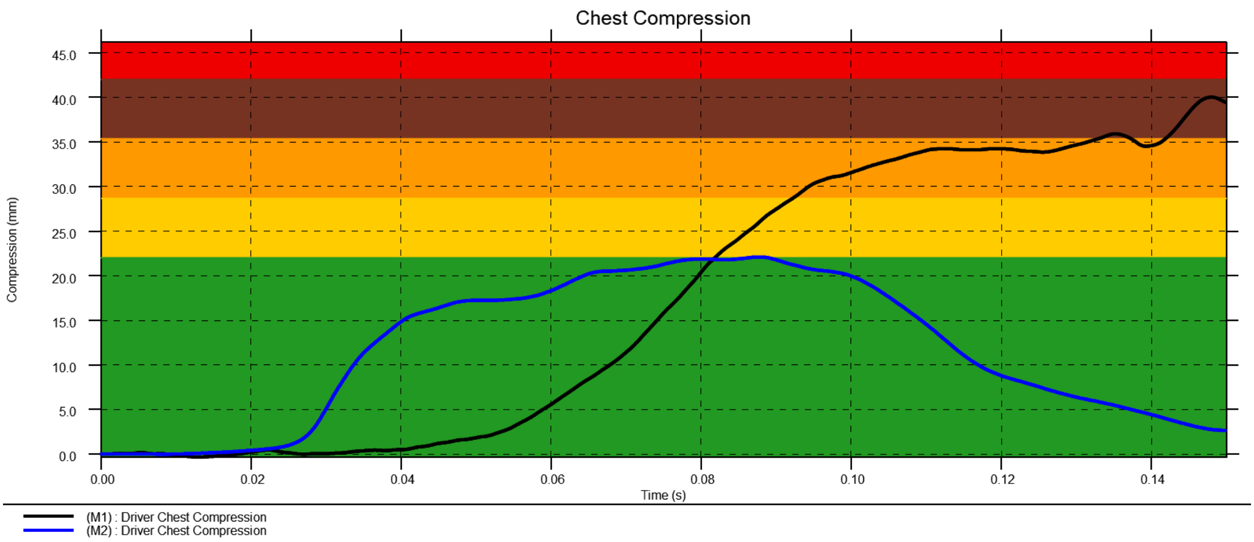


The window will then be populated with the occupants and structures from all the selected models, pre-pending them with the model number (M1, M2, etc)



If you wanted to compare the results for a CHEST_COMPRESSION assessment of the driver you would select the occupants in both models, select the chest body part and the CHEST_COMPRESSION assessment type.





19.2.4. Automotive Assessments REPORTER

Standard Templates

The following standard library templates have been updated so they work with workflow data saved from PRIMER:

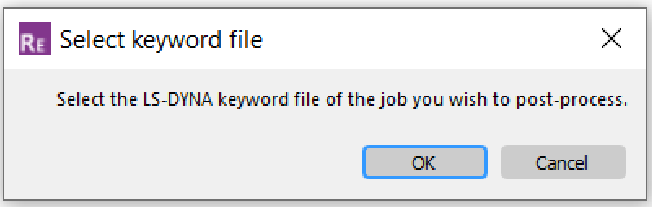
- EuroNCAP Front FFB Impact 2017
- EuroNCAP Front MPDB Impact 2020 Occupant Assessment
- EuroNCAP Front ODB Impact 2017
- EuroNCAP Side MDB Impact 2022
- EuroNCAP Side Pole Impact 2022
- CNCAP Front MPDB Impact 2022 Occupant Assessment

Running the templates interactively

As an example of how to use the templates we'll use the EuroNCAP Side Pole Impact 2022 template, but they all follow the same process:

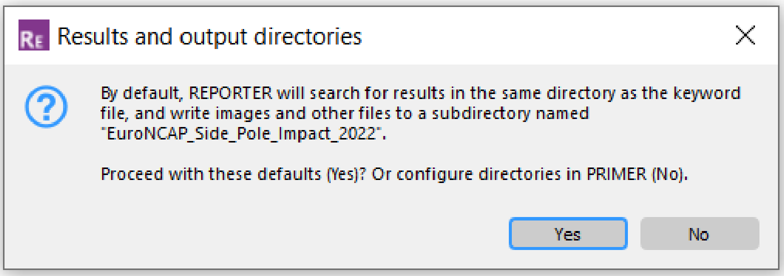
- In PRIMER specify and save the required data using the Automotive Assessments workflow
- In REPORTER use **File → Open Library Template** to select the relevant template. Templates that use workflow data are indicated by the  icon:

- On opening the template you will be prompted to select the keyword file of the job you want to post-process

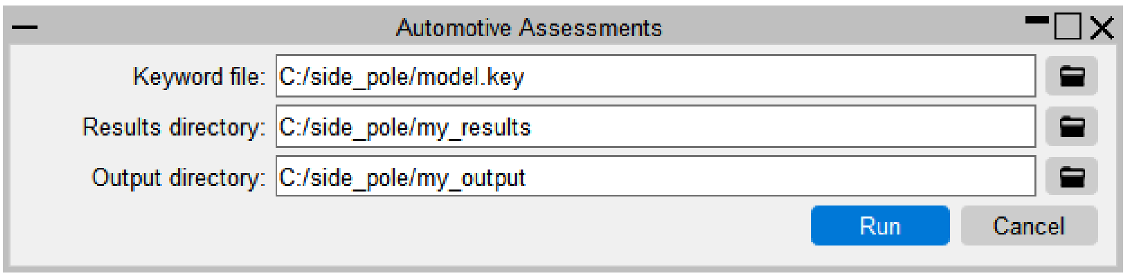


- After pressing **OK** a file selector is mapped for you to select the keyword file.

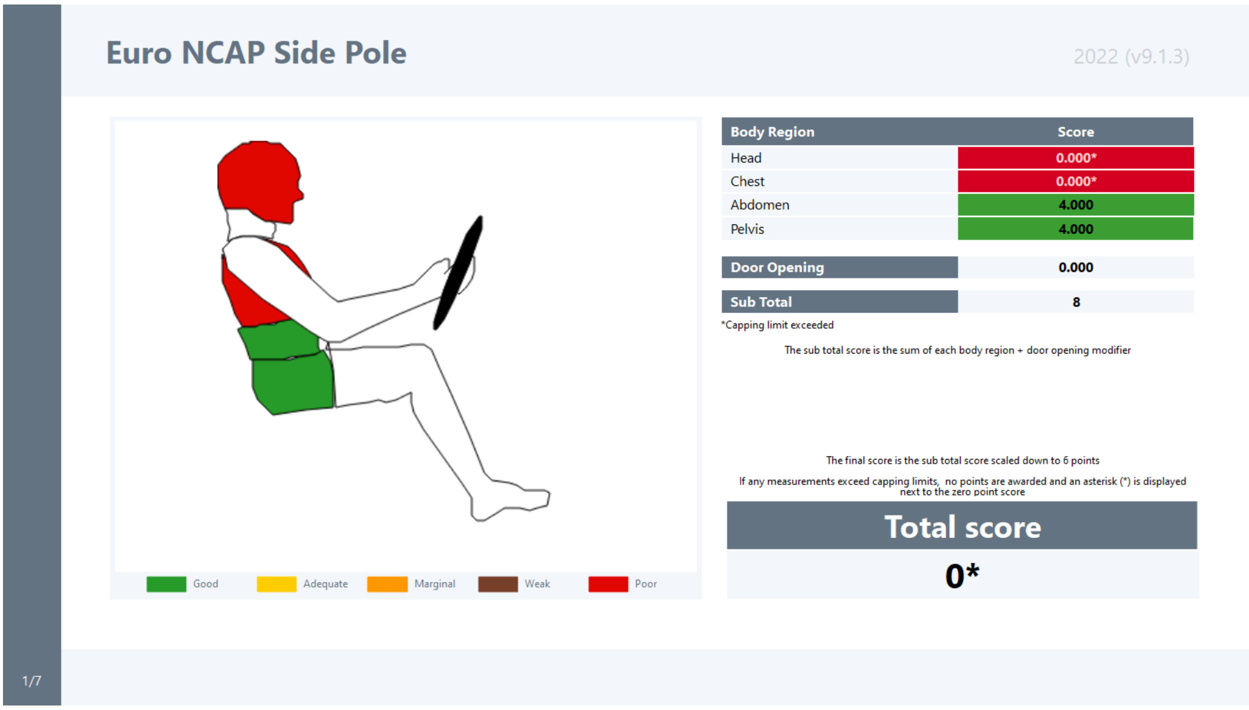
- After selecting the keyword file a prompt will ask if you want to proceed with some default directories to search for model results and for writing images and other files. The defaults assume:
    1. The results are in the same directory as the keyword file
    2. REPORTER will use a subdirectory in the keyword file directory named "EuroNCAP_Side_Pole_Impact_2022" to write images and other files to (the name will differ depending on the template). If the directory doesn't exist REPORTER will create it.



- If you are happy with the defaults press **Yes** and you can skip the next steps. T/HIS will start to post-process the results according to the protocol, generating the required graphs.
- If you want to change the directories press **No**. This will open PRIMER with a window where you can select the directories (and the model keyword file if you want to change this):



- Once selected, press Run. This will close PRIMER and start T/HIS to post-process the results according to the protocol, generating the required graphs
- Once finished, T/HIS will close and the template will be generated:



Running the templates in batch

The templates can also be run in batch mode, specifying the required information through command line arguments.

If your results are in the same directory as the keyword file then you only need to specify the keyword file on the command line:

<*reporter_exe*> -batch -file=<*template_name*> -varKEYWORD_FILE=<*keyword_file*> -exit

[Add the -pdf, -html, -pptx command line arguments to write the report out in the format you want]

Where:

reporter_exe	The full path and filename to the REPORTER executable
template_name	The full path and filename of the template you want to use. The workflow templates can be found in ***$OA_INSTALL/workflows/templates/automotive_assessments***
keyword_file	The full path and filename of the keyword file

If the results are in a different folder to the keyword file, you will need to add an extra argument to specify it:

```
<reporter_exe> -batch -file=<template_name> -
varKEYWORD_FILE=<keyword_file> -varRESULTS_DIR=<results_dir> -exit
```

Where:

results_dir	The full path to the results directory

Similarly if you want to output the images and other files generated by REPORTER to a different folder than the default, you will need to add an extra argument:

```
<reporter_exe> -batch -file=<template_name> -
varKEYWORD_FILE=<keyword_file> -varOUTPUT_DIR=<output_dir> -exit
```

Where:

output_dir	The full path to the output directory

Reasons for migrating the templates to the workflow framework

Migrating the standard templates to use data saved from the Automotive Assessment workflow has the following benefits:

- Setting up the data and generating the templates is simpler with fewer steps required
- The same data can be used in the Automotive Assessment workflow in T/HIS to interactively plot and interrogate results
- It makes it easier to add templates for new protocol versions and protocols not currently supported

19.3. SimVT

What is SimVT?

SimVT is a powerful tool for correlating curves from different data sources: LS-DYNA models, ISO-MME data and CSV data. SimVT removes the need to run the T/HIS CORA tool manually, repeatedly. Pairs of curves are matched automatically using tags. Many correlations can be performed and presented at once.

Why is it called "SimVT"?

The name "SimVT" comes from the fact that its primary purpose is to correlate **sim**ulation **v**ersus **t**est data. Additionally, the main motivation for the tool was to provide support for vehicle manufacturers who need to correlate their simulation crash results against their physical test results as part of the Euro NCAP Far Side **V**irtual **T**esting protocol.

Consequently, SimVT supports the Euro NCAP implementation of the ISO/TS 18571 rating method used by Euro NCAP's VTC Server.

Over time, SimVT will be enhanced to add support for new Virtual Testing protocols that are released. If there is a feature or protocol that you would like to see added to SimVT, please contact us.

19.3.1. Quick-start Guides

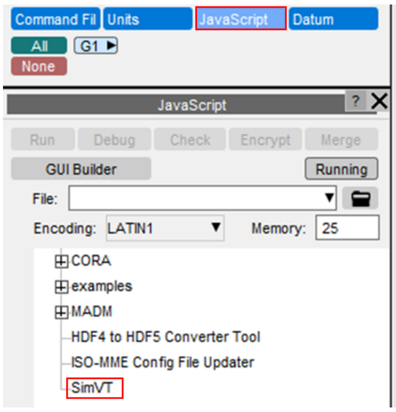
Quick-start Guides

SimVT has been designed to be versatile to support different use cases. To help you get started we have provided a number of quick-start guides which should cover the most common use cases:

1. I have ISO-MME test data from the lab and I want to compare my LS-DYNA simulation to see how well I perform for the Euro NCAP Virtual Far Side protocol: Using SimVT for Virtual Testing Protocols
2. I want to conduct a sensitivity study to see how the new simulations I have run compare against my baseline model: Using SimVT for Sensitivity Studies
3. I have processed simulation results curves myself and I want to compare them against physical test data: Using SimVT Without LS-DYNA Results

Using SimVT Without LS-DYNA Results

The SimVT tool can can be run directly from the **SimVT** JavaScript. This is useful if you want to use the tool without loading LS-DYNA results into T/HIS.



Steps in T/HIS

In this example we will correlate CSV data vs. ISO-MME data to demonstrate using both types of data.

1. Open T/HIS
2. Select **Tools** → **JavaScript** → **SimVT**
3. On the Correlation Setup window click **Import ISO-MME or CSV...** and select an ISO-MME index file (the extension should be ".mmi", ".chn" or ".iso").
4. On the Correlation Setup window click **Import ISO-MME or CSV...** and select a CSV data file.
5. On the Correlation Setup window, select **T1** as the "Reference test" and select **T2** as the simulation.
6. Select the method you want to use for correlating.
7. Add/remove any channel matching rules that you require to match T1 channels to T2.
8. Select the channels you want to correlate in the channel list (or select all of them by clicking on the **All**).
9. Click **Correlate**. The setup window will disappear, and it will be replaced by three windows:
    a. Progress window
    b. Correlation Table
    c. Plotting Controls
10. Once the progress window has disappeared you can inspect the results on the Correlation Table and plot the correlation graphs by using the buttons on the table.
11. You can also perform corrective operations to the input curves.

Using SimVT for Sensitivity Studies

Three models are used for this example (but you can use as many as you like):

1. M1 – the baseline model
2. M2 – a variant of the baseline
3. M3 – another variant of the baseline

The purpose of this sensitivity study is to determine if the changes in M2 and M3 compared to M1 have significantly affected any of the results curves (output channels). Before following the steps in T/HIS, make sure that each model has Automotive Assessment Workflow Data (AAWD) defined. You can optionally choose to share the same AAWD for all the models in the study by placing the AAWD JSON file in a directory that is an ancestor to all the model results.

Steps in T/HIS

1. **Open T/HIS**
2. **Load the LS-DYNA results** for the baseline model
3. **Load the LS-DYNA results** for one or more models that you want to compare against the baseline.
4. Click **Tools** → **Workflows**. Note, if you have “Open Menu Automatically” checked you can skip this step.
5. Select **SimVT** and select all the models **M1**, **M1**, **M3** etc.,
6. Click on **Run** (this will open the "Correlation Setup" window)
7. On the Correlation Setup window, select the baseline model (**M1**) as the "Reference test".
8. Select all the other models as the simulation(s) (hold down shift when clicking to select multiple i.e. Shift + **M2** + **M3**).
9. Select the method you want to use for correlating.
10. Add/remove any channel matching rules that you require to match M1 channels to M2 and M3 etc. (note if you are sharing the same AAWD then the channels will all match without the need for matching rules)
11. Select the channels you want to correlate in the channel list (or select all of them by clicking on the **All**).
12. Click **Correlate**. The setup window will disappear, and it will be replaced by three windows:
    a. Progress window
    b. Correlation Table
    c. Plotting Controls
13. Once the progress window has disappeared you can inspect the results on the Correlation Table and plot the correlation graphs by using the buttons on the table.

14. You can also perform corrective operations to the input curves.

Using SimVT for Virtual Testing Protocols

SimVT has been designed to perform correlation analysis for Virtual Testing protocols. Initially, only support for the Euro NCAP Virtual Far Side Simulation & Assessment Protocol v1.0 has been added, but the intention is to add support for other protocols as well.

Euro NCAP Virtual Far Side Protocol

For this process you will first need Automotive Assessment Workflow data (AAWD) for the LS-DYNA model you wish to process. You can can follow these instructions to create the AAWD in PRIMER before proceeding to the steps below in T/HIS. Note that you can create the AAWD before running a simulation to help ensure that your model will output all of the required results data for processing according to the Euro NCAP Virtual Far Side protocol.

SimVT can be opened from the Workflows menu in T/HIS provided that you have at least one LS-DYNA model loaded in to the T/HIS session.

Steps in T/HIS

1. **Open T/HIS** and **read in the LS-DYNA** results (e.g. binout000 or .thf files) you wish to use with the SimVT Workflow.
2. Select **Tools → Workflows**. Note, if you have “Open Menu Automatically” checked you can skip this step.
3. Filter for "Virtual Testing" and select **SimVT**
4. In the Correlation Setup window, click **Import ISO-MME or CSV...** to select an ISO-MME index file (the extension should be “.mmi”, “.chn” or “.iso”).
5. Select **T1** as the "Reference test" and select **M1** as the simulation (both should be set to this automatically).
6. In the "Protocol" section of the setup window select **Euro NCAP VTC Simulation and Assessment v1.0**. This will automatically configure the following settings that are specific to the protocol:
    a. The correlation method will be set to "ISO/TS 18571 Euro NCAP v1.0",
    b. The curves will be automatically regularised 10kHz before correlating,
    c. The channels list will change to show "protocol channels" rather than "selectable channels" which makes it easier to identify which channels may be expected, by the protocol, but are missing from either the simulation or test data,
    d. Protocol channels that specify the filter class will be automatically derived from the corresponding unfiltered channel if they are not already defined,

   e. The "max =" button will become active which can be clicked to set the maximum evaluation window time to the time of the maximum head excursion + 20%.
7. Click on **max =** to set the maximum evaluation window time to the time of the maximum head excursion + 20% (provided that the head excursion can be computed from the selected simulation channels and the analysis curves have a duration that exceeds the cut off).
8. Inspect the head excursion plot that appears and read the message in the information window to check that the simulation passes (i.e. ensure that the simulation has run for long enough). If it fails, you will need to re-run the analysis for a longer time before repeating these steps.
9. To correlate between simulation and test data, the data must containing matching pairs of ISO-MME channel codes. Add/remove any channel matching rules that you require to match T1 channels to M1. Note that test data typically uses filter class "P" whereas simulation data uses filter class "0" so the filter class equivalence ISO rule **"P|0"** is added by default to facilitate matching test data to simulation data.
10. Select the channels you wish to correlate in the channel list (or select all of them by clicking **All**). Any channels that are greyed out are missing from the provided simulation and/or test data.
11. Click **Correlate**. The setup window will disappear, and it will be replaced by three windows:
    a. Progress window
    b. Correlation Table
    c. Plotting Controls
12. Once the progress window has disappeared you can inspect the results in the Correlation Table and plot the correlation graphs by using the buttons in the table.
13. You can also perform corrective operations to the input curves.
14. Finally, you can export a summary of the results as a CSV file, and save your SimVT settings for a future session.

19.3.2. Data Sources

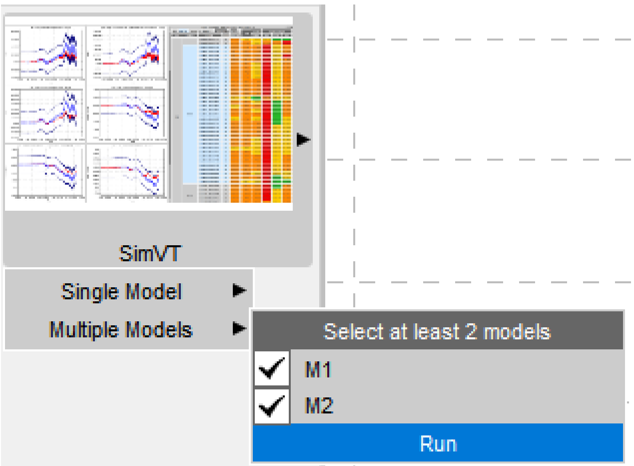
Data Sources

SimVT offers flexibility by supporting channel data from a range of different data sources:

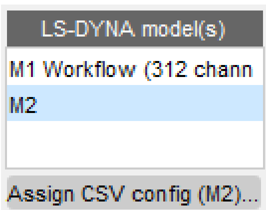
- LS-DYNA with Automotive Assessment Workflow Data
- LS-DYNA with CSV Config File
- CSV Data
- ISO-MME Data

LS-DYNA with Automotive Assessment Workflow Data

When you run SimVT from the workflow window you have the option of selecting which LS-DYNA models you would like to load into the tool. For each selected model ("M1" and "M2" in the image below), SimVT will attempt to find the associated Automotive Assessments workflow data (AAWD) because it requires this data to extract labelled channels from the model. The AAWD also contains additional meta-data such as model units, crash test protocol, and the vehicle drive side, which is used by SimVT to convert curves to SI units and to determine the protocol channels.



If SimVT successfully finds the AAWD for a model, it will appear in the model list along with the number of channels that are available for correlating (e.g. "M1" below). If the AAWD cannot be found, only the model tag will appear in the model list and you will have the option to assign channel labels using the CSV Config file (e.g. "M2" below).



Note that the model tags for all the LS-DYNA models that are present in the T/HIS session will be visible in the model list (even if you did not select them from the workflow window before launching SimVT).

Creating Automotive Assessments Workflow Data (AAWD)

Follow the steps here to create Automotive Assessments workflow data (AAWD) for an LS-DYNA model. Note that it is possible to reuse the same workflow data for multiple models provided that the entity-id mappings and additional meta data (e.g. unit system, crash test protocol, occupants and vehicle drive side) are the same for each model.

LS-DYNA with CSV Config File

LS-DYNA models without associated Automotive Assessment Workflow Data (AAWD) can still be used with SimVT, but a CSV Config File is required to assign channel labels to the entities in the model. **Note that it is recommended to use AAWD but the CSV Config File Format is provided for scenarios which AAWD does not support e.g. custom channel names.**

CSV Config File Format

- An example CSV configuration file format showing a mixture of the supported inputs is shown below.
- The header row is required, but the order of the columns can vary.
- The configuration file supports ISO-MME format channel codes.
- If you are not working with ISO-MME data, you can provide any name you like in the channel column.

	A	B	C	D	E	F	G
1	Object	Location	Channel	Entity type	ID	ISO Comp	T/HIS Component
2	D3	HEAD	HEAD Acceleration X	node	10001	ACX	
3	D3	HEAD	HEAD Acceleration Y	node	10001		AY
4	D3	HEAD	HEAD Acceleration Z	node	10001		AZ
5			D3HEAD0000WSAVXP	node	10006		
6			D3HEAD0000WSAVYP	node	10006		
7			D3HEAD0000WSAVZP	node	10006		
8	D3	SACR	Left Sacrum Force X	beam basic	10005	FOX	
9	D3	SACR	Left Sacrum Force Y	beam basic	10005	FOY	
10	D3	SACR	Left Sacrum Force Z	beam basic	10005	FOZ	
11	D3		SACRLE00WSMOX	beam basic	10005	MOX	
12	D3		SACRLE00WSMOY	beam basic	10005	MOY	
13	D3		SACRLE00WSMOZ	beam basic	10005	MOZ	
14	D3		AIRBAG_KE	part	[1 2 3]		KE
15	D3		AIRBAG_IE	part	1		IE
16	D3		AIRBAG_HE	part	1		HG
17	D3		BPILLAR Acceleration Magnitude	node	11001		AM
18	D3		CONTACT_DUMMY_AIRBAG	contact	1041		FM
19	D3		CONTACT_DUMMY_CC	contact	1041		FM
20	D3		CONTACT_DUMMY_SEAT	contact	1041		FM
21	D3		CONTACT_DUMMY_SEATBELT	contact	1041		FM
22	D3		LAP_BELT	section	1		FM

Assigning CSV Config File

When "Assign config" is clicked on the Correlation Setup window, the CSV Configuration window will appear. Follow these steps to assign a CSV config file.

CSV Data

CSV channel data can be imported into SimVT. Once in SimVT it is handled in the same way as other data sources, allowing you to correlate curves with matching channel names.

CSV Data file format

The CSV file format is based on the CSV format that can be exported from T/HIS using the CSV X,Y,Y,Y,Y,Y write option.

- The **first header row is required** and contains the names of the channels that will be imported
- The first column contains the time values (X points) and the Y value at the time is given in the adjacent columns. Note that if the Y value is empty or not a number the row will be skipped
- The word **"Time" must appear in one of the cells in the first column**. It marks the end of the header rows (i.e. the row below should contain time-value data)
- The "Time" row can optionally be used to defined the units of the non-ISO-MME channels.
- The "Object" and "Location" header rows are optional and are used to group non-ISO-MME channels in the SimVT Correlation Table.

ISO-MME CSV Channel Data Example

The table below shows some example CSV channel data with ISO-MME channels defined. Note that:

- The first cell (A1) is empty.
- The first row contains valid 16 character ISO-MME channel codes for each column.
- The first column contains the word "Time" in the second row and the data values begin on the row below.
- The second row has no unit data defined as the units can be determined from the ISO-MME channel codes' physical dimension (e.g. AC is acceleration so will have units of m/s^2)

	11HEAD0000WSDCX0	11HEAD0000WSDCY0	11HEAD0000WSDCZ0	11HEAD0000WSAVX0	11HEAD0000WSAVY0	11HEAD0000WSAVZ0	11HEAD0000WSACX0	11HEAD0000WSACY0	11HEAD0000WSACZ0	11HEAD0000WSVEX0
Time										
0	0	0	0	-3.22E-27	7.77E-25	8.65E-27	0	0	0	0
0.000100001	0	0	0	5.00E-10	7.34E-11	-5.11E-10	-2.87E-08	-7.70E-08	1.36E-08	1.09E-12
0.000200002	0	0	0	3.03E-10	5.90E-11	-3.36E-10	-5.22E-09	-2.54E-08	1.34E-07	5.87E-14
0.000300003	0	0	0	1.46E-10	2.56E-11	-2.05E-10	3.38E-09	-1.59E-08	8.26E-08	1.90E-14
0.000400004	0	0	0	8.40E-11	2.34E-11	-8.60E-11	-5.79E-09	-8.33E-09	9.71E-08	9.91E-15
0.000500005	0	0	0	9.56E-10	6.46E-09	5.66E-10	0.000367982	-1.75E-05	-0.000660053	2.95E-09

Non-ISO-MME CSV Channel Data Example

The table below shows some example CSV channel data with non-ISO-MME channels defined. Note that:

- The first cell (A1) contains the word "Channel" - this is optional when the channel names are on the first row.
- The channel names are not valid ISO-MME channel codes so they will not benefit from the ISO-MME channel matching for each column.
- The first column contains the word "Time" (i.e. "Time | Units") in the fourth row and the data values begin on the row below.
- The "Time" row also has unit data defined as the units cannot be determined from non-ISO-MME channel codes.
- The "Object" and "Location" rows are defined, so the channels will be grouped together on the SimVT Correlation table.

Channel	Head X Disp.	Head Y Disp.	Head Z Disp.	Head X Ang. Vel.	Head Y Ang. Vel.	Head Z Ang. Vel.	Head X Accel.	Head Y Accel.	Head Z Accel.	Head X Vel.
Object	Driver	Driver	Driver	Driver	Driver	Driver	Driver	Driver	Driver	Driver
Location	Head	Head	Head	Head	Head	Head	Head	Head	Head	Head
Time|Units	m	m	m	rad/s	rad/s	rad/s	m/s^2	m/s^2	m/s^2	m/s
0	0	0	0	-3.22E-27	7.77E-25	8.65E-27	0	0	0	0
0.000100001	0	0	0	5.00E-10	7.34E-11	-5.11E-10	-2.87E-08	-7.70E-08	1.36E-08	1.09E-12
0.000200002	0	0	0	3.03E-10	5.90E-11	-3.36E-10	-5.22E-09	-2.54E-08	1.34E-07	5.87E-14
0.000300003	0	0	0	1.46E-10	2.56E-11	-2.05E-10	3.38E-09	-1.59E-08	8.26E-08	1.90E-14
0.000400004	0	0	0	8.40E-11	2.34E-11	-8.60E-11	-5.79E-09	-8.33E-09	9.71E-08	9.91E-15
0.000500005	0	0	0	9.56E-10	6.46E-09	5.66E-10	0.000367982	-1.75E-05	-0.000660053	2.95E-09

ISO-MME Data

ISO-MME data is supported (versions 1.6 and 2.0 are supported). All the channels (curves) defined in the index file will automatically be extracted and labelled (tagged) with their ISO channel codes. ISO-MME data will typically be test data obtained from a physical crash test, but the tool will work with ISO-MME data that has been generated from simulation results, for example, by using the LS-DYNA to ISO-MME workflow tool.

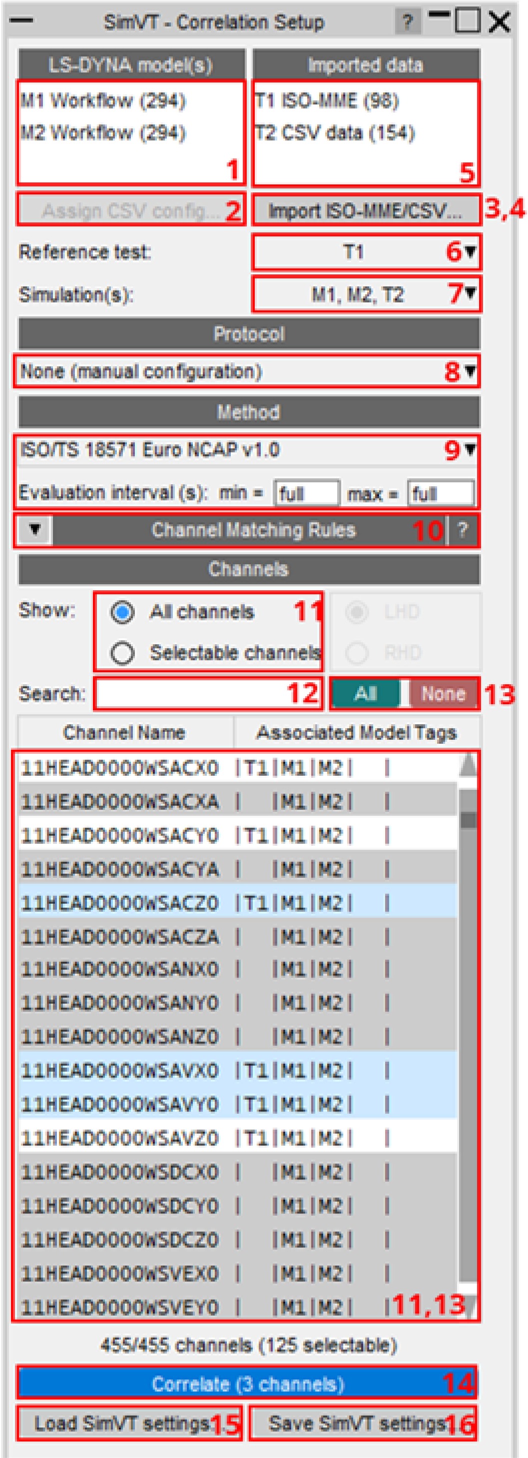
Instructions for importing ISO-MME data in SimVT can be found in this LS-DYNA with ISO-MME Example.

Click on the Import ISO-MME/CSV... button on the Correlation Setup window

IMPORTANT: ISO-MME data is assumed to be in SI units. Some test houses provide angular results in degrees rather than in radians. Please check your ISO-MME test data before importing it to ensure that angles are in radians. This applies to rotations, angular velocities and angular accelerations.

19.3.3. SimVT Windows

Correlation Setup Window



The Correlation Setup Window allows you to prepare your desired correlation(s). The window enables you to:

1. View a list of the available LS-DYNA model(s)
2. Assign a CSV configuration to LS-DYNA models that are missing Automotive Assessments Workflow Data (AAWD)
3. Import ISO-MME data
4. Import CSV data
5. View a list of the imported data
6. Select which model or data source will be used as the reference (this is typically physical test data from a laboratory)
7. Select which model(s) or data source(s) will be compared against the reference test (this is typically LS-DYNA simulation data)
8. Select the virtual testing protocol

9. Select the correlation method that will be used and the maximum and minimum evaluation intervals
10. Apply channel matching rules
11. Inspect the available and selectable channels
12. Search for channels using regular expressions
13. Select the channels to be correlated
14. Perform “Simulation(s)” versus “Reference test” correlations on the selected channels
15. Save SimVT settings files
16. Load SimVT settings files

1. Model List

The models that are present in T/HIS when SimVT is loaded will be listed on the correlation setup window. If T/HIS can find Automotive Assessments Workflow Data (AAWD) for a model, it will appear with "Workflow" and the number of channels that are defined for the model next to the model tag (e.g. "M1 (336 channels)"). Alternatively, if T/HIS cannot find AAWD for a model, only the model tag (e.g. "M2") will be shown.

2. Assign a CSV configuration

Creating AAWD for LS-DYNA models is the recommended method for using LS-DYNA model results with SimVT. However, it is possible to use LS-DYNA results with SimVT without AAWD, but a CSV configuration file needs to be provided to define the channel mapping and the model units and vehicle drive side need to be provided in addition. The "Assign CSV config..." button will become active when a model which does not have

AAWD is selected from the model list. Clicking on the button will open a window where the CSV config can be loaded and checked before it is assigned to the selected model.

3. Import ISO-MME data

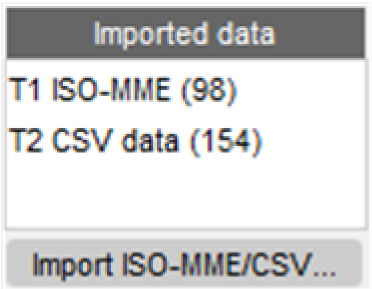
ISO-MME test data (v1.6 and v2.0) can be imported into SimVT for correlating. Clicking on the "Import ISO-MME or CSV..." button will open a file selector window. Navigate to the ".chn" or ".mmi" file in your ISO-MME data and open it to load the data into SimVT.

4. Import CSV data

CSV channel data can be imported into SimVT for correlating. Clicking on the "Import ISO-MME or CSV..." button will open a file selector window. Navigate to the ".csv" file with your channel data and open it to load the data into SimVT.

5. View a list of the imported data

Imported ISO-MME and CSV data will be displayed in the Imported Data list along with the number of channels defined by the data.

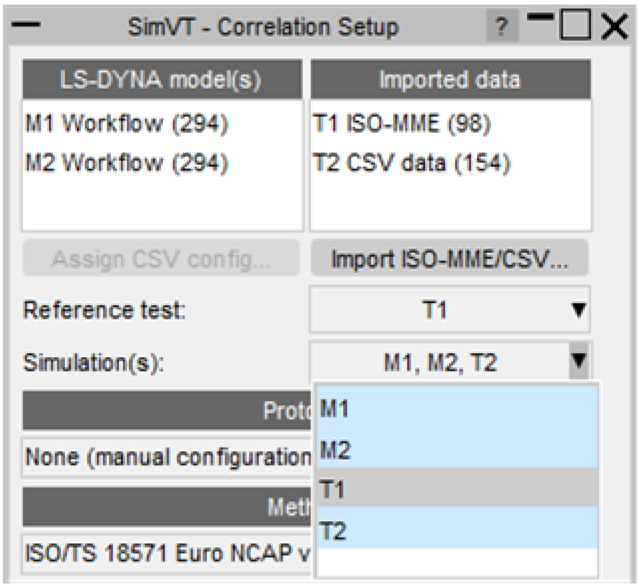


6. Selecting Reference (test) Source

The drop-down can be used to select the reference model or data source. This is typically physical test data from a laboratory, but an LS-DYNA model can also be selected (e.g. a baseline model when conducting a sensitivity study). Note that when ISO-MME or CSV data is imported, the reference test will automatically update to the tag of the newly imported data (e.g. T1). This is because imported data is expected to be test data so it is assumed to be the reference data.

7. Select the Simulation(s)

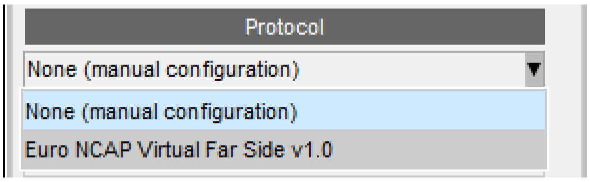
SimVT supports correlating multiple simulations against reference/test data. The simulations that are to be compared against the reference test data can be selected using the drop-down (hold down Shift to select multiple). Note that the reference test data cannot be selected.



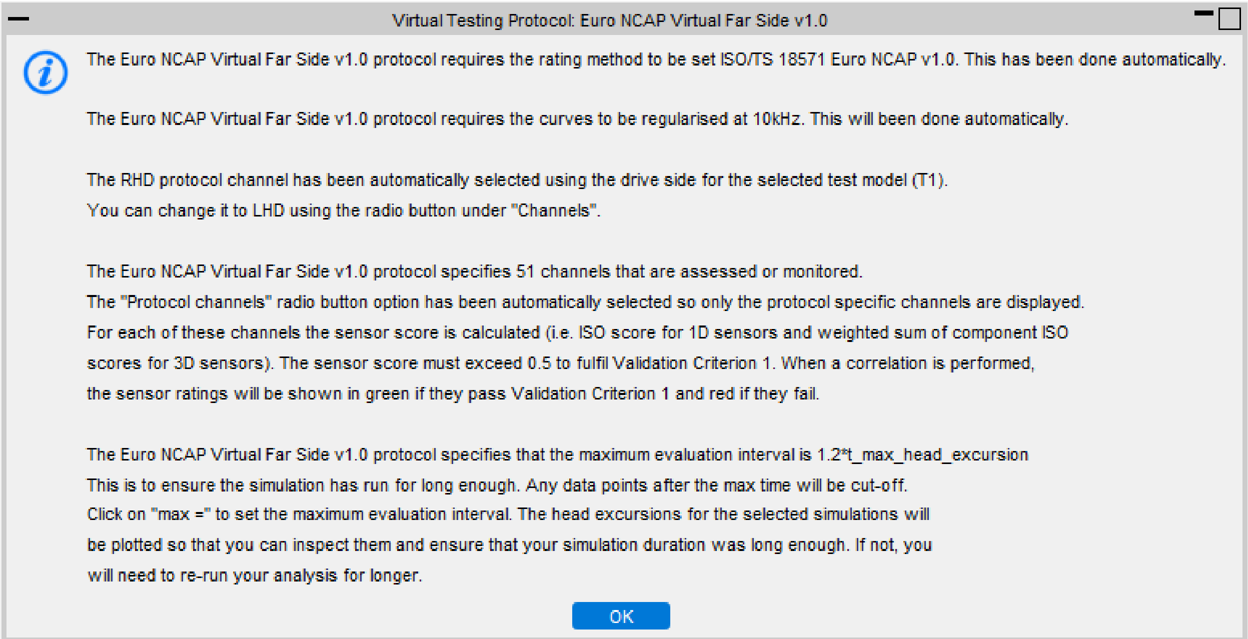
8. Select the Virtual Testing Protocol

The protocol selector allows you to select a protocol so that SimVT will be configured to meet the protocol requirements.

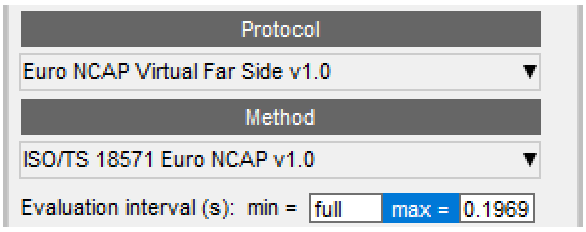
At present, the "Euro NCAP Virtual Far Side v1.0" protocol is supported.



When it is selected an information window will appear to explain the protocol specific configurations that have been applied.

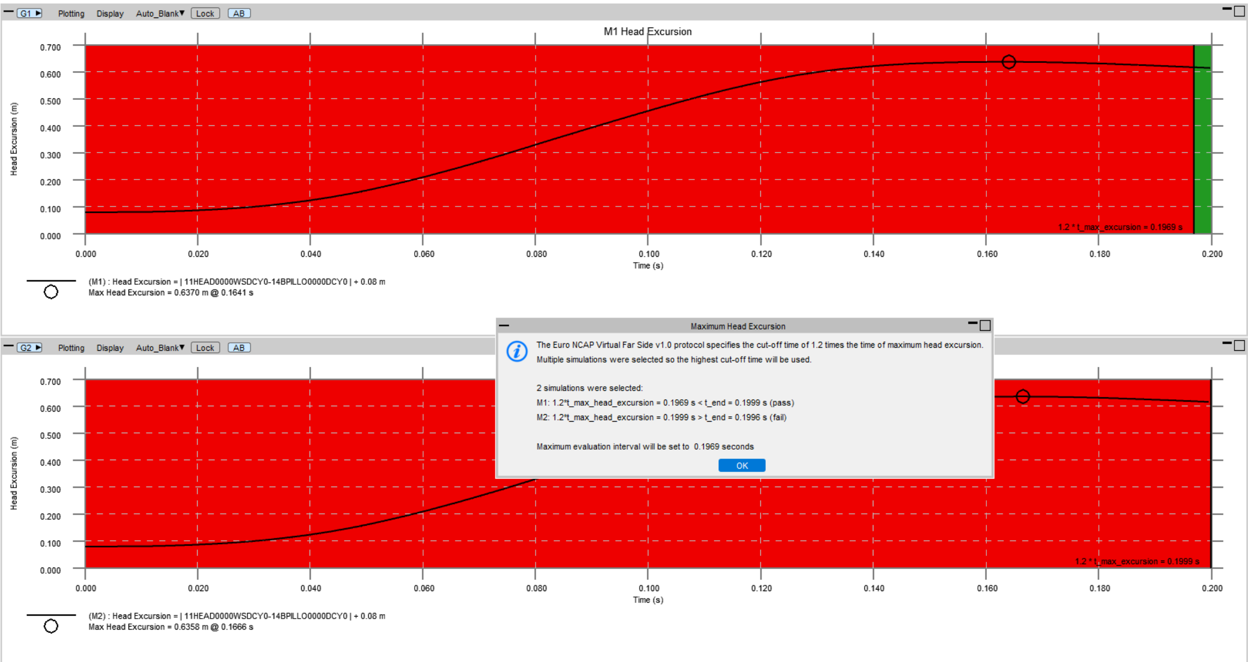


Additionally, the "max =" button will become active.



Pressing **max =** will attempt* to set the maximum evaluation interval based on the time of maximum head excursion + 20% (as defined by the "Euro NCAP Virtual Far Side v1.0" protocol). Additionally, the head excursions for the selected simulations will be plotted and a message will also appear to communicate which simulations passed or failed. Failed simulations should be re-run for longer so that the end time exceeds the cut-off time of 1.2 times the time of maximum head excursion.

*In order to compute the head excursion for a selected simulation, the simulation data source must have the head global Y displacement channel (1?HEAD0000WSDCY0) and the B-Pillar global Y displacement channel (1?BPILLO0000WSDCY0) defined. It is important that both channels should represent **global** Y displacement because using local Y will result in an incorrect head excursion.



Support for additional virtual testing protocols will be added in future releases.

9. Select the Correlation Method

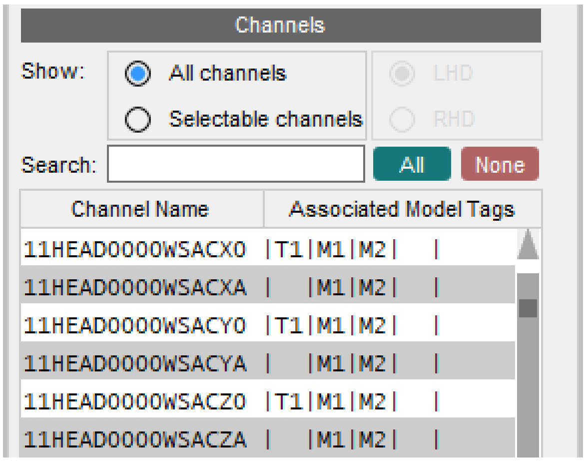
SimVT supports the same correlation methods that CORA uses. The default correlation method is ISO/TS 18572 Euro NCAP v1.0 which is consistent with the correlation method employed by the Euro NCAP Far Side VTC protocol. Other correlation methods can be selected from the drop-down or a CORA config file can be used for full control over the correlation options. This can be created in the CORA tool.

10. Channel Matching Rules

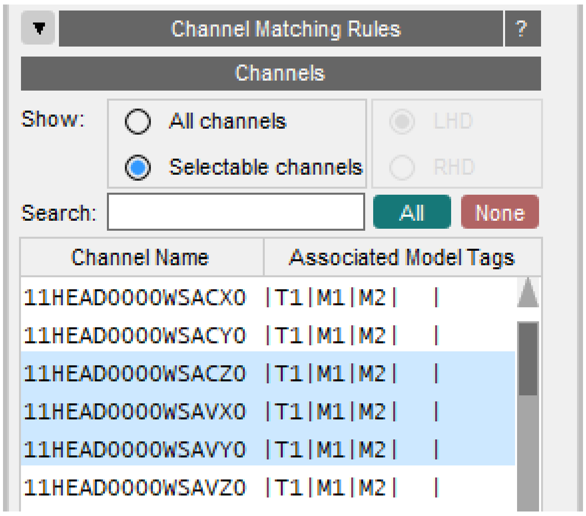
Clicking on the arrow next to "Channel Matching Rules" will expand the rule options. Channel matching rules can be used to help match channels from different data sources which may differ from the standard ISO-MME naming convention. This feature is explained in more detail here.

11. Inspect the available and selectable channels

The channel list shows the channels for the selected simulation(s) versus the reference test. The selected reference test model (T1) is shown in the first column of the associated model tags. Any selected simulations (M1, M2 and T2) are shown next to it separated by “|” provided that the channel is present in the simulation data (e.g."T2" is not shown for for any of the head acceleration channels shown below because it was not defined for the imported CSV data). Some channels may not be selectable because they are either not defined for (or only defined for) the reference model. They are excluded from the list when **Selectable channels** is selected, but shown as greyed out rows when **All channels** is selected to indicate that it is not possible to select them for correlating. For example, in the image below, the filtered head acceleration channels ("11HEAD0000WSACXA", "11HEAD0000WSACYA" and "11HEAD0000WSACZA") are shown as greyed out rows because the channels are not defined for the reference test (T1) so there would be nothing to compare the M1 and M2 simulation curves against.

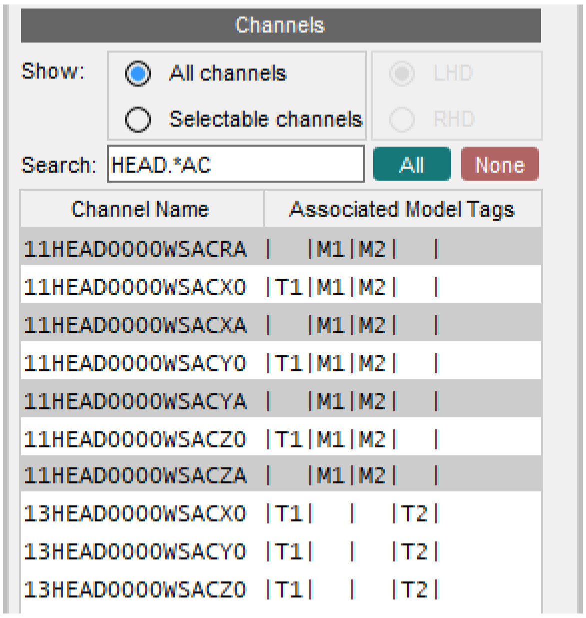


Clicking on the **Selectable channels** radio button will update the list to only show channels which are selectable.

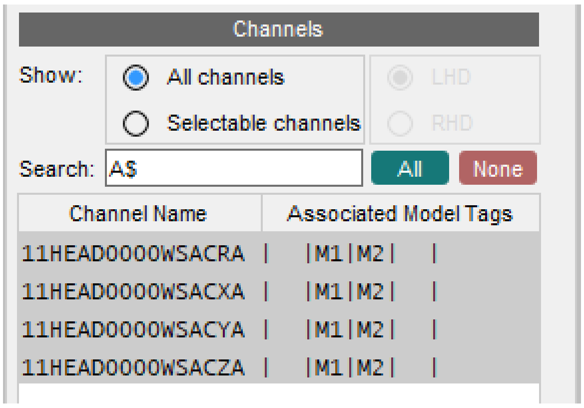


12. Search for channels using regular expressions

The channels shown in the the channel list can be filtered using the search box. The search box uses regular expression matching to determine which channels to show and it is case insensitive. For example searching for "HEAD.*AC" will filter the list to shown only the head acceleration curves.

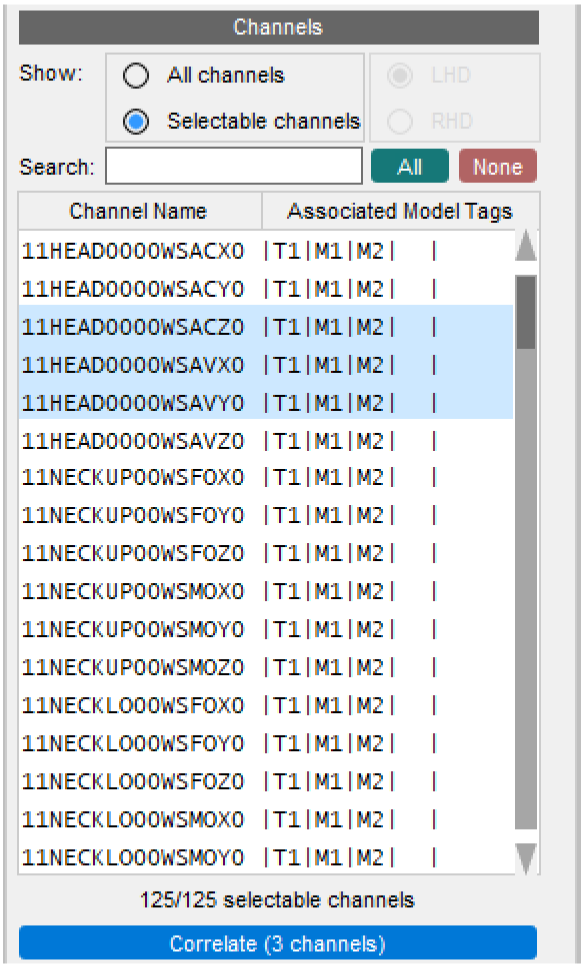


Or searching for "A$" will shown only the channels where the last character is "A" (i.e. ISO-MME channels that have been filtered with CFC1000).



13. Select the channels to be correlated

Channels to be correlated can be selected from the channel list using a combination of Shift+click and Ctrl+click. Clicking on "All" will select all of the channels shown in the list. Clicking on "None" will deselect all the channels. Selected channels will be highlighted in blue and the number of selected channels is indicated on the “Correlate” button.



14. Perform “Simulation(s)” versus “Reference test” correlations on the selected channels

The "Correlate" button will turn become active when 1 or more channels are selected from the channel list. Clicking on the button will commence the correlation.

