Workflows 21.0 W1

Table of Contents

1. Workflows 2
2. Workflow User Data 4
3. Workflow Tools 8
   3.1. Automotive Assessments 11
      3.1.1. Automotive Assessments PRIMER 14
      3.1.2. Automotive Assessments T/HIS 31
      3.1.3. Automotive Assessments D3PLOT 39
      3.1.4. Automotive Assessments REPORTER 42
   3.2. Energy Check 46
   3.3. Entities of Interest 50
   3.4. Eroded Elements 55
   3.5. Intrusion Contour Plot 58
   3.6. Pulse Index Tool 61
   3.7. Seismic 66
      3.7.1. Storey Force 67
      3.7.2. Storey Drift 95
   3.8. Strength Check 124
   3.9. Virtual Testing 130
      3.9.1. Euro NCAP VTC Quality Criteria 133
      3.9.2. Euro NCAP VTC Videos 143
      3.9.3. LS-DYNA to ISO-MME 152
      3.9.4. SimVT 159
      3.9.5. Euro NCAP Virtual Far Side VC1 (ISO Scores) 217
      3.9.6. Euro NCAP Virtual Far Side VC2 (Assessment Criteria) 225
4. Changelog 231
   4.1. Oasys 21.0 W1 (Workflows Update 1) 232

1. Workflows

Workflows 21.0 W1 (Workflows Update 1)

The Oasys Suite contains powerful tools and capabilities that can be used to interrogate and debug your analysis results. However...

1. The basic 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.

Releases

The Workflows tools are constantly being improved and enhanced. If you have any requests for new features or experience issues using Workflows, please contact us and we will aim to address your requests in future releases. This documentation is for 21.0 W1 (Workflows Update 1). A record of all releases is listed below.

Workflows Releases

Date	Release	Description	Documentation	Minimum required Oasys Suite version	Download	Changelog
	21.0 W1	21.0 Workflows Update 1 – Comprehensive support for		21.0		

Workflows Releases

		Euro NCAP Virtual Far Side Protocol v1.0				
15-May-2024	21.0	First version containing Virtual Testing tools		21.0	Oasys Suite download	Oasys 21.0 Release Notes
08-Nov-2023	20.1	Bug fix release	Workflow Tools 20.1	20.1	No longer available	Oasys 20.1 Release Notes
17-Apr-2023	20.0	First release of Workflows	Workflow Tools 20.0	20.0	No longer available	Oasys 20.0 Release Notes

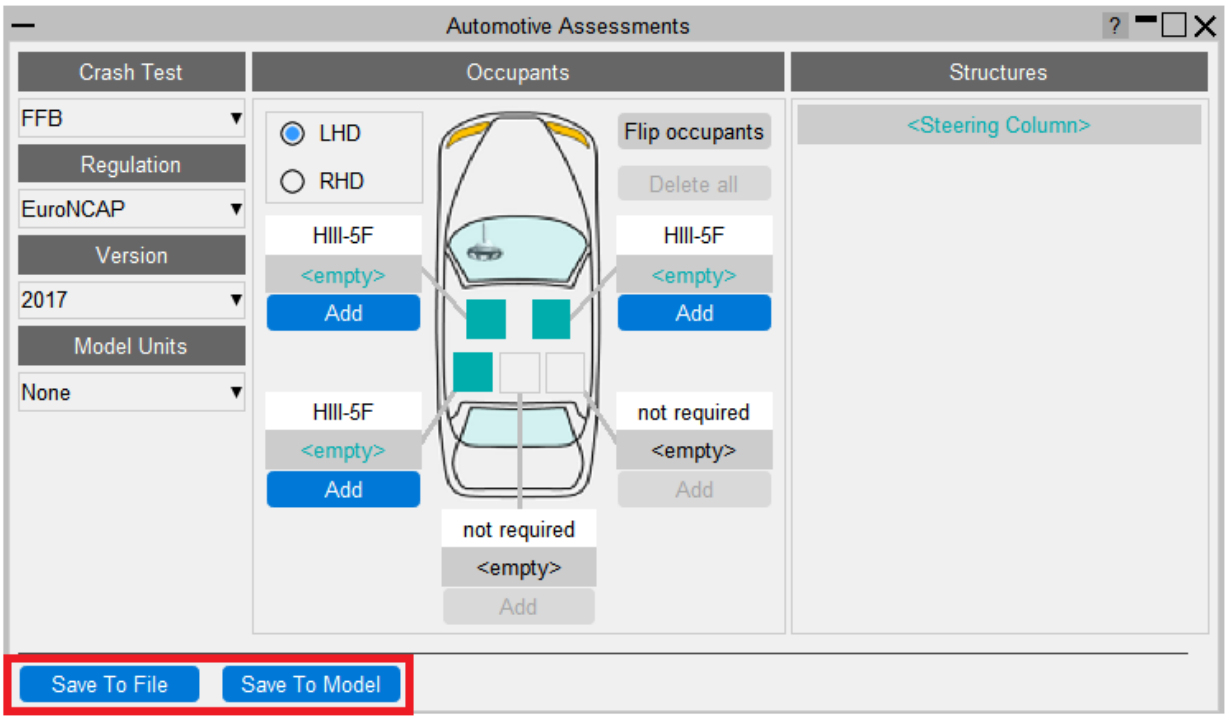
2. Workflow User Data

Saving user data

Workflows enable you to tag your models in PRIMER with user data which can be utilised in D3PLOT, T/HIS, and REPORTER to provide relevant post-processing analysis tools.

The user data can either be saved in a JSON file or as post *END data in the model keyword file.

In PRIMER, each Workflow Tool will have buttons in the 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, the file either 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 folder structure below, 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**. This allows you to store your user data files in a helpfully named directory in your LS-DYNA analysis area.

Writing user data for multiple Workflows

When writing