15. Saving SimVT settings files

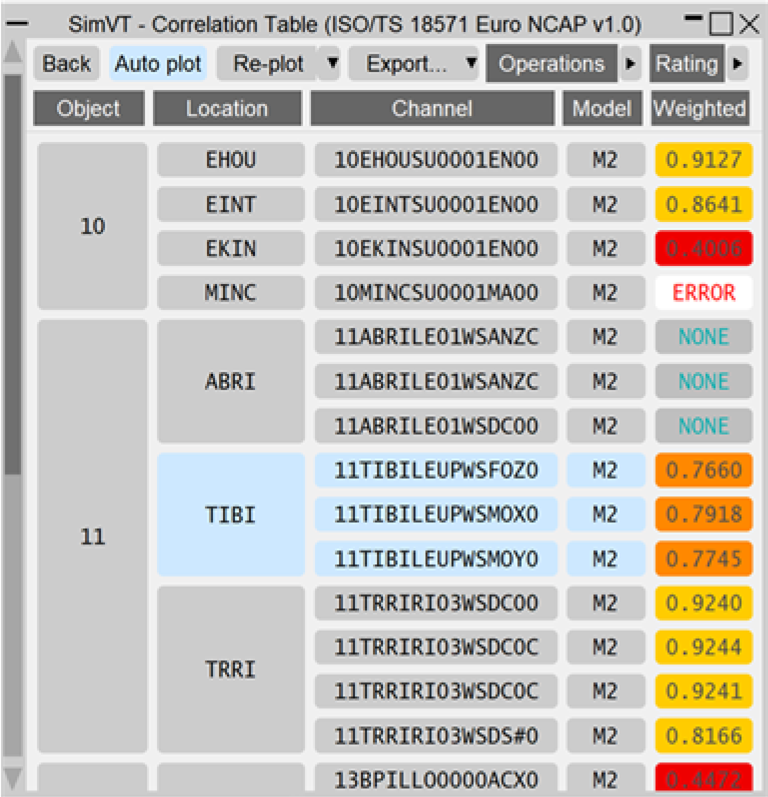
A SimVT settings file (.simvt) can be saved from the correlation setup window by clicking on the "Save SimVT settings..." button. The settings file saves all the information required to restore the session (e.g. data sources, correlation method, selected and plotted channels).

16. Loading SimVT settings files

A SimVT settings file (.simvt) can be loaded from the correlation setup window by clicking on the "Load SimVT settings..." button. The model mapping window will appear which can be used to (re)assign models before loading the settings. When the settings are loaded the correlation will be automatically performed and the correlation table window will appear.

Correlation Table Window

The Correlation Table is used to toggle plotting of the correlations that have been performed. It also shows the results of all the correlations or “ERROR” if there was an error when performing the correlation and “NONE” is some of the input data is missing. The Correlation Table and [Plotting Controls]() are the main way of interacting with the graphs and results.



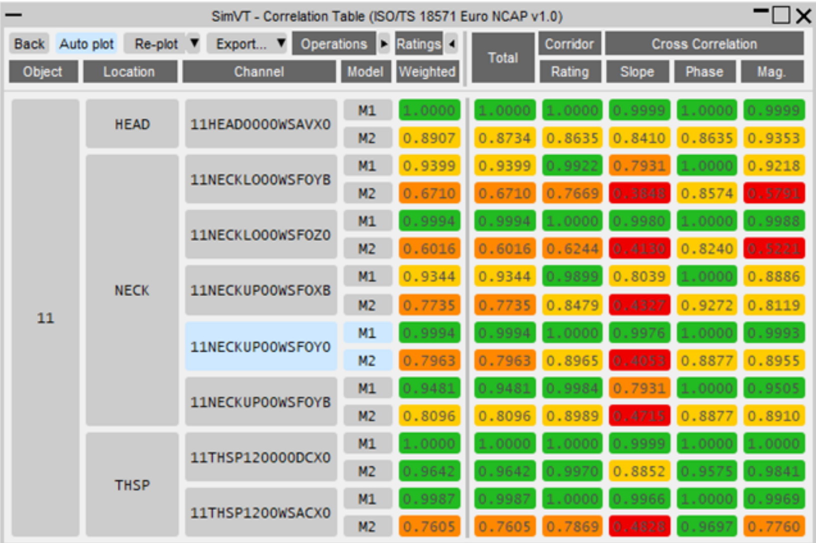
- **Back** - clicking this button goes back to showing the [Correlation Setup Window]()
- **Auto plot** - When auto plot is selected any selections made in the table are automatically (and immediately) plotted. Pending operations are carried out when a selected channel is plotted too.
- **Re-plot** - clicking this button to refreshes the plotting. It switches to saying **Plot** when **Auto-plot** is off and is used to update the plotting to match the new selection.
    - the pop-up arrow next to **Re-plot** opens the [graph options]() pop-up. Note, changing graph options also triggers a re-plot.
- **Export** - Clicking on this button will open the file explorer so that a CSV file containing the results for all the channels shown in the table can be exported to the desired location.
    - The pop-up arrow next to Export reveals the option to export the results to CSV (this is the same as clicking Export directly) and also the option to save a [SimVT settings file]().

- **Operations** - clicking this button (or the expander arrow to the right of it) expands the operations that can be applied to correct curves if they need to be scaled, offset, flipped or filtered.
- **Ratings** - clicking this button (or the expander arrow to the right of it) expands the ratings section to give detailed view of the individual correlation ratings for each channel.

Toggling (selecting) plots

Each row of the Correlation table represents one (simulation versus test) correlation. The rows of the table are determined by the channels that were selected in the Correlation Setup window.

Selected channels appear as buttons in the "Channel" column of the Correlation Table. The "Model" column lists the simulation model tag(s) which are associated with the corresponding channel in the "Channel" column. Note that it is possible to have multiple simulations associated with the same channel. If this is the case, the channel button will occupy multiple rows. For example, in the image below the channel buttons each span 2 rows in the table below because 2 simulation models (M1 and M2) were selected for correlating versus the same test data.


SimVT - Correlation Table (ISO/TS 18571 Euro NCAP v1.0)

Back | Auto plot | Re-plot | Export... | Operations | Ratings

Object	Location	Channel	Model	Weighted	Total	Corridor Rating	Cross Correlation Slope	Cross Correlation Phase	Cross Correlation Mag.
11	HEAD	11HEAD0000WSAVX0	M1	1.0000	1.0000	1.0000	0.9999	1.0000	0.9999
			M2	0.8907	0.8734	0.8635	0.8410	0.8635	0.9353
	NECK	11NECKLO00WSFOYB	M1	0.9399	0.9399	0.9922	0.7931	1.0000	0.9218
			M2	0.6710	0.6710	0.7669	0.3848	0.8574	0.5791
		11NECKLO00WSFOZ0	M1	0.9994	0.9994	1.0000	0.9980	1.0000	0.9988
			M2	0.6016	0.6016	0.6244	0.4130	0.8240	0.5221
		11NECKUP00WSFOXB	M1	0.9344	0.9344	0.9899	0.8039	1.0000	0.8886
			M2	0.7735	0.7735	0.8479	0.4327	0.9272	0.8119
		11NECKUP00WSFOY0	M1	0.9994	0.9994	1.0000	0.9976	1.0000	0.9993
			M2	0.7963	0.7963	0.8965	0.4053	0.8877	0.8955
		11NECKUP00WSFOYB	M1	0.9481	0.9481	0.9984	0.7931	1.0000	0.9505
			M2	0.8096	0.8096	0.8989	0.4715	0.8877	0.8910
	THSP	11THSP120000DCX0	M1	1.0000	1.0000	1.0000	0.9999	1.0000	1.0000
			M2	0.9642	0.9642	0.9970	0.8852	0.9575	0.9841
		11THSP1200WSACX0	M1	0.9987	0.9987	1.0000	0.9966	1.0000	0.9969
			M2	0.7605	0.7605	0.7869	0.4828	0.9697	0.7760

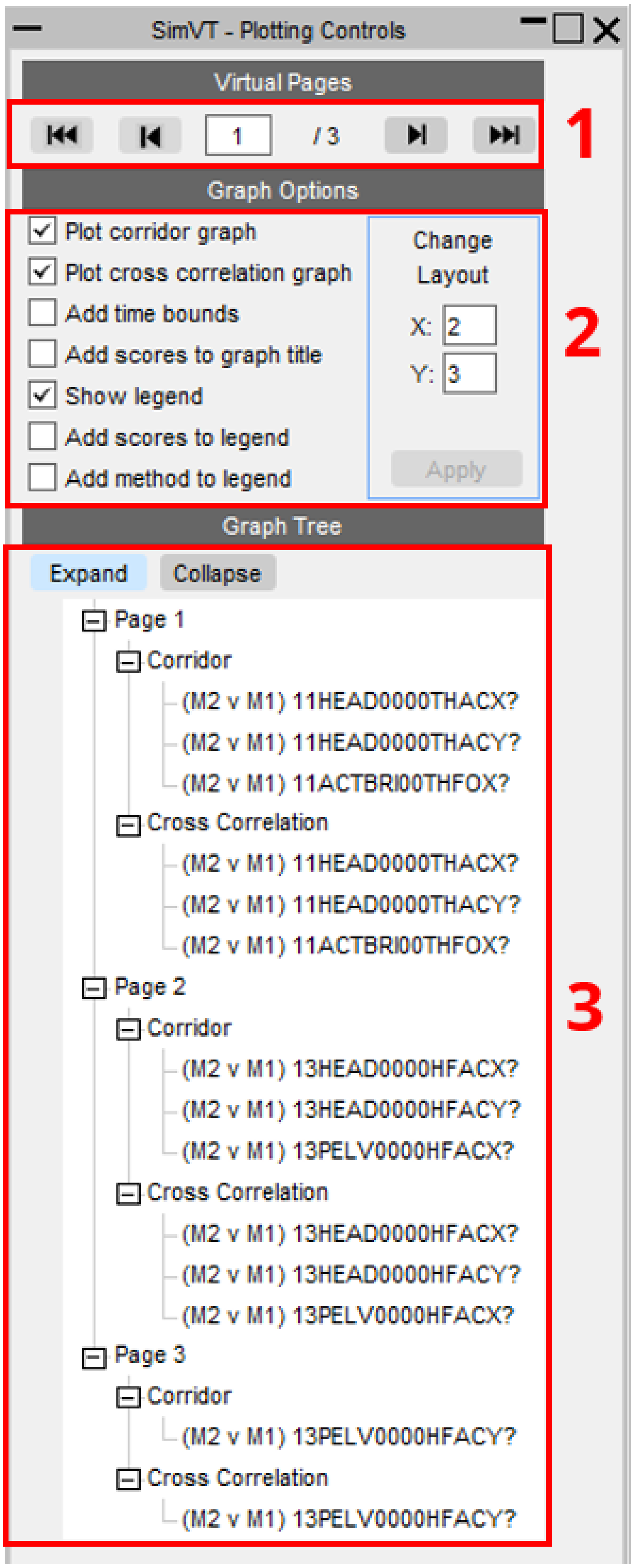
Channels with the same object and location strings are grouped together so that all the channels with the same location and object can be plotted by toggling on the corresponding button (which may span many rows) in the "Location" or "Object" columns (e.g. object "11" which represents the front left occupant in the vehicle.

Any of the buttons in the "Object", "Location", "Channel" and "Model" columns of the Correlation Table can be clicked on to toggle plots on and off. Clicking on a un-toggled (grey) button in one these columns will automatically toggle all the buttons to the right of the button that was clicked on. If the button in the "Model" column is toggled then the correlation plot(s) for that simulation model versus the selected test will be plotted for the channel defined for the row. Conversely, clicking on a button that is already toggled (blue) in any of these columns automatically untoggles all the buttons to the right of the button that was clicked on which will cause the corresponding correlation plots to be 'unplotted'. Note that by default, only the corridor graph is plotted for each correlation, but the graph options can be used to show the cross-correlation plots instead or as well.

Clicking a button in a column and then holding down shift and clicking another button in the same column will change the selection of all the in-between buttons to match the first one clicked.

Clicking one of the ratings under Weighted, Total, Corridor and Cross Correlation columns highlights the plot if it is already plotted.

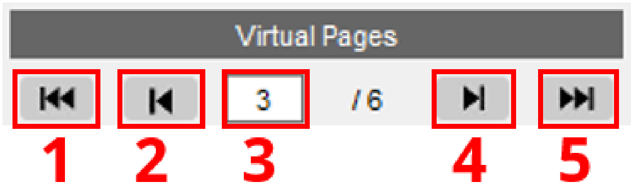
Plotting Controls Window



The Plotting Controls Window in SimVT is used to control what graphs appear for the selected correlation analyses, as well as how those graphs are presented and navigated. The window is divided into three main sections:

- Virtual Pages **[1]**
- Graph Options **[2]**
- Graph Tree **[3]**

Virtual Pages



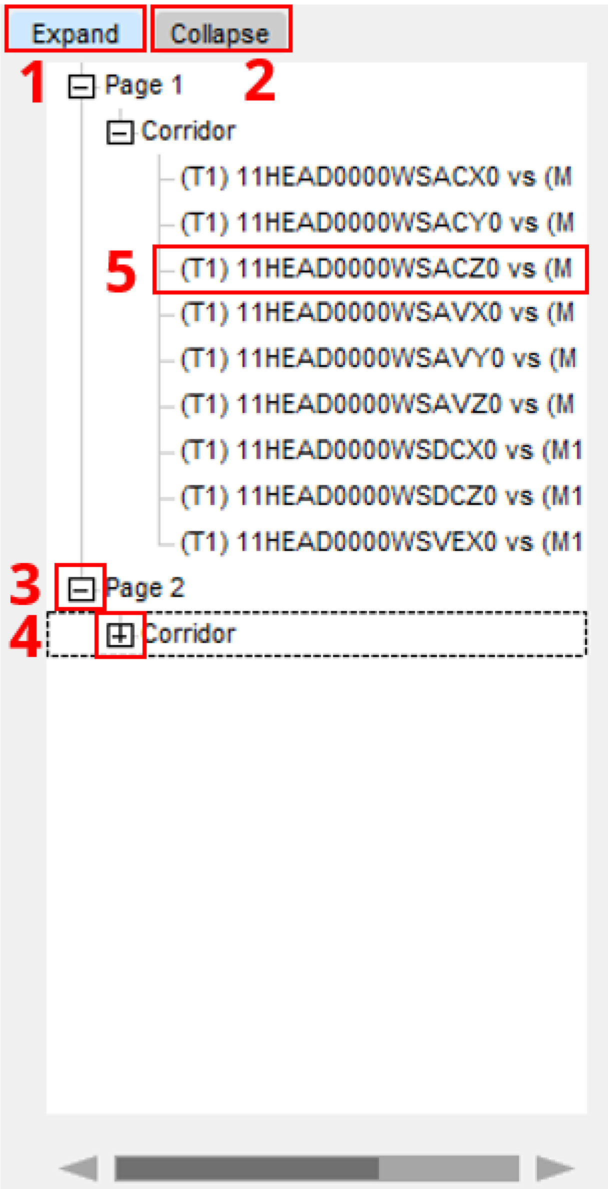
T/HIS currently has a limit of 32 graphs per session. Virtual pages are employed to work around this limitation, allowing SimVT to plot as many graphs as desired. When there is insufficient space to show the graphs for all the selected channels on the current page, additional graphs are added to Virtual Pages. The 'Virtual Pages' section in the Plotting Controls Window allows to navigate through the available virtual pages. The navigation options are as follows:

- Navigate to first virtual page **[1]**
- Navigate to previous virtual page **[2]**
- Navigate to a specific virtual page - requires numerical input **[3]**
- Navigate to next virtual page **[4]**
- Navigate to last virtual page **[5]**

Graph Options

Under the Graph Options section of the Plotting Controls Window the main graph plotting options are located. These options are described in detail in Graph Options.

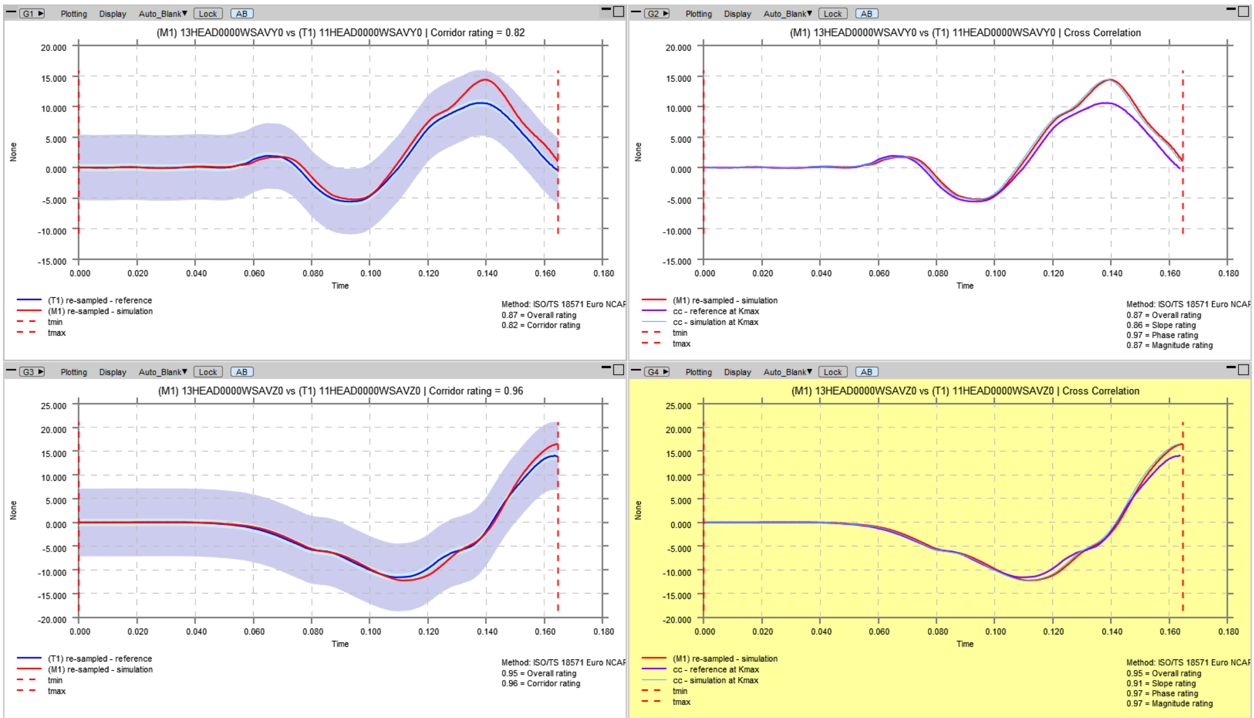
Graph Tree



Under Graph Tree section a list of graph titles grouped by Corridor and Cross Correlation is presented. The functionality in this section includes:

- Expanding the whole graph tree view **[1]**
- Collapsing the whole graph tree view **[2]**
- Expanding an individual section of the graph tree **[3]**
- Collapsing an individual section of the graph tree **[4]**
- Highlighting individual graphs as shown below by clicking graph title in the graph tree view **[5]**

T/HIS


(M1) 13HEAD0000WSAVY0 vs (T1) 11HEAD0000WSAVY0 | Corridor rating = 0.82
(T1) re-sampled - reference
(M1) re-sampled - simulation
tmin
tmax
Method: ISO/TS 18571 Euro NCAP
0.87 = Overall rating
0.82 = Corridor rating
(M1) 13HEAD0000WSAVY0 vs (T1) 11HEAD0000WSAVY0 | Cross Correlation
(M1) re-sampled - simulation
cc - reference at Kmax
cc - simulation at Kmax
tmin
tmax
Method: ISO/TS 18571 Euro NCAP
0.87 = Overall rating
0.86 = Slope rating
0.97 = Phase rating
0.87 = Magnitude rating
(M1) 13HEAD0000WSAVZ0 vs (T1) 11HEAD0000WSAVZ0 | Corridor rating = 0.96
(T1) re-sampled - reference
(M1) re-sampled - simulation
tmin
tmax
Method: ISO/TS 18571 Euro NCAP
0.95 = Overall rating
0.96 = Corridor rating
(M1) 13HEAD0000WSAVZ0 vs (T1) 11HEAD0000WSAVZ0 | Cross Correlation
(M1) re-sampled - simulation
cc - reference at Kmax
cc - simulation at Kmax
tmin
tmax
Method: ISO/TS 18571 Euro NCAP
0.95 = Overall rating
0.91 = Slope rating
0.97 = Phase rating
0.97 = Magnitude rating

CSV Configuration Window

When “Assign config” is clicked on the Correlation Setup window, the CSV Configuration window will appear. The window enables you to load a CSV configuration file and preview any warnings or errors with your CSV file data before you actually assign it to an LS-DYNA model.

Steps to assign a config file:

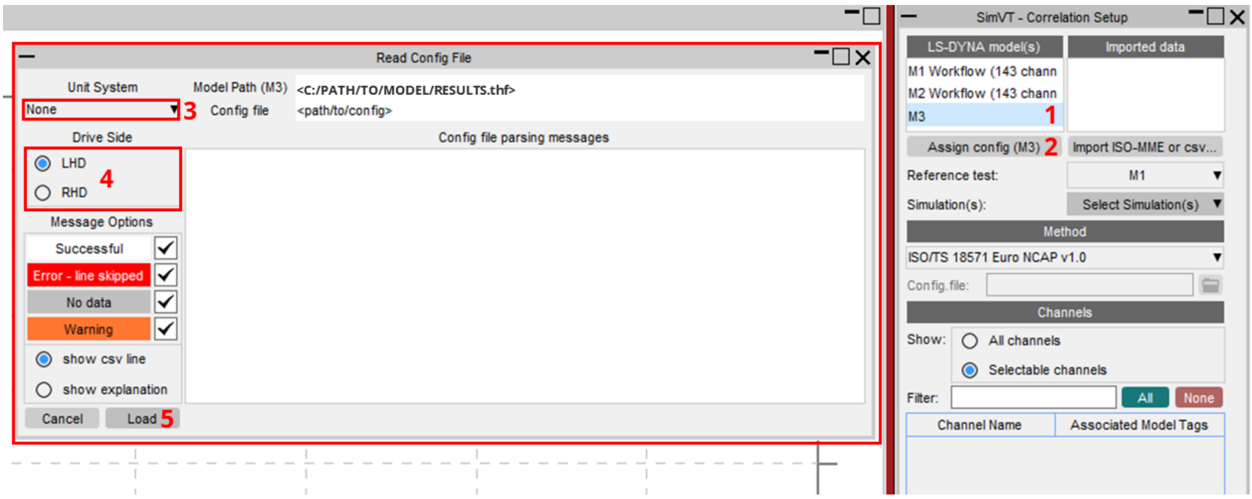
1. Select an LS-DYNA model from the model list that has no Automotive Assessment Workflow Data (AAWD) or CSV config data assigned (i.e. only the model tag will be listed (e.g. "M3"))

2. Click on “Assign config”, CSV Configuration Window will open.

3. Select the unit system from the drop-down

4. Select the vehicle drive side (LHD is default)

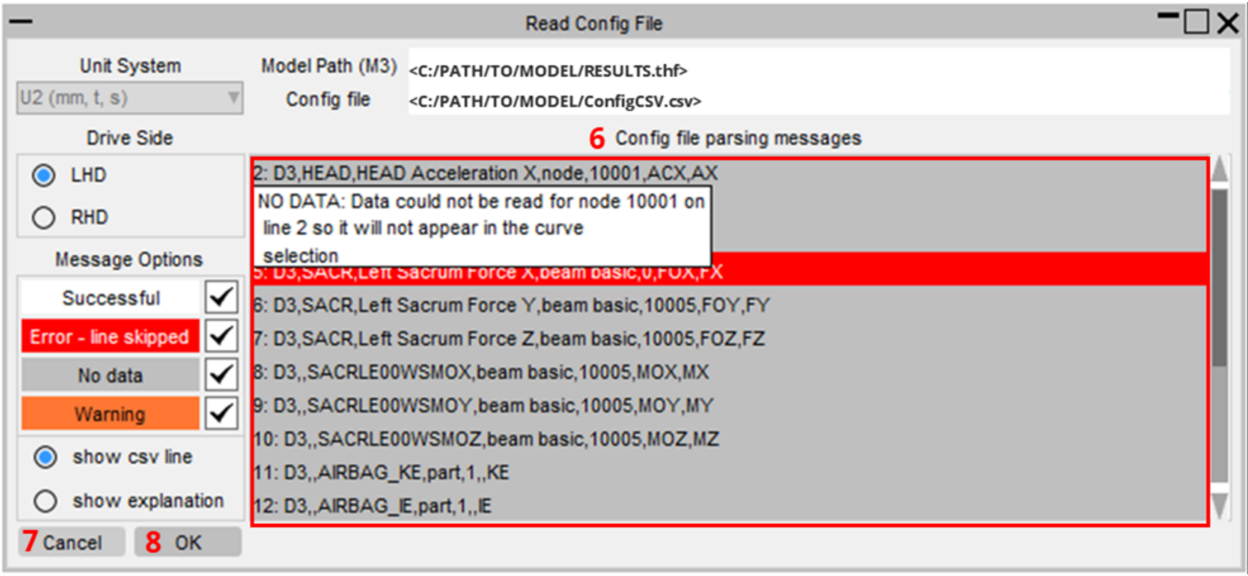
5. Click “Load” and open a CSV file with the correct configuration file format. The "Load" button will change to say "OK" when a config CSV file has been loaded.



6. Check the messages that appear in the window to ensure that the loaded config CSV has no errors and that it has successfully mapped channel data.

7. If some of the parsing messages require attention click "Cancel" and update the CSV before repeating the steps. Note, you can cancel the CSV assignment by pressing “Cancel” at any stage.

8. If you happy to proceed with assigning the CSV config Click on "OK".



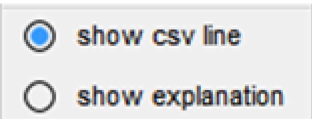
Checking Parsing Messages

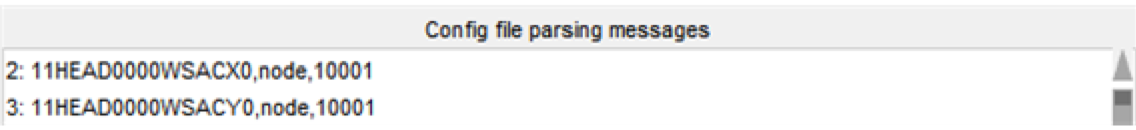
When the configuration file is loaded, messages will appear in the window. Each row is coloured according to the parsing status:

- White means that all the inputs were valid and that the model contains the entity and ID specified
- Red means there was an error with one or more inputs on the line
- Grey means that the model did not contain data for the entity type and ID (e.g., node 1001 below).
- Orange means that there was something wrong with the line, but it may still be possible to parse the line.

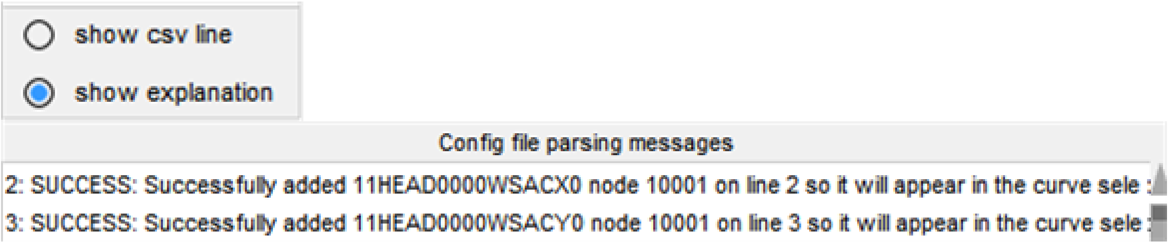
Messages can be hidden by unchecking the corresponding status in "Message Options" to make it easier to find problematic rows.

If “show CSV line” is selected then the messages will show each row of the CSV file.





If “show explanation” is selected then the messages will explain the reason for the status. This is especially useful if there is an error with the CSV data which causes the line to be skipped. It can also help in identifying where an incorrect entity type or id has been provided in the config CSV so that it can be corrected before loading in the CSV again.

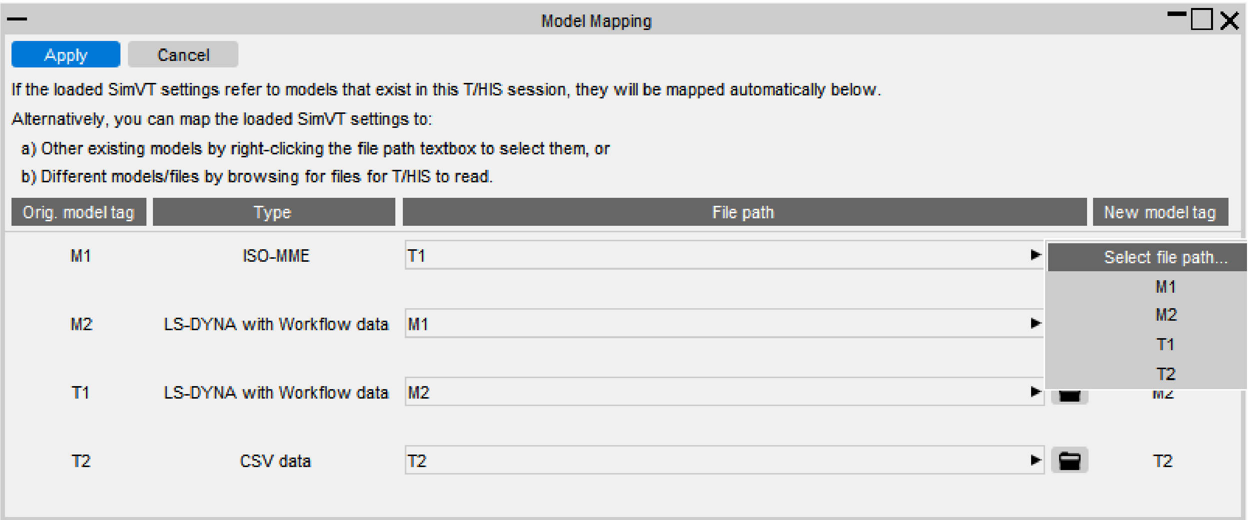


Model Mapping Window

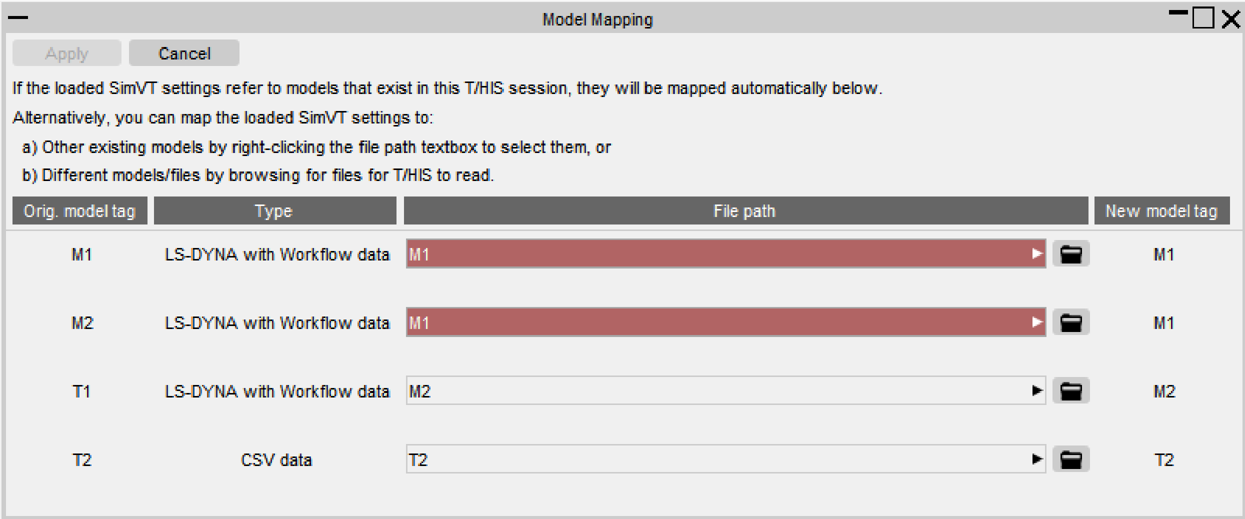
The Model Mapping window appears when loading a SimVT settings file from the Correlation Setup window. It is used to "Map" the model tags defined in the setting file to the models (or data sources) which are defined in SimVT.

Mapping to Existing Data

In SimVT, existing data sources (i.e. those listed in the model list and imported data list) are represented by their model tag. For example, LS-DYNA models will be labelled M1, M2 etc. and ISO-MME and CSV data will be tagged T1, T2 etc. In the model mapping window you can select existing data sources using the selector arrow at the end of the file path textboxes. For example, you may have loaded the LS-DYNA models and imported data source in a different order to the order when the settings file was created. Alternatively, you may want to change which data source is used as the reference test. In the example below T1 was the reference test in the settings file, but it has been mapped to M2 which will become the reference test when **Apply** is clicked.

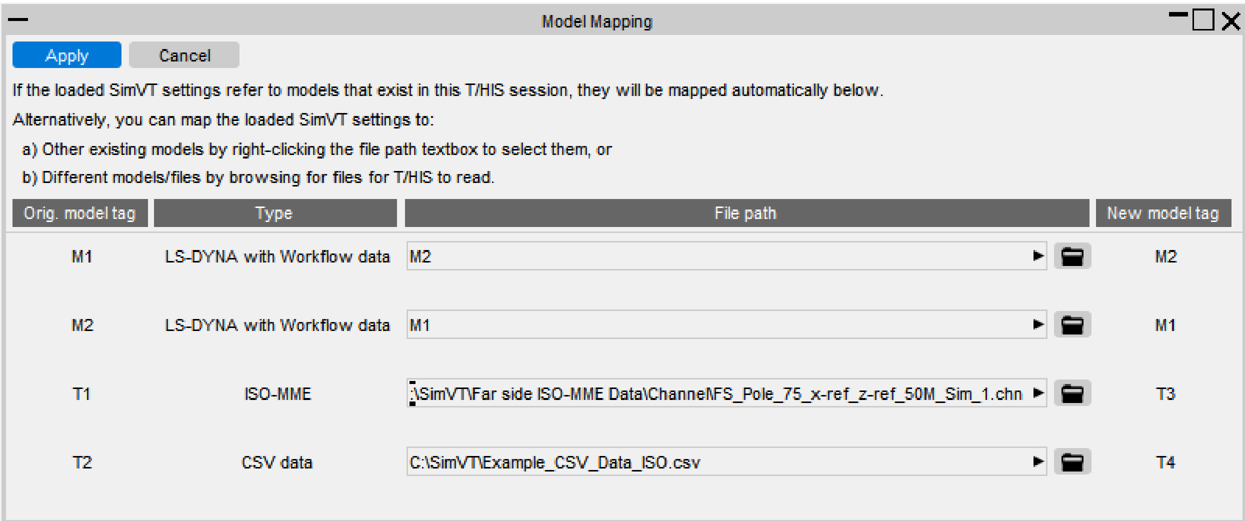


You cannot map the same data source multiple times and the file path textboxes will turn red to indicate the issue (e.g. M1 and M2 in the settings file cannot both map to the existing M1 model).



Mapping to New Data

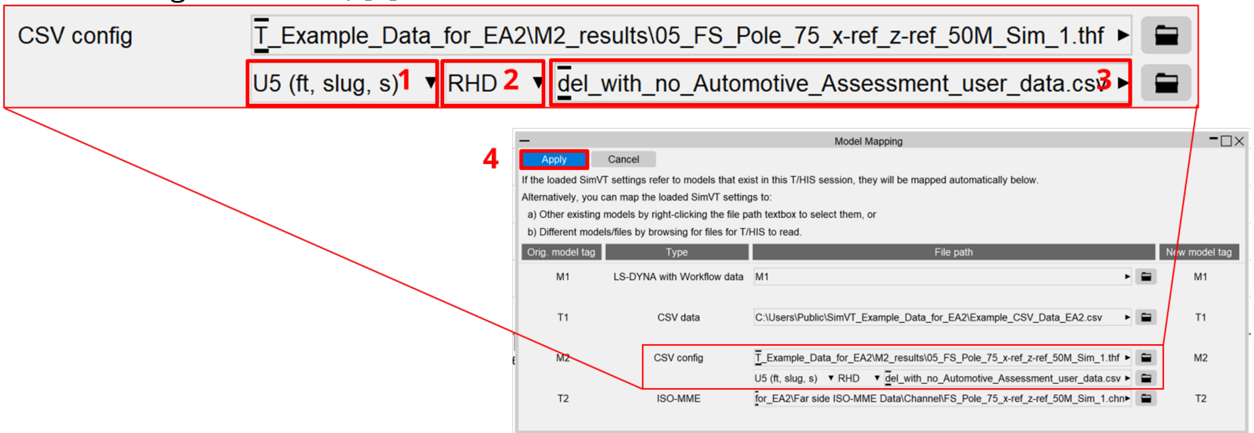
It is possible to import ISO-MME data and CSV data directly from the model mapping window. You can paste the path to the data into the file path textbox or you can click on the file selector and open it from there. In the example below T1 has been remapped to T3 which is a new ISO-MME data source and T2 has been remapped to T4 which is a new CSV data source. Note that the data for T3 and T4 will not be imported until **Apply** is clicked.



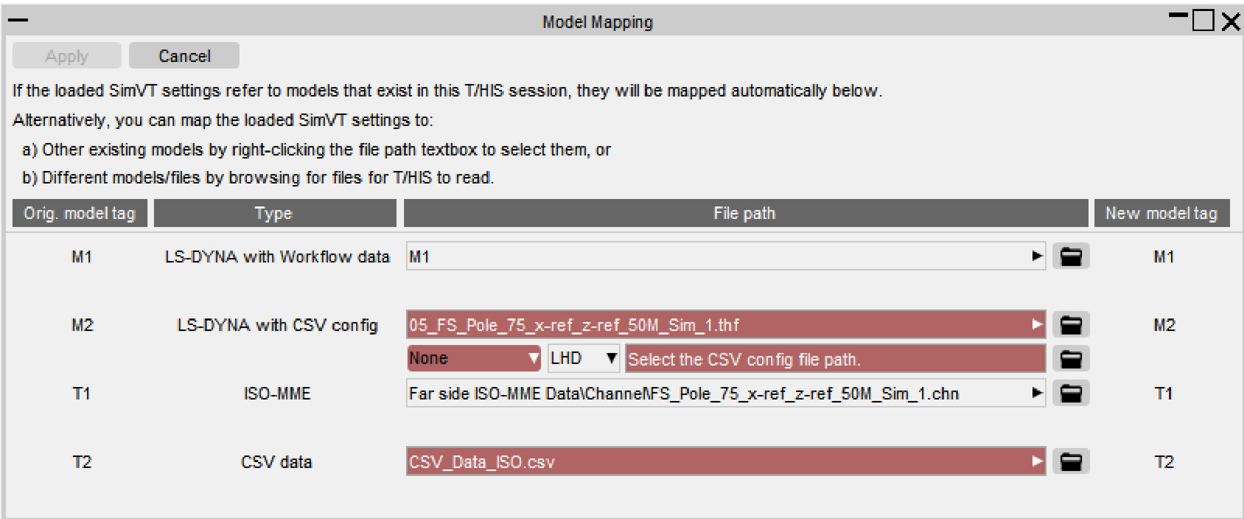
You can also load new LS-DYNA results into T/HIS but you will also need to provide a CSV config which describes the channel entity-id pairings and select the appropriate unit system and vehicle drive side. The steps to follow are:

- Choose which of the original model tags (i.e. from the settings file) you want to remap with new LS-DYNA results.
- Use the file selector to select the LS-DYNA results file (or paste the path into the file path textbox)
- Select the unit system [1]

- Select the vehicle drive side [2] (LHD is default)
- Use the file selector to select the CSV config file (or paste the path into the CSV Config file textbox) [3].

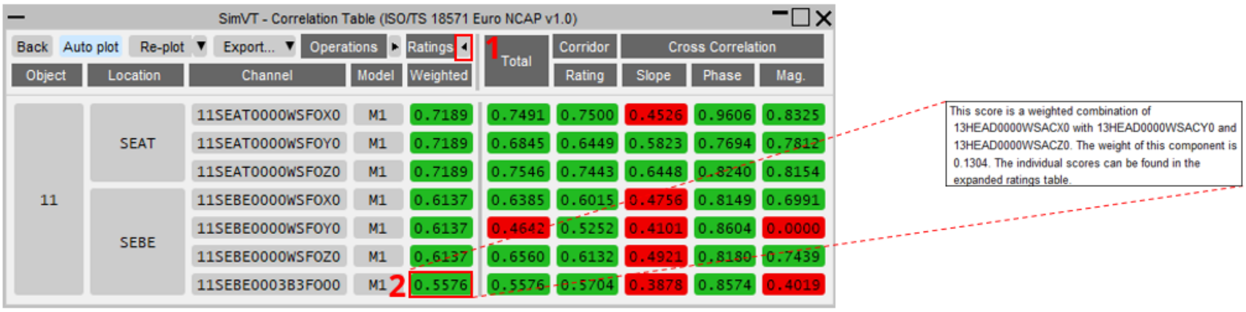


IMPORTANT: Once a SimVT session has been started, you cannot load in or map a model to an LS-DYNA model with Automotive Assessments Workflow Data (AAWD). To do this, you will need to close SimVT, load the model into T/HIS, then restart SimVT, making sure to select the model(s) which you want to use AAWD with. You can then load the settings file and this time the new LS-DYNA model with workflow data will appear as an existing data source in the file path selector.



19.3.4. SimVT Features

Ratings and Weightings



Object	Location	Channel	Model	Weighted	Total	Corridor Rating	Cross Correlation Slope	Cross Correlation Phase	Cross Correlation Mag.
11	SEAT	11SEAT0000WSFOX0	M1	0.7189	0.7491	0.7500	0.4526	0.9606	0.8325
		11SEAT0000WSFOY0	M1	0.7189	0.6845	0.6449	0.5823	0.7694	0.7812
		11SEAT0000WSFOZ0	M1	0.7189	0.7546	0.7443	0.6448	0.8240	0.8154
	SEBE	11SEBE0000WSFOX0	M1	0.6137	0.6385	0.6015	0.4756	0.8149	0.6991
		11SEBE0000WSFOY0	M1	0.6137	0.4642	0.5252	0.4101	0.8604	0.0000
		11SEBE0000WSFOZ0	M1	0.6137	0.6560	0.6132	0.4921	0.8180	0.7439
		11SEBE0003B3FO00	M1	0.5576	0.5576	0.5704	0.3878	0.8574	0.4019

Individual Ratings

By default, the individual Corridor and Cross Correlation ratings are minimised to reduce the amount of space the Correlation Table occupies, but they can be shown by clicking the **Ratings** expander [1].The total rating is the weighted sum of the Corridor and all Cross Correlation ratings, with the weights depending on your selected correlation method. The ratings scores are coloured differently depending on the selected correlation method.

For ISO/TS 18571 Euro NCAP v1.0:

- [0.50, 1.00] - green
- [0.00, 0.50) - red

For CORAplus 4.0.4 and ISO/TS 18571:

- [0.94, 1.00] - green
- [0.80, 0.94) - yellow
- [0.58, 0.80) - orange
- [0.00, 0.58) - red

Weighted Ratings

For all sensor locations where more than one axis is measured, weighting factors are calculated for each axis based on the maximum amplitude of the axis, according to equation below (Equation 2, Section 6.3.3, Euro NCAP VTC Simulation and Assessment Protocol v1.0).

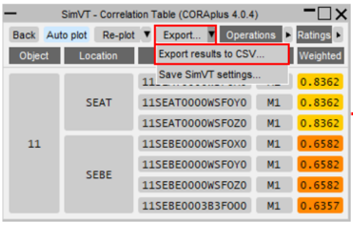
$$w_i = \frac{\max\left(\left|Channel_{test_i}\right|\right)}{\max\left(\left|Channel_{test_x}\right|\right) + \max\left(\left|Channel_{test_y}\right|\right) + \max\left(\left|Channel_{test_z}\right|\right)} \quad with\ i = X, Y, Z$$

The weighting factors are then used to summarise the individual ISO scores for each axis of one sensor to one score per sensor $_{sensor}$ according to equation below (Equation 3, Section 6.3.4, Euro NCAP VTC Simulation and Assessment Protocol v1.0).

$$S_{Sensor} = \sum_{i} w_i * S_i \quad with\, i = X, Y, Z$$

For channels without X, Y, Z siblings, the weighted rating is equal to the total rating. To find out how a single weighted rating was calculated hover over the weighted rating score [**2**] or Export Results.

Exporting Results

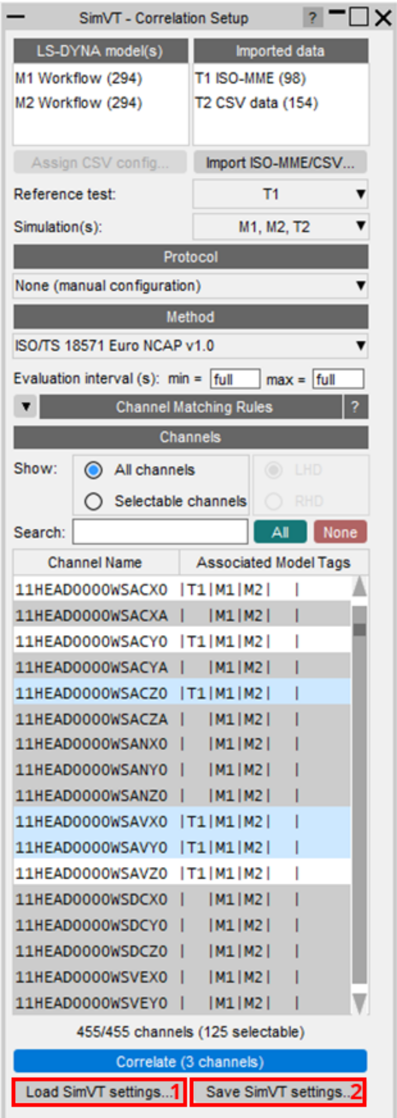


RUN_ID	METHOD	TMIN	TMAX	MAX_CC_VALUE	CC_OFFSET_AT_MAX	CORRELATION_RATING
(M1 vs T1) 11SEAT0000WSFOX0	CORAplus 4.0.4	0.02	0.16	0.98	0.00	0.866271753
(M1 vs T1) 11SEAT0000WSFOY0	CORAplus 4.0.4	0.02	0.102837	0.990044008	0.00	0.85637394
(M1 vs T1) 11SEAT0000WSFOZ0	CORAplus 4.0.4	0.02	0.114368	0.996355489	0.00	0.842724723
(M1 vs T1) 11SEBE0000WSFOX0	CORAplus 4.0.4	0.03	0.148143	0.970110717	-0.01	0.750338828
(M1 vs T1) 11SEBE0000WSFOY0	CORAplus 4.0.4	0.04	0.117111	0.977121679	-0.01	0.705786312
(M1 vs T1) 11SEBE0000WSFOZ0	CORAplus 4.0.4	0.031031	0.155569	0.99	-0.01	0.782937556
(M1 vs T1) 11SEBE0003B3FO00	CORAplus 4.0.4	0.030523	0.140577	0.94	0.00	0.697668837

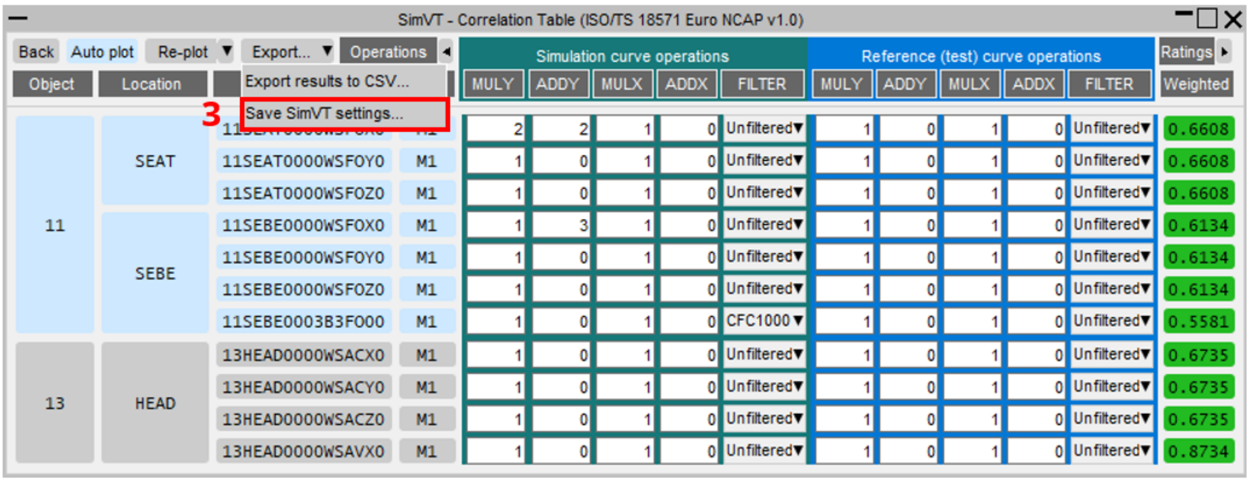
SimVT allows users to export their correlation results from the SimVT tool to a file in CSV format. To export results navigate to **Export.. → Export results to csv** from the Correlation Table Window and save in a desired location. The following fields will be written out:

- RUN_ID - Analysis identificator
- METHOD - Correlation method
- TMIN - Evaluation interval minimum (seconds)
- TMAX - Evaluation interval maximum (seconds)
- MAX_CC_VALUE - Maximum cross correlation value K, K ranges from -1 to 1
- CC_OFFSET_AT_MAX - Time offset at maximum cross correlation value
- CORRELATION_RATING - correlation rating
- PROGRESSION_RATING - correlation rating (specific to CORAplus4.0.4 method)
- PHASE_RATING - phase rating (specific to CORAplus4.0.4 method)
- SIZE_RATING - size rating (specific to CORAplus4.0.4 method)
- SLOPE_RATING - slope rating (specific to ISO18571 methods)
- PHASE_RATING - phase rating (specific to ISO18571 methods)
- MAGNITUDE_RATING - magnitude rating (specific to ISO18571 methods)
- CORRIDOR_RATING - corridor rating
- TOTAL_SIGNAL_RATING - total signal rating i.e. a combination of corridor and correlation ratings
- ISO_RATING_MEANING - a classification for the total rating according to the ISO18571 standard (specific to ISO18571 methods)
- MAX_AMPLITUDE - maximum amplitude of the correlation signal
- WEIGHT - weight calculated based on maximum amplitude
- WEIGHTED_SIGNAL_RATING - weighted signal rating

Settings File



OR



SimVT settings files are a new way for you to load and save your correlation settings from a particular SimVT session. You can load [**1**] and save [**2**] SimVT settings file from the Correlation Setup Window. A SimVT settings file can also be saved from the Correlation Table Window by going to **Export... → Save SimVT settings...** [**3**]. The settings file will have a *.simvt* extension and will contain the following data:

- Model data
- Channels selected for correlation in the channels list
- Operations associated with selected channels
- Channel matching rules
- Protocol (if selected)
- Rating method settings
- Graph layout settings
- Correlations toggled (pale blue) in the Correlation Table window

Once a settings file is loaded, the Model Mapping Window appears. It is populated with the saved models, allowing to map those to existing models or new model data.

Channel Matching Rules

Channel Matching Rules can be defined on the Correlation Setup Window. Creating matching rules facilitates correlating channels that do not have exactly matching channel names.

There are two modes to select from:

1. ISO – allows you to apply rules to specific parts of the ISO-MME channel code
2. General – applies rules to the entire channel string

ISO and General rules can coexist. ISO rules will be applied to everything that qualifies as ISO-MME channel code and General rules will be applied to the rest.

Matching is enabled through two types of matching rules:

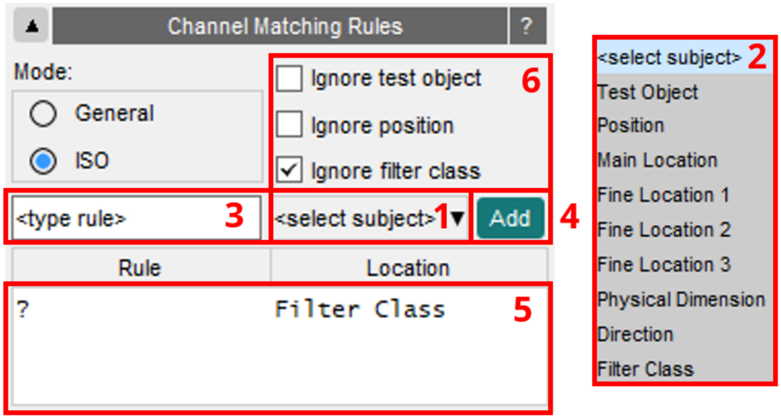
1. Ignore rule – allows you to ignore a part of a string, e.g. the main location (this rule is only applicable in ISO mode)
2. Equivalence rule – allows you to establish equivalency between two or more substrings (options).
    a. options are separated by the "|" e.g. <option1>|<option2>|<option3>|...
    b. there is no limit on the number of options
    c. options are case insensitive (e.g. "LE|RI" is equivalent to "le|ri")
    d. for ISO rules each option must have the same number of characters as the selected subject (e.g. Fine Location 1 has 2 characters so all the options that are treated as equivalent must be 2 characters long).
    e. for General rules options do not need to have the same number of characters as each other.

Adding an ISO rule

- Select the **ISO** mode using the radio button.
- Select a subject by clicking [**1**] and picking from [**2**].
- Type a rule in [**3**]:
    - For the **ignore rule** enter "?"
    - For the **equivalence rule** type in <option1>|<option2>|<option3>|...
- Click **Add** [**4**]. If the rule is successfully validated it will appear in the list [**5**].

Alternatively, use the checkboxes [**6**] to add or delete any of the following rules with a single click:

- Ignore test object
- Ignore position
- Ignore filter class

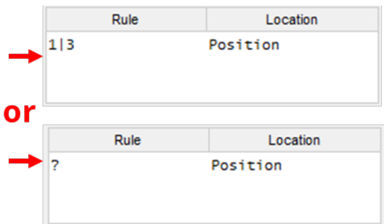


Note: only <u>one</u> rule per ISO subject is allowed.

Examples

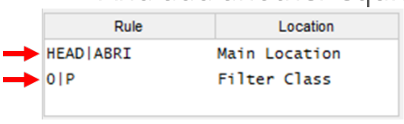
To match channels 1<u>3</u>HEAD000000ACX0 with 1<u>1</u>HEAD000000ACX0:

- Add an equivalence rule by typing in **1|3** and selecting **Position** as the subject.
- Or add an ignore rule by typing in **?** and selecting **Position**.



Similarly, to match 13<u>HEAD</u>000000ACX<u>0</u> with 13<u>ABRI</u>000000ACX<u>P</u>:

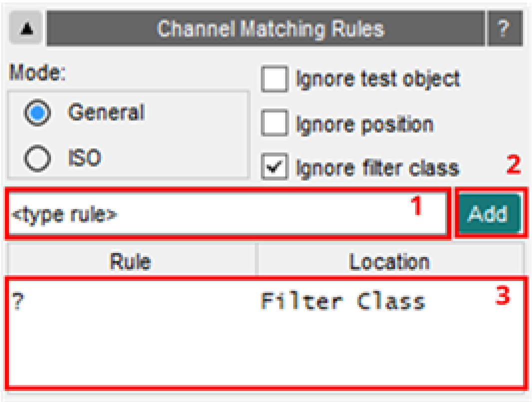
- Add an equivalence rule by typing in **HEAD|ABRI** and selecting **Main Location** as the subject.
- And add another equivalence rule by typing **0|P** and selecting **Filter Class**.



Adding a General Rule

To add a General rule, follow a similar process to ISO rule addition, but only equivalence rules are supported.

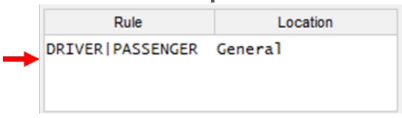
- Select the General mode using the radio button.
- Enter a rule in [**1**] in the format: <option1>|<option2>|<option3>...
- Press **Add** [**2**]. If the rule is successfully validated it will appear in the list [**3**].



Examples

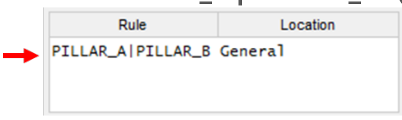
To match channels Driver_Airbag with Passenger_Airbag:

- Add an equivalence rule by selecting **General** in Mode and typing in **DRIVER|PASSENGER**. (Note: matching is case insensitive)



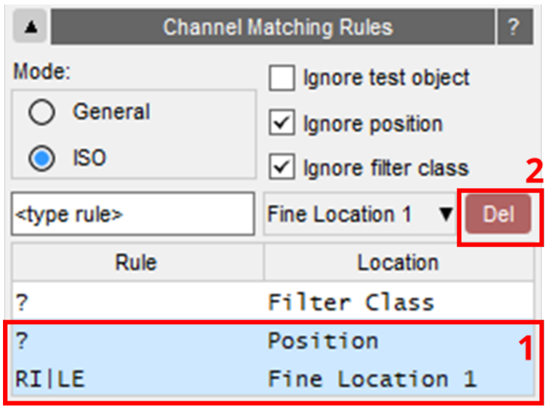
Similarly, to match channels Accel_X_Pillar_A with Accel_X_Pillar_B:

- Add an equivalence rule by selecting **General** in Mode and typing in **PILLAR_A|PILLAR_B**. (Note: matching is case insensitive)



Deleting Rules

Rules can be deleted by selecting one or more from the list [1] and then clicking **Del** [2] to delete them from the list.



Important note

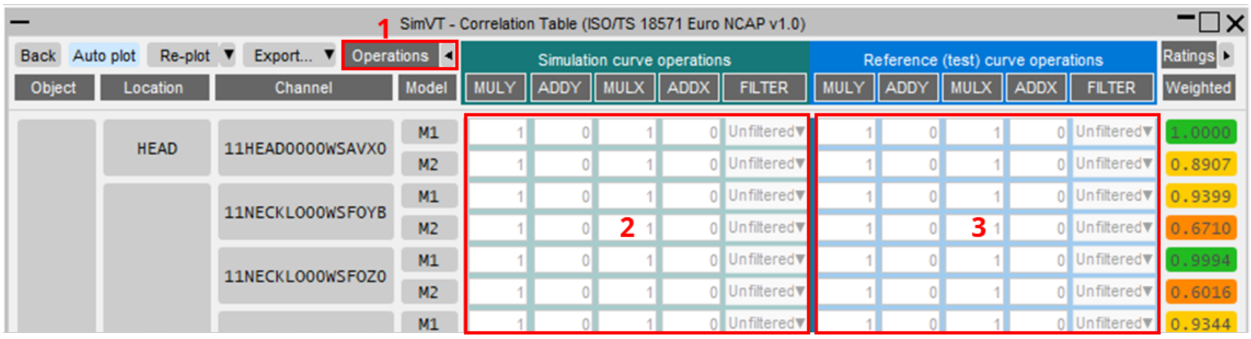
Matching rules are only applied when comparing simulation curves with the reference curves. If you are comparing multiple simulations against test simultaneously, the multiple simulation channels must match each other exactly. The rules allow them to differ from the reference only.

Operations

Corrective operations can be applied to both the simulation and test curves before the correlation is performed.

Applying Operations

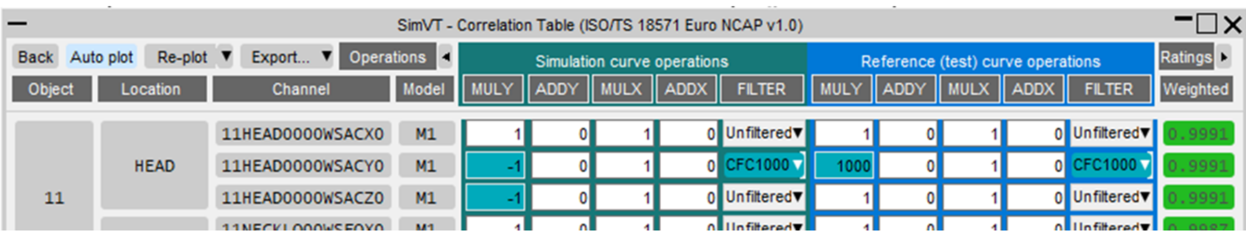
Operations are accessed by clicking **Operations** [1] in the Correlation Table Window which will expanding the window to show the operations that have been applied. This view is minimised by default to reduce the amount of space that the Correlation Table Window occupies.



Operations can be applied to both simulation [2] and reference [3] curves.
The order of application is in the order below: (i.e., from left to right)

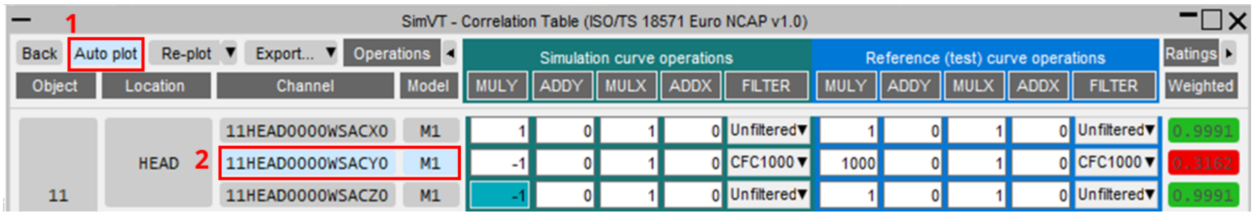
- **MULY** scales the Y coordinates by the value entered (default = 1, no scaling). For example, if you want to invert the sign of a curve, enter “-1”.
- **ADDY** offsets the Y coordinates by the value entered (default = 0, no offset).
- **MULX** scales the X coordinates by the value input (default = 1, no scaling).
- **ADDX** offsets the X coordinates by the value input (default = 0, no offset)
- **FILTER** applies a CFC filter: CFC1000, CFC600, CFC180, CFC60 are supported (default = Unfiltered).

When you change the value of an operation, the textbox will change to a latent colour to indicate that the operation is “pending”:



Pending operations are only executed on channels (rows) that are selected channels.

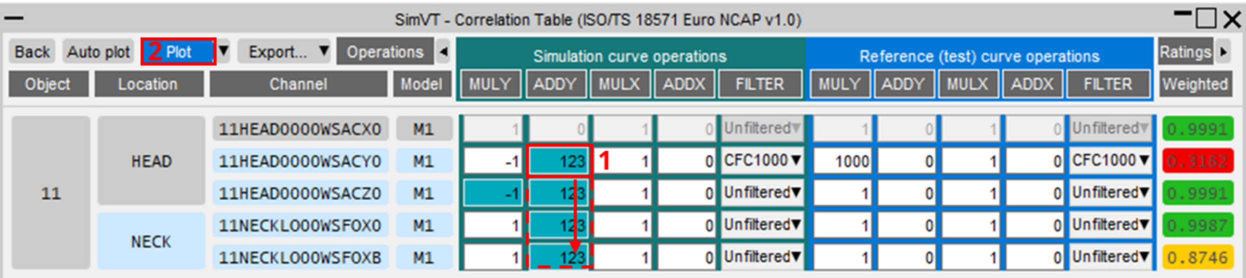
When **Auto plot** is active [1] and a channel is selected [2], operations are executed automatically. Note that the latent coloured textboxes change back to white to indicate that the operations have been applied and ratings will be recomputed.



Applying Multiple Operations (Auto plot off)

If you plan on applying lots of operations, turn **Auto plot** off and apply a batch at a time.

When multiple rows are selected, and **Auto plot** off, you can choose to input a value on any row and it will be propagated to all the active rows (e.g., the input “123” [1] is applied to all the active rows in the ADDY column). Pending operations are again shown as with a latent colour background. Pressing **Plot** [2] will execute pending operations but only for channels (rows) that are selected.



When no rows are selected all operation inputs become active, and you can input operations on a row-by-row basis, but note that pressing **Plot** will not result in the operations being applied, you need to select them first.

Regularisation

After applying the MULX scaling, the curve is regularised to an X axis interval of 0.0001, which corresponds to a sampling frequency of 10kHz. This is consistent with the Euro NCAP Far Side VTC requirement to sample curves are 10kHz. Note that the regularisation happens **after** corrective scaling of the X axis since **curves are assumed to be in SI units** (i.e. time in seconds) and if MULX is used to scale a curve it is assumed to be converting the time seconds.

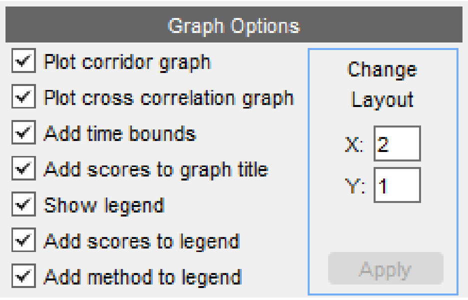
Filtering Operations

Care must be taken when using the filtering operations as filtering a channel curve which has already been filtered will result in different curve and therefore different results compared to filtering the unfiltered channel curve. In most cases the difference is small, but if you wish to be consistent with the Euro NCAP Far Side VTC protocol then you need to make sure that you do not apply another filter to channel curves which have already been filtered (i.e. their filter class in A, B, C, D etc.).

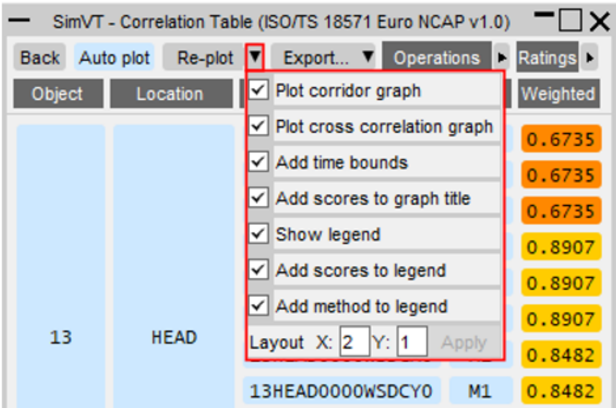
Saving Operations

Operations are saved in the SimVT settings file and they will be restored when loading SimVT settings. This can save a lot of time if you plan on reusing the operations you have defined.

Graph Options

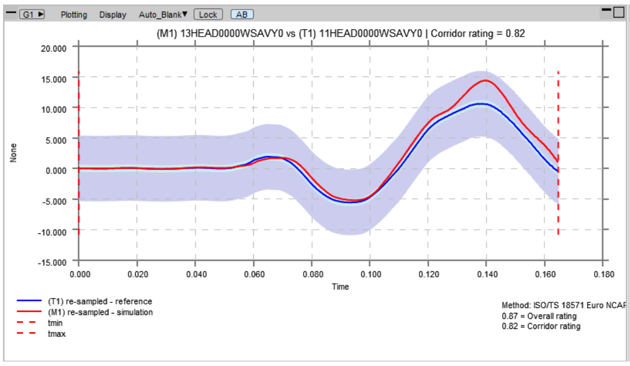


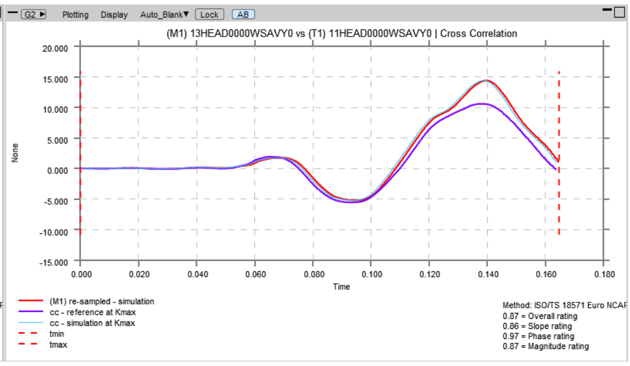
OR



The graph options controls can be found under the Graph Options section of the Plotting Controls Window or in the Plot button dropdown of the Correlation Table Window. You can specify any combination of the following options, all of which are enabled on the view below:

- **Plot corridor graph** - controls if the corridor graph is plotted (Note: one of the graphs must be plotted).
- **Plot cross correlation graph** - controls if the cross correlation graph is plotted (Note: one of the graphs must be plotted).
- **Add time bounds** - controls if vertical lines (dashed red) representing the evaluation interval are displayed on the graph.
- **Add scores to graph title** - controls if the rating score is added to the title of the graph (Note: this only applies to corridor graphs).
- **Show legend** - controls if a legend is displayed on the graph.
- **Add scores to legend** - controls if the rating scores are written onto the legend of the graph.
- **Add method to legend** - controls if the rating method is written onto the legend of the graph.
- **Change layout** - controls the graph layout of the T/HIS session.


(M1) 13HEAD0000WSAVY0 vs (T1) 11HEAD0000WSAVY0 | Corridor rating = 0.82
Time
None
(T1) re-sampled - reference
(M1) re-sampled - simulation
tmin
tmax
Method: ISO/TS 18571 Euro NCAP
0.87 = Overall rating
0.82 = Corridor rating


(M1) 13HEAD0000WSAVY0 vs (T1) 11HEAD0000WSAVY0 | Cross Correlation
Time
None
(M1) re-sampled - simulation
cc - reference at Kmax
cc - simulation at Kmax
tmin
tmax
Method: ISO/TS 18571 Euro NCAP
0.87 = Overall rating
0.86 = Slope rating
0.97 = Phase rating
0.87 = Magnitude rating

19.3.5. SimVT FAQ

SimVT FAQ

How can I change the layout of the graphs?

By default the layout is set to 3x3, but you can change the layout from either the using graph options which can be accessed from both the Correlation Table window and the Plotting Controls window.

How can I flip the sign of my simulation or test curve so that they are consistent?

You can flip the sign of either curve using the "MULY" operation. Assuming the curves are in the same units you simply need to enter "-1" in the "MULY" column for the curve that needs corrected.

Why are the operations I have entered not being applied?

Operations that have not been applied yet will show with a turquoise background. They are only applied when the row is selected **and** plotted. In "Auto plot" mode, selecting a row is equivalent to plotting it so to force pending operations to be applied you simply need to select the row by pressing one of the object/location/channel/model buttons that corresponds to the row in question. When not in "Auto plot" mode you need to press the **Plot** button to plot the selected row(s) and force the pending operations to be applied.

Why do the ratings say "NONE"?

The ratings will say none when it was not possible to perform the correlation. The main reason for this is that one or both of the curves do not lie within the min and max time window (a.k.a. evaluation interval) or there is a time offset between them such that they do not have points in the same X range. You should check the values of the "ADDX" and "MULY" operations to see if they could explain why the curves do not overlap.

19.4. Energy Check

Energy Check

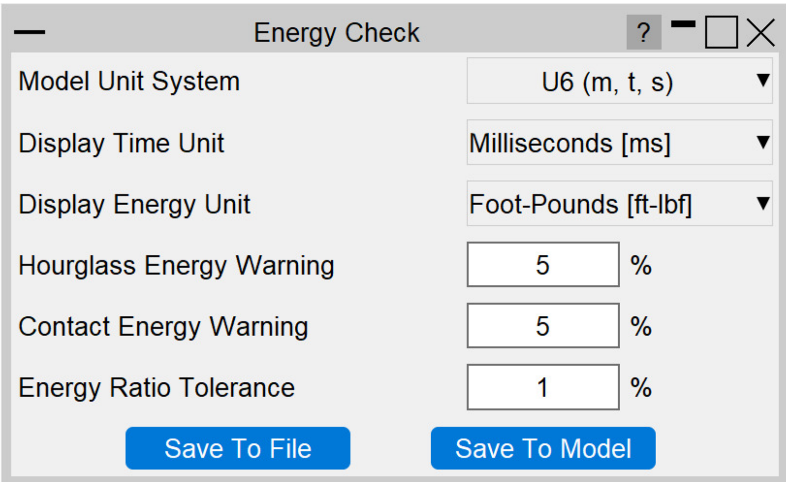
Tools → Workflows → Energy Check

The Energy Check Workflow is a quick tool to help plot global energies for your model and perform checks.
In PRIMER we can set the tool up, by selecting the model unit system, selecting the desired time and energy units and selecting the tolerances and percentages allowed for the energy checks.

In T/HIS, this tool displays the Kinetic Energy, Internal Energy, Hourglass Energy, Total Energy, Absolute Total Contact Energy, External Work and Total System Energy. Multiple checks are completed on these energies such as the percentage of Hourglass Energy compared to Total Energy, the Absolute Total Contact Energy compared to Internal Energy and Energy Ratio (Total System Energy).

Setup in PRIMER

When this tool is initially launched, PRIMER will ask you to select which model you want to use to configure for Energy Check. You can only configure a single model at a time.



Model Unit System

You can select the unit system used for the model from the drop-down menu.
Once you have selected the unit system, the Display Time Unit and Display Energy Unit will automatically update to match the model unit system.

Display Time Unit

You can select the display time unit to use: Seconds or Milliseconds.

Display Energy Unit

You can select the display energy unit to use: Joules, Millijoules, Kilojoules or Foot-Pounds.

Hourglass Energy Warning

You can change the warning value of which Hourglass Energy as a percentage of Total Energy will be flagged.
Hourglass Energy should be less than 5% of Total Energy, therefore the default for this check is 5%.

Contact Energy Warning

You can change the warning value of which absolute Total Contact Energy as a percentage of Internal Energy will be flagged.
Contact Energy should be less than 5% of Internal Energy, therefore the default for this check is 5%.

Energy Ratio Tolerance

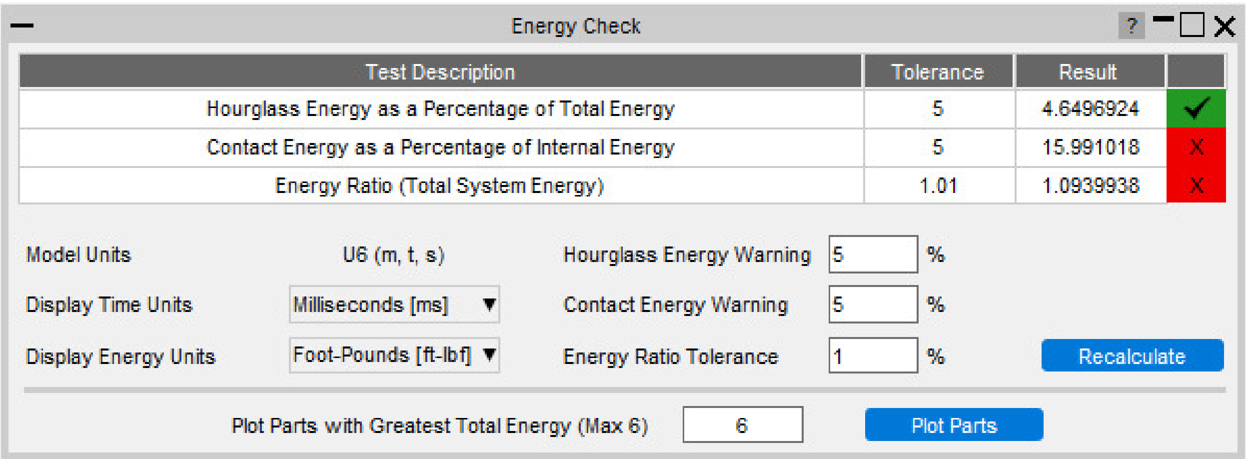
You can change the warning value of which Energy Ratio (Total Energy minus External Work or Total System Energy) will be flagged at if the curve has exceeded the tolerance. Total System Energy should remain constant (Energy Ratio should stay at 1.0), although this is not realistic therefore the default for this check is 1%.

Saving

Save the Workflow data to a .json file or save it to your model and then write the keyword file from PRIMER.

Use in T/HIS

When this tool is initially launched, the tool will complete a first run of the script by producing the aforementioned energies and checks. Once the run has completed the GUI will look something like this by default:



Energy Checks

The first check determines if Hourglass Energy is below the percentage you specified of Total Energy. If this check is a fail, the maximum percentage will be displayed in the GUI. On its graph, the Hourglass Energy as a percentage of Total Energy is displayed with datums used to visualise the warning threshold. If Hourglass Energy does not exist in the model or have the same number of points on the graph as Total Energy, the check will not be displayed.

The second check determines if absolute Total Contact Energy is below the percentage you specified of Internal Energy. If this check is a fail, the maximum percentage will be displayed in the GUI.
On its graph, the Contact Energy as a percentage of Internal Energy is displayed with datums used to visualise the warning threshold. If time intervals for contacts are not consistent with other energies, the check will not be displayed. Please edit the DT field for GLSTAT within the *DATABASE_ASCII card to enable fill accuracy of this check.

The third check is Energy Ratio, which determines if the Total System Energy (Total Energy minus External Work) is constant. If this check is a fail, the energy ratio value will be displayed in the GUI.
On its graph, if TER - total/initial (Energy Ratio) is available as a curve in your model then it is displayed with datums to visualise the tolerance threshold.
If it is not available then (Total Energy - External Work) / Initial Total Energy is displayed, please note this check is calculated manually so may not be as accurate. Please request GLSTAT in your model to enable full accuracy of this check. If External Work is not available in your model, the check will not be displayed.

Model Unit System

The unit system that has been selected in PRIMER for this model.

Setup

You have the option to change the Display Time Unit, Display Energy Unit, Hourglass Energy Warning, Contact Energy Warning and Energy Ratio Tolerance just like in PRIMER, however any adjustments made here won't be saved upon re-load of the workflow, unlike in PRIMER if it was saved to a .json or to the model. Press 'Recalculate' to reproduce the graphs and energies with the updated setup options.

Recalculate

You can recalculate the energies and checks with the any changes made to the display time/energy units and warning tolerances taken into effect.

Plot Parts with Greatest Total Energy

Plots the Kinetic, Internal, Hourglass and Total Energies of the biggest N parts in the users model calculated by their greatest Total Energies. N is a value specified by the user, from 1 to 6.

19.5. Entities of Interest

Entities of Interest

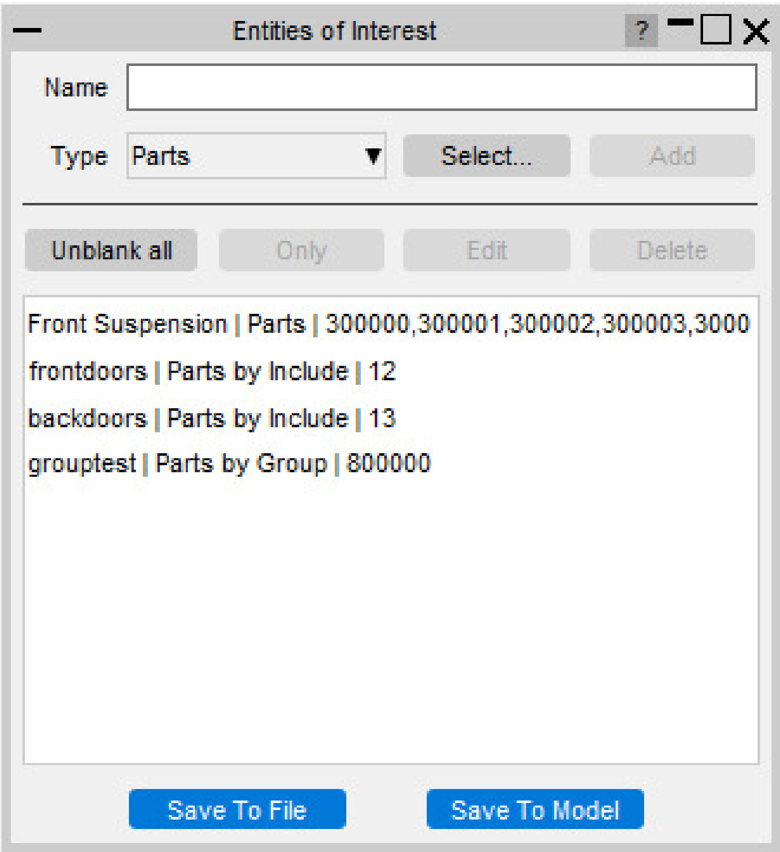
Tools → Workflows → Entities of Interest

The Entities of Interest tool allows you to visualise specific groups of entities quickly in D3PLOT and action them. Actions include:

- Only
- Zoom In
- GLB Export
- Mixed-Mode Plot
- Highlight
- Colour By

Setup in PRIMER

In PRIMER, open Entities of Interest from the Workflows menu (**Tools → Workflows → Entities of Interest**). In the menu that appears, you can a dd groups of entities to a list, and save it to a Workflows .json file or add the data to your model in PRIMER and then write the keyword file.



Adding Entries to the List

For each group of entities, make sure that you complete the following steps:

1. **Name**
   You must give your entry a name so it can be identitfied
2. **Type**
   Select the entity type for your entry. Current options are **Parts, Parts by Set, Parts by Include or Parts by Group**
3. **Select...**
   Select the entities for your entry by using the menu that appears on the right-hand side.
4. **Add**
   Once you have completed the above steps, **Add** your entry to the list.

Only and Unblank all

You can select as many entries from the list as you want and then click **Only** to show only the selected entries. You can click **Unblank all** at any time to unblank the entire model.

Edit

You can only have one entry selected to **Edit** an entry. Much like adding an entry to the list, you then update its name and/or the entities in the entry. You can not change entity type.

Delete

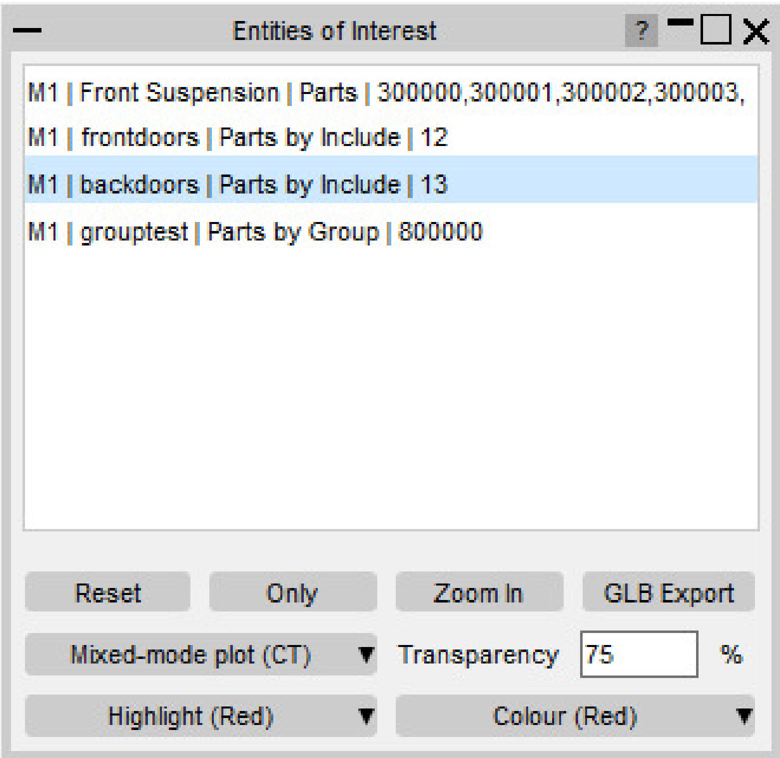
Select one or more entries from the list and click **Delete** to delete them.

Saving

You can either save your list of Entities of Interest to a .json file or directly to the model. The user data from the file will then be picked up when the Workflow is selected in D3PLOT.

Use in D3PLOT

When you open Entities of Interest in D3PLOT, your list of entries will be displayed, along with action buttons:



Reset

You can **Reset** the selected models to default.

Only

Select one or more entries and **Only** them.

Zoom In

Select one or more entries and **Zoom In** on them.

GLB Export

Select one or more entries and **GLB Export** them for use in D3PLOT Viewer. This will open the GLB Export Options window, which has the following options:

- **Directory**
  Provide a valid directory location for saving GLB files
- **Output**
  Select the output type – either **Current Frame** or **Animation**
- **Frame Rate**
  If Animation is chosen for output, select the default **Frame Rate**
- **Export**
  Once the above options are all valid, click **Export** to export one GLB file for each of the selected entries. The exported GLB filenames will correspond to the entry names.

Mixed-Mode Plot

Select one or more entries and perform a **Mixed-mode plot** on them.
The selected entities will be plotted by a method of the users choice. The default is CT however you can select from the following options using the dropdown menu:
Continous tone solid contour (CT), colour contours with lightning model (SI), line contour plot (LC), cloud points plot (CL), Iso-surface contour plot (ISO), velocity arrows plot (VEL) and interface stresses and forces (INT).
Any entities that are not selected will turn transparent and grey. The transparency can be set by the user using the transparency textbox, the default is 75%.

Highlight

Select one or more entries and **Highlight** them.
The highlighted entities will turn opaque and a colour of your choice from the dropdown menu, the default is red.

The non-highlighted entities will turn grey and transparent of which the value can be set the user using the transparency textbox, the default is 75%.

Transparency

The **Transparency** textbox is used to control the transparency value used in Mixed-Mode Plotting and Highlighting. The default is 75%.

Colour

Select one or more entries and **Colour** them.
The selected entities will become opaque and turn a colour of your choice from the dropdown menu, the default is red.

19.6. Eroded Elements

Eroded Elements

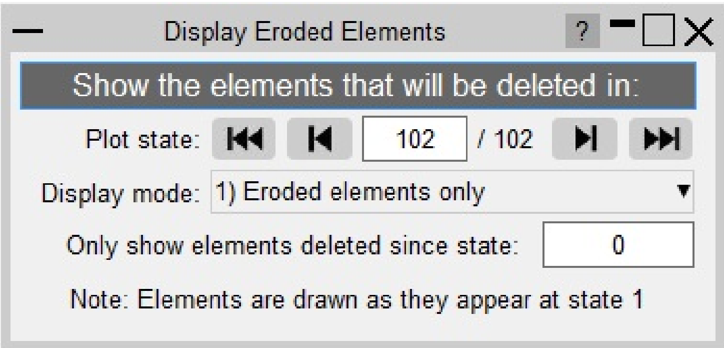
Tools → Workflows → Eroded Elements

The Eroded Elements tool allows you to visualise eroded (deleted) elements in your LS-DYNA simulation.
Note that while using this tool, elements are drawn as they appear at state 1.

Use in D3PLOT

You don't need to set up anything in PRIMER to use the Eroded Elements Workflow. Simply open it in D3PLOT (**Tools → Workflows → Eroded Elements**) to visualise Eroded Elements for any set of results.

When you open Eroded Elements, the elements deleted in the final state are displayed. The following menu will appear:



The menu provides several options to control the visualisation of eroded elements.

Plot state

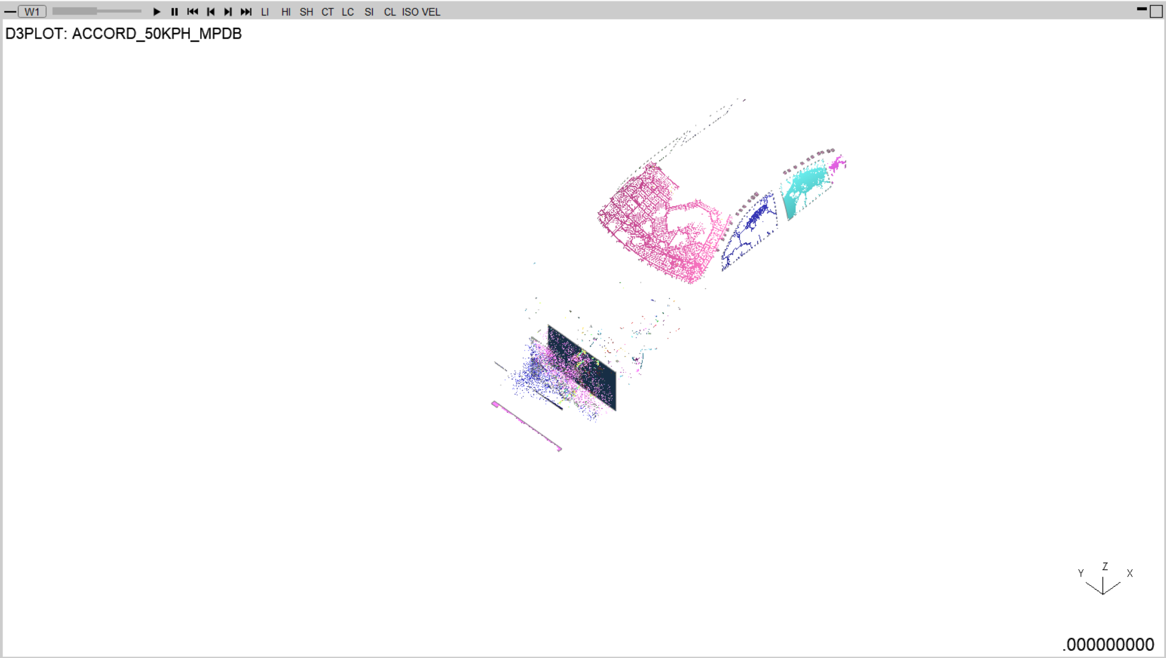
You can choose to display the elements deleted at any plot state. Use the controls in the menu to change plot state, rather than D3PLOT's main controls. Only elements deleted between the comparison state and the plot state will be displayed. Note that the plot state cannot be before the comparison state.

Display mode

There are three display modes:

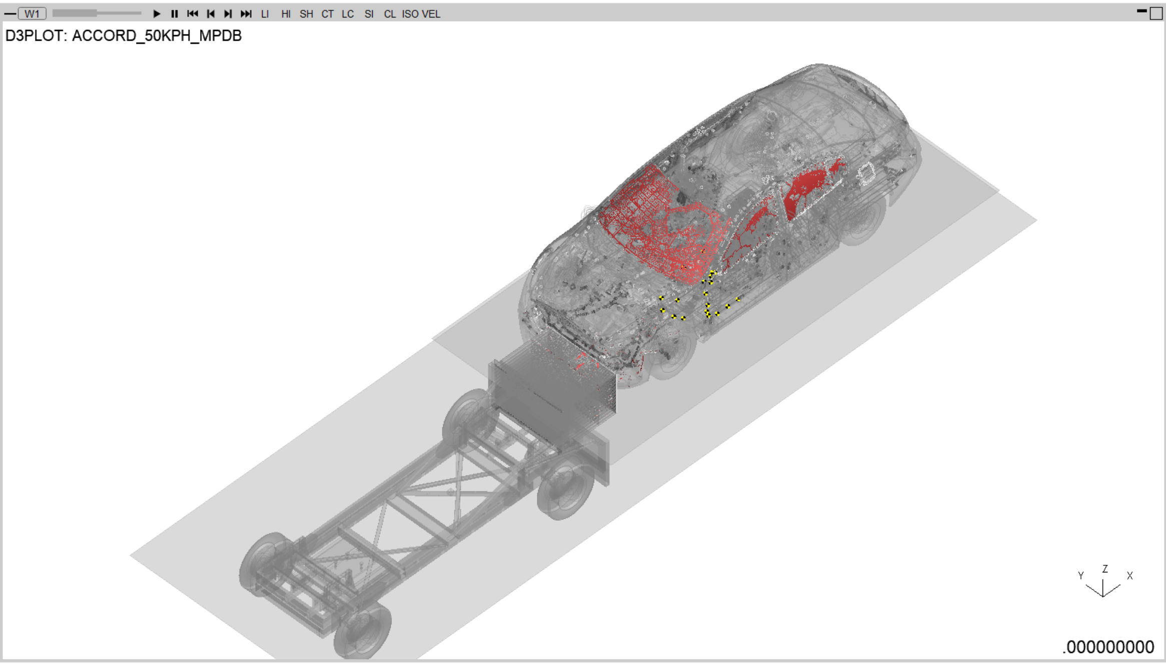
1. Eroded elements only (default)

> Only the elements deleted between the comparison state and the plot state are shown (all other elements are blanked)



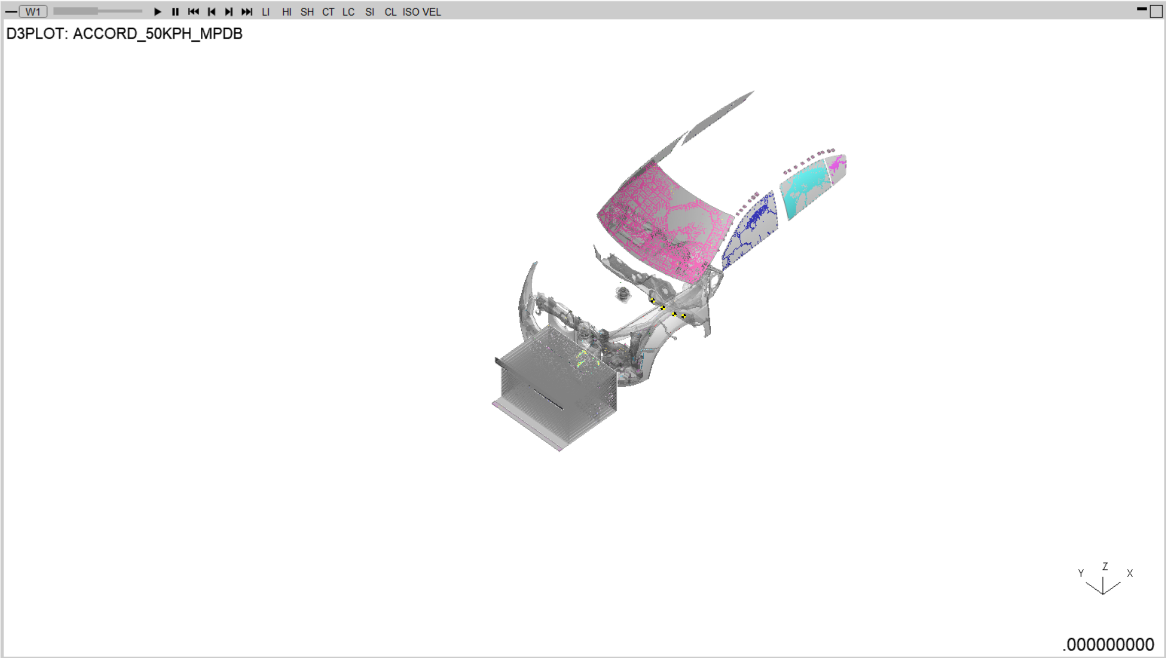
2. Eroded elements in red

Displays the elements deleted between the comparison state and the plot state in red and all other elements in transparent-grey.



3. Parts with eroded elements

Only the parts with elements deleted between the comparison state and the plot state are shown (all other parts are blanked). Elements on these parts that are not deleted will be shown in transparent-grey.



Comparison state

You can choose to display the elements deleted between the comparison state and the main selected state. The comparison state can be modified via the textbox. Only elements deleted after the comparison state will be displayed. Note that the comparison state cannot exceed the plot state.

Properties

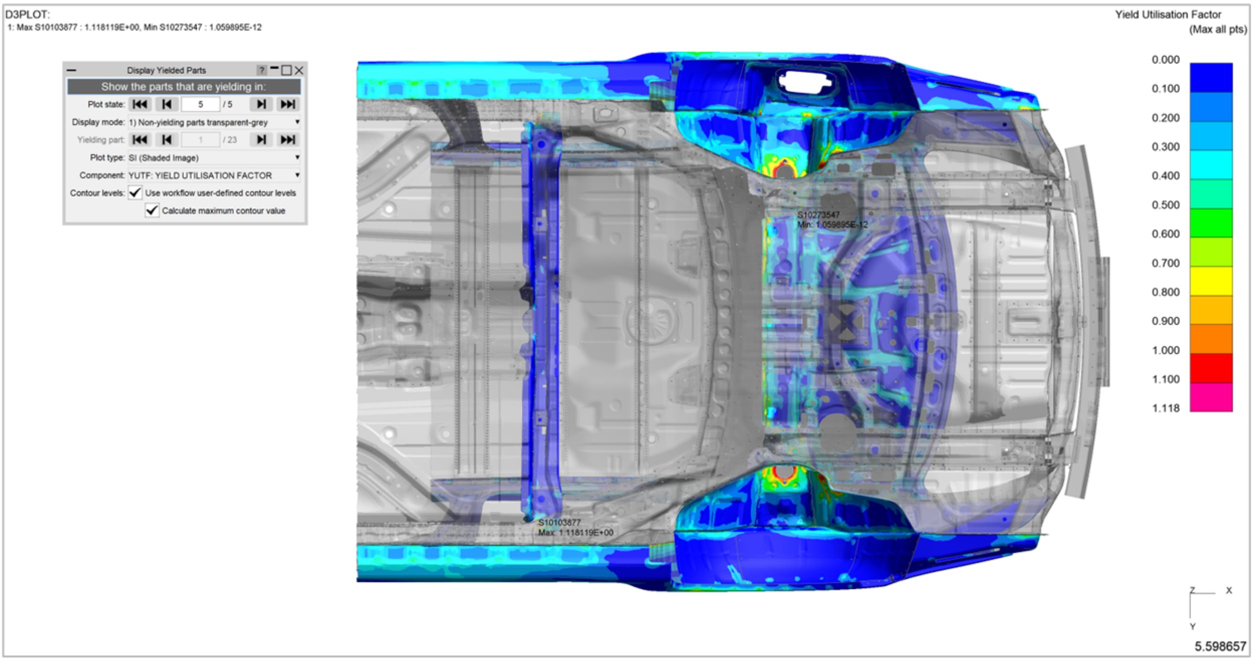
When Eroded Elements is opened, a temporary properties file is saved. When you exit the tool, you can choose to restore the model properties (view, blanking, colours, etc.) to their appearance before you opened the tool.

19.7. Strength Check

Strength Check

Tools → Workflows → Strength Check

The Strength Check tool allows you to visualise yielding shell, thick shell, and solid parts in D3PLOT. Note: Yielding parts are parts containing at least one yielding element, measured as an element with a Yield Utilisation Factor greater than one (or Yield Utilisation Percentage greater than 100%).



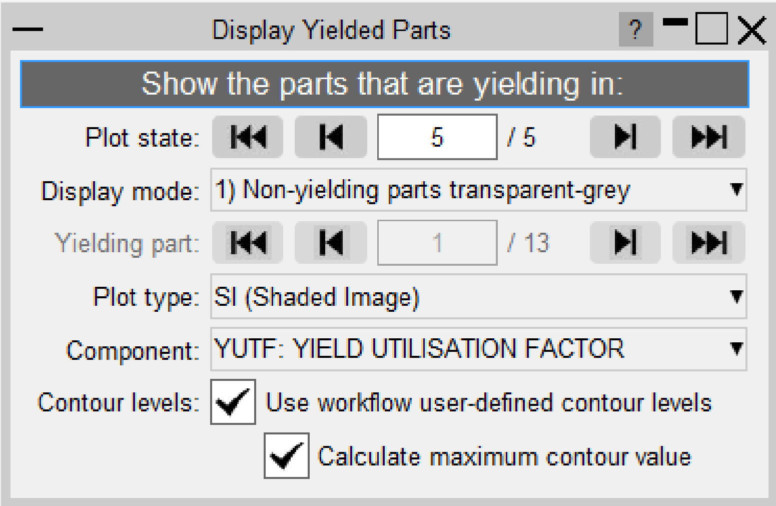
Setup in PRIMER

You don't need to set up anything in PRIMER to use the Strength Check Workflow, but you do need to make sure that you have generated a ZTF file to accompany your LS-DYNA results. D3PLOT will read the ZTF file along with the d3plot/PTF files. The ZTF file contains materials data that D3PLOT needs to determine the yield strength of the parts in your model.

Use in D3PLOT

Open the Strength Check Workflow in D3PLOT (**Tools → Workflows → Strength Check**) to visualise yielding parts for any set of results with a ZTF file.

When you open Strength Check, D3PLOT will perform a **SI (Shaded Image)** plot of the **YUTF: Yield Utilisation Factor** component of the model's final plot state with all non-yielding parts shown in transparent-grey. A menu will appear with further controls:



Plot state

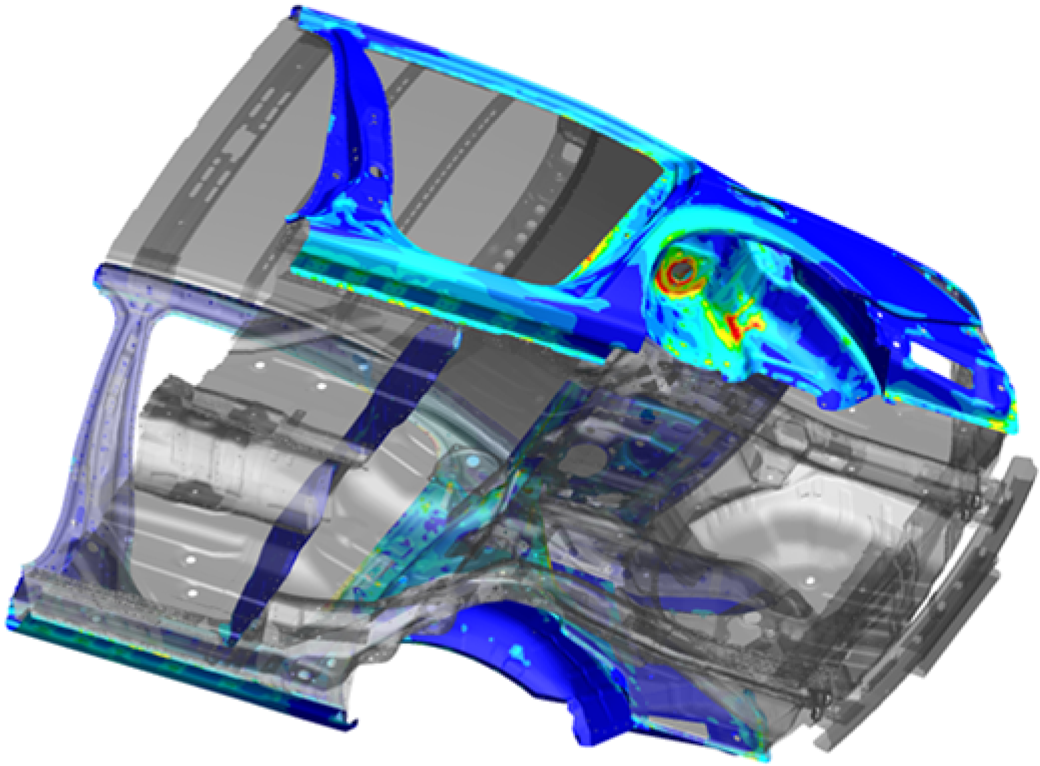
You can display the intrusion contour plot at any state. Use the controls in the menu to change plot state, rather than D3PLOT's main controls.

Display mode

This tool has three display modes:

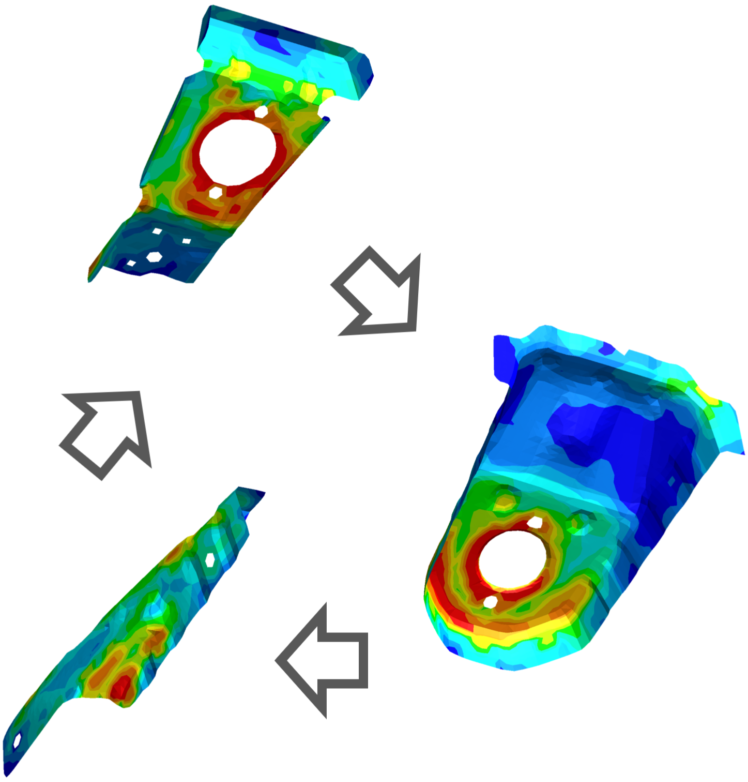
1. Non-yielding parts transparent-grey (default)

A CT (Continuous Tone) or SI (Shaded Image) contour plot of the YUTF/YUTP component of the model with the non-yielding parts displayed as transparent-grey:



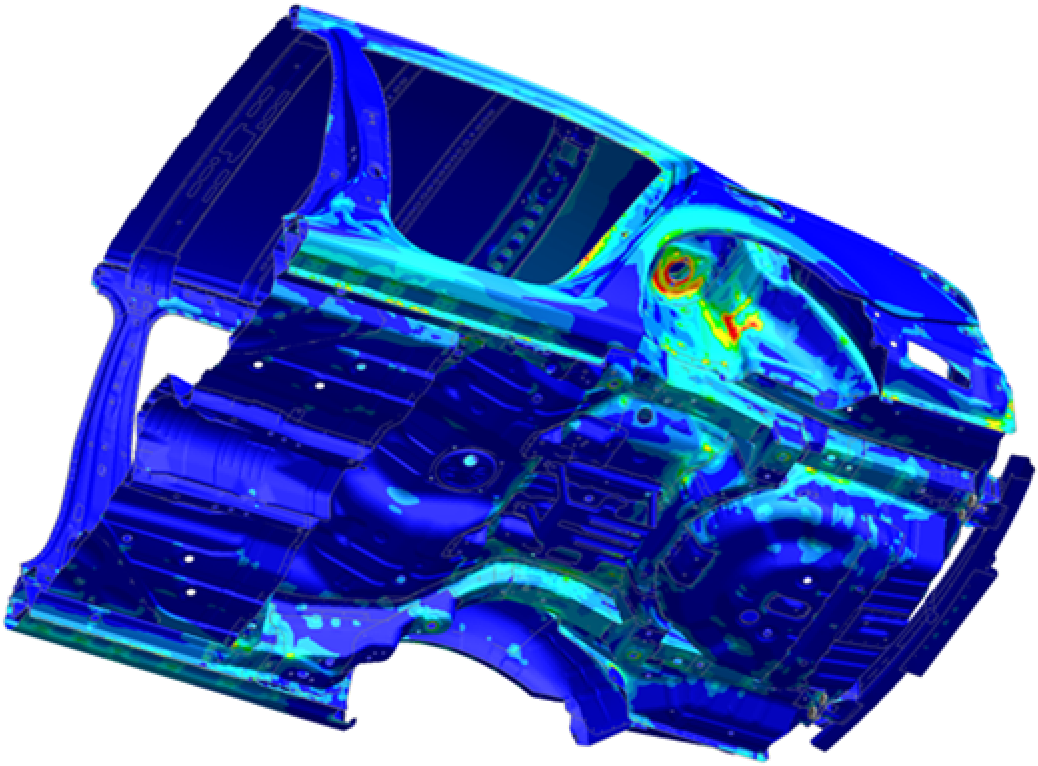
2. Cycle through yielding parts

A CT/SI plot of the YUTF/YUTP component of the specified yielding part. Cycle through yielding parts using the controls provided.



3. Plot of entire model

A normal CT/SI plot of the YUTF/YUTP component of the entire model:



Plot type

Choose between a SI (Shaded Image) contour plot (default) or a CT (Continuous Tone) contour plot.

Component

Choose to plot either the YUTF: Yield Utilisation Factor data component (default) or the YUTP: Yield Utilisation Percentage data component.

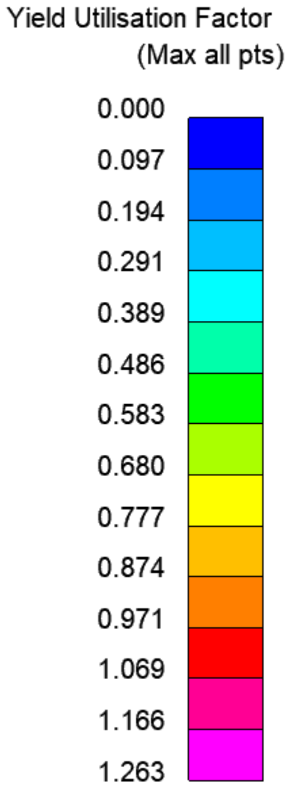
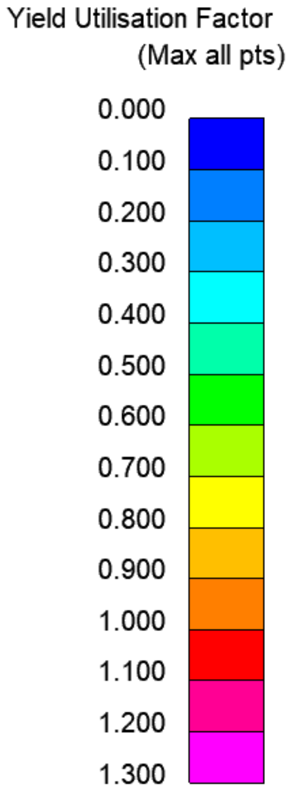
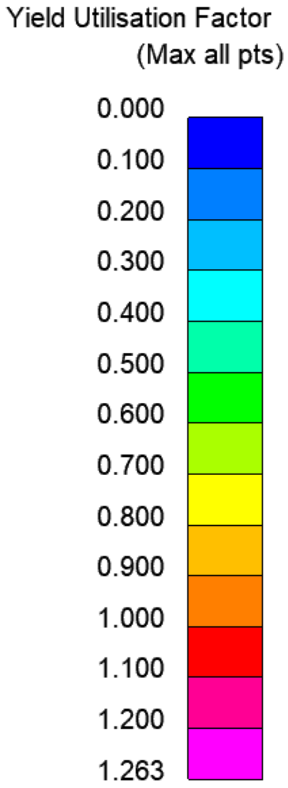
Contour Levels

There are several options regarding the contour levels:

- **Use workflow user-defined contour levels**
  To better visualise yielding parts, this option is enabled by default. The contour bar has values from 0.0 to 1.0 (blue to dark orange) in increments of 0.1, and then three further contour levels in red and magenta, so that all yielding material is clearly indicated in red/magenta. If this option is unchecked, automatic contour levels will be used.
  - **Calculate maximum contour value**
    With the above option enabled then by default, the actual maximum data value will be calculated for the maximum contour level. For example, if

the maximum yield utilisation factor is 1.263 then the contour levels above 1.0 will be 1.1, 1.2 and 1.263. This makes the maximum yield utilisation clearer. You can uncheck this option to speed up the plot (skips the maximum value calculation). In this case, the contour levels above 1.0 will be fixed to 1.1, 1.2 and 1.3.

Examples for different contour level settings

Non-user-defined	**User-defined without maximum calculation**	**User-defined with maximum calculation (Default)**
		

Properties

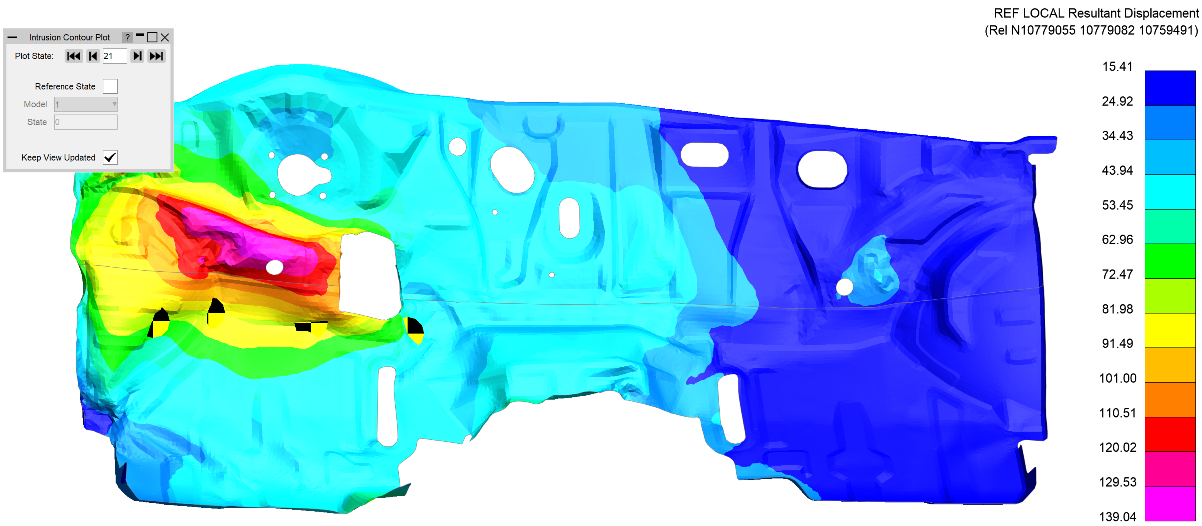
When Strength Check is opened, a temporary properties file is saved. When you exit the tool, you can choose to restore the model properties (view, blanking, colours, etc.) to their appearance before you opened the tool.

19.8. Intrusion Contour Plot

Intrusion Contour Plot

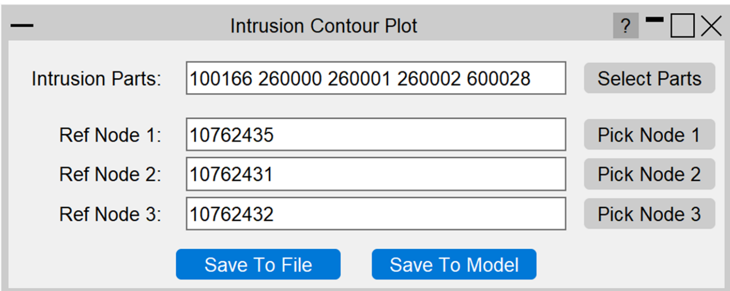
Tools → Workflows → Intrusion Contour Plot

The Intrusion Contour Plot tool creates a contour plot of intrusion displacements for selected parts, relative to specified reference coordinates:



Setup in PRIMER

In PRIMER, open Intrusion Contour Plot from the Workflows menu (**Tools → Workflows → Intrusion Contour Plot**). In the menu that appears, select intrusion parts and reference nodes , and then save the data to a Workflows .json file or add the data to your model in PRIMER and then write the keyword file:



Intrusion Parts

Select which parts you wish to include in the intrusion plot. The specified parts will be unblanked in D3PLOT by default, and the camera will adjust to point at the selected parts.

Reference Nodes

Select three reference nodes that will be used to define a reference coordinate system and for setting up the camera in D3PLOT. The intrusion contours are calculated relative to this reference system.

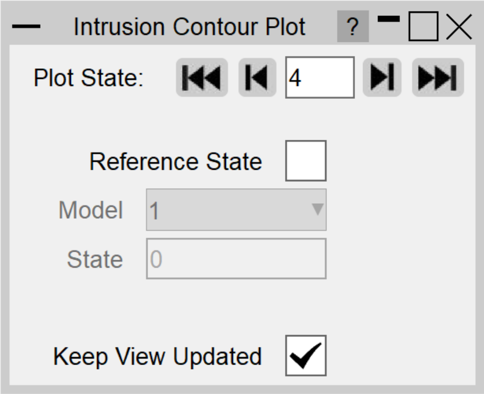
Saving

You can either save the setup data to a .json file or directly to the model. The user data from the file will then be picked up when the Workflow is selected in D3PLOT.

Use in D3PLOT

When you open Intrusion Contour Plot in D3PLOT, the plot will appear immediately. A menu will appear giving you further controls:

- **Plot State**
  You can display the intrusion contour plot at any state. Use the controls in the menu to change plot state, rather than D3PLOT's main controls.
- **Reference State**
  You can adjust the plot to show intrusion relative to a reference state (and, when using with multiple models, from a specified model rather than from the model itself).
- **Keep View Updated**
  If the checkbox is ticked then each time a state or reference state change is made, the camera resets. Uncheck this option if you wish to control the view manually.


Intrusion Contour Plot
Plot State:
4
Reference State
Model
1
State
0
Keep View Updated

19.9. Pulse Index Tool

Pulse Index

Tools → Workflows → Pulse Index

During the early stages of vehicle development, it can be useful to understand occupant acceleration without needing to include a complex and computationally expensive occupant model.

The Pulse Index Workflow allows you to estimate the acceleration that would be experienced by a vehicle occupant in a crash test scenario.

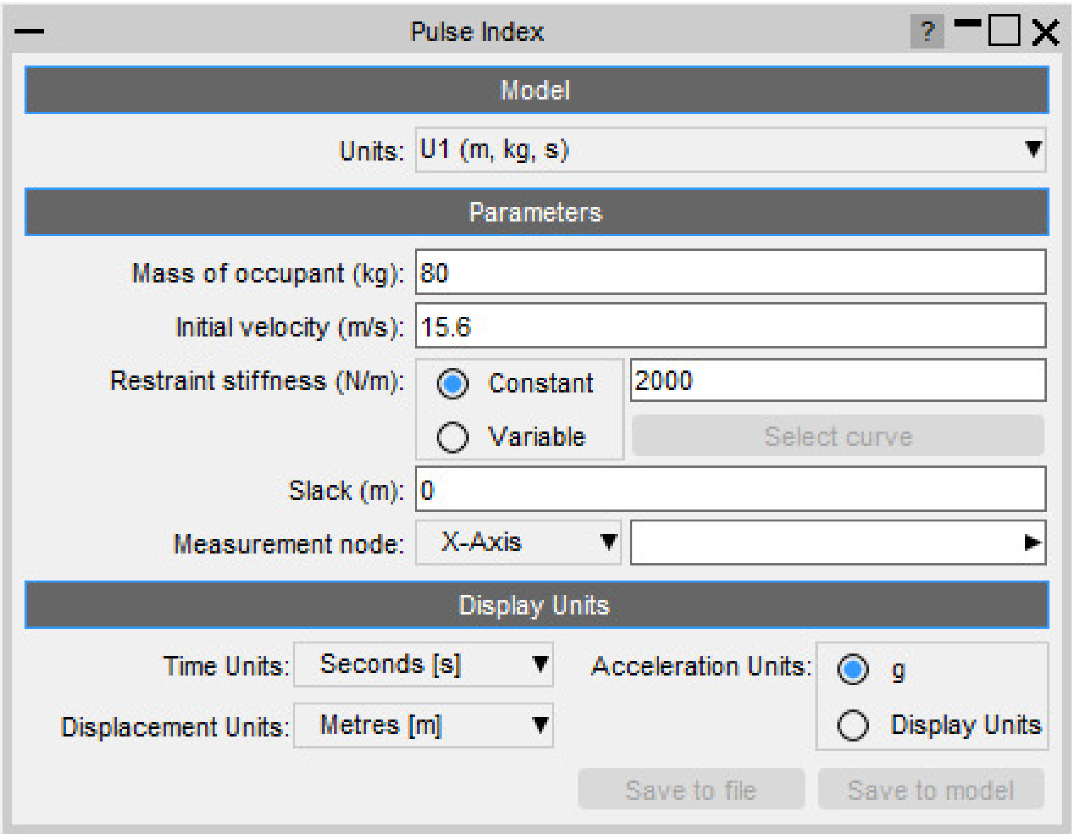
The tool assumes a virtual single-degree-of-freedom mass-spring system in which the occupant is represented by a **mass** and the seatbelt is represented by a **spring**. See below for more details of the pulse index calculation.

This virtual mass-spring system is effectively attached to a selected **node** moving with a set **initial velocity**.

The tool optionally takes a **slack** input to account for seatbelt engagement.

How to use the Pulse Index tool in PRIMER

Access the Pulse Index tool from the Workflows menu in PRIMER. Upon selecting the Pulse Index tool, the following menu will appear:



Units System

Select the appropriate units system for your model. All of the input **Parameters** will be expecting an input in the selected units system. When the selected unit system is changed, all existing inputs are automatically converted to the new units system.

Mass of occupant

Enter a mass (must be a non-zero positive value) for the virtual vehicle occupant. (This mass will not be added to the model mass and so will not have an effect on the LS-DYNA results.)

Initial velocity

Enter the initial velocity of the vehicle.

Restraint stiffness

The Pulse Index can either be calculated with constant or variable restraint stiffness (must be a non-zero positive value). For **Constant** stiffness, enter a non-zero stiffness value. For **Variable** stiffness, select a *DEFINE_CURVE keyword that defines the variable stiffness in terms of the model units (you may need to create a new keyword before making the selection).

Slack (optional)

You can choose to add slack to the seatbelt. This option allows the virtual vehicle occupant to move freely for a specified distance before the restraint stiffness begins to take effect. Slack works with both constant and variable stiffness options.

Measurement node

Select a *DATABASE_HISTORY_NODE that will be used as the node on the vehicle structure to which the virtual single-degree-of-freedom mass-spring system will be attached.

Display Units

Select time (Seconds or Milliseconds) and displacement units (Metres, Millimetres or Feet) to use on the graphs using the dropdowns. For the acceleration graph use the radio buttons to calculate in g or the display units. Please note if the Unit System is changed, the Time and Displacement Units will default to the matching ones for the Unit System, for example changing to U5 would default the Time units to Seconds and the Displacement units to Feet.

Save

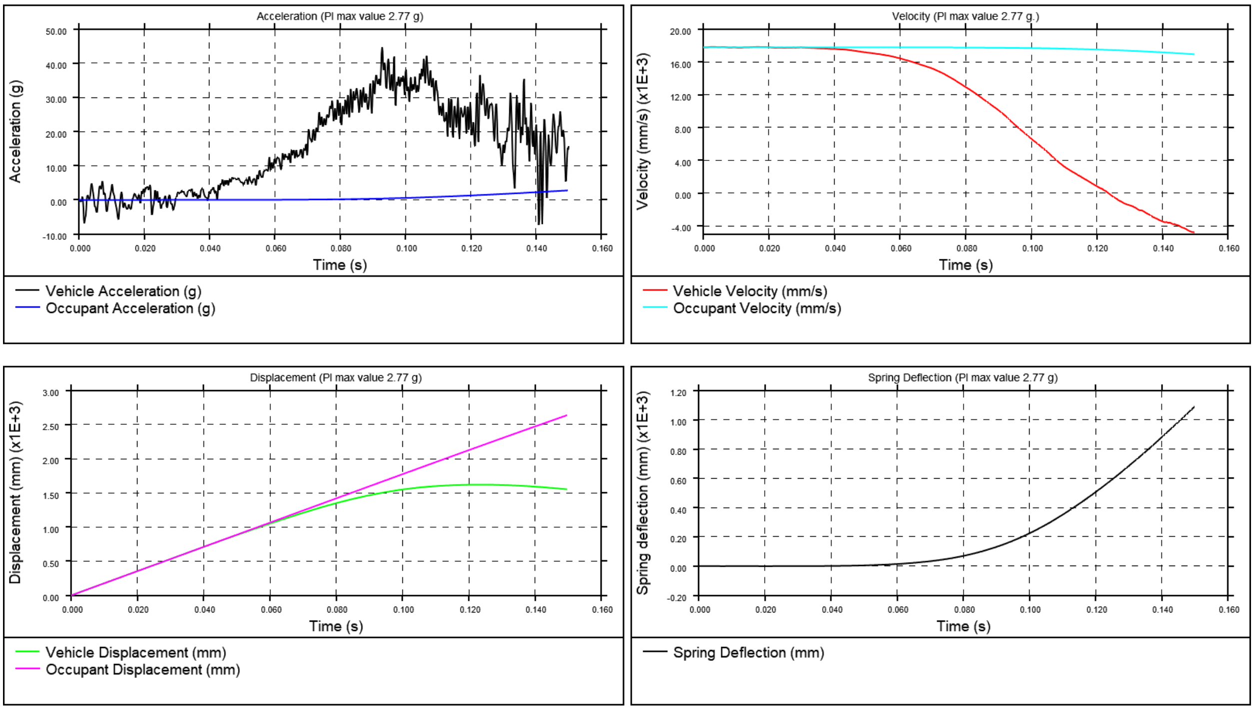
You can save the Workflow Definition to a .json file or to the model (as post-*END data). When saving to the model, you will need to write the model from PRIMER in order to save the changes to the keyword file.

How to use the Pulse Index tool in T/HIS

Access the Pulse Index tool from the Workflows menu in T/HIS. Upon selecting the Pulse Index tool, the same menu appears as in PRIMER. This allows you to adjust some of the input parameters when performing the pulse index calculation. The Measurement Node and Variable Stiffness settings can only be modified in PRIMER.

Calculate

If the input parameters are valid, the **Calculate** button will become active. Clicking **Calculate** executes the pulse index calculation and produces a four-graph layout comparing Vehicle and Virtual Occupant results, such as in this example:



Pulse Index Calculation

Initial Conditions

The vehicle and virtual occupant both start with zero initial displacement:

$$s_t^{veh} = s_t^{occ} = 0$$

The vehicle and the virtual occupant are both given the same initial velocity, defined by you in PRIMER. In PRIMER, you also specify the restraint system stiffness k and the measurement node for the vehicle acceleration a_t^{veh}.

Iterative Calculation

The Pulse Index Workflow performs an iterative calculation to determine the displacement s_t^{occ}, velocity v_t^{occ} and acceleration a_t^{occ} experienced by the virtual occupant over time. First, the vehicle's velocity v_{t+1}^{veh} and displacement s_{t+1}^{veh} at time $t+1$ are calculated from its acceleration a_t^{veh}:

$$v_{t+1}^{veh} = v_t^{veh} + a_t^{veh} * dt$$

$$s_{t+1}^{veh} = s_t^{veh} + v_t^{veh} * dt$$

Then the change in displacement dx_{eff} between the vehicle and virtual occupant is calculated:

$$dx = s_t^{occ} - s_t^{veh}$$

$$dx_{eff} = (|dx| - slack) \frac{dx}{|dx|}$$

The occupant acceleration is then calculated from the change in displacement, the occupant's mass m and the restraint stiffness k:

$$a_t^{occ} = \frac{k}{m}dx_{eff}$$

Finally, the occupant's resultant velocity and displacement are calculated, ready for the next time iteration:

$$v_{t+1}^{occ} = v_t^{occ} + a_t^{occ} * dt$$

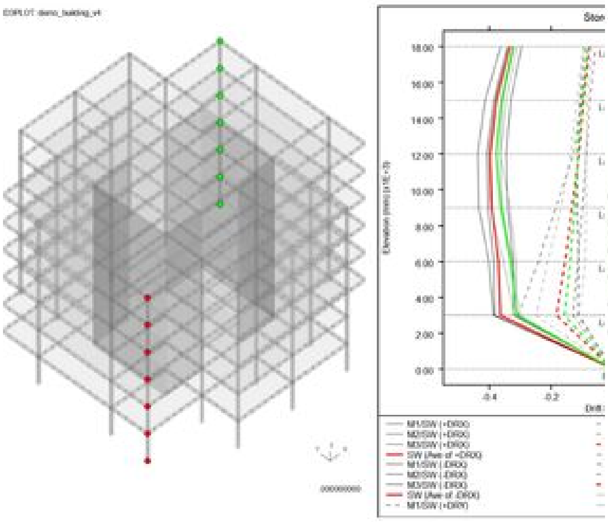
$$s_{t+1}^{occ} = s_t^{occ} + v_t^{occ} * dt$$

19.10. Seismic

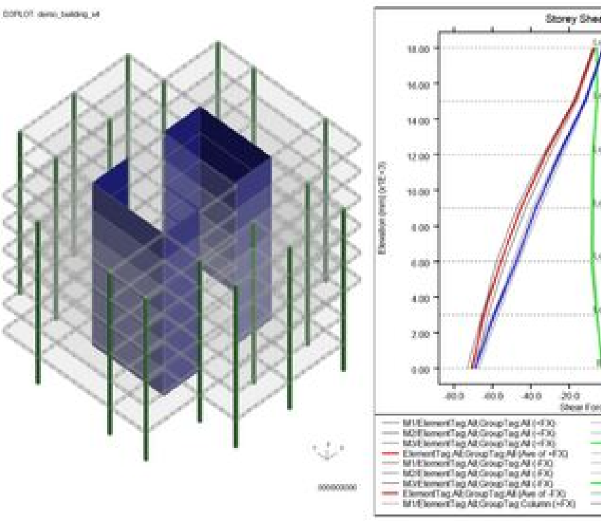
Seismic Workflows

These are a collection of workflows catered to help you interrogate the results of your seismic analysis and generate automated reports.

Currently, there are two workflows available for generating global structural results:



[Storey Drift]



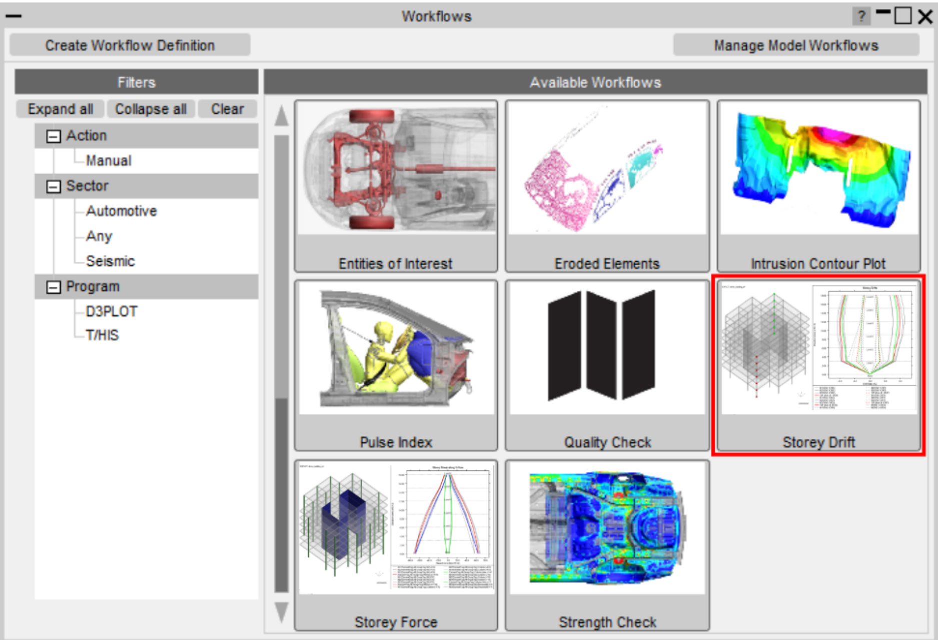
Storey Force

19.10.1. Storey Drift

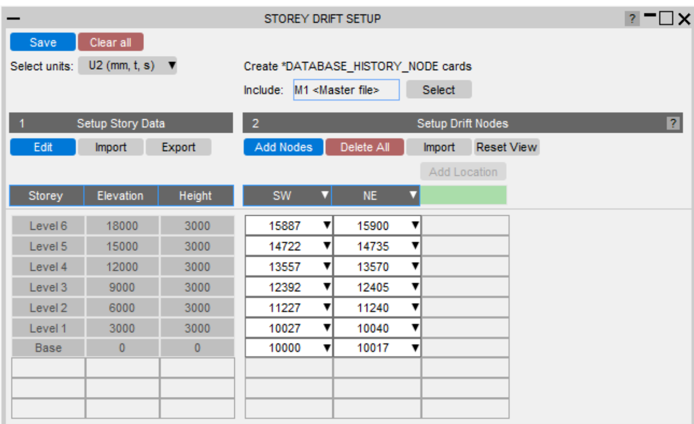
Storey Drift

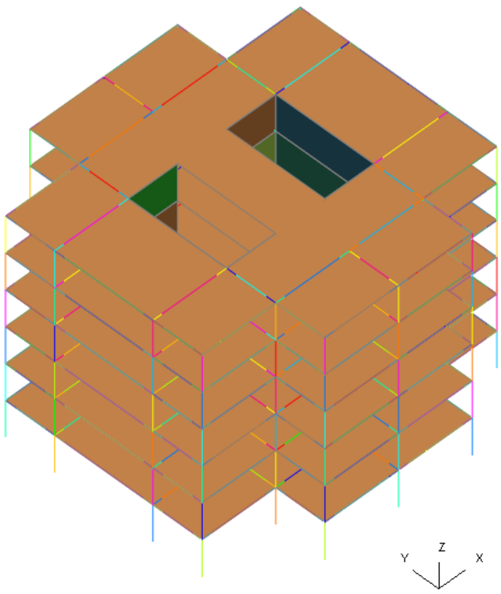
Tools → Workflows → Storey Drift

The Storey Drift workflow tool is used to post-process building drifts on various locations in the structure which can be used to check compliance against relevant building standards.



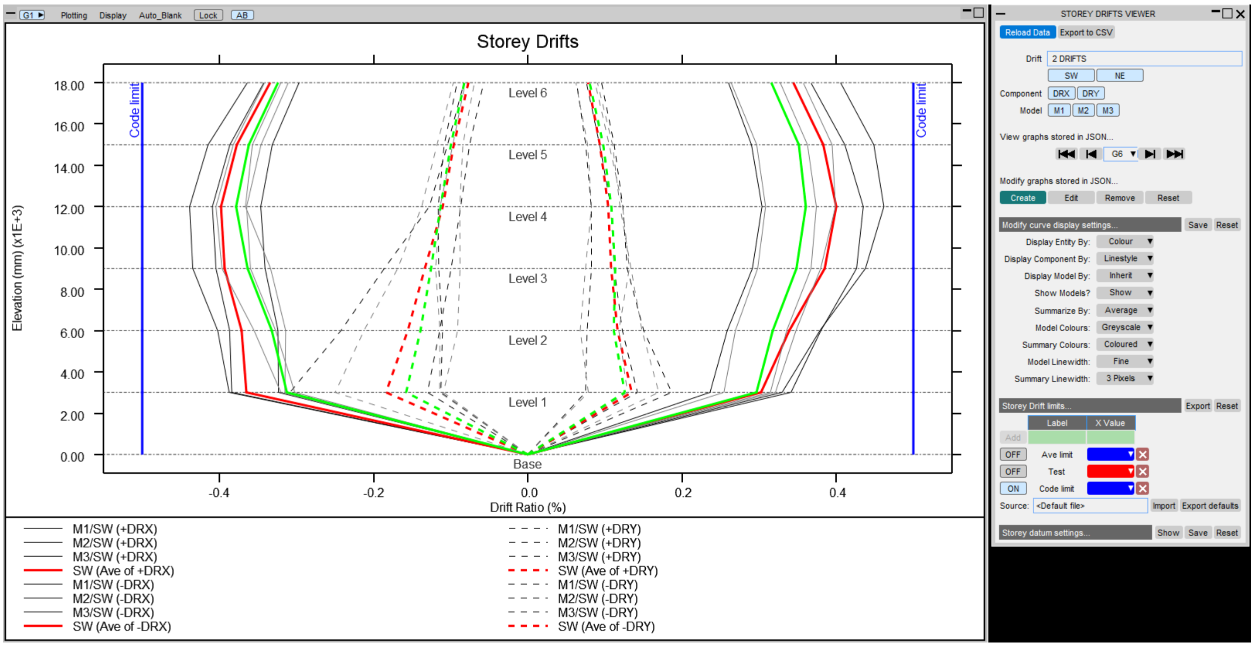
In PRIMER, you can setup drift locations, defining nodes for each storey of the building.





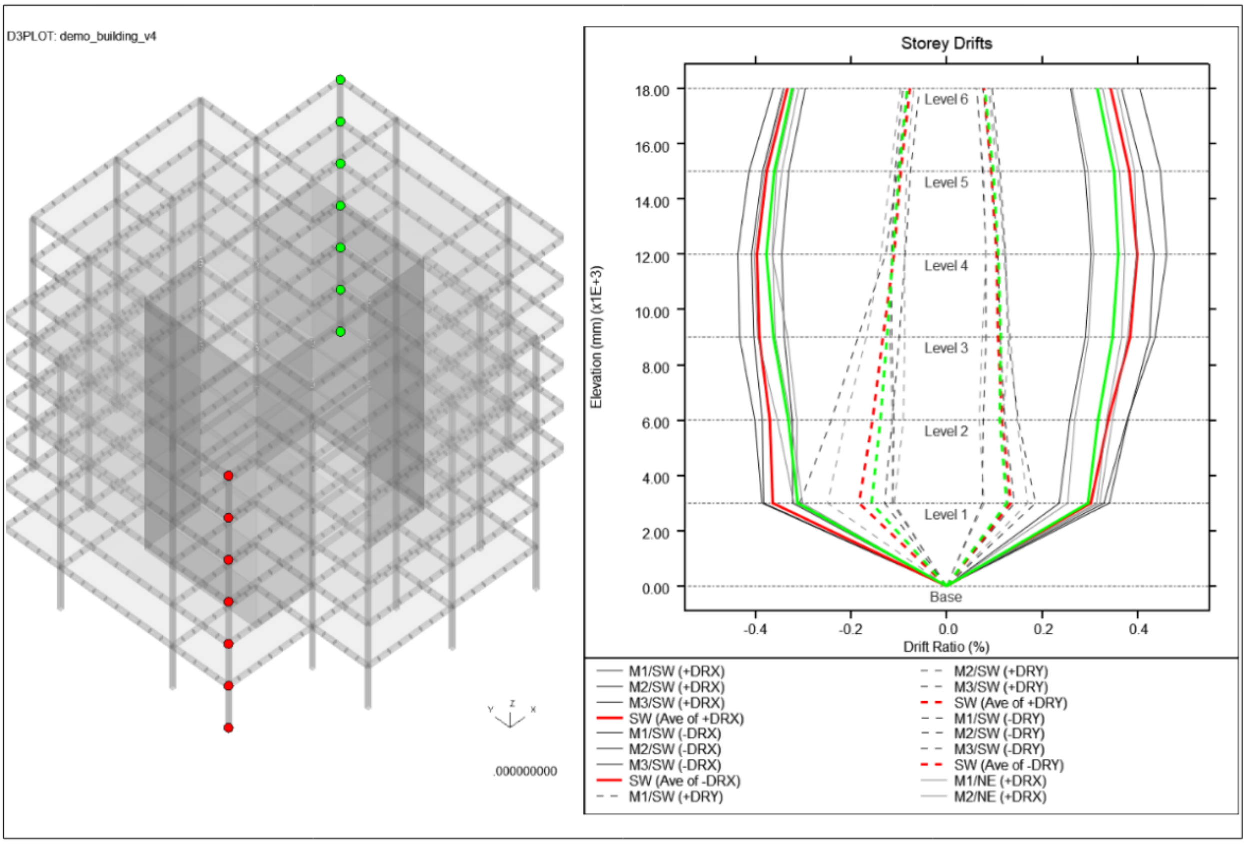
In T/HIS, storey drifts are calculated for each of the locations you defined in PRIMER and then storey curves are generated – plotted on graphs.

This allows you to interrogate the global behaviour of the structure and make changes to member designs or structural layout if necessary.



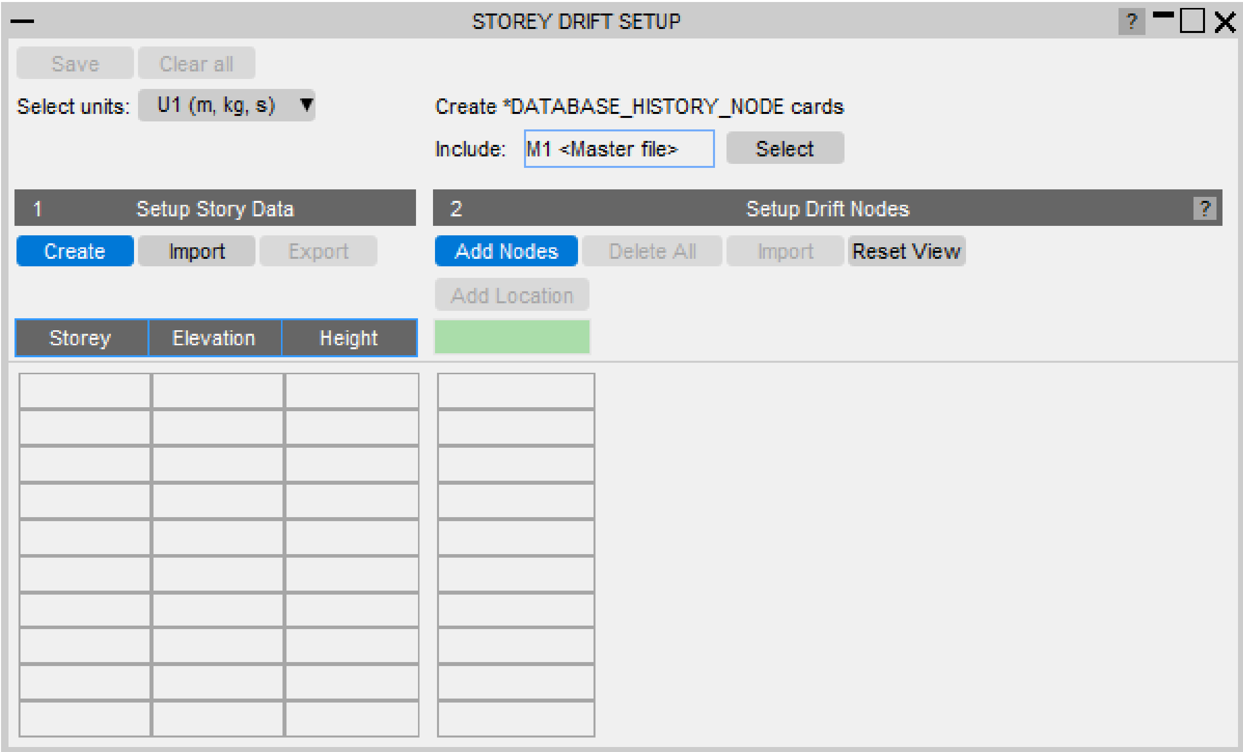
Finally, you can generate automated reports with the REPORTER templates provided.

In the report, corresponding D3PLOT views are paired with each T/HIS plot to visually locate the drifts in the model.

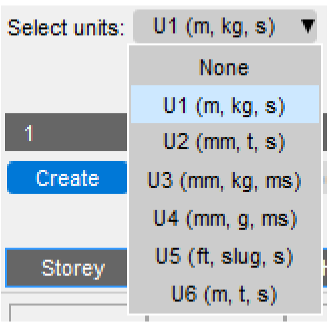

D3PLOT: demo_building_v4
.000000000
Storey Drifts
Elevation (mm) (x1E+3)
Drift Ratio (%)
Level 6
Level 5
Level 4
Level 3
Level 2
Level 1
Base
M1/SW (+DRX)
M2/SW (+DRX)
M3/SW (+DRX)
SW (Ave of +DRX)
M1/SW (-DRX)
M2/SW (-DRX)
M3/SW (-DRX)
SW (Ave of -DRX)
M1/SW (+DRY)
M2/SW (+DRY)
M3/SW (+DRY)
SW (Ave of +DRY)
M1/SW (-DRY)
M2/SW (-DRY)
M3/SW (-DRY)
SW (Ave of -DRY)
M1/NE (+DRX)
M2/NE (+DRX)

Storey Drift Setup

When the tool is launched in PRIMER, a window appears for you to set up the drift definitions you wish to process:

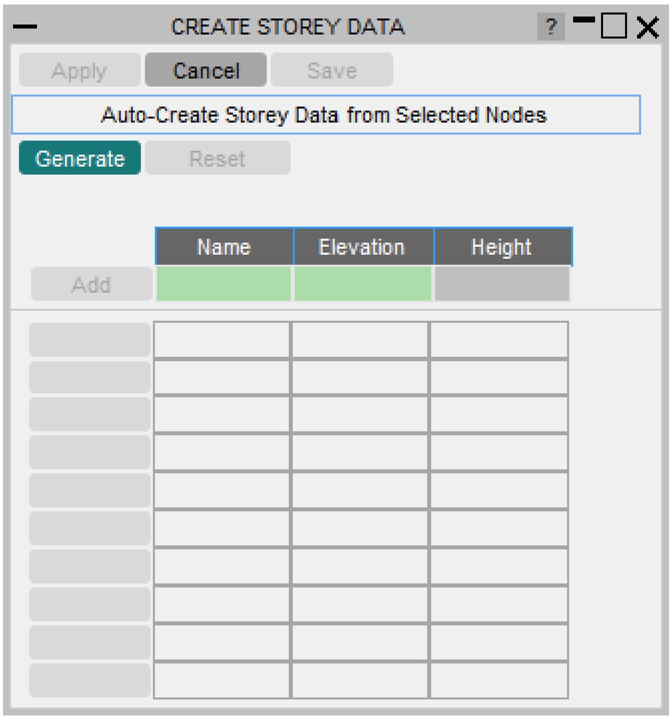


First, you need to choose the appropriate unit system from the dropdown menu:



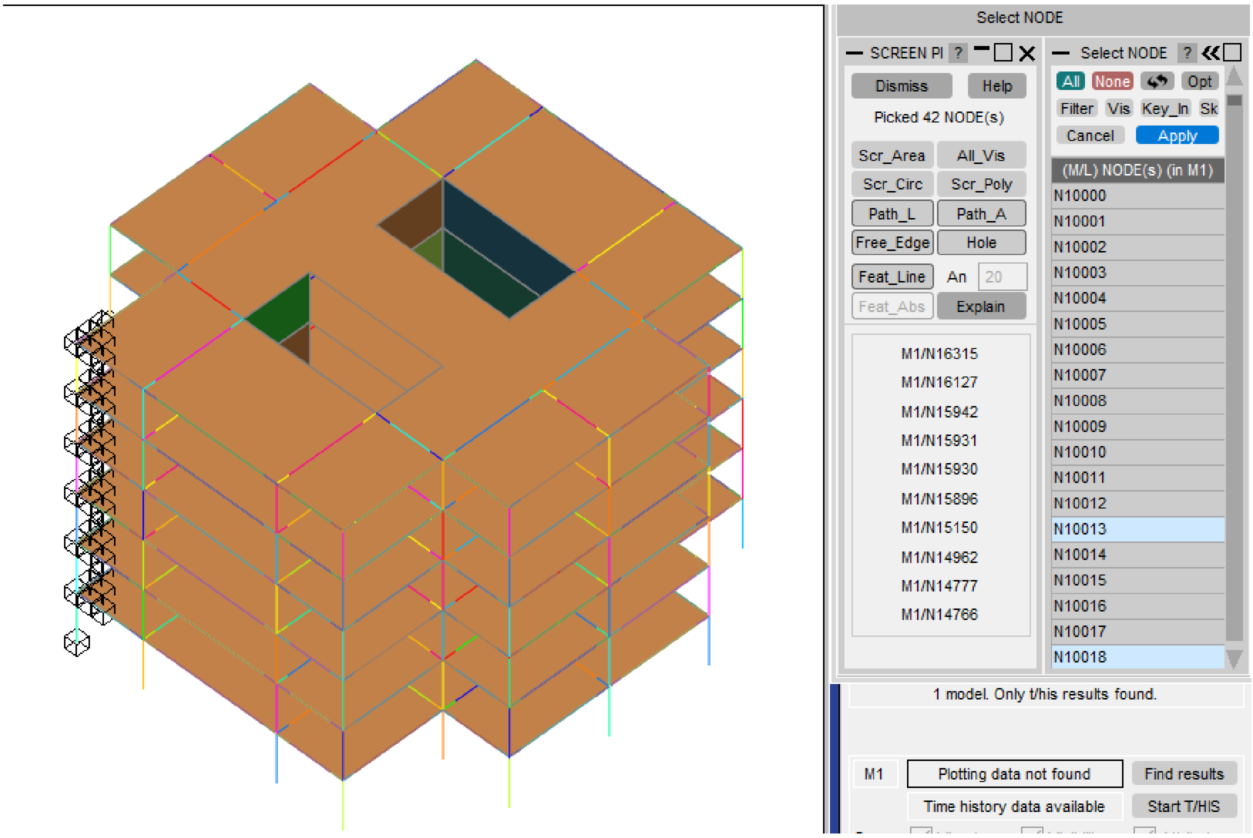
Defining Storey Data

You can define the storey data for the structure either by clicking the **Create** button or the **Import** button under the Setup Storey Data section. **Import** allows you to import previously saved storey definitions (e.g. those created for the Storey Force workflow). When you click **Create**, the Storey Data window appears:



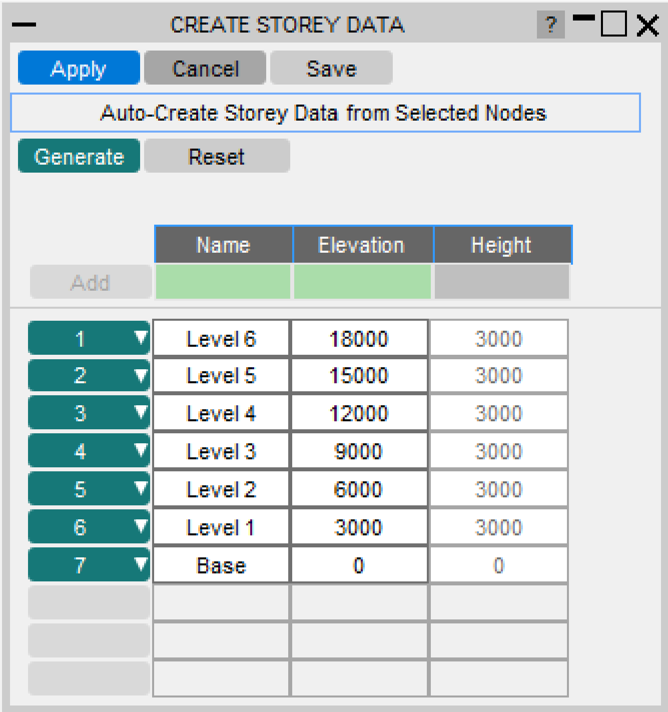
Each storey can be defined manually by populating **Name** and **Elevation** textboxes and then clicking the **Add** button which will be activated if the inputs are valid.

Alternatively, you can define multiple storeys automatically by clicking **Generate**. You will be prompted to select nodes in the model. This will then generate storeys for each unique elevation (z-coordinate) among the nodes you have selected. Finally, you can then modify the labels of each generated storey to be more informative for your project.



Click **Apply** to import the storey data back to the main setup window.

You can optionally save this data by clicking **Save**. This will write it to a separate JSON file, which you can **Import** when you are starting a new setup. Normally, storey data would be applicable to multiple Seismic workflows, so saving this data will be useful to those other workflows too.

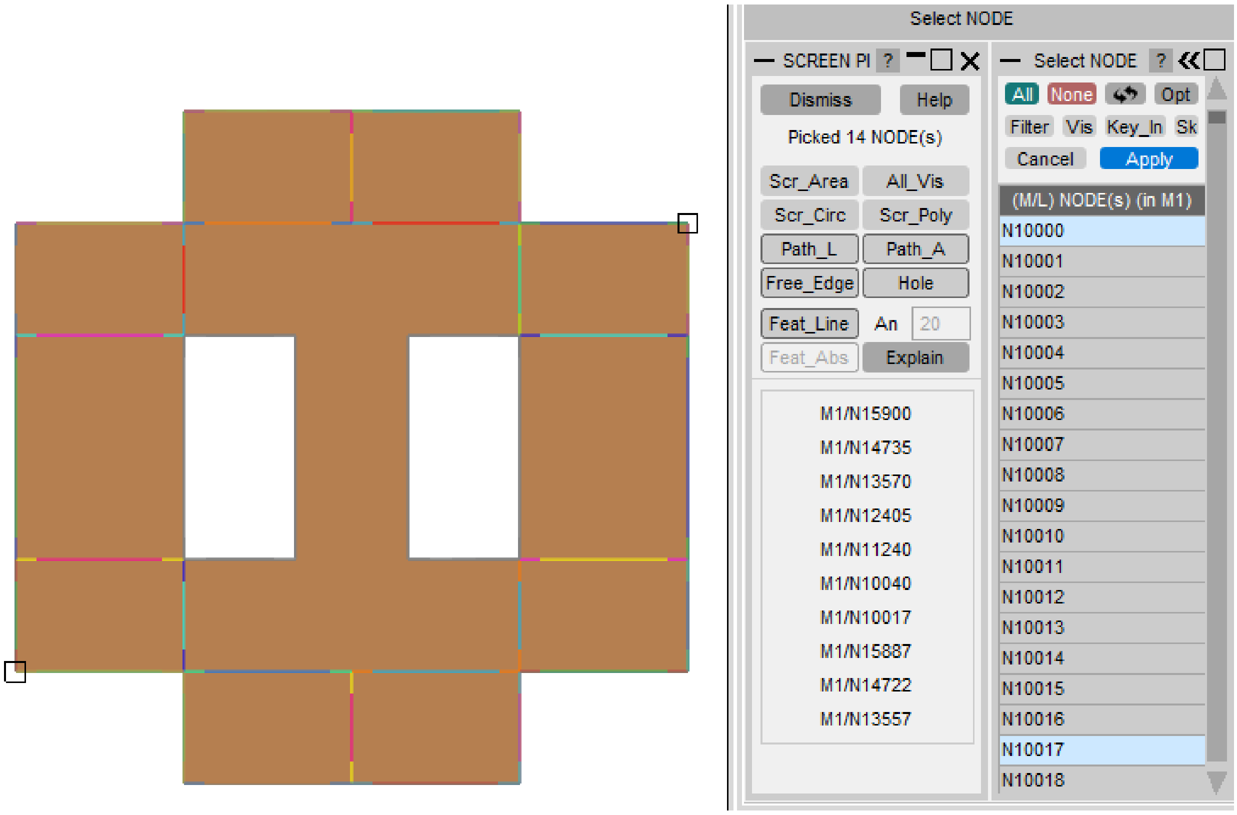


Defining Drift Locations

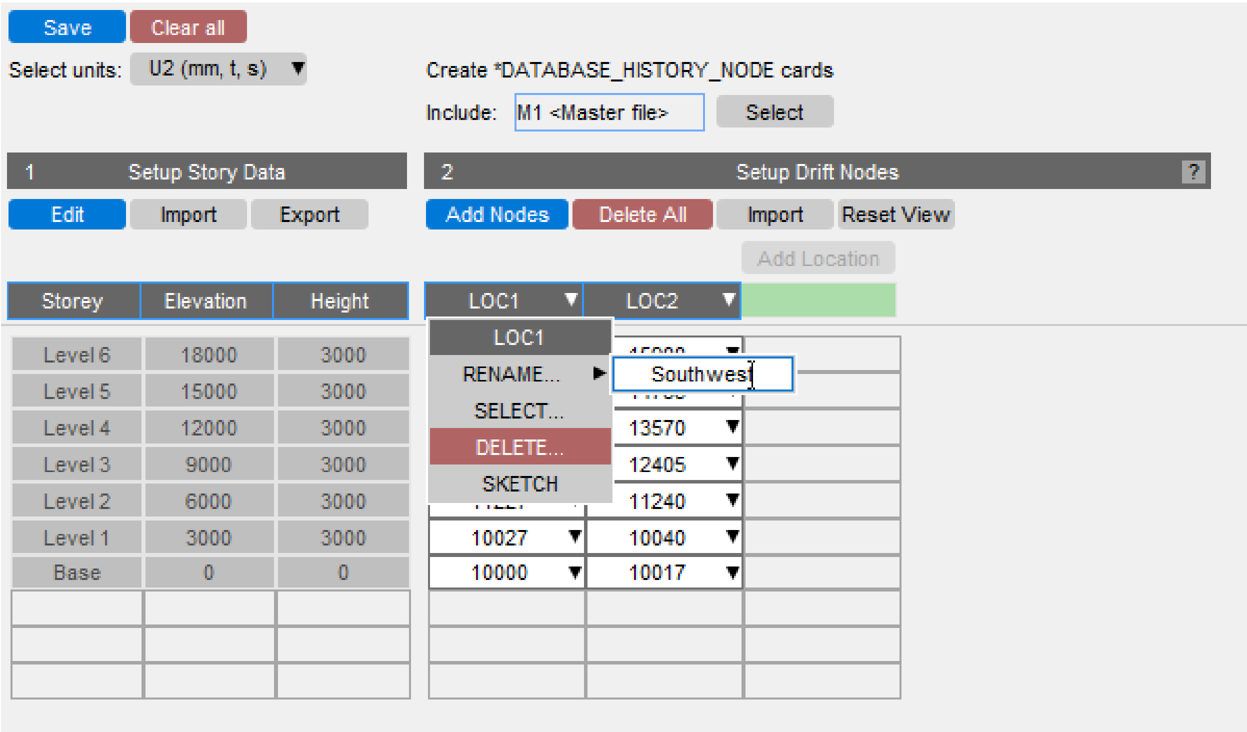
There are two ways to define drift locations.

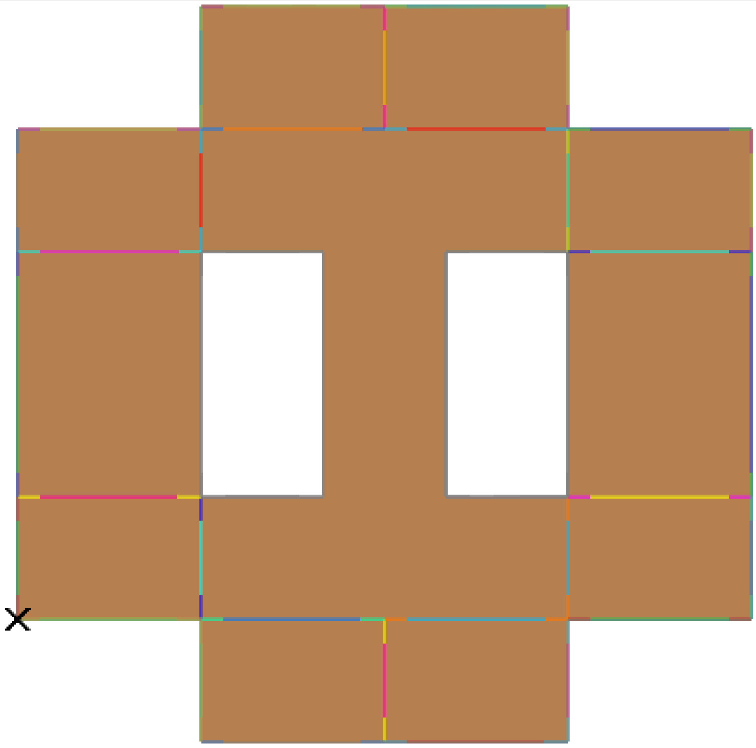
1. Create several at once using **Add Nodes**.
2. Create one at a time by defining the drift label in the column header text box and then clicking **Add Location**.

You can define multiple drift nodes at once by clicking **Add Nodes**. You will then be prompted to select nodes in the model. It is recommended to select nodes in plan view to do this quickly:

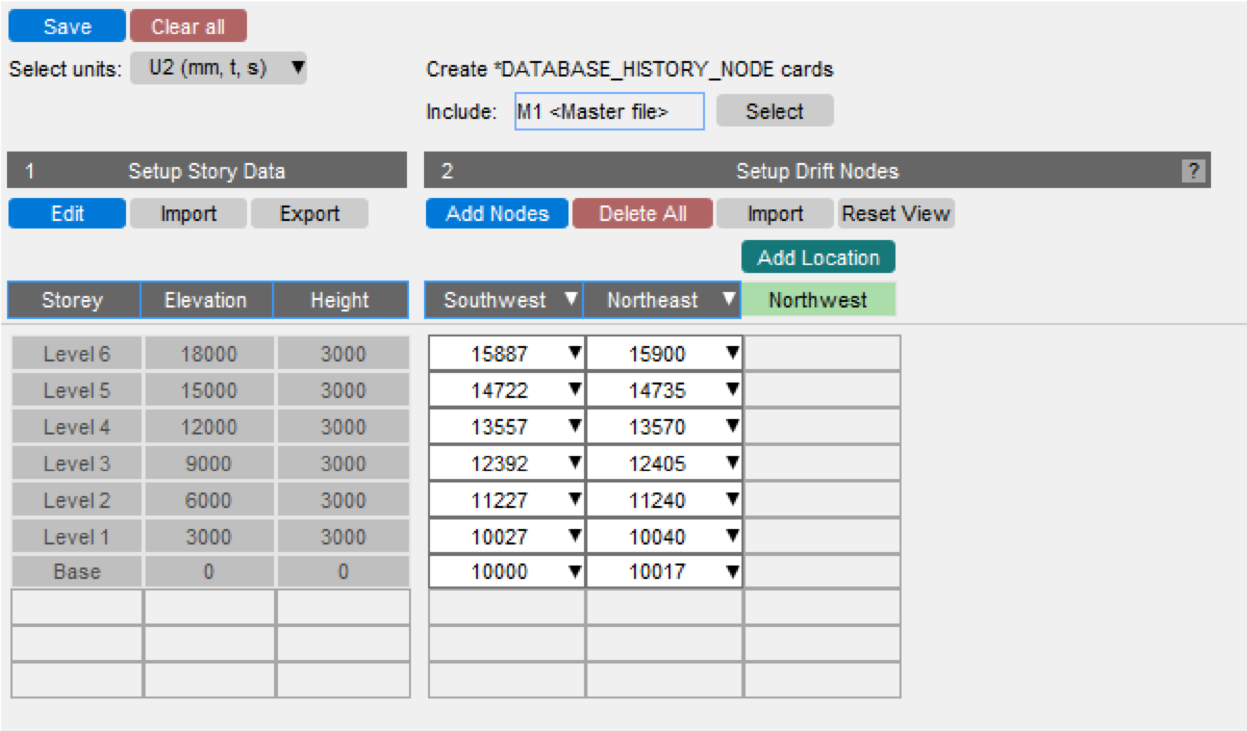


The drifts will be assigned with default labels. By right-clicking the drift header, you can rename the drift with a more informative label, as shown below. You can click **Sketch** in this popup menu to locate the drift nodes in the model, helping you to define an appropriate drift label. You can also redefine new drift nodes for an existing drift via **Select**, and even delete a current drift via **Delete**.



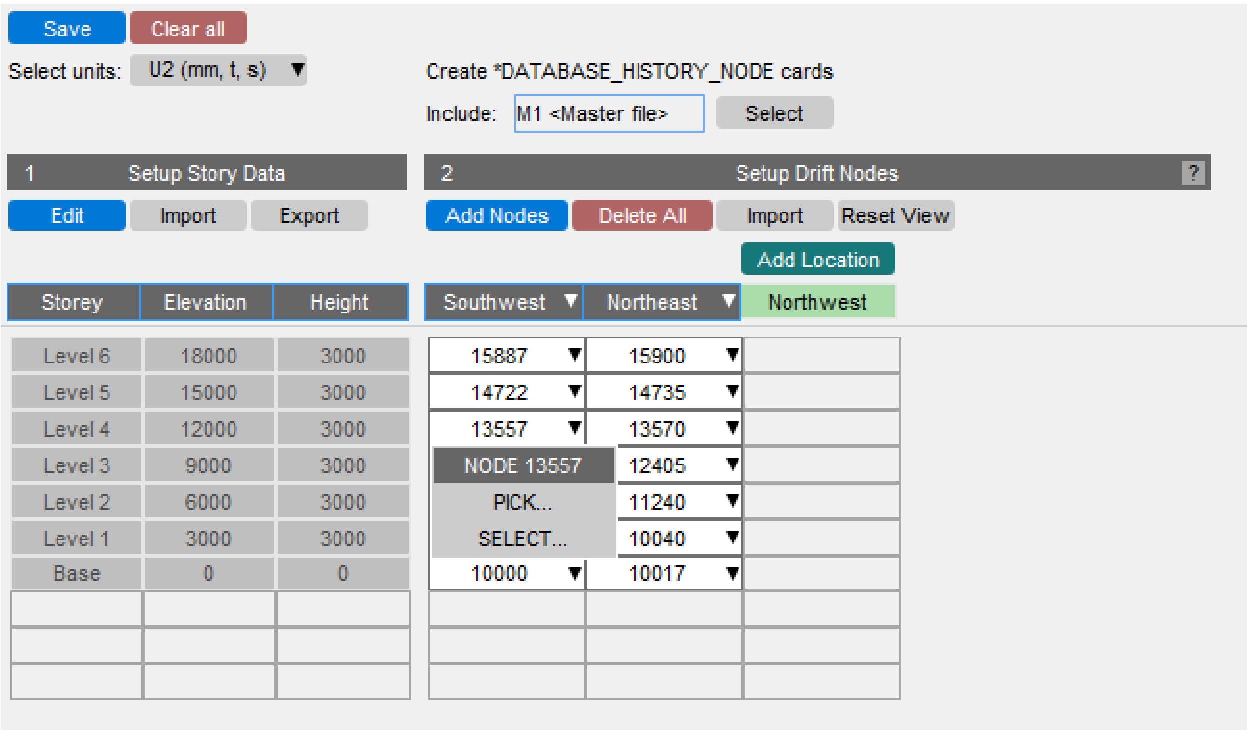


To add drift locations individually, define the drift label in the column header text box and click **Add Location**. A new blank drift column will be added to the table. You can then add nodes by right-clicking the drift header and then clicking **Select**.



You may wish to update specific nodes on each drift manually. To do this, right-click the desired drift node on the table and use either **Pick** or **Select** in the popup menu.

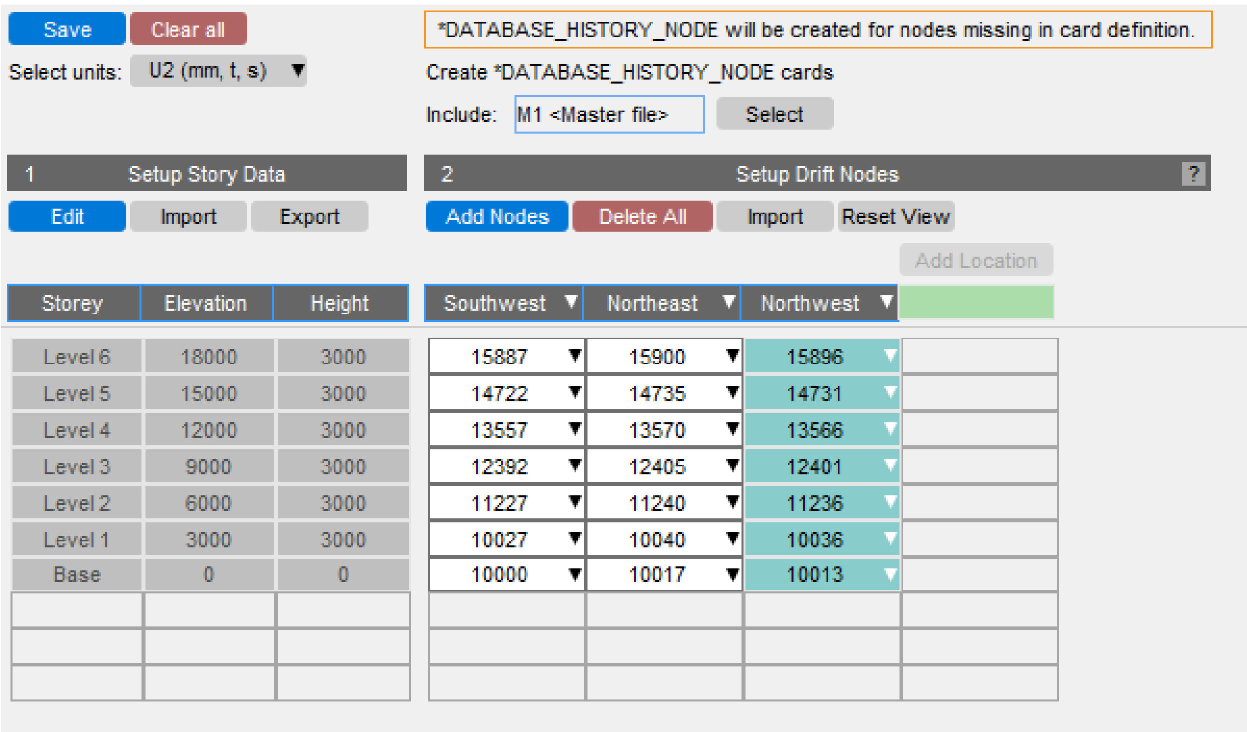
To delete a node for a particular storey in the drift, just delete the contents of the cell in the table.



Writing the Workflow File

Once all data has been defined, save the drift setup by clicking **Save**. This will write a Workflow file in JSON format. This file will be used to post-process the defined drifts in T/HIS and create a report in REPORTER.

The Storey Drift Workflow tool has been designed to be used on a sweep of LS-DYNA runs with different ground motions applied to the same model. It is advised to save the Workflow file in the parent folder (the folder containing several child folders, each containing one set of ground motion results). Currently, this Workflow will only work properly if only **one Workflow file exists** in the parent folder, including its child folders. If you save this file in the folder of an individual model, then there is a risk to duplicate the Workflow file, which might cause problems later. This will most probably happen when you duplicate the original model to create a new model with a different ground motion input.



Database history output

For this workflow, **DATABASE_HISTORY_NODE(s)** will be generated for each drift node. Remember to save the .key file and rerun the model if necessary. As shown above, some defined nodes will be latent (highlighted in light blue). This means that the DATABASE_HISTORY_NODE(s) do not exist in the model yet. You would need to rerun the model so the results will be available in T/HIS.

Before saving the drift setup, you may also wish to select an include file for the DATABASE_HISTORY_NODE(s). You can choose an include file by clicking **Select** above the Setup Drift Nodes header. The tool will add any DATABASE_HISTORY_NODE keywords created to your selected include file.

Resetting the data

To reset all data, click **Clear all** and start the whole process again to define a new drift setup. Alternatively, use **Delete All** under the Setup Drift Nodes section to reset only the drift nodes while retaining the storey data.

Importing existing Workflow Data

When an existing Workflow file is present in the root folder, the storey data and drift nodes are automatically imported when you run this Workflow.

After removing all data in a current session, you can import the storey data and the drift nodes by clicking the **Import** buttons on each sub-section. Storey data must be imported first before importing the drift nodes. Every node on each drift specified in the Workflow file are validated. If a node does not exist, it will be highlighted in the table, flagged as an error. For further details on importing storey data, please refer to the following section.

Importing existing Storey Data

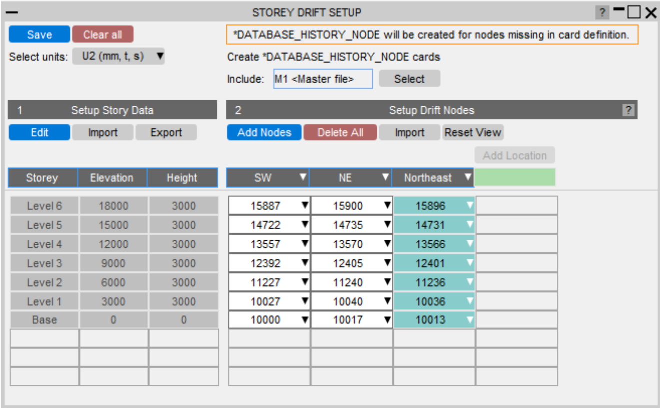
As mentioned on the section above, you can import pre-defined storey data to quickly define storeys. The storey data may exist in an **external JSON file** or in the **Workflow file**. If it is present, you will be prompted to use an existing Workflow file. If you **choose not to**, then a file selector popup will appear so you can select an external JSON file.

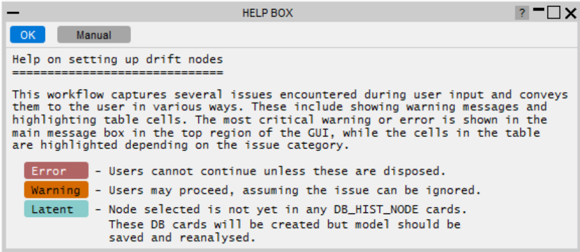
Dealing with input Errors/Warnings

You might encounter errors or warnings when populating the drift table.

If errors exists, the **Save** button will be disabled so you cannot proceed unless the errors are addressed. On the other hand, warnings will not disable the **Save** button so you may still proceed with caution. Make sure the warnings are expected and intended. For example, drift nodes on one drift location might sit on different XY coordinates. If the difference is outside the tool's tolerance, this tool will show you a warning. You may then proceed or update the selection.

The **most critical warnings and errors** will be shown at the top of Setup Drift Nodes section for your information. The cells related to input errors/warnings will be **colour-coded**. More details on these is available on the **Help** button on the right side of the Setup Drift Nodes section header.


STOREY DRIFT SETUP
Save
Clear all
*DATABASE_HISTORY_NODE will be created for nodes missing in card definition.
Select units:
U2 (mm, t, s)
Create *DATABASE_HISTORY_NODE cards
Include:
M1 <Master file>
Select
1 Setup Story Data
2 Setup Drift Nodes
Edit
Import
Export
Add Nodes
Delete All
Import
Reset View
Add Location
Storey
Elevation
Height
SW
NE
Northeast
Level 6 18000 3000 15887 15900 15896
Level 5 15000 3000 14722 14735 14731
Level 4 12000 3000 13557 13570 13566
Level 3 9000 3000 12392 12405 12401
Level 2 6000 3000 11227 11240 11236
Level 1 3000 3000 10027 10040 10036
Base 0 0 10000 10017 10013

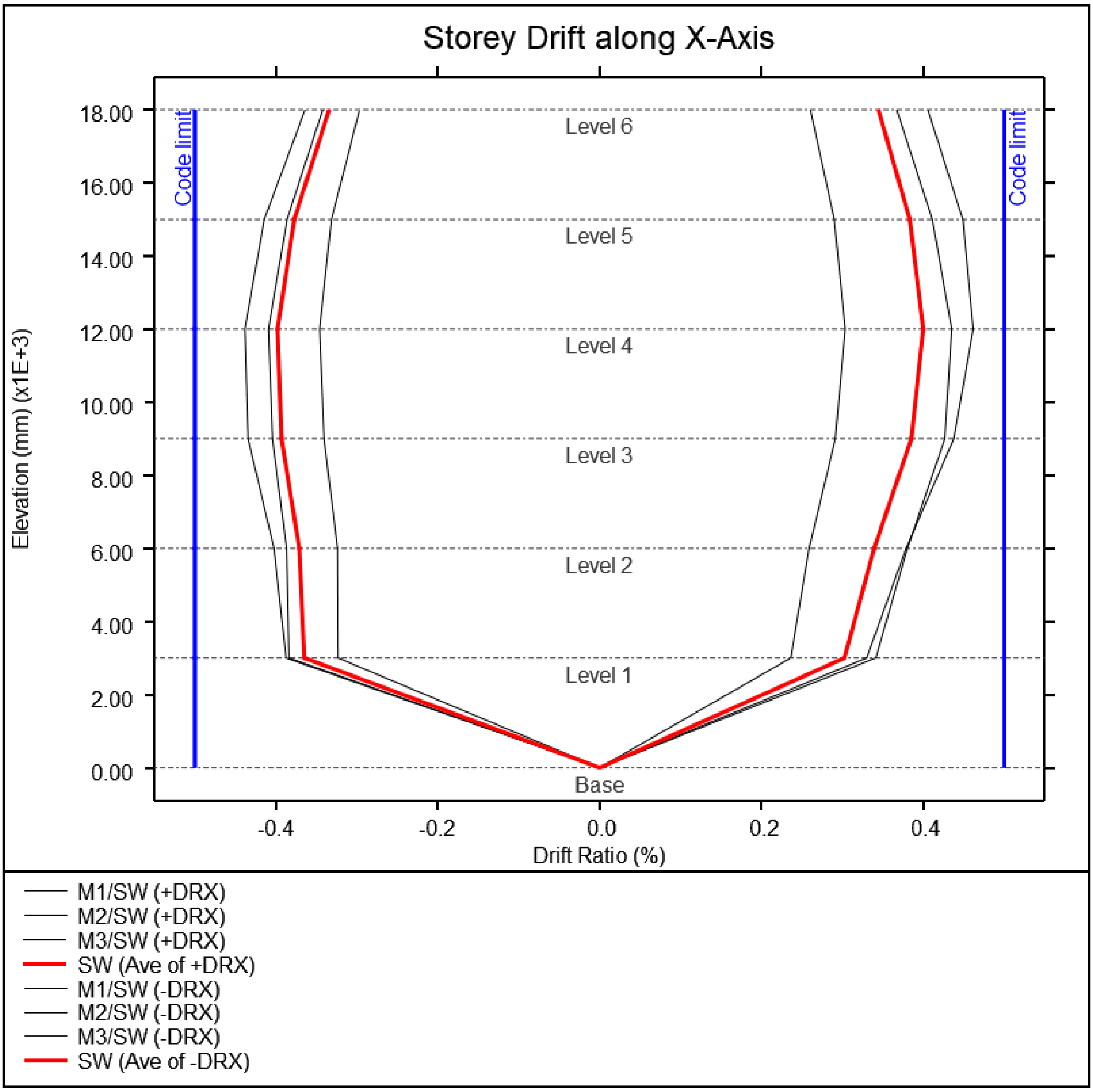

HELP BOX
OK
Manual
Help on setting up drift nodes
==============================
This workflow captures several issues encountered during user input and conveys them to the user in various ways. These include showing warning messages and highlighting table cells. The most critical warning or error is shown in the main message box in the top region of the GUI, while the cells in the table are highlighted depending on the issue category.
Error - Users cannot continue unless these are disposed.
Warning - Users may proceed, assuming the issue can be ignored.
Latent - Node selected is not yet in any DB_HIST_NODE cards. These DB cards will be created but model should be saved and reanalysed.

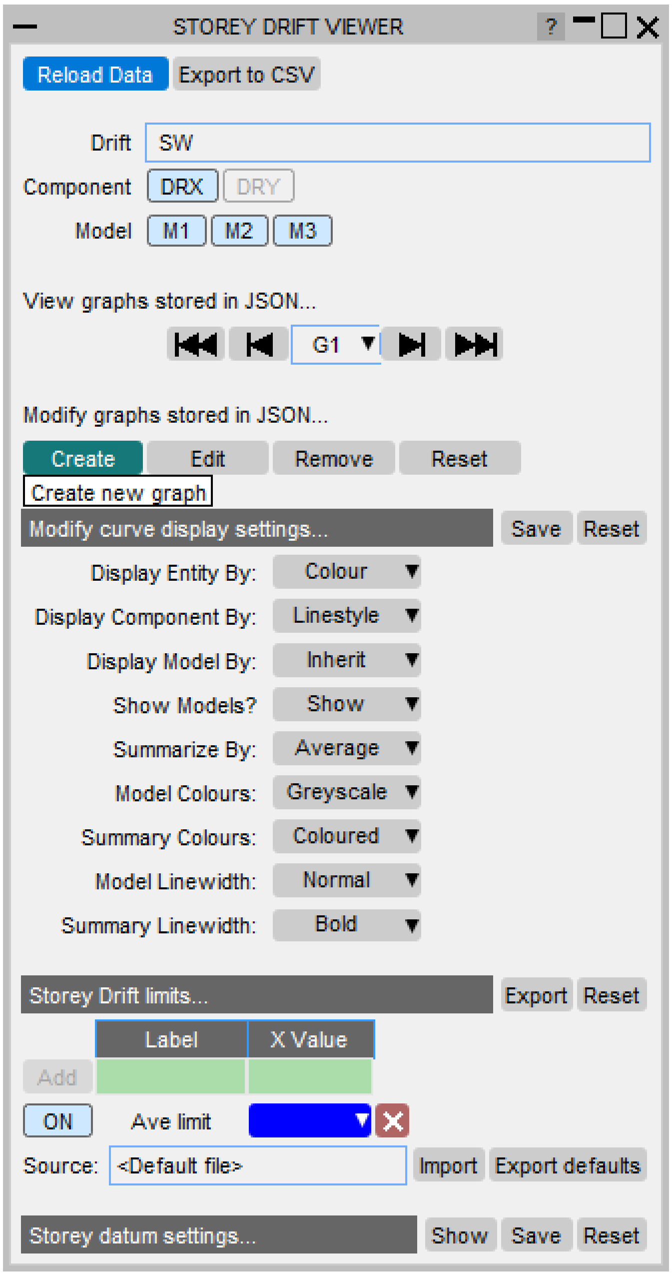
Storey Drift Viewer

When the tool is launched in T/HIS, the storey drift curves will be generated for each graph setup existing in the Workflow file. Then you will be presented with this window below.

When the Workflow file is initially created from PRIMER, default graph setups are included – one for each direction component, for each drift location defined. The storey drift curves will be created for each of these graph setups and the first graph setup will be plotted in T/HIS and will be active in the Viewer GUI:

T/HIS


Storey Drift along X-Axis
Elevation (mm) (x1E+3)
18.00
16.00
14.00
12.00
10.00
8.00
6.00
4.00
2.00
0.00
Code limit
Code limit
Level 6
Level 5
Level 4
Level 3
Level 2
Level 1
Base
-0.4
-0.2
0.0
0.2
0.4
Drift Ratio (%)
M1/SW (+DRX)
M2/SW (+DRX)
M3/SW (+DRX)
SW (Ave of +DRX)
M1/SW (-DRX)
M2/SW (-DRX)
M3/SW (-DRX)
SW (Ave of -DRX)

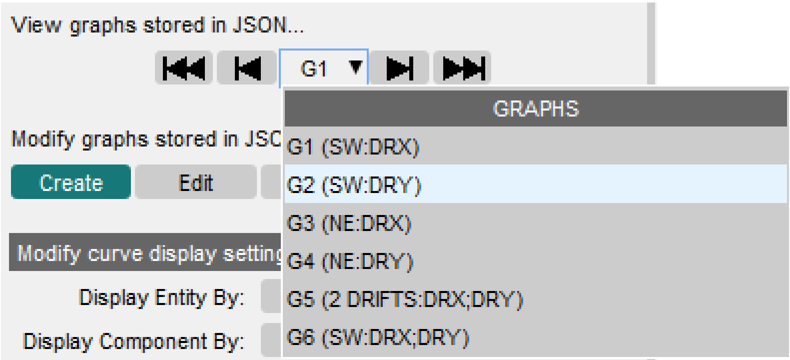


The Viewer GUI is generally split into four sections listed below:

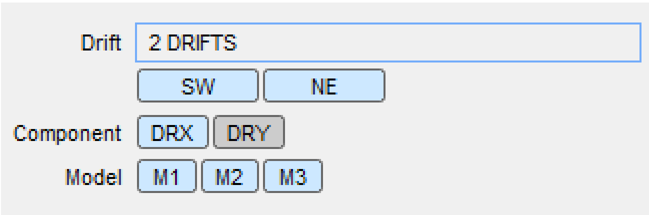
1. Graph selection/creation panel
2. Curve display settings
3. Storey drift limits definition
4. Storey datum settings

Graph selection/creation panel

This panel allows you to cycle through the graphs you have generated. You can use the navigation buttons to view the graphs sequentially or you can select a graph from the combo box.



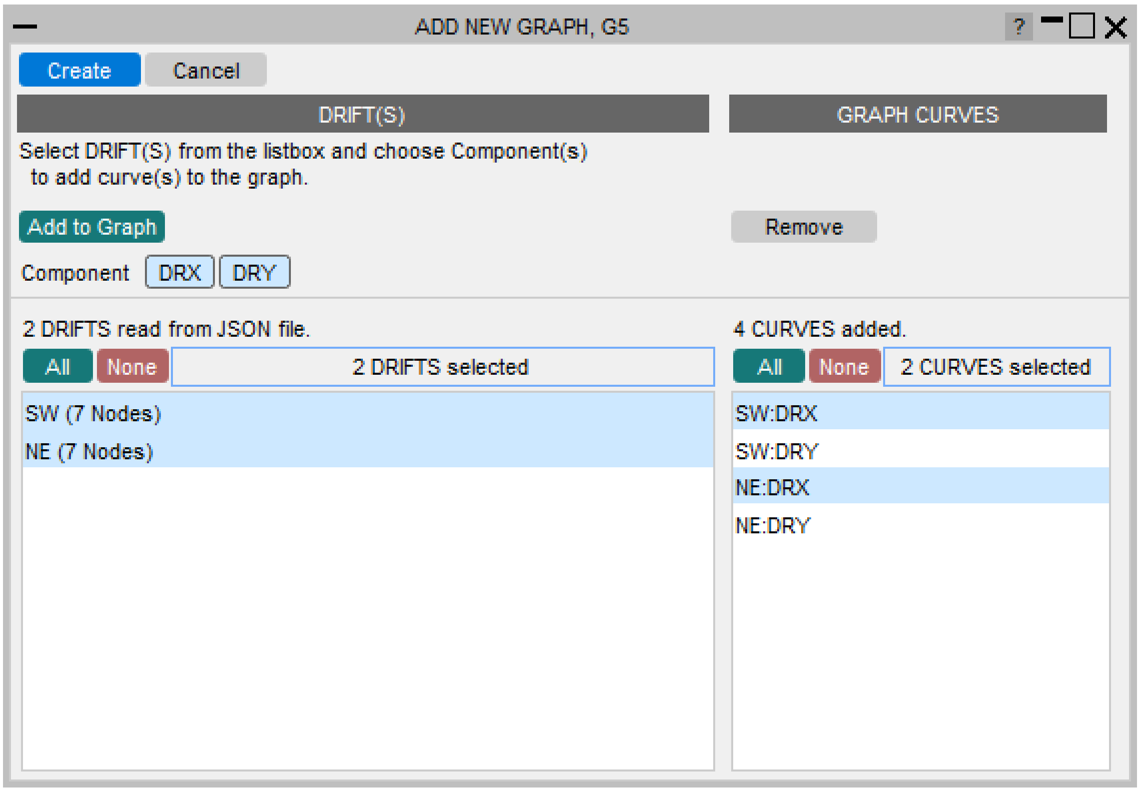
You will be provided with three toggles: **drift locations**, **direction components**, and **model**. The toggle for drift locations will only be shown if more than one drift is included in the current graph setup. All direction component toggles will be shown, but only those included will be active. Finally, the model toggles will only be visible if more than one model is loaded in the current T/HIS session.



In this panel, you are provided with control buttons allowing you to create or modify graph setups.

To create a new graph, click **Create**. You will then be presented with a new window as shown below. Select the drifts and direction components you want to include. Once the selection is made, the **Add to Graph** button will be active. Click **Add to Graph** to generate the list of curves that will be added to the graph, which will be shown on the list box on the right. You may then do some final selection adjustments (e.g. you can remove some of the curves listed by selecting them and clicking **Remove**).

Once you have finalised the curves you wish to include, click **Create** to generate the new graph and return to the **Plot Viewer** window.



Other commands available to you are as follows:

1. **Edit** allows you to modify the currently active graph setup in your **Plot Viewer**. You will be shown with a similar window as for **Create**.
2. **Remove** allows you to delete the currently active graph setup. This will not delete the T/HIS curves associated with the graph.
3. **Reset** deletes every graph setup and recreates the defaults set in PRIMER.

Modifications made in the graph selection panel will be automatically saved to the Workflow JSON file.

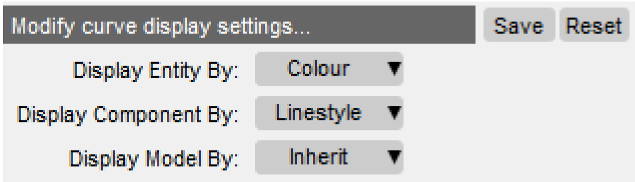
You can also export the current T/HIS curves to an external file. You can do this by clicking **Export to CSV**.

Curve display settings

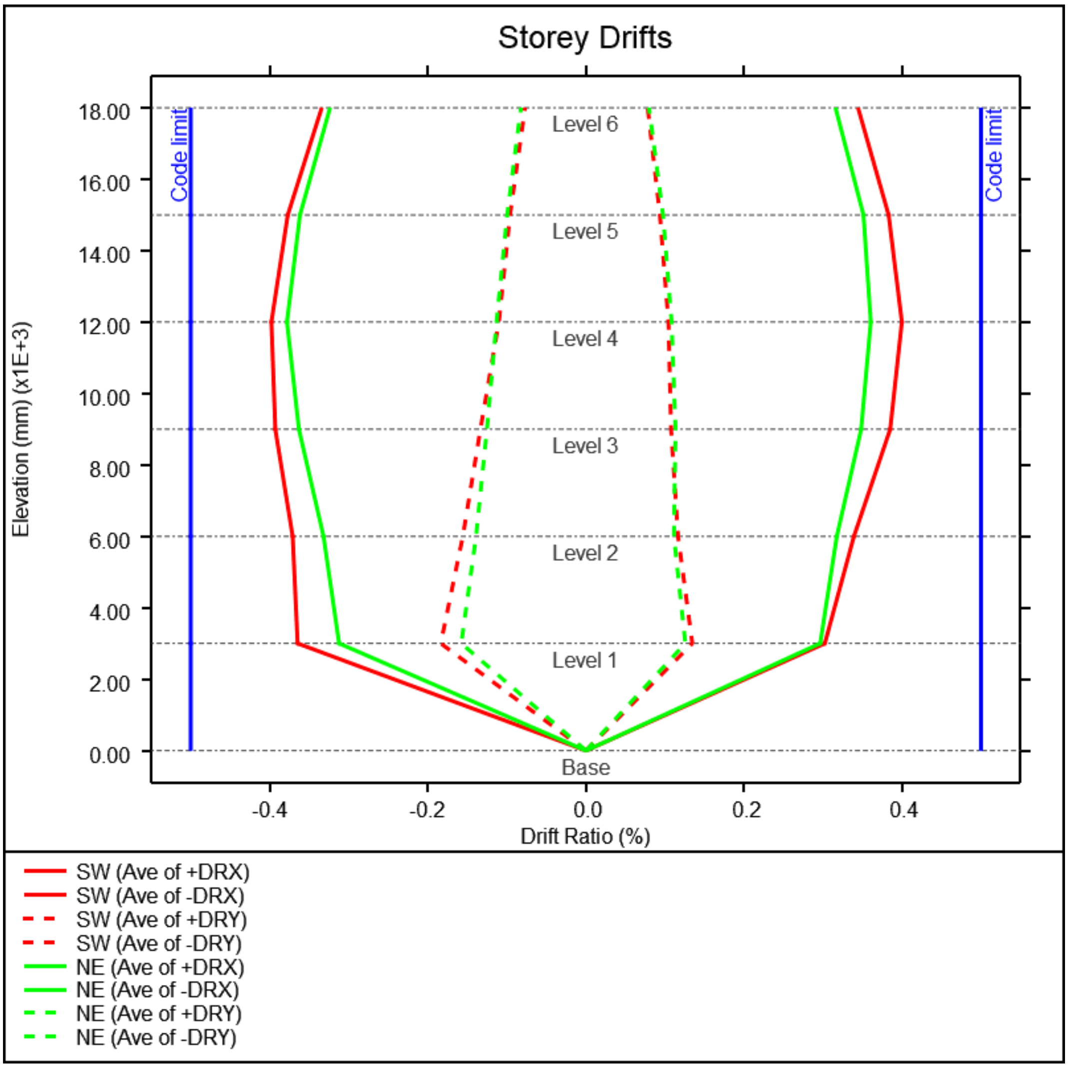
This panel allows you to define the formatting of the curves in the T/HIS graph. These settings will be applied to all graph setups stored in your Workflow file. Later, when you generate the report, REPORTER will read these settings and apply the styling you have defined.

The Workflow file will hold two separate sets of settings for **single model mode** and **multiple model mode**. This is because you may want to have different settings when you are plotting results for only one model and when you are plotting results for multiple models. If you are intending to generate reports containing results from a single model and from multiple models, you need to define the settings for these two modes separately.

The first three settings are responsible for categorising your curves by drift location, direction component and model – in the following hierarchy order:

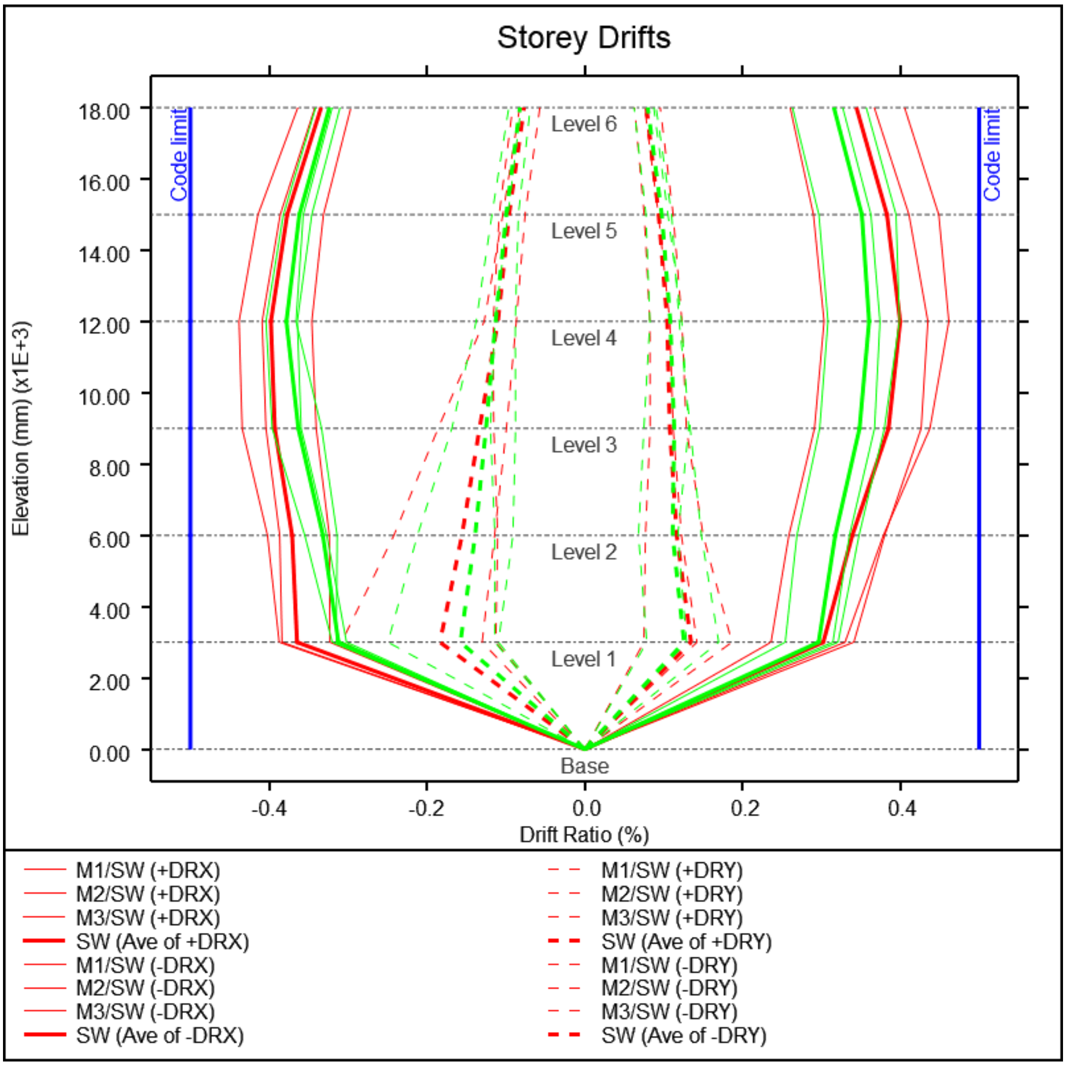


You can categorise the drift locations and direction components by **Colour** or **Line style.** For example, if you display the drift locations by colour and the direction components by line style, the tool will then assign one colour for all curves under a drift location and will assign one line style for all curves under a direction component. As shown in the example below, all curves under **Drift SW** are red and all the curves representing **drifts along the X direction** (DRX) have solid lines:



You can also categorise the models by Colour or Line style. For models, there is a third option called **Inherit** (which is set by default). This option essentially tells the tool that the curves **will not be categorised by model**. Instead, they will just follow the formatting of the first two categories. This is particularly useful if you are more concerned with the aggregate curves and you are just displaying the model curves to see if there is an outlier compared to the aggregate curve. If you use this option, you can quickly identify visually which model curves are associated with an aggregate curve.

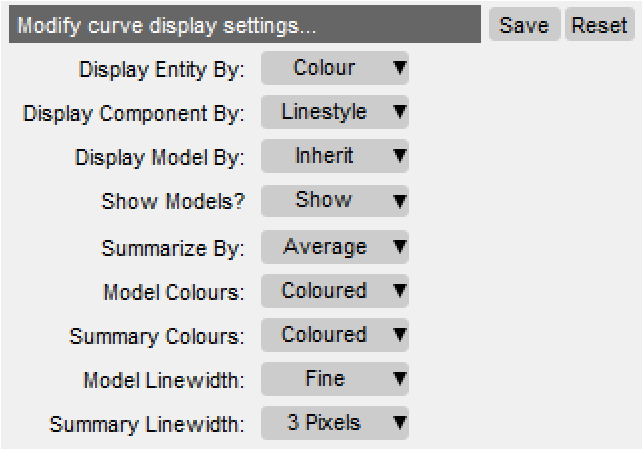
In the example below, the curves under **Drift SW** along **DRX** are solid lines in red colour. The curve representing the mean storey drifts follows the same format but with a thicker line width to differentiate it from the rest of the individual model curves under the same categories:



This current implementation of curve categorisation may not work for all scenarios, and could be improved further in future. Please contact us with your feedback.

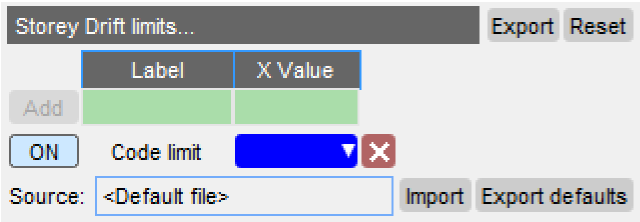
The other curve settings available to you are described below:

1. **Show Models** allows you to set if the model curves are shown or hidden in the plot. This is only relevant for **multiple model mode**.
2. **Summarize by** allows you to choose which aggregate curve is shown. You have the following options: **None, Average, Envelope**.
3. **Model Colours** allows you to choose if the model curves will be in **Colour** or **Greyscale**.
4. **Summary Colours** allows you to choose if the aggregate curves will be in **Colour** or **Greyscale**.
5. **Model Line width** allows you to set the line width for the model curves.
6. **Summary Line width** allows you to set the line width for the aggregate curves.



Any modifications made in this settings panel will not be automatically saved to the Workflow file. Click **Save** to write these settings to the Workflow file. You may also revert back to default settings by clicking **Reset**, which will simultaneously update these settings on the Workflow file.

Storey Drift limits



This panel allows you to define vertical curve limits on the positive and negative X-axis. These limits normally represent acceptable code standard drift limits. They are typically included in building design reports to demonstrate compliance.

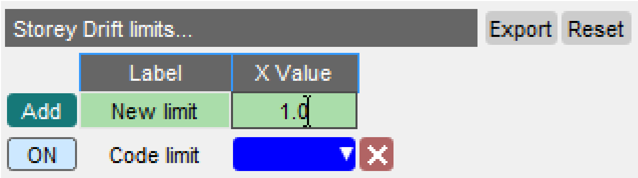
There are two types of vertical storey curve limits that you can define:

1. Constant curve limit along the structure elevation
2. Stepped curve limit, where the desired limit per storey extent varies

You can define a constant curve limit using this panel. In order to define a stepped curve limit, you need to import a CSV file. You can download an example plot limit input file by clicking **Export defaults**.

You can also import a constant curve limit using an external file and this file may contain multiple curve limits of different types. Theoretically, you can store all your curve limits into one file to quickly generate them later.

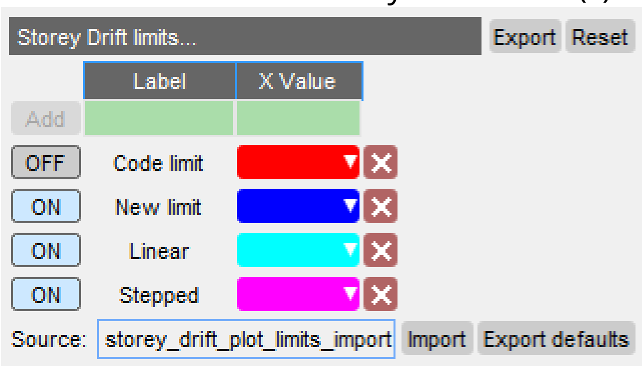
To define a constant curve limit, you would need to define a label and the X-axis value on the text boxes provided. Then, click **Add**.

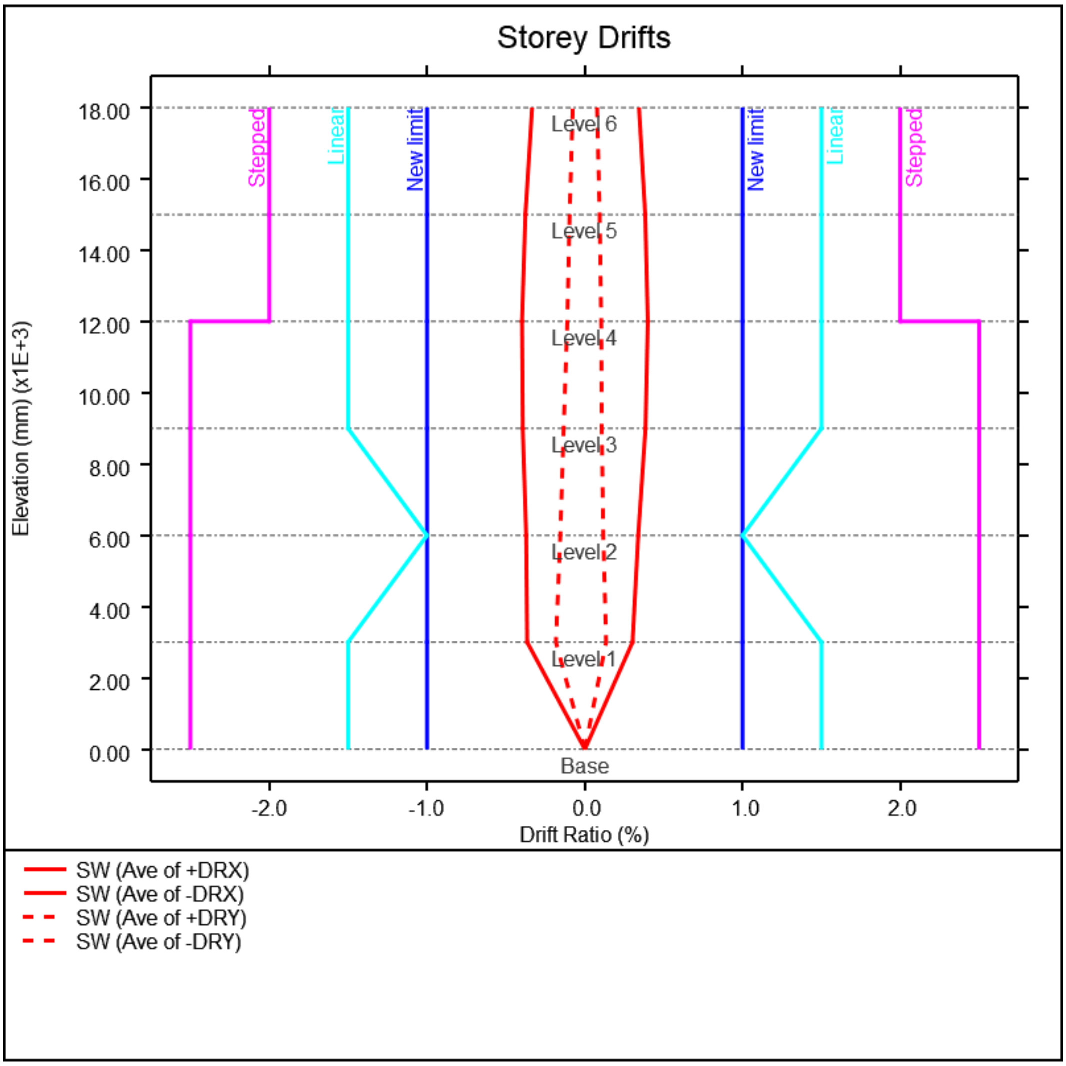


To define a stepped curve limit, create a CSV file following the data format of the exported default file. Click **Import** to add the data to the plot.

The limits created will be listed below along with some control buttons to manipulate them:

1. Show or hide the curve limits using the **ON/OFF** toggles
2. Change the colour of the curve limits using the colour selection dropdown.
3. Delete a curve limit using the delete **(X)** button. Currently, this panel does not allow you to edit an existing curve limit. You may need to recreate a curve limit to modify the X-value(s) along the storeys.





The storey curve limits will be automatically saved to the Workflow file upon creation. Curve colour and visibility settings will also be automatically updated in the Workflow file upon changing them in this panel. You may wish to store these data separately for future use. You can do so by clicking **Export** located on the right side of the panel header.

You may also revert back to default storey curve limits by clicking the **Reset** button.

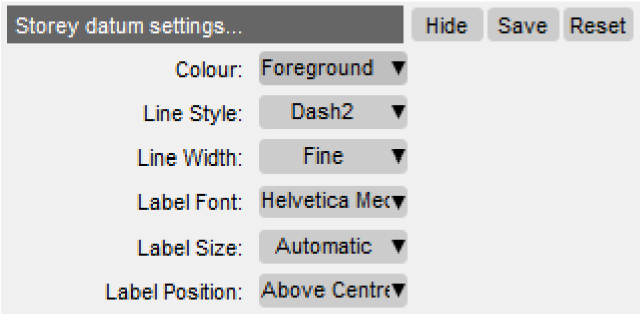
Each Workflow will have a different set of default limits.

Storey datum settings

This panel allows you to define the formatting of the storey datums shown in the plot. This panel is hidden by default. Click **Show** to reveal this panel.

The settings available to you are as follows:

1. **Colour** allows you to choose the colour of the storey datums
2. **Line Style** allows you to choose the line style of the storey datums
3. **Line Width** allows you to choose the line width of the storey datums
4. **Label Font** allows you to choose the font of the storey datum labels
5. **Label Size** allows you to choose the font size of the storey datum labels
6. **Label Position** allows you to define the location of the labels relative to the storey datums



Any modifications made on this settings panel will not be automatically saved to the Workflow file. Click **Save** to write these settings to the Workflow file. You may also revert back to default settings by clicking **Reset**, which will simultaneously update these settings on the Workflow file.

Storey Drift Report

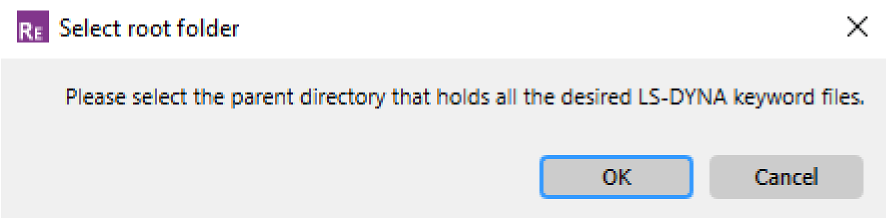
This workflow provides you with REPORTER templates to automatically generate report documents. The template compiles all T/HIS graphs you have set in PRIMER and T/HIS along with a model view from D3PLOT to show you the locations of drifts you have specified on each graph.

There are currently two templates with different report layouts available.

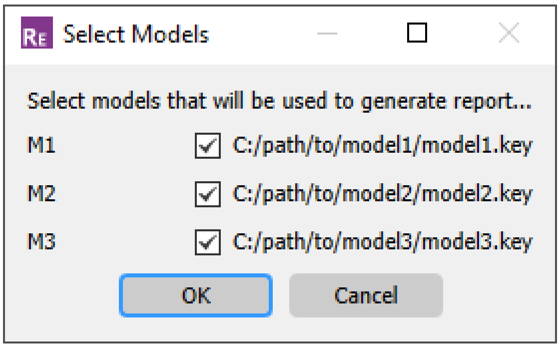
- **1x1** layout showing one T/HIS-graph/D3PLOT-model-view pair per page, split vertically.
- **2x1** layout showing two T/HIS-graph/D3PLOT-model-view pairs per page.

Running the template

Upon opening the template, you will be prompted to select the parent/root folder where all your model keyword files sit. If you have followed the recommendations for Writing the Workflow File from PRIMER, this should be the same directory where you have saved the Workflow file.



When multiple models are detected, the template will show you another window where you can choose which models to include in the report. By default, all models are selected assuming that the root folder only contains the relevant model analysis runs that you wish to process and report.



After this, the template generation should commence, running T/HIS and D3PLOT items to generate the report images. These images will also be saved into a subfolder named "reporter" that will be created when this template is generated. A sample page from a successful template run is shown below.

The REPORTER variables hold a record of the paths of models you have chosen to run. This can serve as a way to validate that you have run the models you intended.

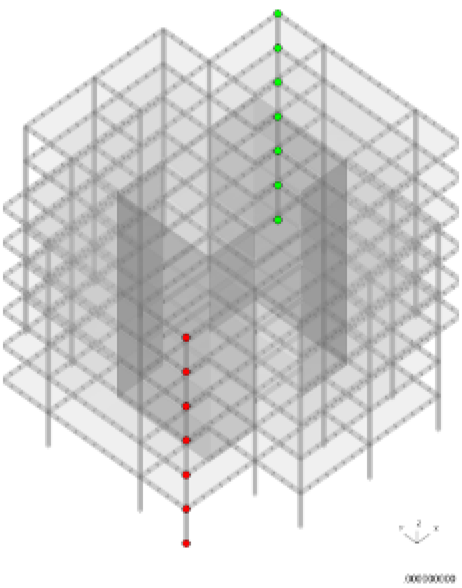
Storey Drift

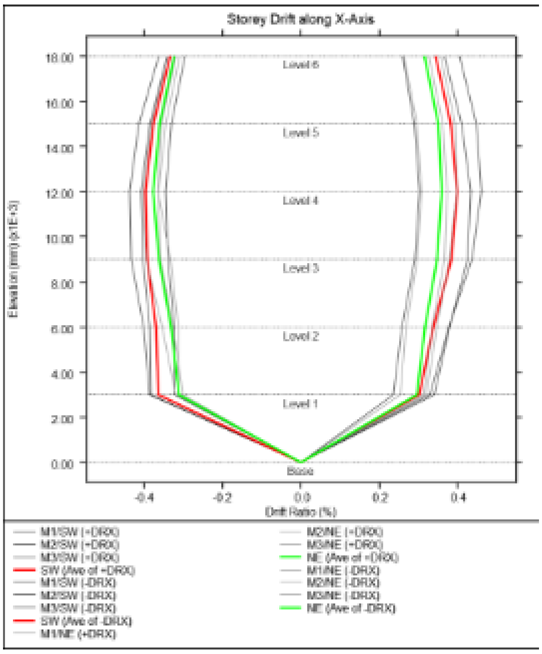
Seismic Analysis

2 DRIFTS

Drift along X

D3PLOT: demo_building_v4

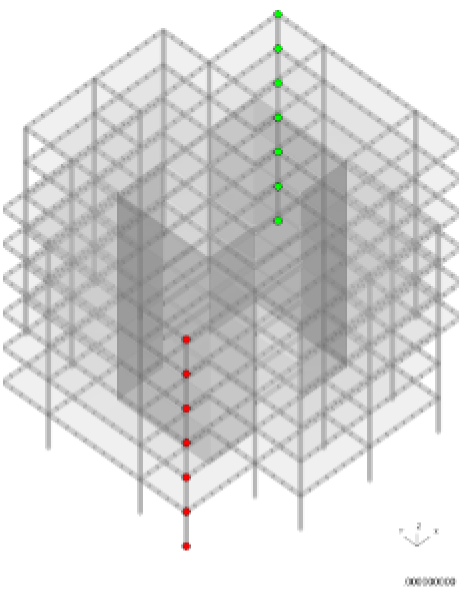


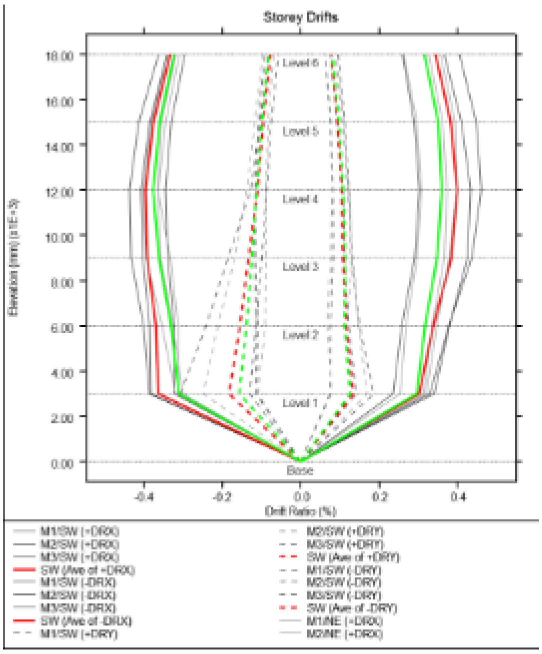


2 DRIFTS

Drift along X, Y

D3PLOT: demo_building_v4





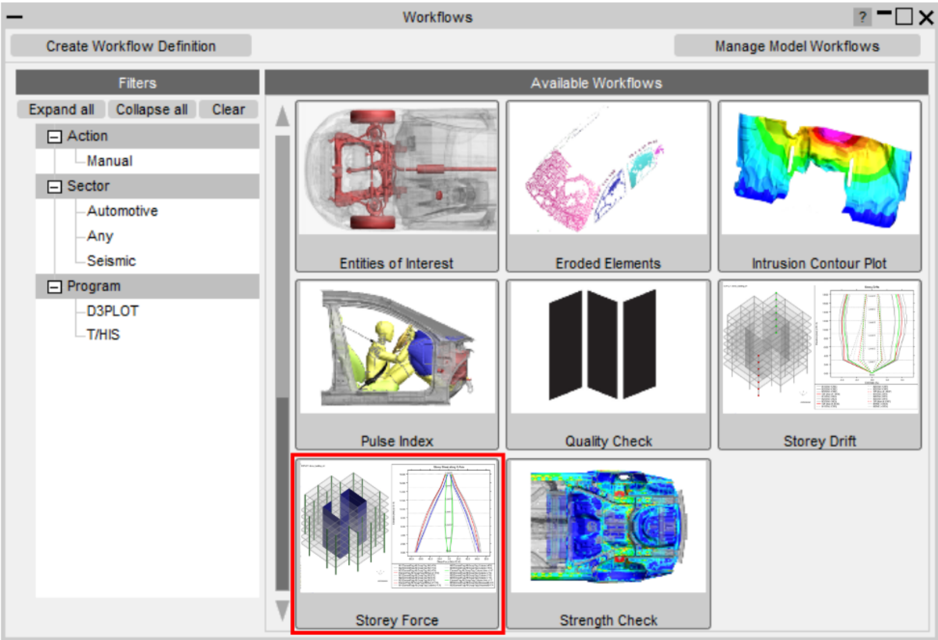
Model Directory | C:/Users/Ian-Keith.Relucio/OneDrive - Arup/Arup UK/LS-Dyna_Dev/DEMO_MDL/01_MULTI_MDL_TEST

19.10.2. Storey Force

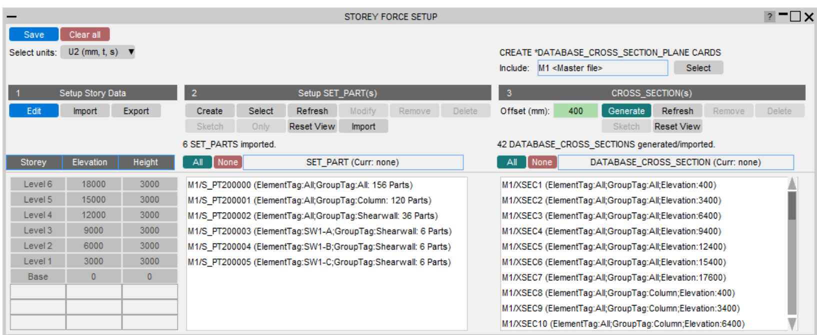
Storey Force

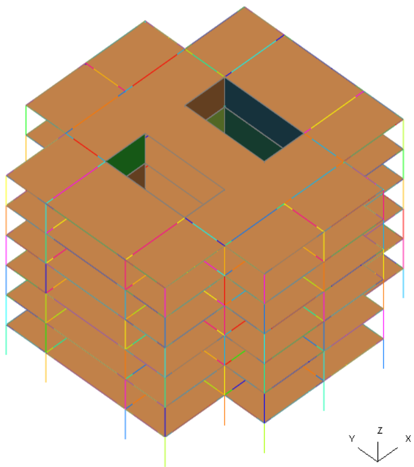
Tools → Workflows → Storey Force

The Storey Force workflow tool is used to show forces on each storey of the building to investigate the flow of force through the entire structure or on selected elements grouped into SET_PARTs.



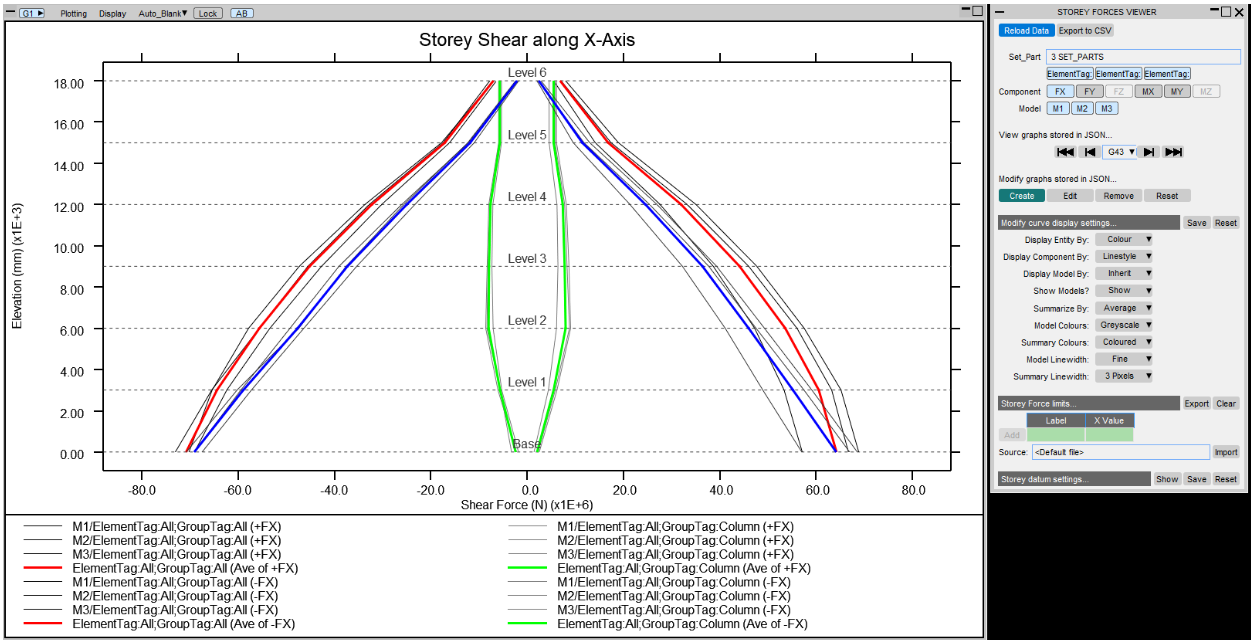
In PRIMER, you can define cross sections for each SET_PART, for every storey of the building.





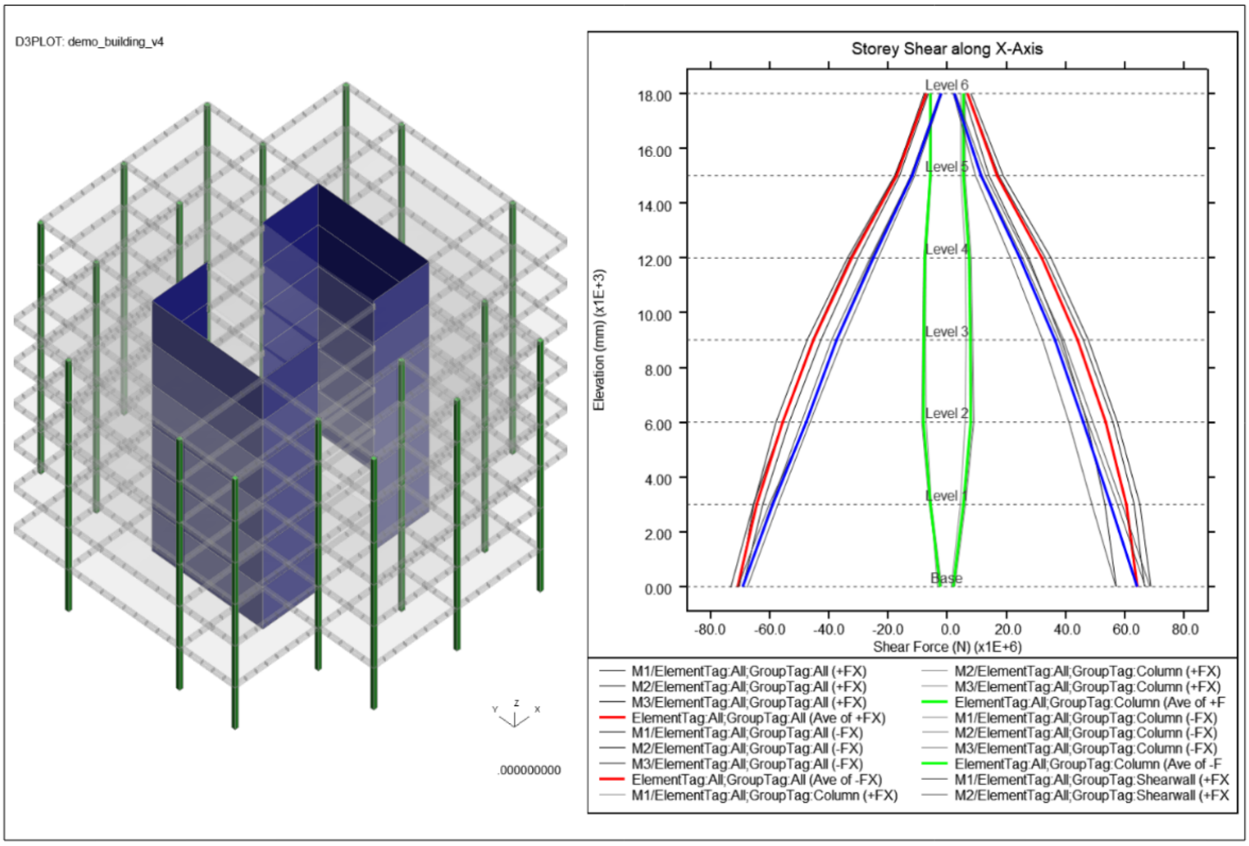
In T/HIS, storey forces are extracted for each of the cross sections you defined in PRIMER and then storey curves are generated – plotted on graphs.

This allows you to interrogate the global behaviour of the structure and make changes to member designs or structural layout if necessary.



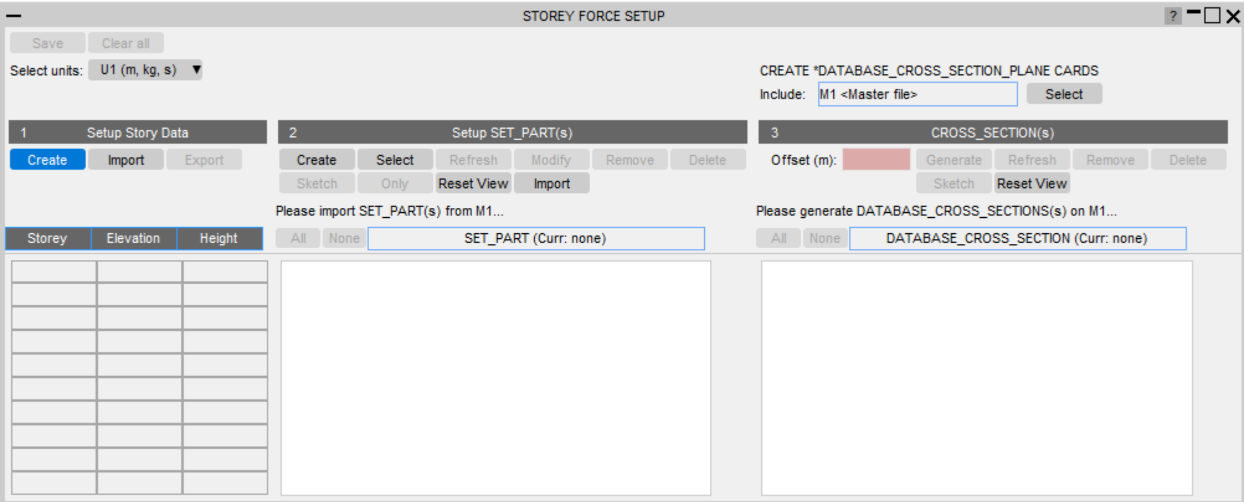
Finally, you can generate automated reports with the REPORTER templates provided.

In the report, corresponding D3PLOT views are paired with each T/HIS plot to highlight the corresponding SET_PART(s) in the model.

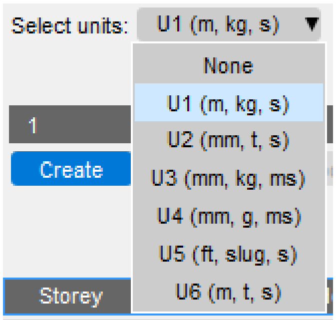

D3PLOT: demo_building_v4
.000000000
Storey Shear along X-Axis
Level 6
Level 5
Level 4
Level 3
Level 2
Level 1
Base
18.00
16.00
14.00
12.00
10.00
8.00
6.00
4.00
2.00
0.00
Elevation (mm) (x1E+3)
-80.0
-60.0
-40.0
-20.0
0.0
20.0
40.0
60.0
80.0
Shear Force (N) (x1E+6)
M1/ElementTag:All;GroupTag:All (+FX)
M2/ElementTag:All;GroupTag:All (+FX)
M3/ElementTag:All;GroupTag:All (+FX)
ElementTag:All;GroupTag:All (Ave of +FX)
M1/ElementTag:All;GroupTag:All (-FX)
M2/ElementTag:All;GroupTag:All (-FX)
M3/ElementTag:All;GroupTag:All (-FX)
ElementTag:All;GroupTag:All (Ave of -FX)
M1/ElementTag:All;GroupTag:Column (+FX)
M2/ElementTag:All;GroupTag:Column (+FX)
M3/ElementTag:All;GroupTag:Column (+FX)
ElementTag:All;GroupTag:Column (Ave of +F
M1/ElementTag:All;GroupTag:Column (-FX)
M2/ElementTag:All;GroupTag:Column (-FX)
M3/ElementTag:All;GroupTag:Column (-FX)
ElementTag:All;GroupTag:Column (Ave of -F
M1/ElementTag:All;GroupTag:Shearwall (+FX
M2/ElementTag:All;GroupTag:Shearwall (+FX

Storey Force Setup

When the tool is launched in PRIMER, the setup window opens. This where you select the SET_PARTs and cross-sections you wish to process:

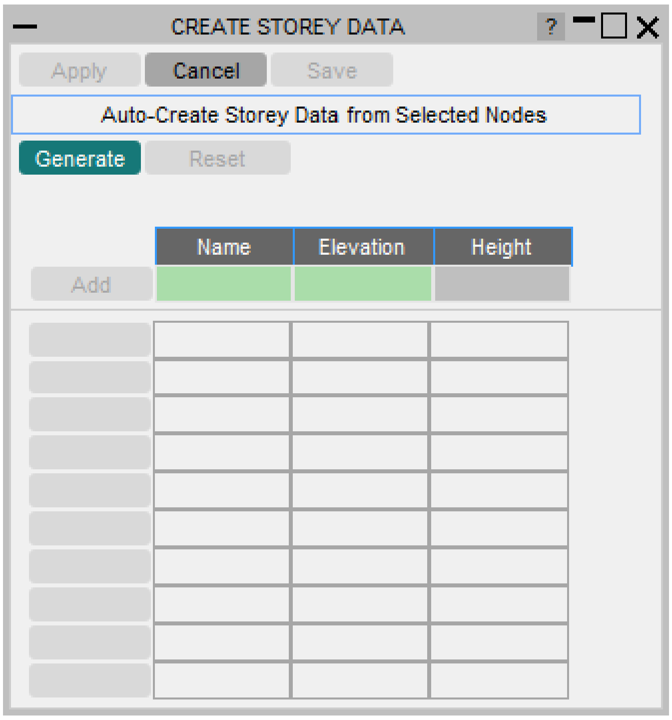


First, you need to choose the appropriate unit system from the dropdown menu:



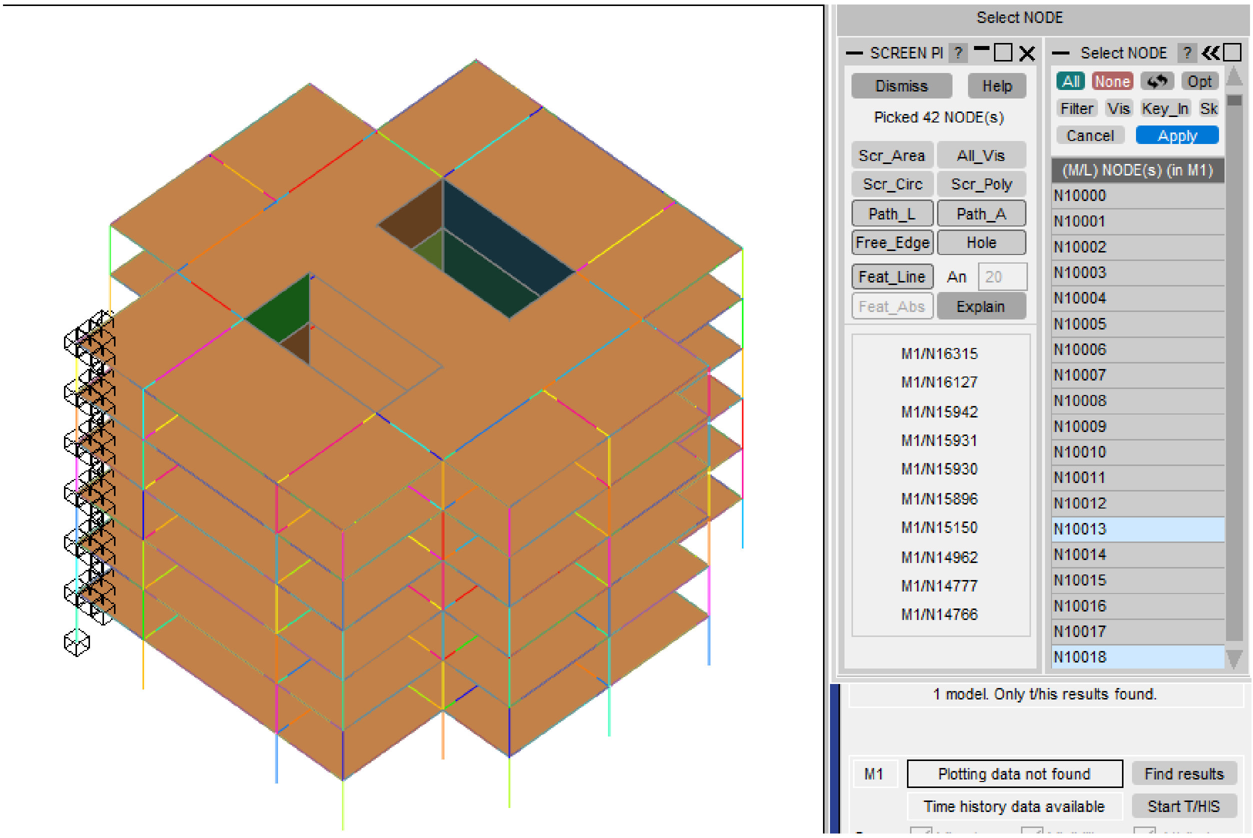
Defining Storey Data

You can define the storey data for the structure either by clicking the **Create** button or the **Import** button under the Setup Storey Data section. **Import** allows you to import previously saved storey definitions (e.g. those created for the Storey Drift workflow). When you click **Create**, the Storey Data window appears:



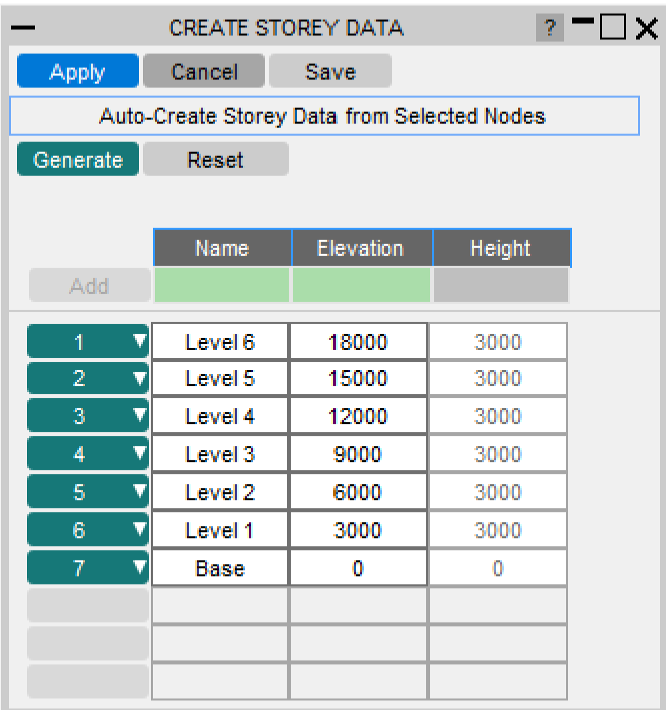
Each storey can be defined manually by populating **Name** and **Elevation** textboxes and then clicking the **Add** button which will be activated if the inputs are valid.

Alternatively, you can define multiple storeys automatically by clicking **Generate**. You will be prompted to select nodes in the model. This will then generate storeys for each unique elevation (z-coordinate) among the nodes you have selected. Finally, you can then modify the labels of each generated storey to be more informative for your project.



Click **Apply** to import the storey data back to the main setup window.

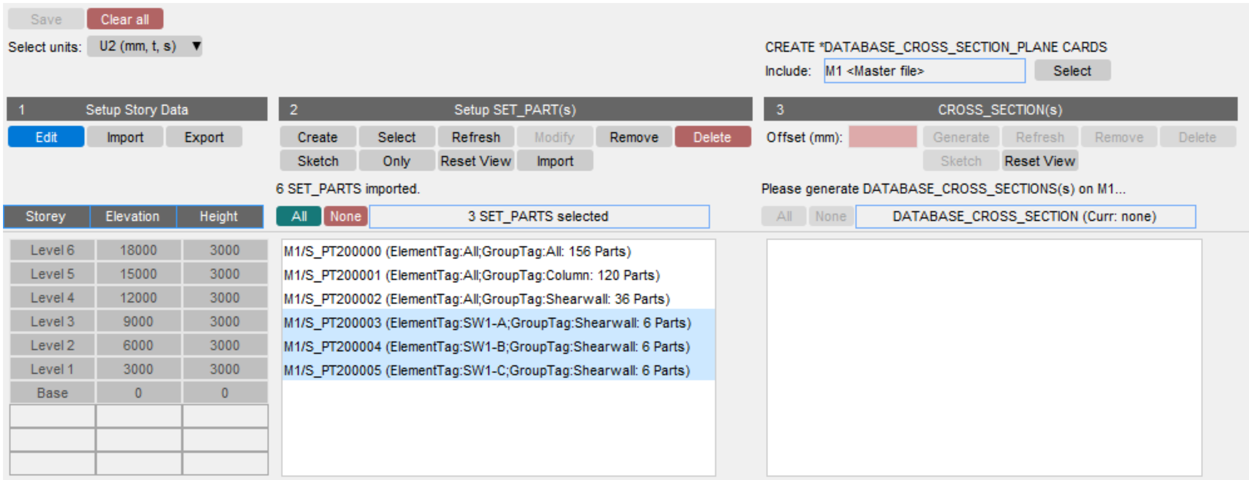
You can optionally save this data by clicking **Save**. This will write it to a separate JSON file, which you can **Import** when you are starting a new setup. Normally, storey data would be applicable to multiple Seismic workflows, so saving this data will be useful to those other workflows too.

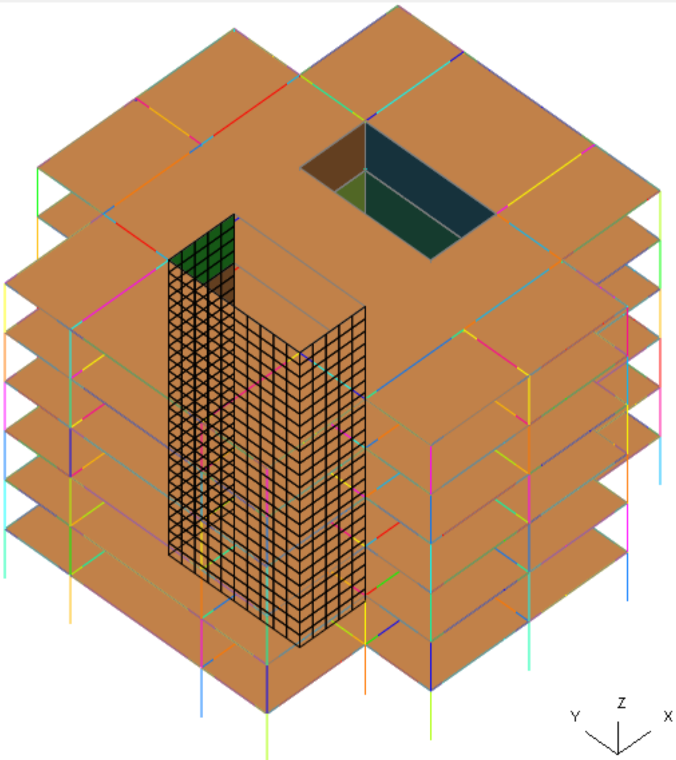


Defining SET_PARTs

If you have not defined any SET_PARTs prior to running this Workflow, you can use the **Create** button under the **Setup SET_PART(s)** section of the setup window. A popup window will appear allowing you the create a new SET_PART. This window is the same as PRIMER's usual Create SET_PART menu (**Volumes I & II→ SET→ PART→ Create**).

If you have defined some SET_PARTs beforehand, you can use them by clicking **Select**. A selection window will appear, prompting you to choose SET_PARTS in the model.



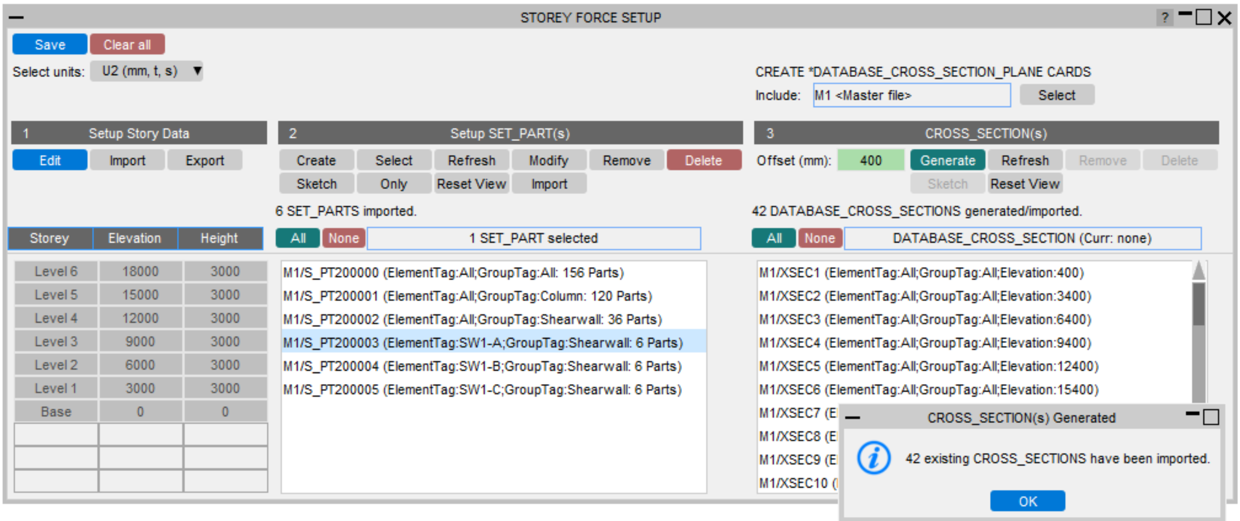


Some section controls will be active when you select SET_PARTs in the list box as shown above. You will have access to SET_PART commands that will allow you to modify a SET_PART or update your list.

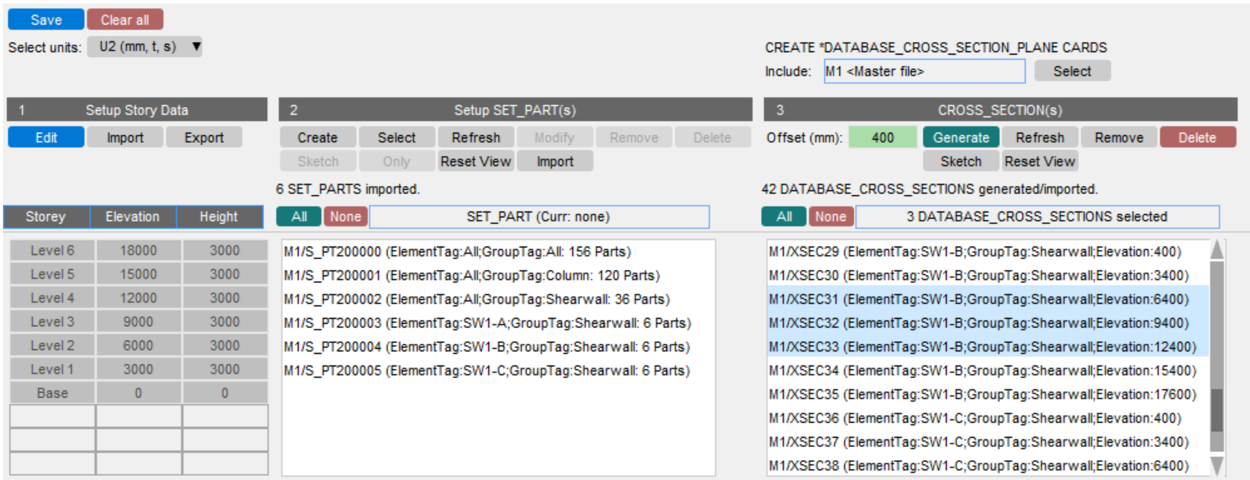
1. **Modify** – modify SET_PARTs one at a time. A popup window will appear to allow you to modify the properties of the SET_PART and add or remove PARTs from it.
2. **Remove** – remove existing SET_PARTs from the list. This will not delete them from the model.
3. **Delete** – remove existing SET_PARTs from the list and delete them from the model. The deletion is **not recursive** so the PARTs along with their components will still remain.
4. **Sketch** – highlight the contents of one or more SET_PARTs in the model.
5. **Only** – isolate the components of one or more SET_PARTs in PRIMER, blanking everything except those components.
6. **Reset view** – reset the state of the model in view and remove sketches.
7. **Refresh** – re-validate the SET_PARTs listed. If any SET_PARTs have been deleted using other PRIMER menus, this command will remove them from the list too.

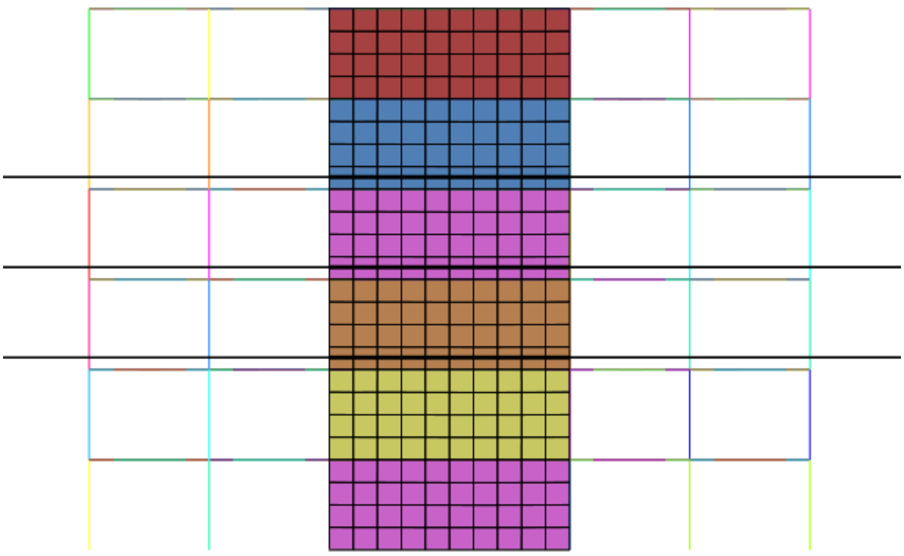
Defining Storey DATABASE_CROSS_SECTIONs

DATABASE_CROSS_SECTIONs can be derived from the storey and SET_PART definitions by clicking **Generate**. This button will be active only after you specify a positive offset value. The offset value applies a vertical offset for the cross-sections from each storey level z-coordinate. The purpose of the offset is to ensure that DATABASE_CROSS_SECTIONs intersect beam, shell and solid elements, preferably at their midpoint, rather than aligning with nodes at their ends (which would typically be the case without an offset, since storey levels are typically defined at node locations).



A popup window will appear upon successful cross-section generation. It will show you how many new DATABASE_CROSS_SECTIONs have been created in the model, and/or how many existing ones have been added to the CROSS_SECTION(s) sub-section list box.





Similar to the SET_PARTs, some commands will be available to you when you select DATABASE_CROSS_SECTIONs in the list box.

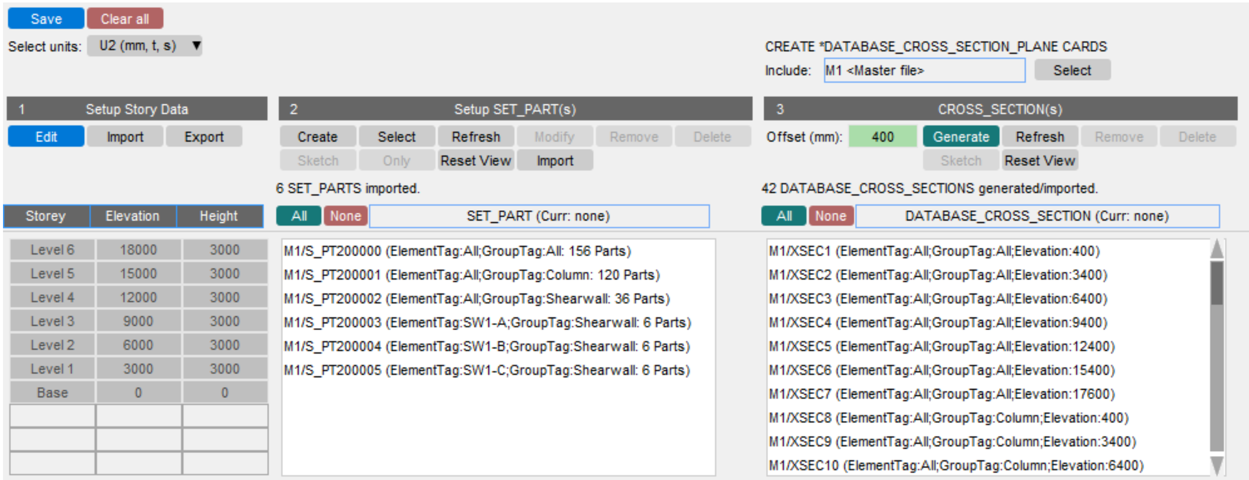
1. **Remove** – remove existing DATABASE_CROSS_SECTIONs from the list. This will not delete them from the model.
2. **Delete** – remove existing DATABASE_CROSS_SECTIONs from the list and delete them from the model.
3. **Sketch** – highlight the DATABASE_CROSS_SECTIONs you selected, along with the SET_PARTs they refer to.
4. **Reset view** – reset the state of the model in view and remove sketches.
5. **Refresh** – re-validate the DATABASE_CROSS_SECTIONs listed. If any DATABASE_CROSS_SECTIONs have been deleted via other PRIMER menus, this command will remove them from the list too.

Tip: You may want to delete an entire set of DATABASE_CROSS_SECTIONs from the model that were created by previously running this Workflow. To quickly delete unwanted DATABASE_CROSS_SECTIONs, repopulate the setup window with the previous storey data and SET_PARTs you have used, and then generate the CROSS_SECTIONs with the same offset value. Once these CROSS_SECTIONs are listed, you can select them all and click **Delete**.

Writing the Workflow File

Once all data has been defined, save the storey force setup by clicking **Save**. This will write a Workflow file in JSON format. This file will be used to post-process the defined DATABASE_CROSS_SECTIONs in T/HIS and create a report in REPORTER.

The Storey Force Workflow tool has been designed to be used on a sweep of LS-DYNA runs with different ground motions applied to the same model. It is advised to save the Workflow file in the parent folder (the folder containing several child folders, each containing one set of ground motion results). Currently, this Workflow will only work properly if only **one Workflow file exists** in the parent folder, including its child folders. If you save this file in the folder of an individual model, then there is a risk to duplicate the Workflow file, which might cause problems later. This will most probably happen when you duplicate the original model to create a new model with a different ground motion input.



For this workflow, one DATABASE_CROSS_SECTION will be generated for each storey, for each SET_PART definition. Remember to save the .key file and rerun the model if new DATABASE_CROSS_SECTIONs have been created, so their results will be available in T/HIS.

Before saving the drift setup, you may also wish to select an include file for the DATABASE_CROSS_SECTION(s). You can choose an include file by clicking **Select** above the DATABASE_CROSS_SECTION(s) header. The tool will add any DATABASE_CROSS_SECTION keywords created to your selected include file.

Resetting the data

To reset all data, click **Clear all** and repeat the whole process again to define a new storey force setup. Alternatively, you can select all items in sections 2 and 3 of the setup window and click the **Remove** buttons on each sub-section to remove the data defined on those sections only.

Importing existing Workflow Data

When an existing Workflow file is present in the root folder, the storey data and SET_PARTs are automatically imported when you run this Workflow.

After removing all data in a current session, you can import the storey data and the SET_PARTs by clicking **Import** on each sub-section. The SET_PARTs stored in the Workflow file are then validated, and only those existing in the model will be displayed. For further details on importing storey data, please refer to the following section of this manual.

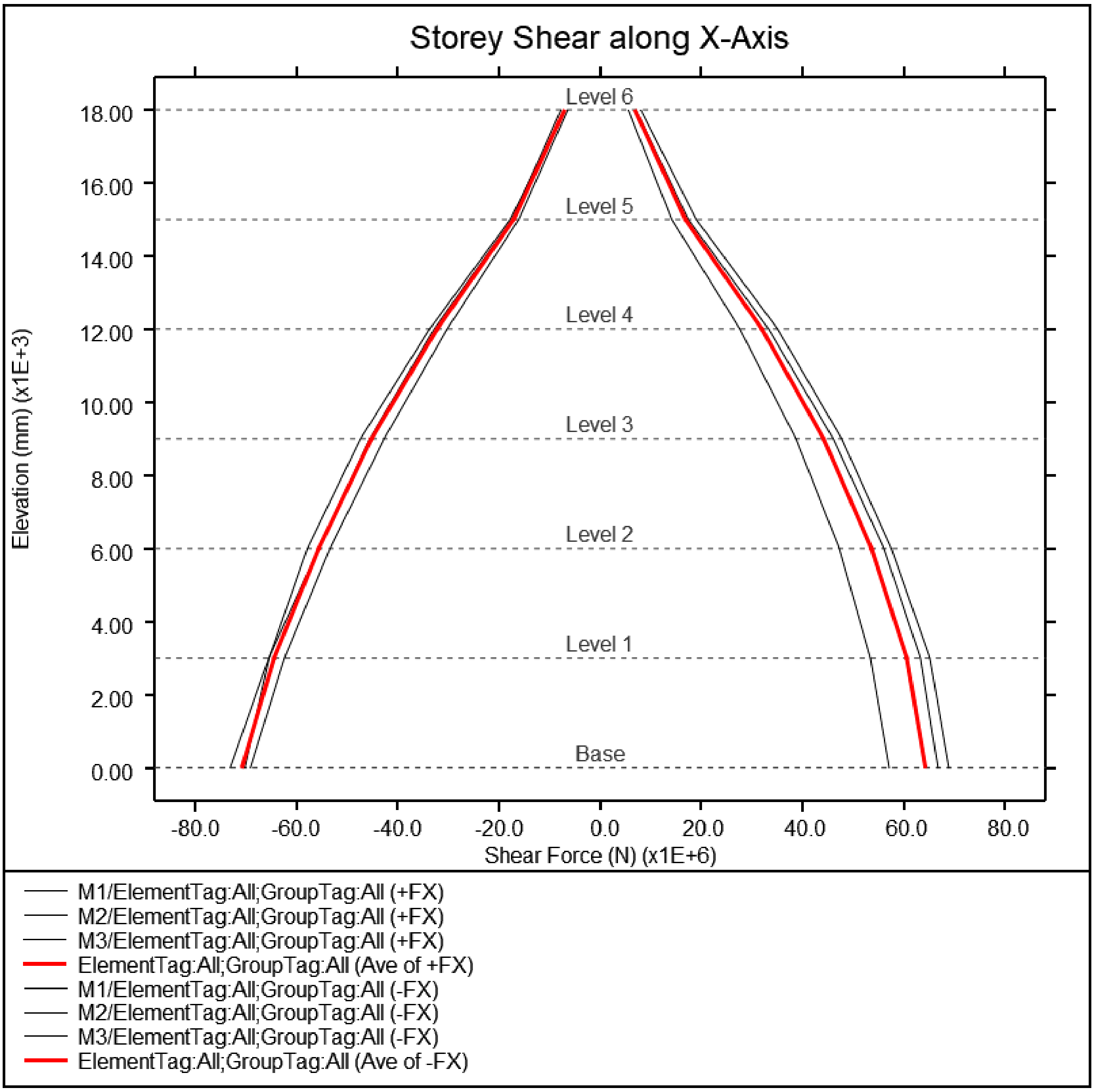
Importing existing Storey Data

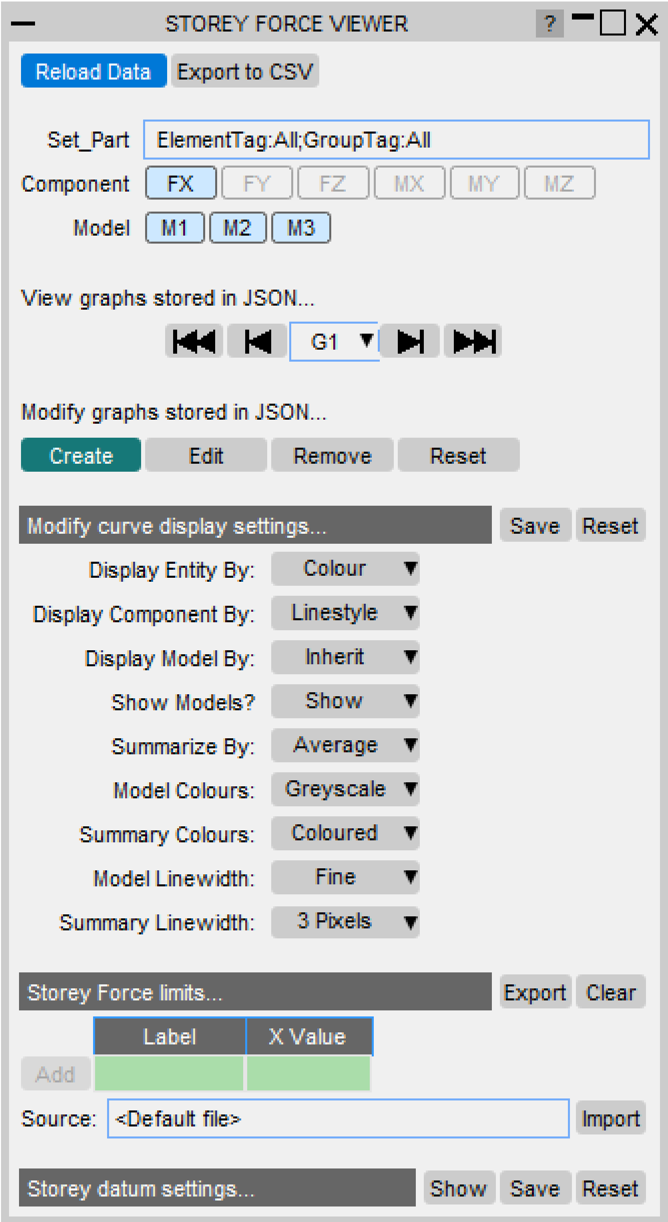
As mentioned on the section above, you can import pre-defined storey data to quickly define storeys. The storey data may exist in an **external JSON file** or in the **Workflow file**. If it is present, you will be prompted to use an existing Workflow file. If you **choose not to**, then a file selector popup will appear so you can select an external JSON file.

Storey Force Viewer

When the tool is launched in T/HIS, the storey force curves will be generated for each graph setup existing in the Workflow file. Then you will be presented with the window below.

When the Workflow file is initially created from PRIMER, default graph setups are included, one for each force component, for each SET_PART defined. The storey force curves will be created for each of these graph setups and the first graph setup will be plotted in T/HIS and will be active in the Viewer GUI.


Storey Shear along X-Axis
Elevation (mm) (x1E+3)
18.00
16.00
14.00
12.00
10.00
8.00
6.00
4.00
2.00
0.00
Level 6
Level 5
Level 4
Level 3
Level 2
Level 1
Base
-80.0
-60.0
-40.0
-20.0
0.0
20.0
40.0
60.0
80.0
Shear Force (N) (x1E+6)
M1/ElementTag:All;GroupTag:All (+FX)
M2/ElementTag:All;GroupTag:All (+FX)
M3/ElementTag:All;GroupTag:All (+FX)
ElementTag:All;GroupTag:All (Ave of +FX)
M1/ElementTag:All;GroupTag:All (-FX)
M2/ElementTag:All;GroupTag:All (-FX)
M3/ElementTag:All;GroupTag:All (-FX)
ElementTag:All;GroupTag:All (Ave of -FX)

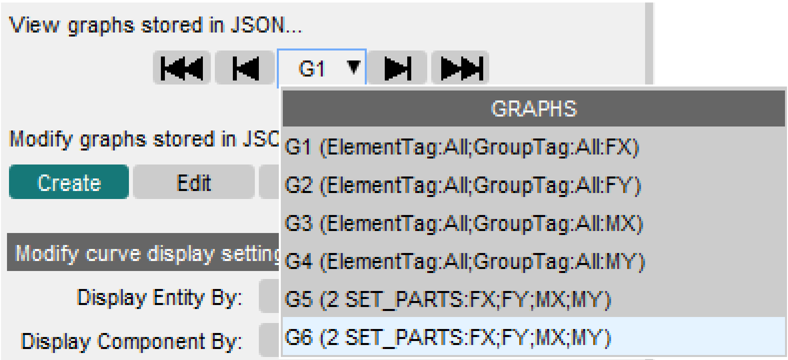


The Viewer GUI is generally split into four sections listed below:

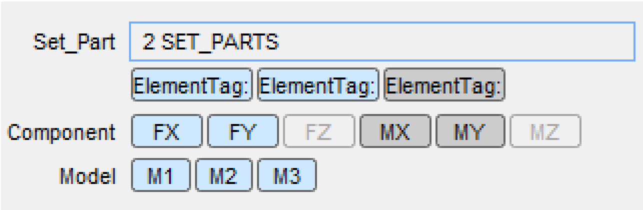
1. Graph selection/creation panel
2. Curve display settings
3. Storey force limits definition
4. Storey datum settings

Graph selection/creation panel

This panel allows you to cycle through the graphs you have generated. You can use the navigation buttons to view the graphs sequentially or you can select a graph from the drop-down list.



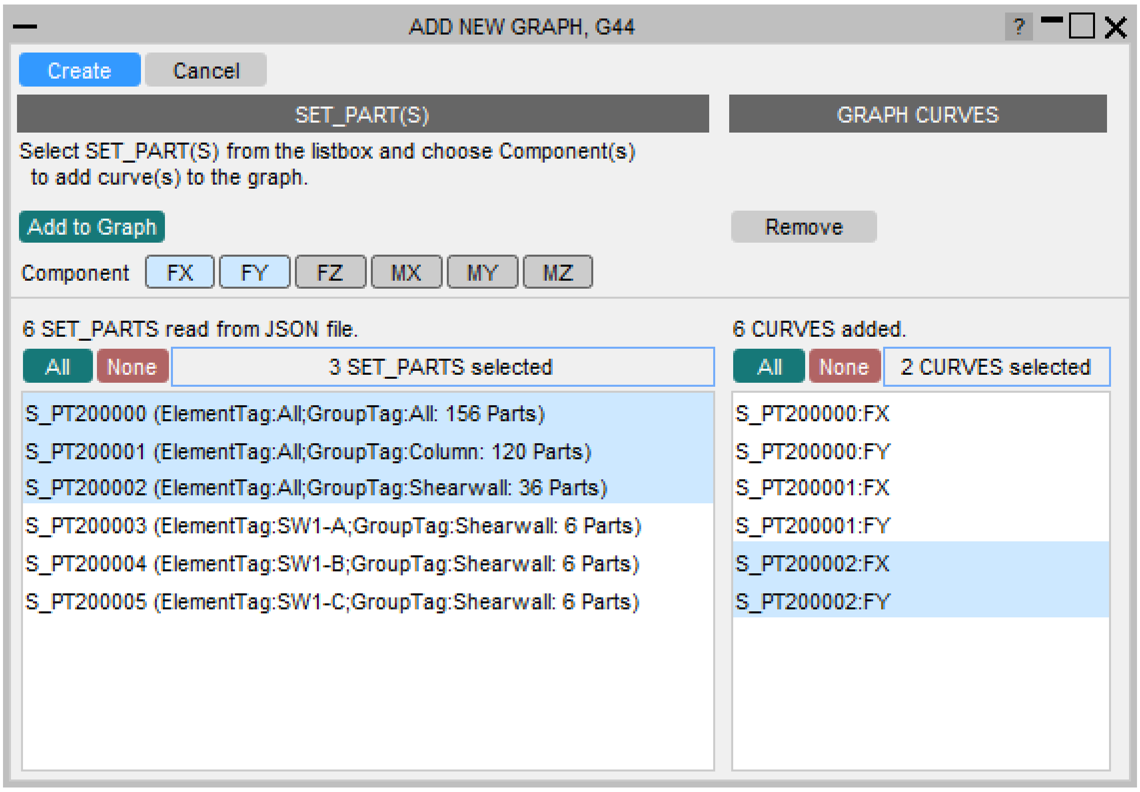
You will be provided with three toggles: **SET_PARTs**, **force components**, and **model**. The toggle for SET_PARTs will only be shown if more than one SET_PART is included in the current graph setup. All force component toggles will be shown, but only those included will be active. Finally, the model toggles will only be visible if more than one model is loaded in the current T/HIS session.



In this panel, you are provided with control buttons allowing you to create or modify graph setups.

To create a new graph, click **Create**. You will then be presented with a new window as shown below. Select the SET_PARTs and force components you wish to include. Once you have made your selections, the **Add to Graph** button will be active. Click **Add to Graph** to generate the list of curves that will be added to the graph, which will be shown on the list box on the right. You can then do any final selection adjustments (e.g. you can remove some of the curves listed by selecting them and clicking **Remove**).

Once you have finalised the curves you wish to include, click **Create** to generate the new graph and return to the **Plot Viewer** window.



Other commands available to you are as follows:

1. **Edit** – modify the currently active graph setup in your **Plot Viewer**. You will be shown a similar window as for **Create**.
2. **Remove** – delete the currently active graph setup. This will not delete the T/HIS curves associated with the graph.
3. **Reset** – delete every graph setup and recreate the defaults set in PRIMER.

Any modifications made in the graph selection panel will be automatically saved to the Workflow JSON file.

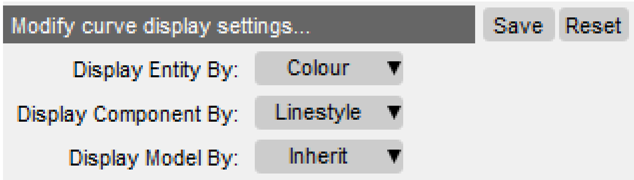
You may also wish to export the current T/HIS curves to an external file. You can do this by clicking **Export to CSV**.

Curve display settings

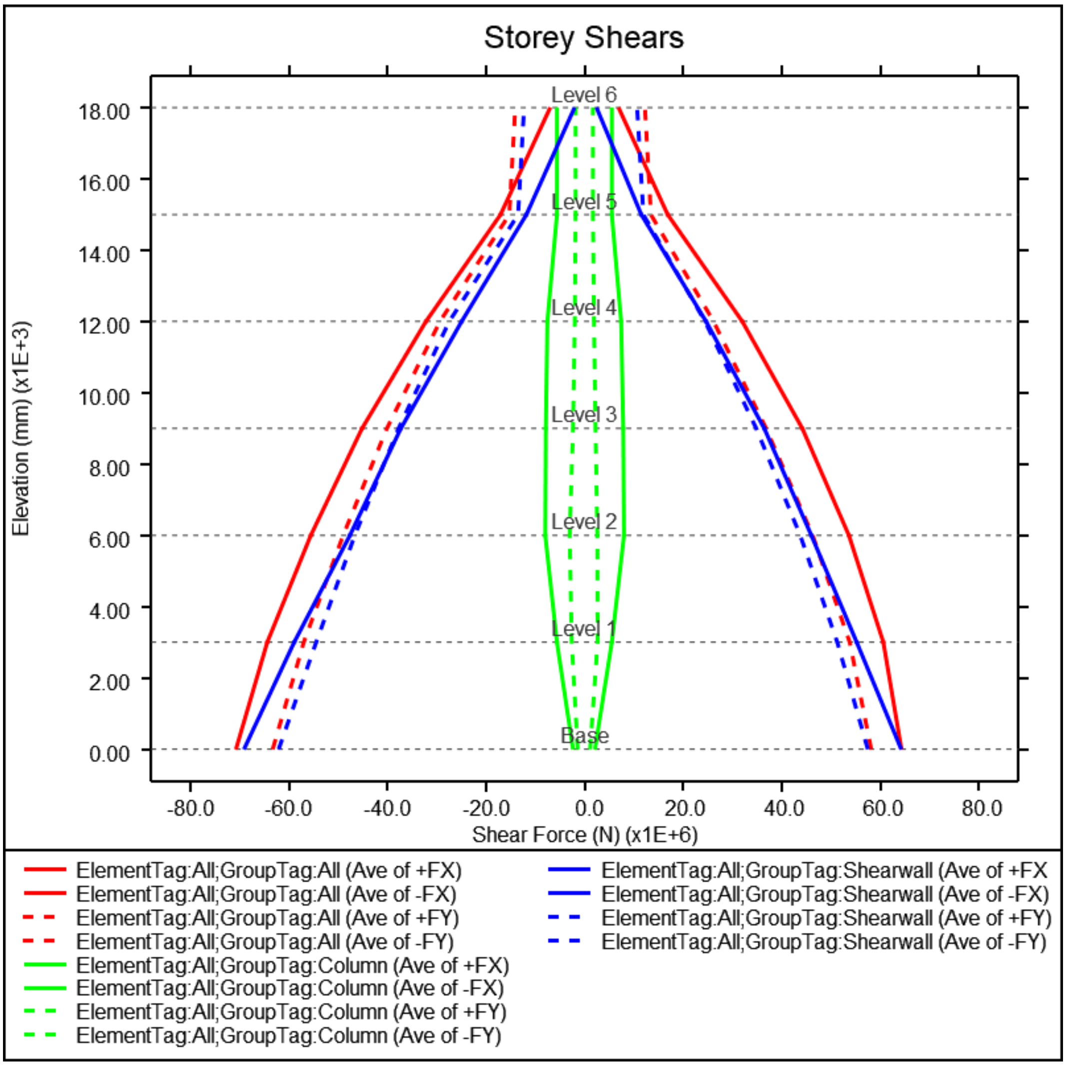
This panel allows you to define the formatting of the curves in the T/HIS graph. These settings will be applied to all graph setups stored in your Workflow file. Later when you generate the report, REPORTER will read these settings and apply the styling you have defined.

The Workflow file will hold two separate sets of settings for **single model mode** and **multiple model mode**. This is because you may want to have different settings when you are plotting results for only one model and when you are plotting results for multiple models. If you are intending to generate reports containing results from a single model and from multiple models, you need to define the settings for these two modes separately.

The first three settings are responsible for categorising your curves by SET_PART, force component and model – in the following hierarchy order:

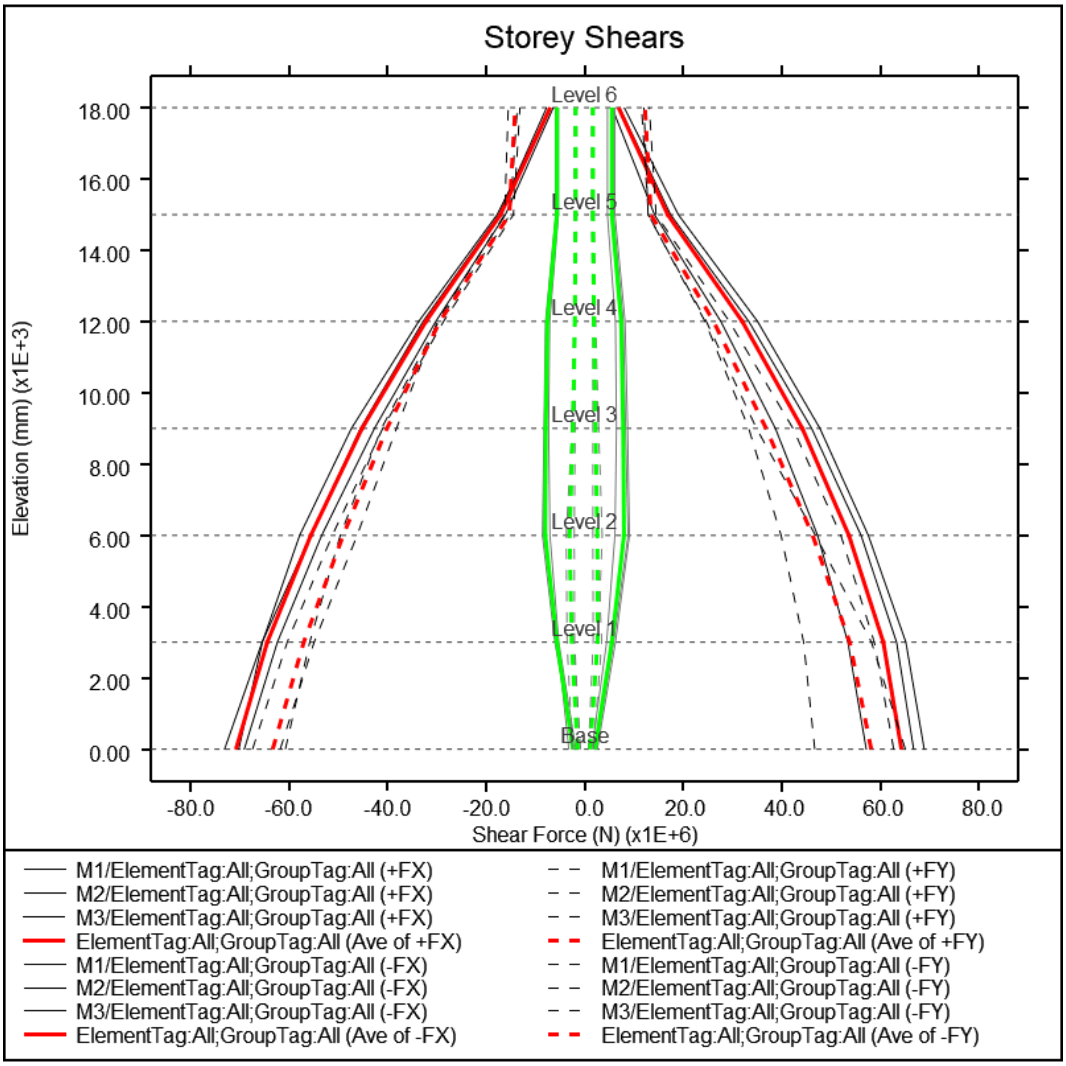


You can categorise the SET_PARTs and force components by **Colour** or **Line style.** For instance, if you display the SET_PARTs by colour and the force components by line style, the tool will then assign one colour for all curves under a SET_PART and will assign one line style for all curves under a force component. As shown in the example below, all curves under **ElementTag:All;GroupTag:All** are red and all the curves representing **shear force along the X direction** (FX) have solid lines:



You can also categorise the models by Colour or Line style. However, there is a third option called **Inherit** (which is set by default). This option essentially tells the tool that the curves **will not be categorised by model**. Instead, they will just follow the formatting of the first two categories. This is particularly useful if you are more concerned with the aggregate curves and you are just displaying the model curves to see if there is an outlier compared to the aggregate curve. If you use this option, you can quickly identify visually which model curves are associated with an aggregate curve.

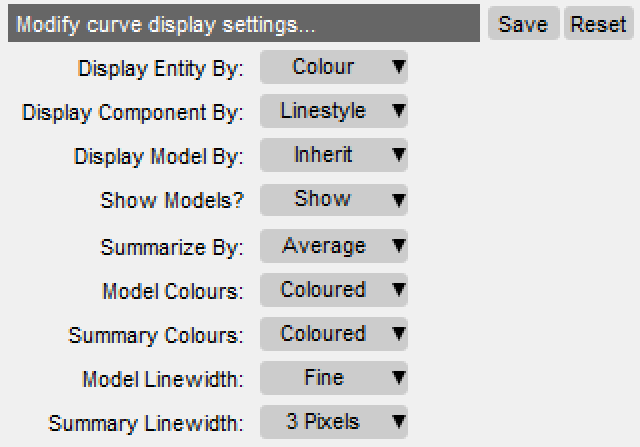
In the example below, the curves representing the **shear forces (FX)** of **ElementTag:All;GroupTag:All** are solid lines in red colour. The curve representing the mean storey forces follows the same format but with a thicker line width to differentiate it from the rest of the individual model curves under the same categories.



This current implementation of curve categorisation may not work for all scenarios, and could be improved in future. Please contact us with any feedback.

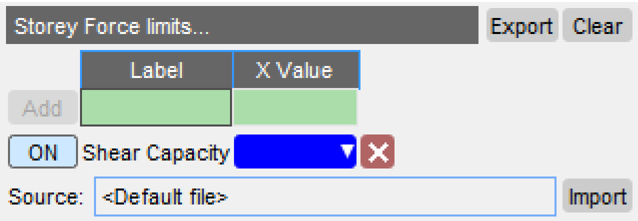
The other curve settings available to you are as follows:

1. **Show Models** – set whether the model curves are shown or hidden in the plot. This is only relevant for **multiple model mode**.
2. **Summarise by** – choose which aggregate curve is shown. You have the following options: **None, Average, Envelope**.
3. **Model Colours** – choose whether the model curves will be in **Colour** or **Greyscale**.
4. **Summary Colours** – choose whether the aggregate curves will be in **Colour** or **Greyscale**.
5. **Model Line width** – set the line width for the model curves.
6. **Summary Line width**– set the line width for the aggregate curves.



Any modifications made in this settings panel will not be automatically saved to the Workflow file. Click **Save** to write these settings to the Workflow file. You can also revert back to default settings by clicking **Reset**, which will simultaneously update these settings in the Workflow file.

Storey Force limits



This panel allows you to define vertical curve limits on the positive and negative X-axis. One scenario where this feature will be useful is when you are analysing member design utilisation – for a shear wall segment, for example. You can import the design capacity of the wall and plot it against the wall forces to illustrate whether the current wall design is acceptable.

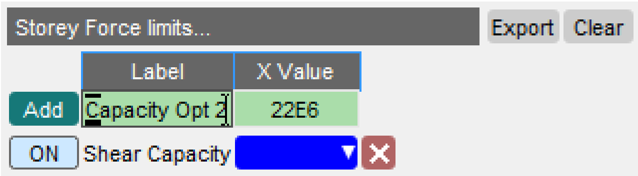
There are two types of vertical storey curve limits that you can define:

1. Constant curve limit along the structure elevation
2. Stepped curve limit, where the desired limit per storey extent varies

You can define a constant curve limit using the panel. In order to define a stepped limit curve, you need to import an external CSV file. There are no default curve limits for the Storey Force Workflow. One way to get an example curve limit input file is to create constant curve limits in the panel and then click the **Export** button to write them to a file which will show you how these data are structured.

You can also import a constant curve limit using an external file and this file can contain multiple curve limits of different types. Theoretically, you can store all your curve limits in one file to quickly generate them later.

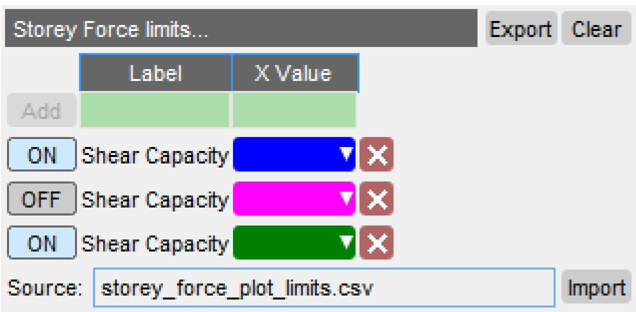
To define a constant curve limit, you need to define a label and the X-axis value in the textboxes provided. Then, click **Add**.

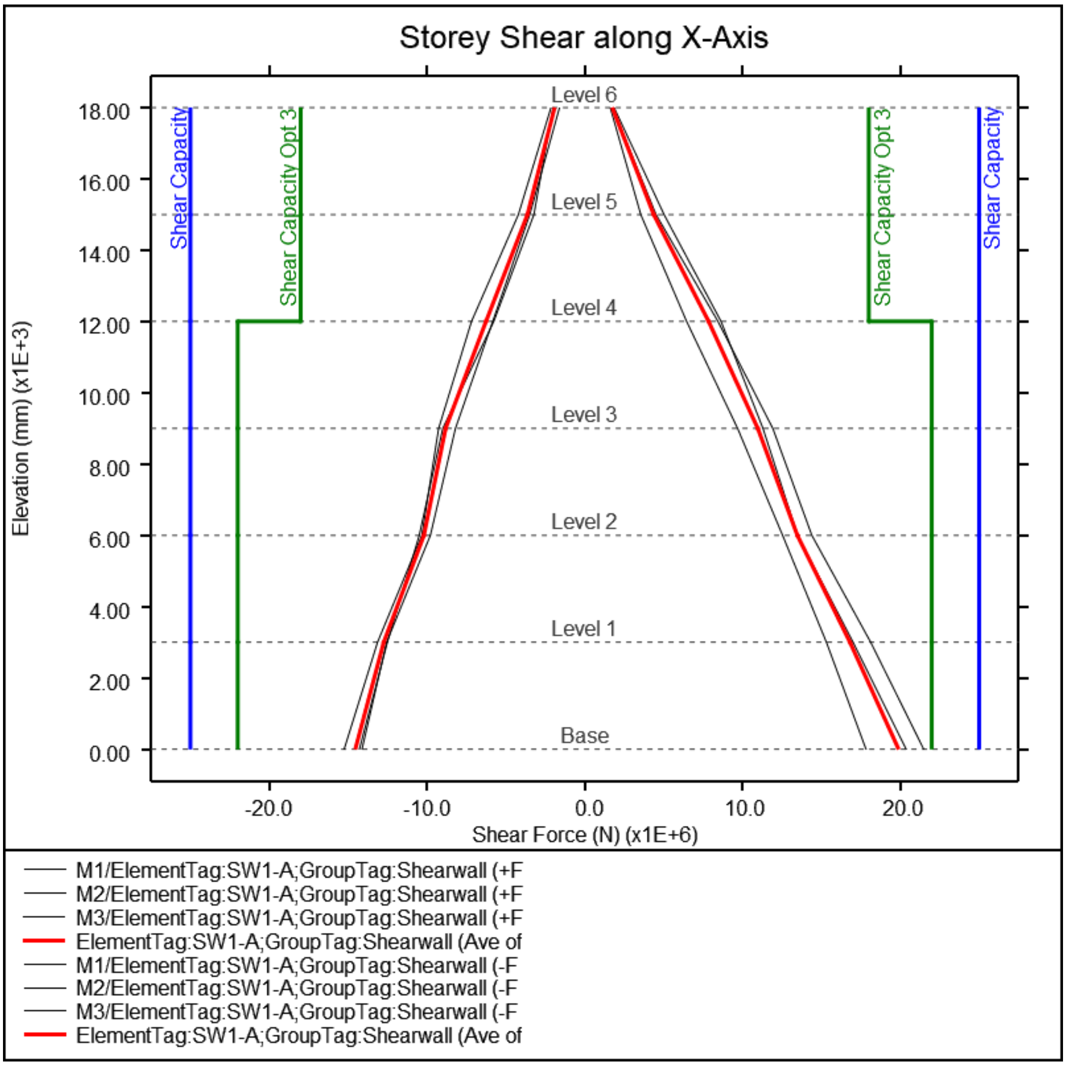


To define a stepped curve limit, create a CSV file following the data format of the exported sample file, as described above. Click **Import** to add the data to the plot.

The limits created will be listed below along with some control buttons to manipulate them:

1. Show or hide the curve limit using the **ON/OFF** toggle button
2. Change the colour of the curve limit using the provided colour selection drop-down
3. Delete a curve limit using the delete **(X)** button provided. Currently, this panel does not allow you to edit an existing curve limit. You may need to recreate a curve limit to modify the X-value(s) along the storeys.





The storey curve limits will be automatically saved to the Workflow file upon creation. Curve colour and visibility settings will also be automatically updated in the Workflow file. You may wish to store these data separately for future use. You can do so by clicking **Export** located on the right side of the panel header.

You can also revert back to default storey curve limits by clicking **Reset**.

Each Workflow will have a different set of default limits.

Storey datum settings

This panel allows you to define the formatting of the storey datums shown on the plot. This panel is hidden by default. Click the **Show** button to expand this panel.

The settings available to you are as follows:

1. **Colour** – choose the colour of the storey datums

2. **Line Style** – choose the line style of the storey datums
3. **Line Width** – choose the line width of the storey datums
4. **Label Font** – choose the font of the storey datum labels
5. **Label Size** – choose the font size of the storey datum labels
6. **Label Position** – define the location of the labels relative to the storey datums

Any modifications made in this settings panel will not be automatically saved to the Workflow file. Click **Save** to write these settings to the Workflow file. You can also revert back to the default settings by clicking **Reset**, which will simultaneously update these settings in the Workflow file.

Storey Force Report

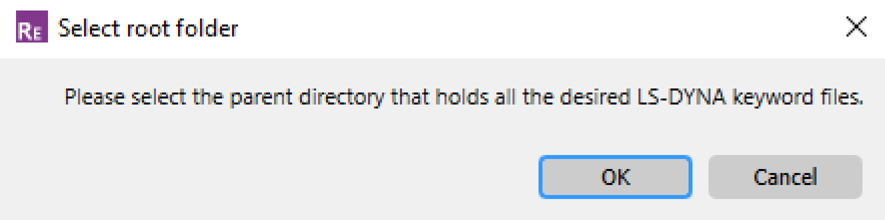
This workflow provides you with REPORTER templates to automatically generate report documents. The templates compile all T/HIS graphs you have set in PRIMER and T/HIS along with a model view from D3PLOT to show you the SET_PART(s) you have specified on each graph.

There are currently two templates with different report layouts available:

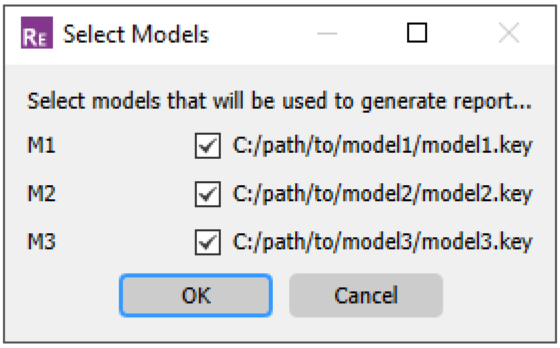
- **1x1** layout showing one T/HIS-graph/D3PLOT-model-view pair per page, split vertically.
- **2x1** layout showing two T/HIS-graph/D3PLOT-model-view pairs per page.

Running the template

Upon opening the template, you will be prompted to select the parent/root folder where all your model keyword files sit. If you have followed the recommendations for [Writing the Workflow File] from PRIMER, this should be the same directory where you have saved the Workflow file.

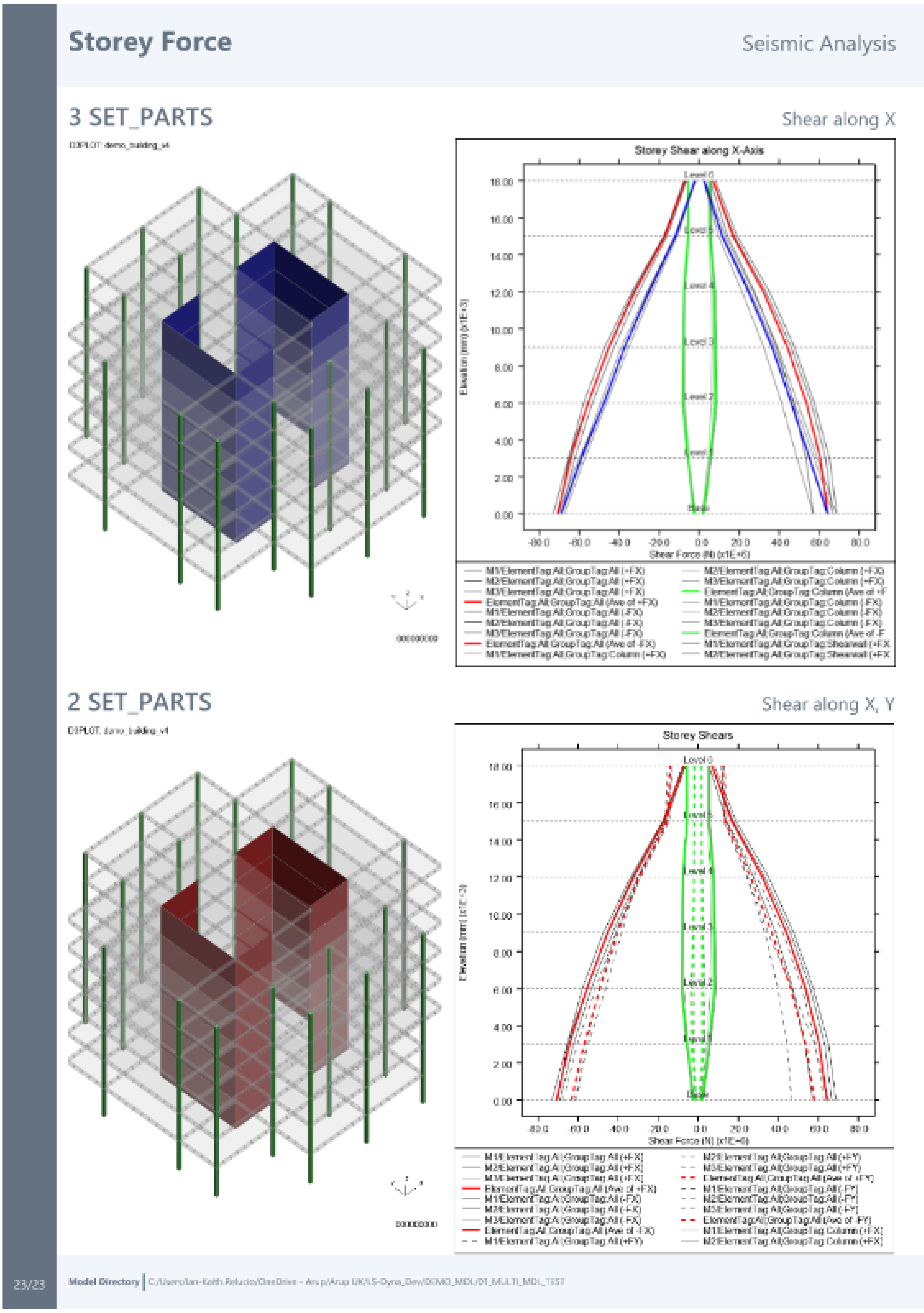


When multiple models are detected, the template will show you another window where you can choose which models to include in the report. By default, all models are selected assuming that the root folder only contains the relevant model analysis runs that you wish to process and report.



After this, the template generation should commence, running T/HIS and D3PLOT items to generate the report images. These images will also be saved into a subfolder named "reporter" that will be created when this template is generated. A sample page from a successful template run is shown below.

The REPORTER variables hold a record of the paths of models you have chosen to run. This can serve as a way to validate that you have run the models you intended.



19.11. LS-DYNA to ISO-MME

Tools → Workflows → LS-DYNA to ISO-MME

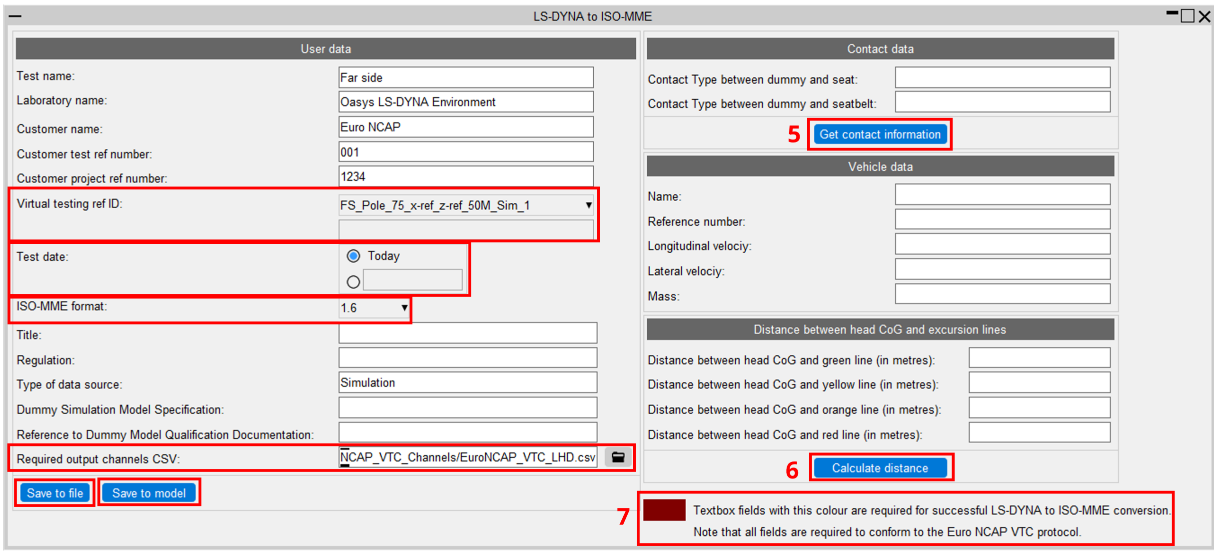
The LS-DYNA to ISO-MME workflow tool is one of the Virtual Testing tools. It helps you automatically export LS-DYNA results data to ISO-MME format as specified by the Euro NCAP Virtual Far Side Simulation & Assessment Protocol v1.0, ready for upload to the VTC Server. Automotive Assessments workflow user data removes the need to manually map LS-DYNA entities to ISO-MME channel codes.

The LS-DYNA to ISO-MME Workflow involves the following steps:

1. Create Automotive Assessments user data for the "Far Side + VTC" crash test.
2. Complete the LS-DYNA to ISO-MME setup in PRIMER
3. Perform the LS-DYNA to ISO-MME export in T/HIS
4. Optionally, automate the process in REPORTER

Setup in PRIMER

When this tool is initially launched, the GUI will look something like this by default:



All the inputs are divided in different categories e.g. User data, Contact data, Vehicle data. Some inputs are already given for reference purposes. e.g. Test name, Laboratory name, Customer name. You can see what input can be given for each textbox by hovering over it.

1. Virtual testing ref ID

Select the Virtual Testing Reference ID from the dropdown. If 'Other' is selected, the textbox below will become active to write your own Reference ID.

2. Test date

If you select "Today", the ISO-MME export will use the current date each time. If you want to enter the test date manually you can select the other radio button which will enable manual text entry.

3. ISO-MME format

You can choose between ISO-MME version 1.6 and 2.0. The Euro NCAP Virtual Far Side Simulation & Assessment Protocol v1.0 specifies version 1.6.

4. Required output channels CSV

This is the list of channels required as per the Euro NCAP Far Side VTC protocol. It will be loaded automatically. You can modify it or provide your own CSV list, but make sure to have it in same format. On each line of the CSV file, the first 16 characters need to be the ISO-MME channel code you wish to output.

5. Get contact information

You can retrieve the contact information required by the Euro NCAP Far Side VTC protocol automatically from the Automotive Assessments user data. Make sure you have added valid contact IDs in the Automotive Assessments user data to get it working. You can still input or modify information manually by editing the textbox values.

6. Calculate distance

PRIMER calculates the distance between the head centre of gravity (CoG) and green, yellow and orange lines using head node information from Automotive Assessments user data. We assume the vehicle is symmetric and centred on y = 0 and hence that the orange seat centreline y-coordinate is symmetrically opposite the occupant's head CoG y-coordinate.

7. Required inputs

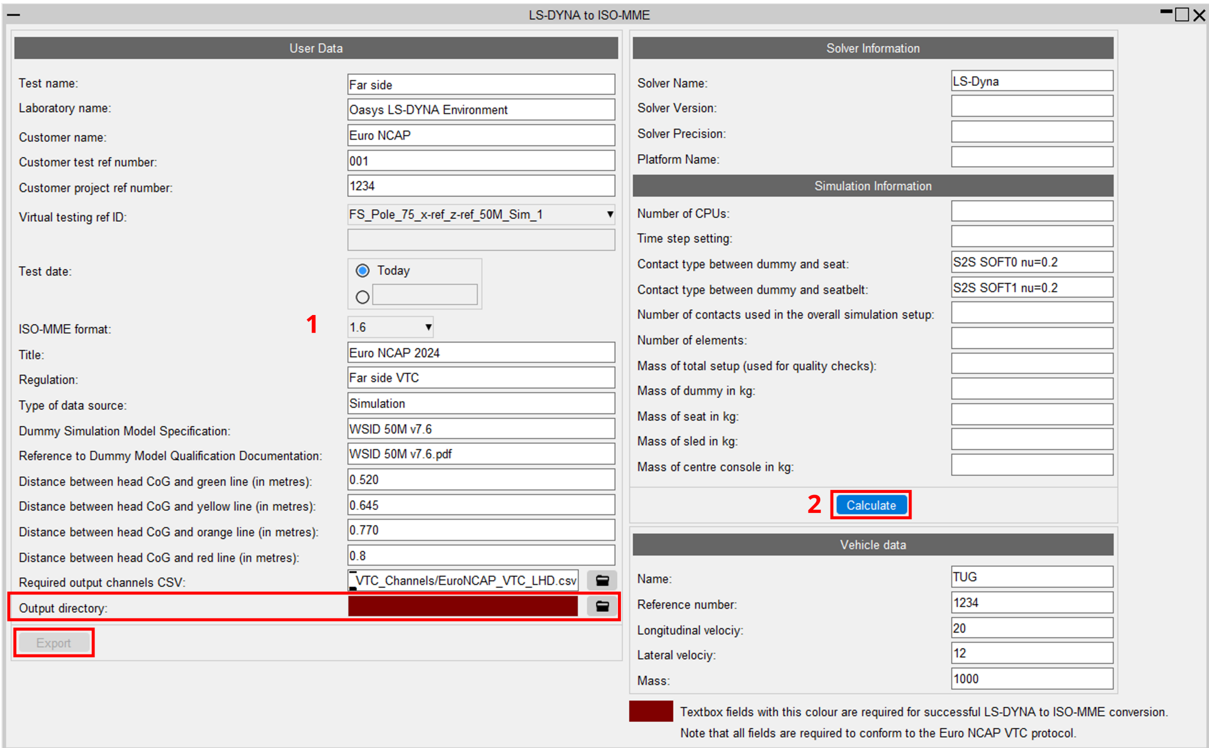
Only “Test name” and “Required output channels CSV” are required for exporting channels. However, note that technically, all inputs are required to conform to the Euro NCAP Far Side VTC protocol.

8. Saving

Save the Workflow data to a .json file or save it to your model and then write out the keyword file from PRIMER.

LS-DYNA to ISO-MME export in T/HIS

When this tool is initially launched, the GUI will look something like this by default (provided you have filled all information in PRIMER workflow panel):



1. Modify descriptors

Before performing the export in T/HIS, you can modify any of the descriptors you defined in PRIMER.

2. Calculate

Calculate can be used to automatically populate “Solver Information” and “Simulation Information” from the OTF/d3hsp file and Automotive Assessment user data.

T/HIS will automatically populate the following fields:

1. Solver Version
2. Solver Precision
3. Platform Name
4. Number of CPUs
5. Time step setting
6. Number of contacts used in the overall simulation setup
7. Number of elements

8. Mass of total setup (used for quality checks)
9. Mass of dummy in kg
10. Mass of seat in kg
11. Mass of sled in kg
12. Mass of centre console in kg

3. Output directory

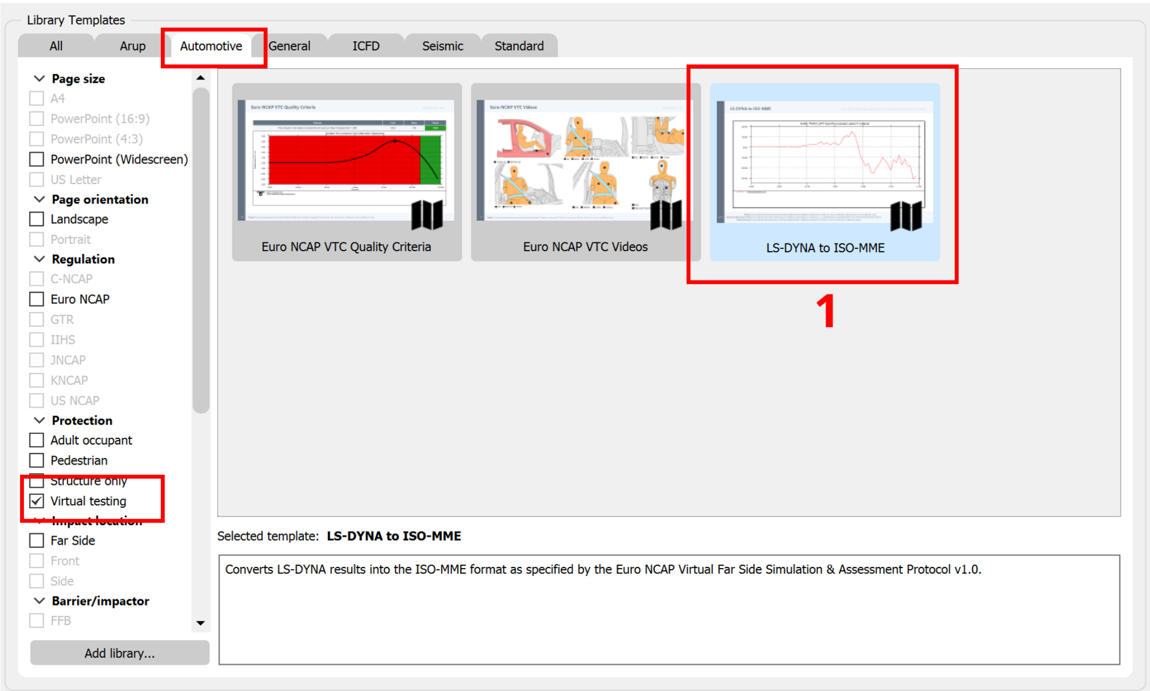
Select the output directory where you want to export channels in ISO-MME format.

4. Export

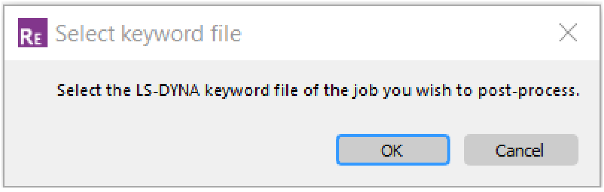
Once the output directory is selected, the **Export** button will be enabled. To perform the export, the LS-DYNA to ISO-MME workflow tool generates a configuration file from all the data and runs a separate T/HIS session in batch mode to export ISO-MME channels in the selected output directory.

Automation in REPORTER

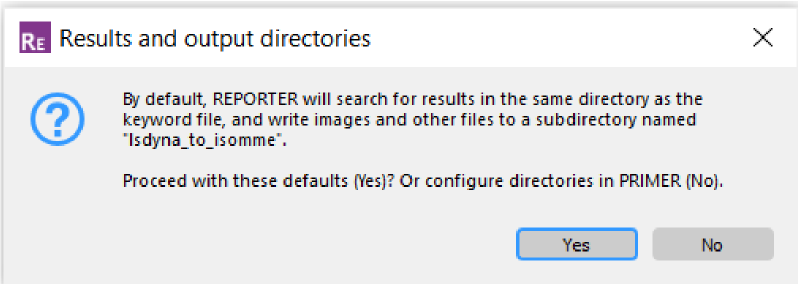
1. Within the Automotive tab in REPORTER, you will be able to select the LS-DYNA to ISO-MME Template. Filter by 'Virtual Testing' to easily find it.



2. Once open you will be prompted to select the LS-DYNA keyword file of the job you wish to post-process.

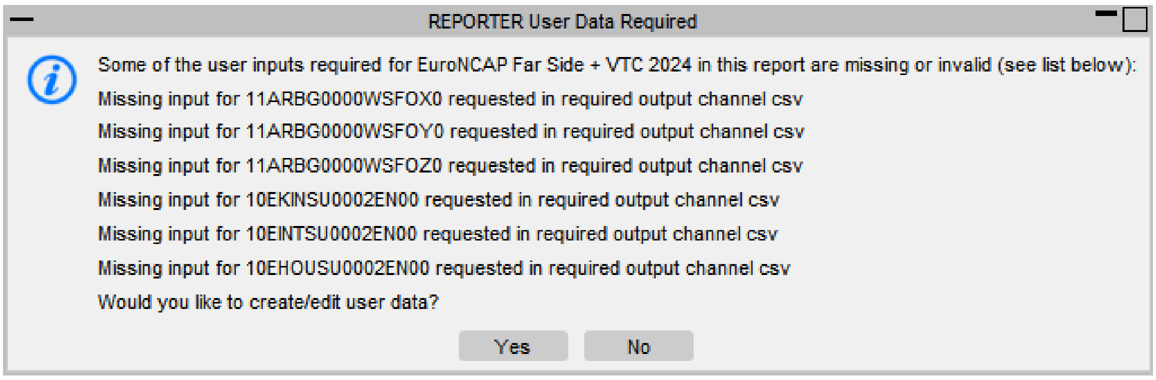


3. After you have selected your keyword file, you will then be asked if you want to continue with the default options of REPORTER searching for results in the same directory of the keyword file and writing images and outputs into a subdirectory called "lsdyna_to_isomme". If **No** is selected then PRIMER will be launched and a GUI will be displayed to configure the options. If **Yes** is selected the default options will be used.



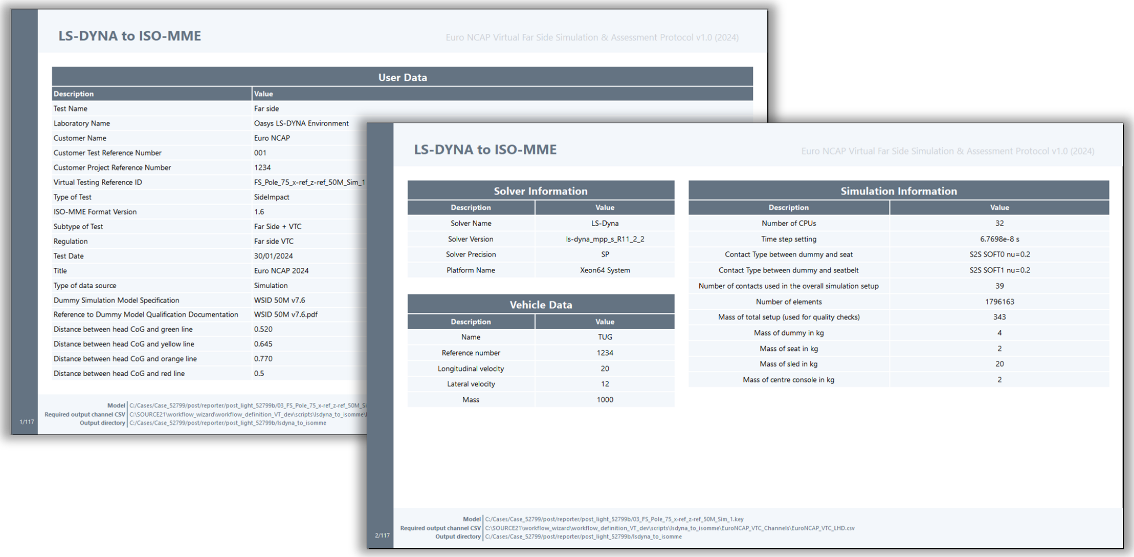
4. T/HIS will then launch automatically to produce the output files for the report.

5. If any of the required inputs are missing or invalid, T/HIS will prompt a message window (see example below) asking if you would like to create/edit user data. If you select **Yes**, PRIMER will be launched and panels will open for you to enter the required information. If you select **No** then T/HIS will try to generate the report with the data available.

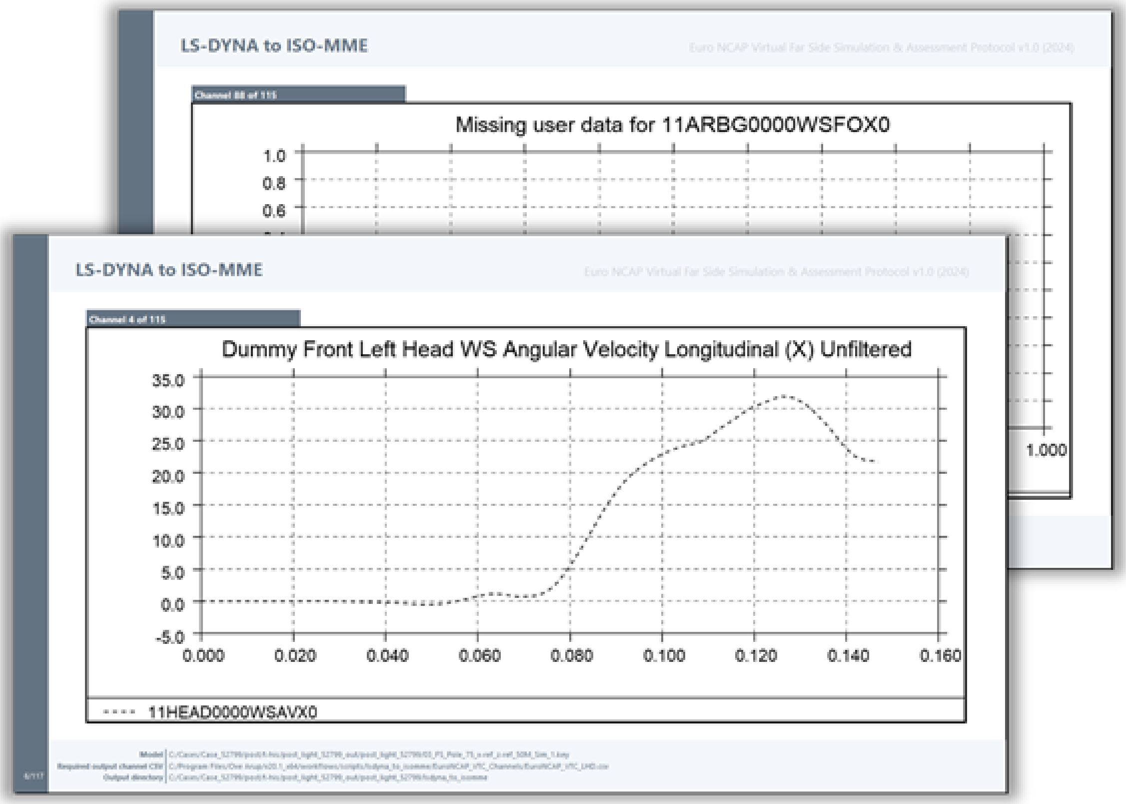


6. Once it has completed, T/HIS will close and return back to REPORTER.

7. On the first two pages, summary information is shown in table format much like the GUI output when running the Workflow manually in T/HIS:



8. On the remaining pages you can see each channel image requested in "Required output channels CSV". If the input entities were missing or invalid for a given channel, an empty graph image with a missing or invalid graph title message will be shown in the report:



19.12. Euro NCAP VTC Quality Criteria

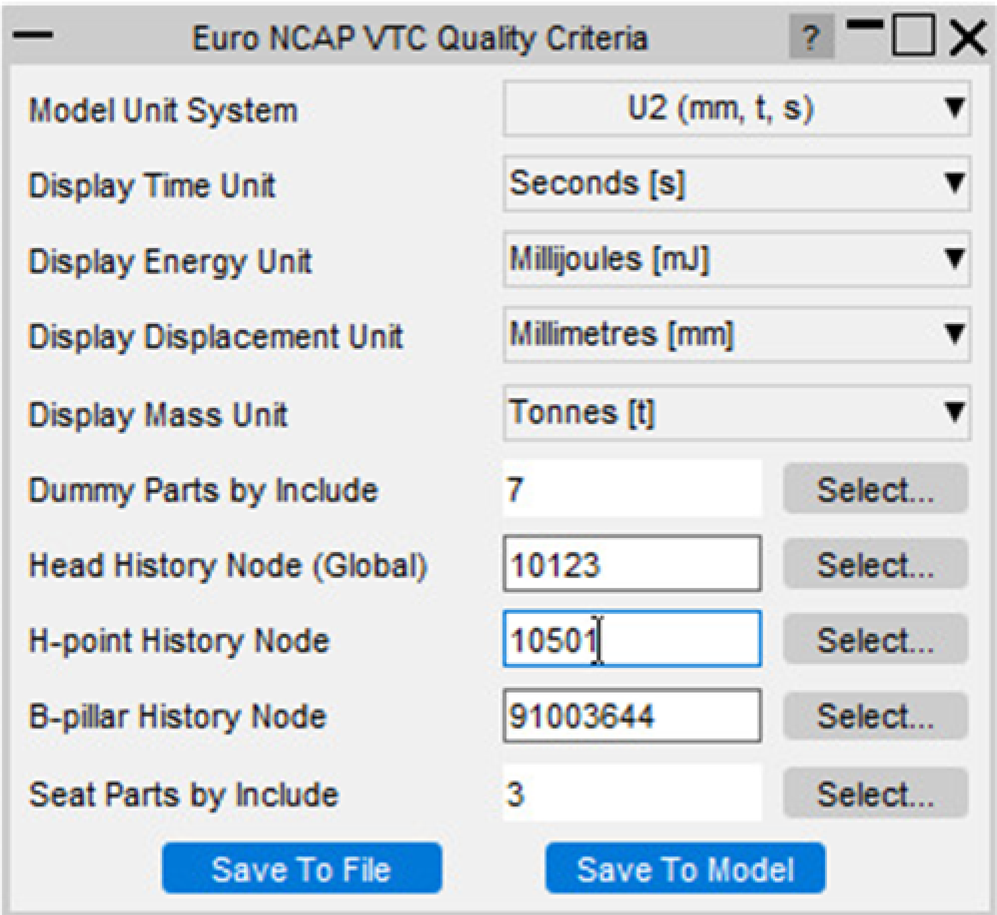
Tools → Workflows → Euro NCAP VTC Quality Criteria

The Euro NCAP VTC Quality Criteria workflow tool is part of the virtual testing protocol and allows you to perform the quality checks outlined in part 6.1 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol.

In T/HIS, this tool displays the results and graphs required for the Simulation Set-Up. In PRIMER we can set the tool up, by selecting the model unit system, selecting the WSID Dummy Include, Head Node, H-point Node and Seat Include.

How to use the Workflow Tool in PRIMER

When this tool is initially launched, the GUI will look something like this by default:



Model Unit System

Select the unit system of your model.

Display Time Unit

Select the display time unit for the graph outputs, either Seconds or Milliseconds.

Display Energy Unit

Select the display energy unit for the graph outputs, either Joules, Millijoules, Kilojoules or Foot-Pounds.

Display Displacement Unit

Select the display displacement unit for the graph outputs, either Metres, Millimetres or Feet.

Display Mass Unit

Select the display mass unit for the graph outputs, either Kilograms, Tonnes, Grams or Slugs.

WSID Dummy Include

Select the include file containing the WorldSID Dummy by pressing the Select... button.

Head Node

Select the DATABASE_HISTORY_NODE matching the Global Head Node of the WorldSID Dummy by pressing the Select... button or manually typing in the textbox. The default is 10123.

H-point Node

Select the DATABASE_HISTORY_NODE matching the H-point Node of the WorldSID Dummy by pressing the Select... button or manually typing in the textbox. The default is 10501.

B-pillar Node

Select the DATABASE_HISTORY_NODE matching the B-pillar Node of the Vehicle by pressing the Select... button or manually typing in the textbox.

Seat Include

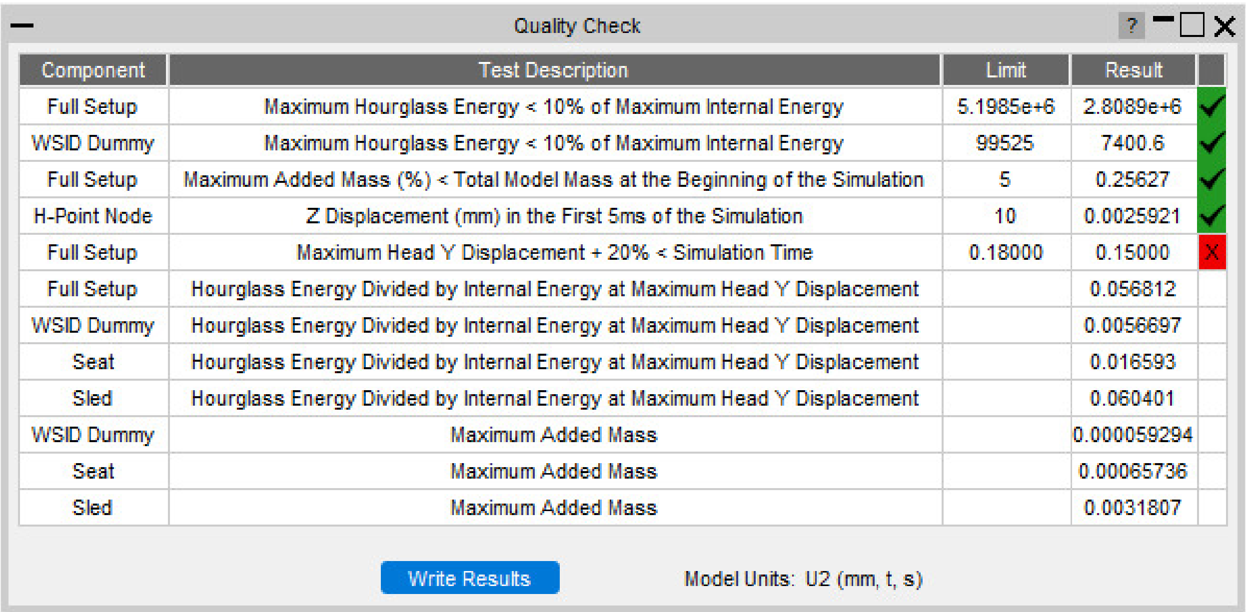
Select the include file containing the Seat of the model by pressing the Select... button.

Saving

Save the Workflow data to a .json file or save it to your model and then write the keyword file from PRIMER.

How to use the Workflow Tool in T/HIS

When this tool is initially launched, the tool will perform the quality checks automatically.
Once the run has completed the GUI will look something like the following image by default, with 7 checks presented on it's own graph on a single page.
For a full breakdown of each graph and it's results please see 'Understanding Each Graph and the Results' further down this manual.



Component	Test Description	Limit	Result	
Full Setup	Maximum Hourglass Energy < 10% of Maximum Internal Energy	5.1985e+6	2.8089e+6	✓
WSID Dummy	Maximum Hourglass Energy < 10% of Maximum Internal Energy	99525	7400.6	✓
Full Setup	Maximum Added Mass (%) < Total Model Mass at the Beginning of the Simulation	5	0.25627	✓
H-Point Node	Z Displacement (mm) in the First 5ms of the Simulation	10	0.0025921	✓
Full Setup	Maximum Head Y Displacement + 20% < Simulation Time	0.18000	0.15000	X
Full Setup	Hourglass Energy Divided by Internal Energy at Maximum Head Y Displacement		0.056812	
WSID Dummy	Hourglass Energy Divided by Internal Energy at Maximum Head Y Displacement		0.0056697	
Seat	Hourglass Energy Divided by Internal Energy at Maximum Head Y Displacement		0.016593	
Sled	Hourglass Energy Divided by Internal Energy at Maximum Head Y Displacement		0.060401	
WSID Dummy	Maximum Added Mass		0.000059294	
Seat	Maximum Added Mass		0.00065736	
Sled	Maximum Added Mass		0.0031807	

Write Results

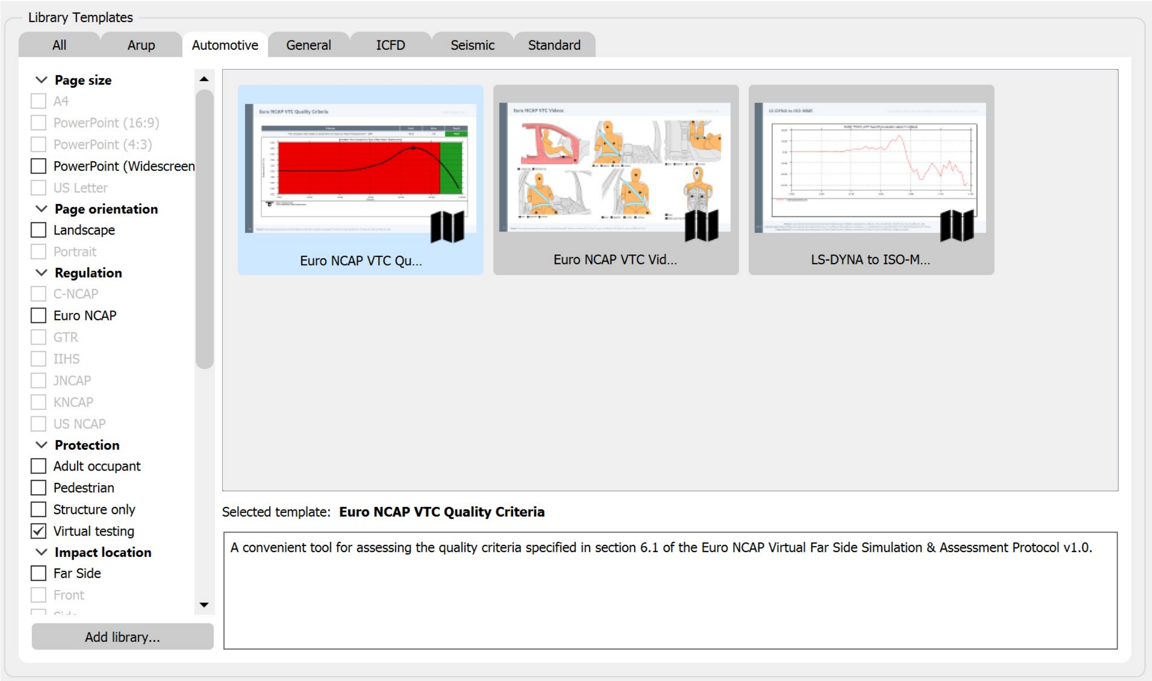
Writes the results out as displayed in the table in CSV format.

Model Unit System

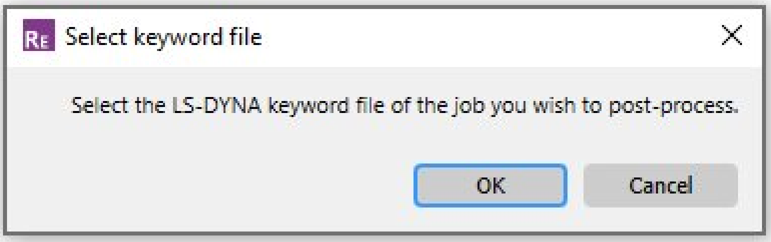
Displays the unit system that has been selected in PRIMER for this model.

How to use the Workflow Tool in REPORTER

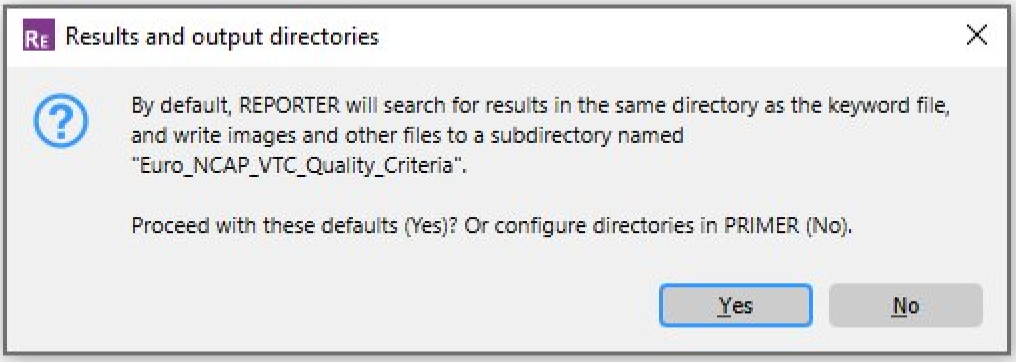
1. Within the Automotive tab in REPORTER, select the Euro NCAP VTC Quality Criteria template. It can be found by filtering for 'Virtual Testing'.



2. Once open you will be prompted to select the LS-DYNA keyword file of the job you wish to post-process.



3. You will then be asked whether you want to continue with the default results and output directories or configure them in PRIMER.



4. On the first page an overview of the results will be presented in a table format much like the GUI output when running the Workflow manually in T/HIS. On the remaining pages you can see each 'Check' one by one with its results in more detail.

Euro NCAP VTC Quality Criteria

2024 (Version 1.0)

Summary

Component	Test Description	Value	Limit	Result
Full Setup	Maximum Hourglass Energy < 10% of Maximum Internal Energy	2.8089e+6	5.1985e+6	PASS
WSID Dummy	Maximum Hourglass Energy < 10% of Maximum Internal Energy	7400.6	99525	PASS
Full Setup	Maximum Added Mass (%) < Total Model Mass at the beginning of the simulation	0.25627	5	PASS
H-Point Node	Z Displacement (mm) in the first 5 ms of the simulation	70.006	10	FAIL
Full Setup	(Time of Maximum Head Y Displacement) + 20% < Simulation Time	0.15	0.18	FAIL
Full Setup	Hourglass Energy divided by Internal Energy at Time of Maximum Head Y Displacement	0.056812	[monitored]	[monitored]
WSID Dummy	Hourglass Energy divided by Internal Energy at Time of Maximum Head Y Displacement	0.0056697	[monitored]	[monitored]
Seat	Hourglass Energy divided by Internal Energy at Time of Maximum Head Y Displacement	0.016593	[monitored]	[monitored]
Sled	Hourglass Energy divided by Internal Energy at Time of Maximum Head Y Displacement	0.060401	[monitored]	[monitored]
Dummy	Maximum Added Mass	5.9294e-5	[monitored]	[monitored]
Seat	Maximum Added Mass	0.00065736	[monitored]	[monitored]
Sled	Maximum Added Mass	0.0031807	[monitored]	[monitored]

1/8 Model | C:/Users/harry.graham/Documents/Work/QualityCheck/far_side_Pole_2022.key

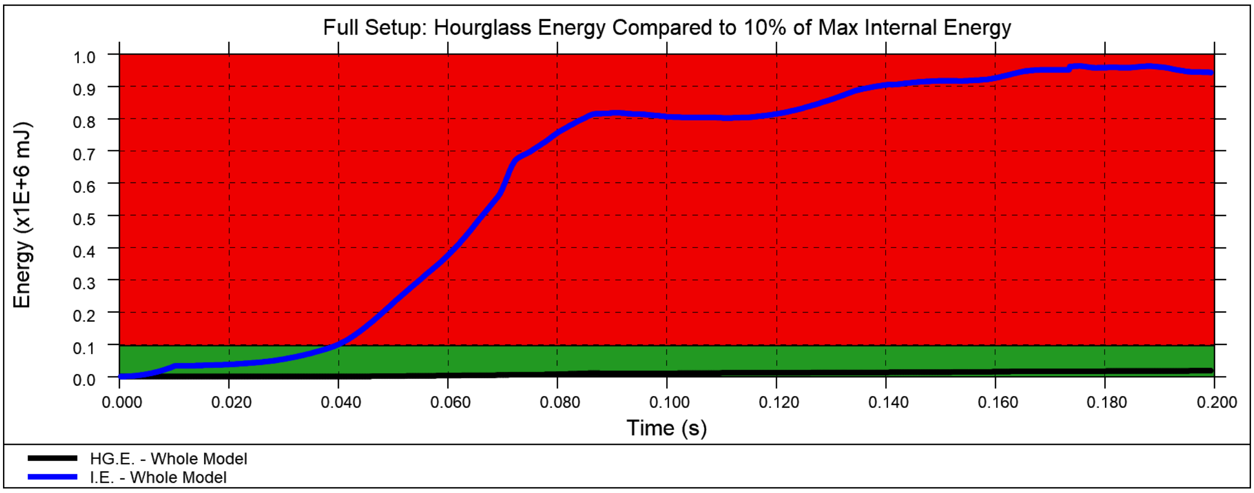
Understanding Each Graph and the Results

Full Setup: Maximum Hourglass Energy < 10% of Maximum Internal Energy

The first graph displays the quality check satisfying the following criteria from part 6.1.2 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol:

Max. Hourglass Energy of full setup must be < 10% of max. internal energy.

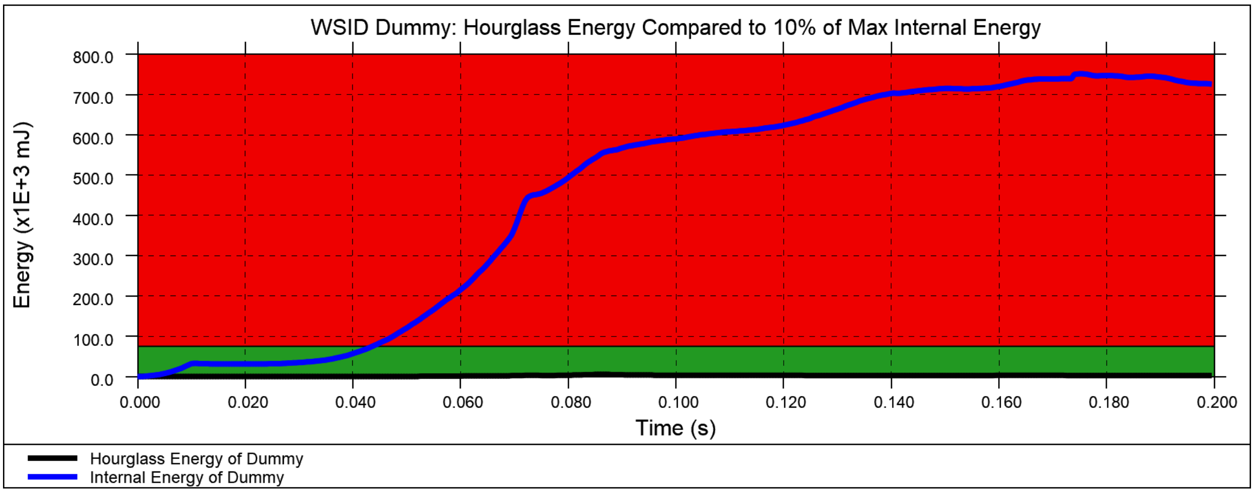
In blue colour, the internal energy of the full setup is displayed.
The datum line is drawn at 10% of the maximum internal energy.
In foreground colour, the hourglass energy of the full setup is displayed.
For this check to pass, the peak of the hourglass energy curve must be within the green zone.
The limit and result are displayed in the table.



WSID Dummy: Maximum Hourglass Energy < 10% of Maximum Internal Energy

The second graph displays the quality check satisfying the following criteria from part 6.1.2 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol:
Max. Hourglass Energy of all WSID components must be < 10% of max. internal energy of WSID

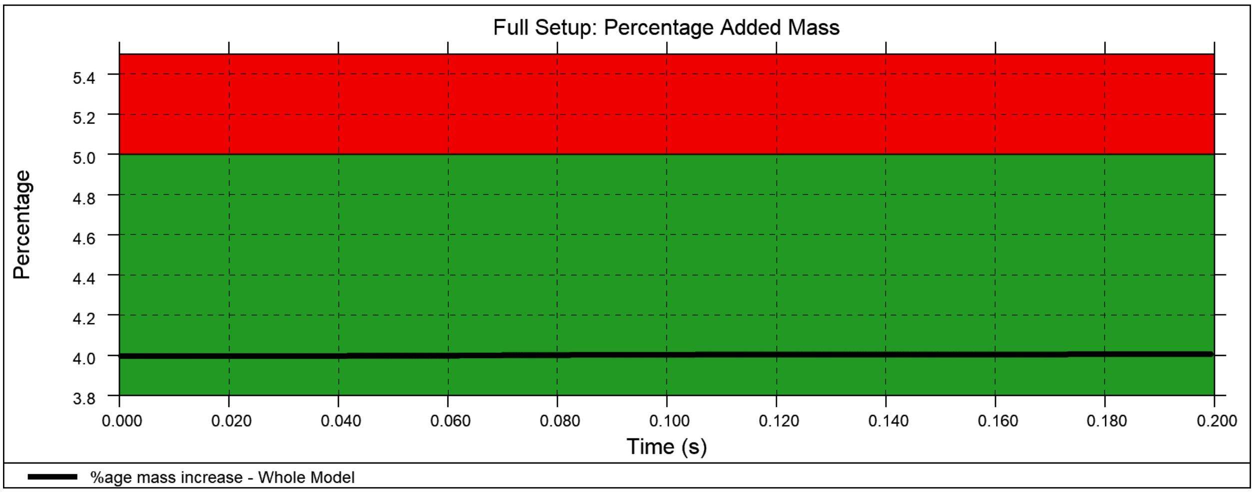
In blue colour, the internal energy of the WorldSID Dummy is displayed.
The datum line is drawn at 10% of the maximum internal energy.
In foreground colour, the hourglass energy of the WorldSID Dummy is displayed.
For this check to pass, the peak of the hourglass energy curve must be within the green zone.
The limit and result are displayed in the table.



Full Setup: Maximum Added Mass (%) < Total Model Mass at the Beginning of the Simulation

The third graph displays the quality check satisfying the following criteria from part 6.1.2 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol:
Max. mass added due to mass scaling to the total model is less than 5 % of the total model mass at the beginning of the run.

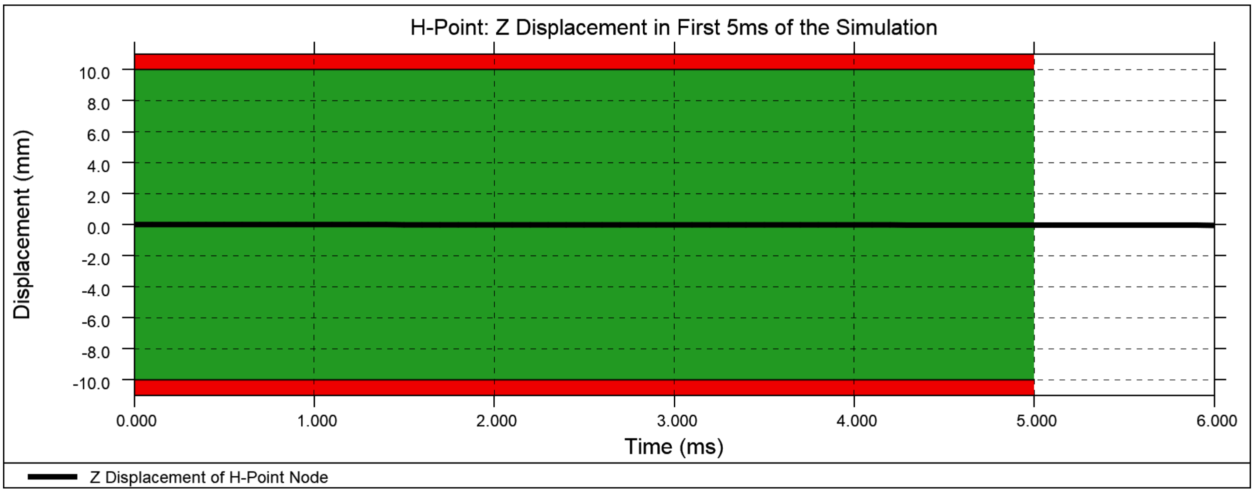
In foreground colour, the percentage mass increase is displayed.
The datum line is drawn at 5%.
For this check to pass, the peak of the percentage mass increase curve must be within the green zone.
The limit and result are displayed in the table.



H-Point Node: Z Displacement (mm) in the First 5ms of the Simulation

The fourth graph displays the quality check satisfying the following criteria from part 6.1.2 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol:
Less than 10 mm H-point z-displacement recorded in first 5 ms of the simulation (5ms after t0).

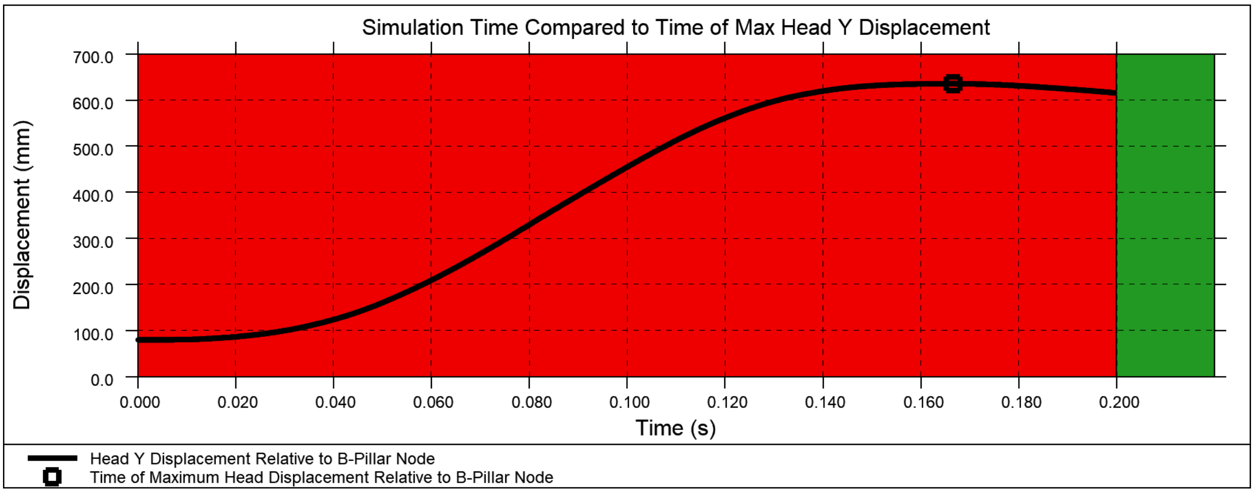
In foreground colour, the Z displacement of the H-Point Node is displayed, zoomed in to the first 6ms.
The datum line is drawn at 10mm.
For this check to pass, the peak of the Z displacement curve must be within the green zone within the first 5ms.
The limit and result are displayed in the table.



Full Setup: Maximum Head Y Displacement + 20% < Simulation Time

The fifth graph displays the quality check satisfying the following criteria from part 6.1.2 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol:
The simulation time needs to exceed time of maximum head y displacement + 20% (Equation 1).
The Head Y Displacement is calculated by taking the relative displacement compared to the B-Pillar Node Y Displacement, plus 80mm for the approximate Head diameter.

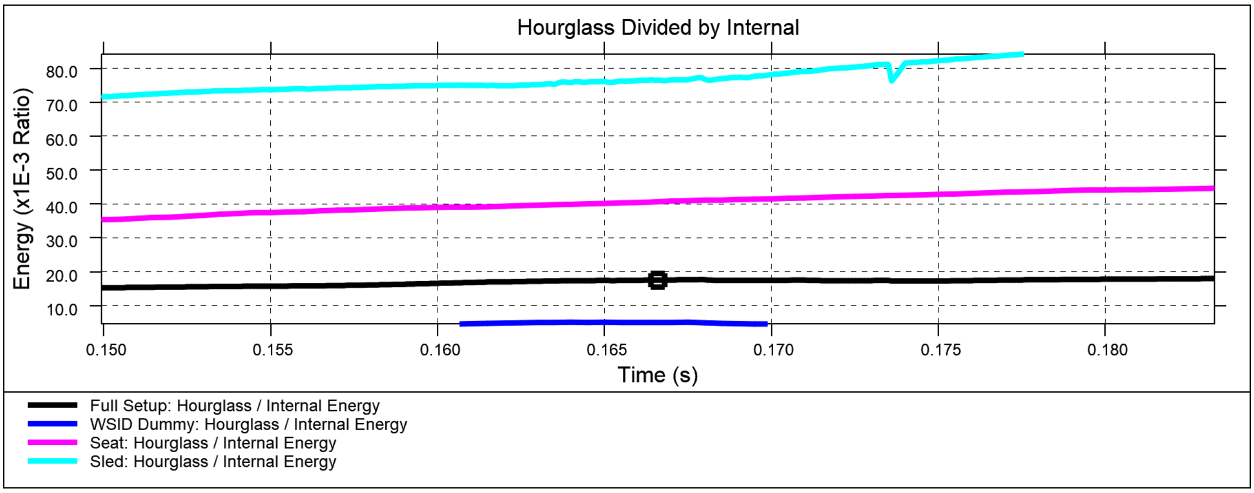
In foreground colour, the Head Y Displacement is displayed.
The datum line is drawn at Maximum Head Y Displacement Relative to B-Pillar Node Time + 20%.
For this check to pass, the Head Y Displacement curve should finish in the green zone.
The limit and result are displayed in the table.



Hourglass Energy Divided by Internal Energy at Maximum Head Y Displacement

The sixth graph displays the quality check satisfying the following criteria from part 6.1.3 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol:
Hourglass energy / internal energy at time of max. y head excursion for setup, dummy, sled and seat.

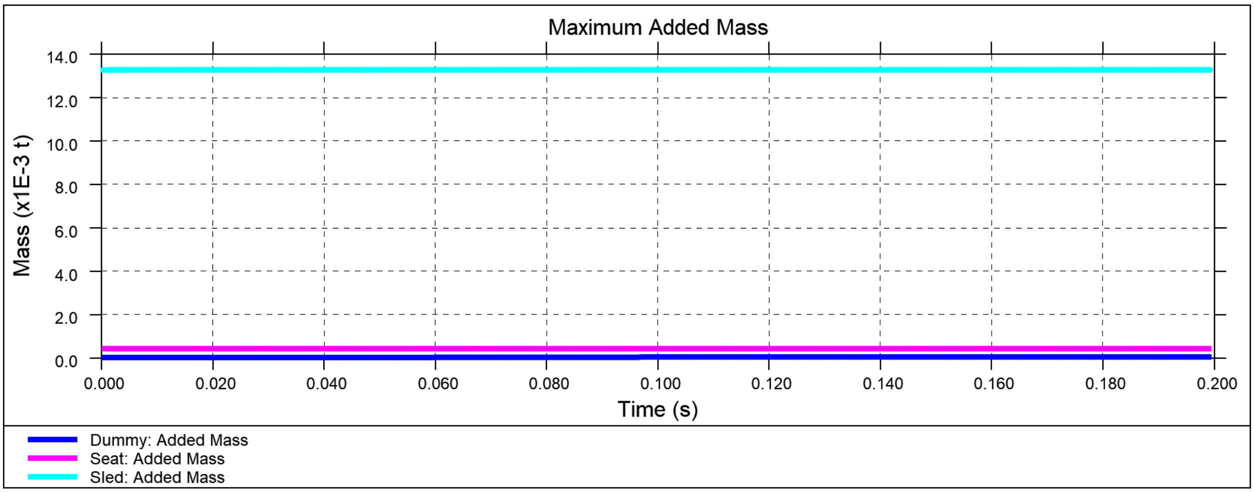
In foreground colour, the Hourglass divided by Internal Energy of the full setup is displayed.
In blue colour, the Hourglass divided by Internal Energy of the WorldSID Dummy is displayed.
In magenta colour, the Hourglass divided by Internal Energy of the Seat is displayed.
In cyan colour, the Hourglass divided by Internal Energy of the Sled is displayed.
In foreground colour, the Maximum Head Y Displacement time is displayed as a square.
There is no pass criteria for this check, it is just calculated and monitored.
The result of each curve at the Maximum Head Y Displacement is displayed in the table.



Maximum Added Mass

The seventh graph displays the quality check satisfying the following criteria from part 6.1.3 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol:
Max. added mass (Dummy, seat, sled).

In blue colour, the Added Mass of the WorldSID Dummy is displayed.
In magenta colour, the Added Mass of the Seat is displayed.
In cyan colour, the Added Mass of the Sled is displayed.
There is no pass criteria for this check, it is just calculated and monitored.
The result of each curve at the peak is displayed in the table.



T/HIS

19.13. Euro NCAP VTC Videos

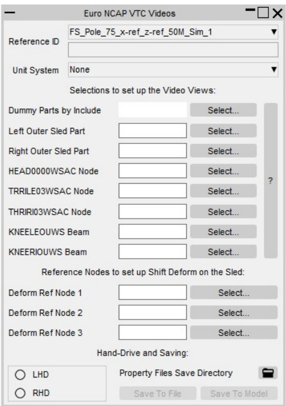
Tools → Workflows → Euro NCAP VTC Videos

The Euro NCAP VTC Videos workflow tool is part of the Virtual Testing Protocol and allows users to calculate the views and export the videos outlined in part 5.2.1 of the EuroNCAP Virtual Far Side Simulation & Assessment Protocol.

The tool attempts to calculate the camera positions automatically based on model entities you define in PRIMER. You can then adjust and save the views in D3PLOT to be reused to capture the videos for future LS-DYNA runs.

How to use the Workflow Tool in PRIMER

When this tool is initially launched, the GUI will look something like this by default:



Reference ID

Select the Virtual Testing Reference ID from the dropdown, if 'Other' is selected the textbox below will become active to write your own Reference ID.

Unit System

Select the Unit System of the model from the dropdown.

Selections to set up the video views

The following required selections are used to calculate the views. Press the '?' help buttons in the GUI for a further understanding of each selection.

Dummy Parts by Include

Press 'Select...' to Select or Pick the include for the Dummy.

Left Outer Sled Part

Press 'Select...' to Select or Pick a part for the Left Outer Sled or type the Part ID into the textbox. It should contain the door frame.

Right Outer Sled Part

Press 'Select...' to Select or Pick a part for the Right Outer Sled or type the Part ID into the textbox. It should contain the door frame.

HEAD0000WSAC Node

Press 'Select...' to Select or Pick a *DATABASE_HISTORY_NODE for the Head or type the ID into the textbox. For the WSID Dummy this is likely to be 10001.

TRRILE03WSAC Node

Press 'Select...' to Select or Pick a *DATABASE_HISTORY_NODE for the Left Lower Thorax Rib or type the ID into the textbox. For the WSID Dummy this is likely to be 10013.

THRIRI03WSAC Node

Press 'Select...' to Select or Pick a *DATABASE_HISTORY_NODE for the Right Lower Thorax Rib or type the ID into the textbox. For the WSID Dummy this is likely to be 10023.

KNEELEOUWS Beam

Press 'Select...' to Select or Pick a *DATABASE_HISTORY_BEAM for the Left Outboard Knee Load Cell or type the ID into the textbox. For the WSID Dummy this is likely to be 10012.

KNEERIOUWS Beam

Press 'Select...' to Select or Pick a *DATABASE_HISTORY_BEAM for the Right Outboard Knee Load Cell or type the ID into the textbox. For the WSID Dummy this is likely to be 10013.

Shift Deform Reference Nodes

Press 'Select...' to Select or Pick a *NODE for three Shift Deform Reference Nodes. The purpose of this is to hold the sled in position during the videos.

LHD/RHD

Using the Radio buttons, select LHD or RHD for Left Hand Drive or Right Hand Drive Occupant.

Property Files Save Directory

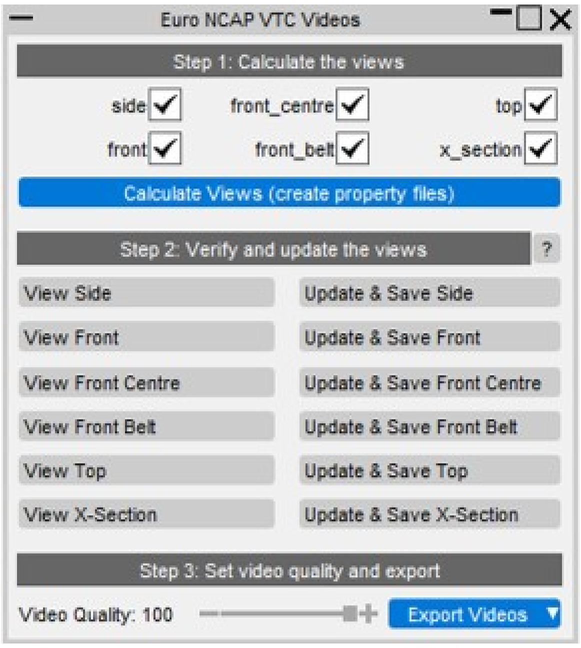
Select the Directory by pressing on the Directory icon to save the view files (Property (.prp) and Cut-Section (.cut)).

Saving

Save the Workflow data to a .json file or save it to your model and then write out the keyword file from PRIMER.

How to use the Workflow Tool in D3PLOT

When this tool is initially launched, the GUI will look something like this by default:



Step 1: Calculate the Views

Once the Workflow is clicked on, step 1 is to calculate the views. Use the checkboxes to control which views you wish to calculate.
When you click Calculate View, properties files are generated and saved in the directory you defined in PRIMER.

Step 2: Verify & Update the views

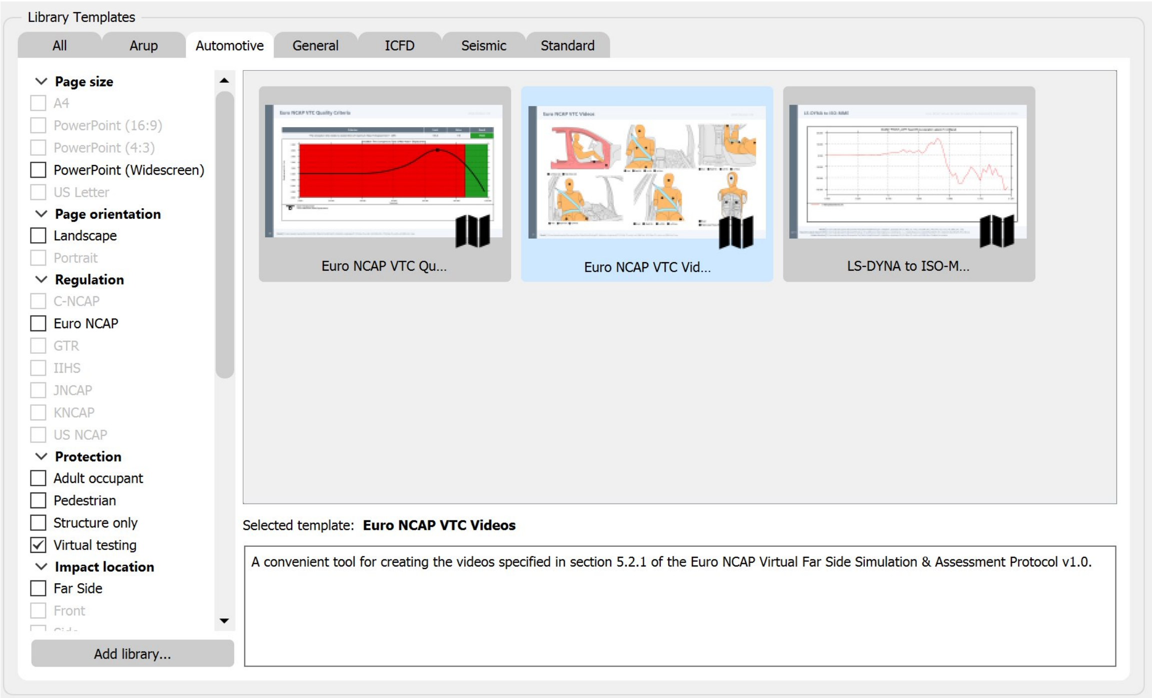
Once the views have been calculated, click the “View” buttons to see each view. If you are not satisfied with the view calculated, you can manually adjust the view by moving the camera position.
Click the “?” button to remind yourself of what the views should look like according to the Euro NCAP specification.
Once you are satisfied with the new camera position, click “Update & Save”.

Step 3: Set the video quality and export

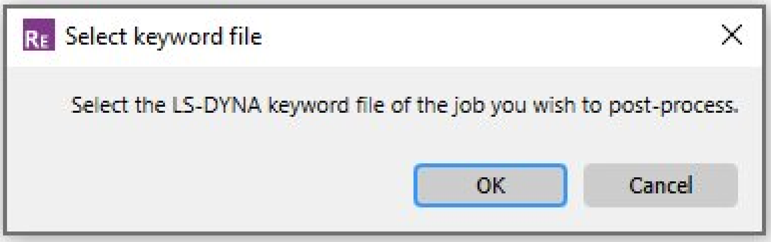
Once you have verified your views, set the video quality using the slider between 10 and 100. According to the Euro NCAP specification, the videos should be 1-10 MB in size. “Export Videos” will export all six videos by default to the directory you defined in PRIMER – you can change the views to be exported via the dropdown.

How to use the Workflow Tool in REPORTER

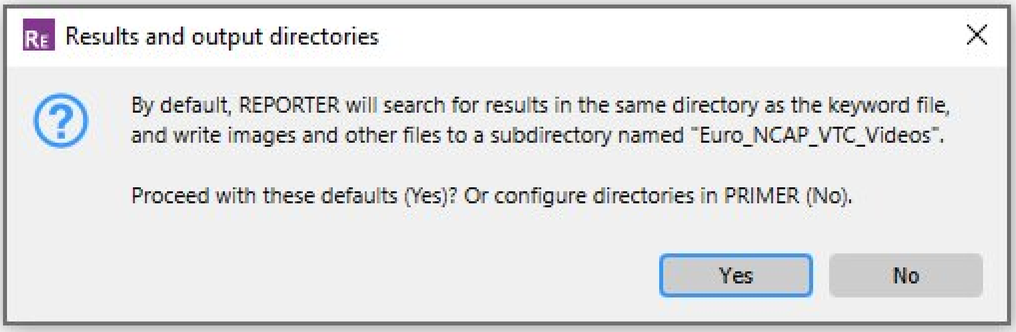
1. Within the Automotive tab in REPORTER, select the Euro NCAP VTC Videos template. It can be found by filtering for 'Virtual Testing’.



2. Once open you will be prompted to select the LS-DYNA keyword file of the job you wish to post-process.

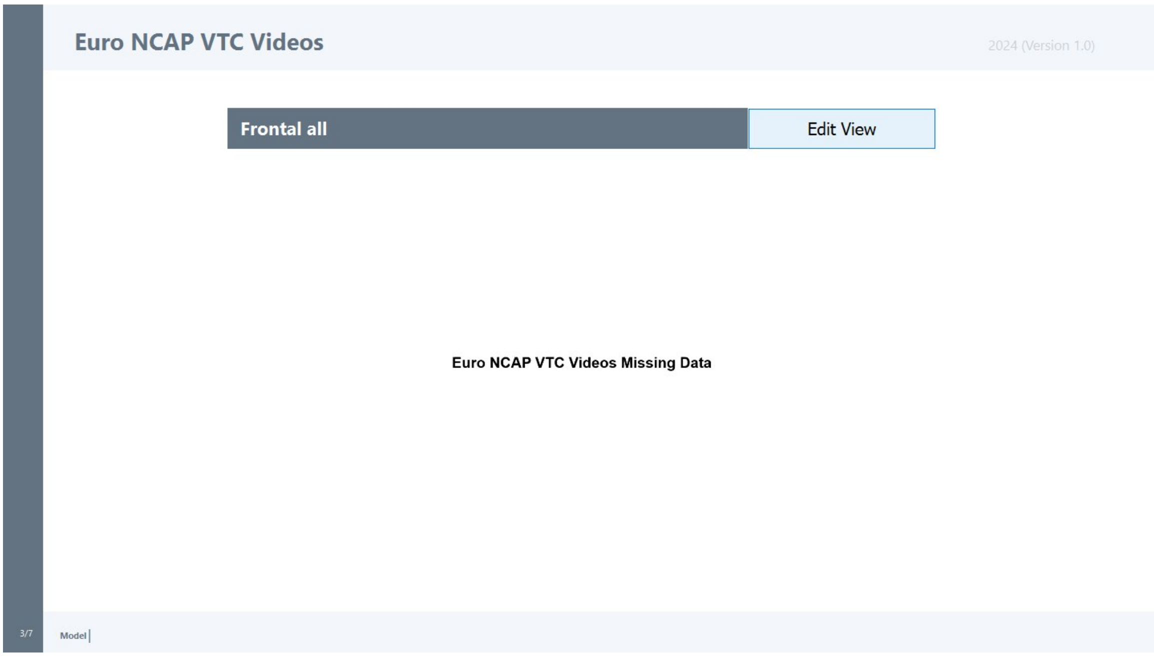


3. You will then be asked whether you want to continue with the default results and output directories or configure them in PRIMER.

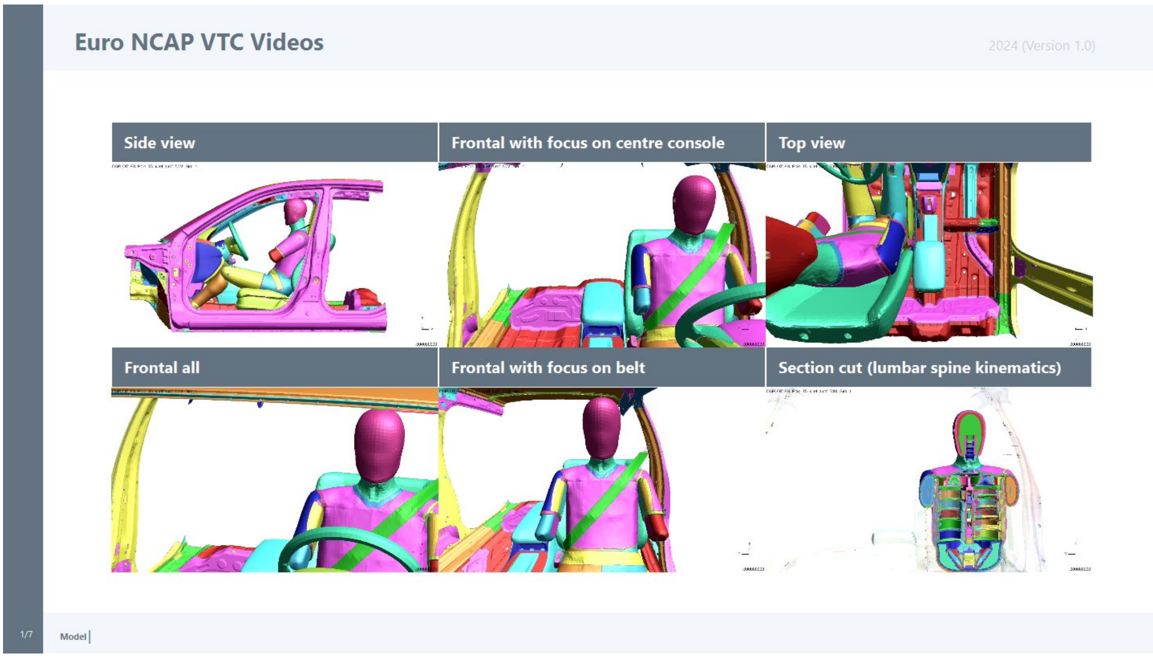


4. REPORTER will then proceed to generate the report. If there is any missing data, PRIMER will be launched for you to edit the setup. From then on, REPORTER will attempt to generate the report with the data it has available.

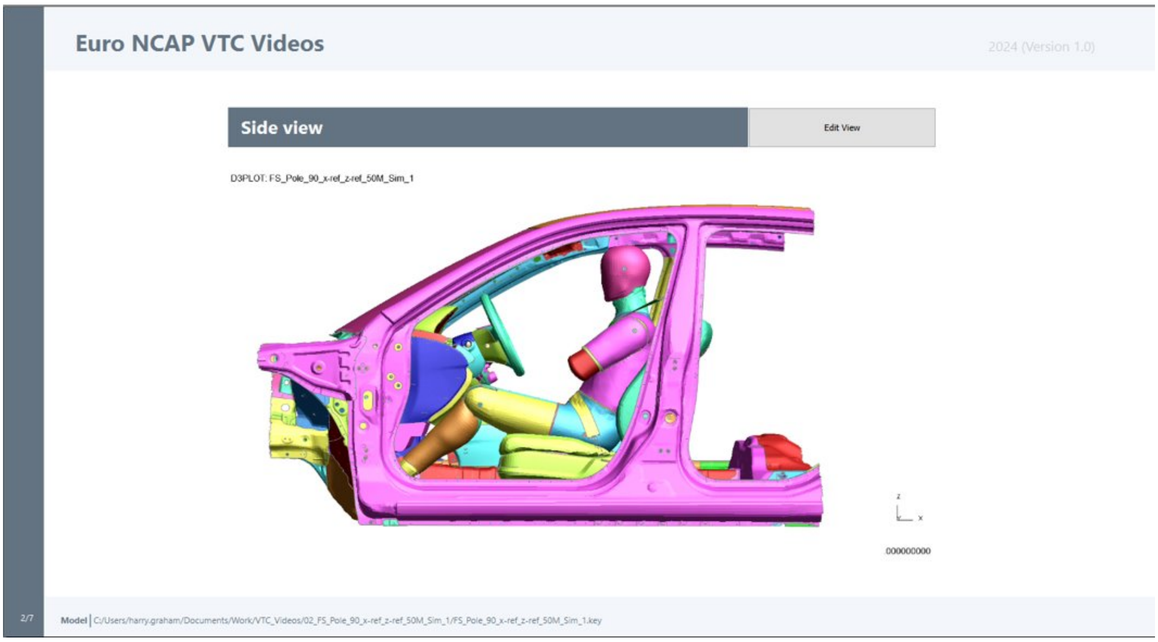
5. Where videos cannot be produced, a “missing data” image will be displayed.



6. REPORTER automatically calculates the views and exports the videos. If you have previously adjusted any of the views, REPORTER will use the saved views. On the first page of the report, an overview of the results is presented.



7. The following pages show each view in more detail. Click "Edit View" to update the camera positions if you are not satisfied with a specific view.



8. Then use the simplified D3PLOT GUI to edit the view.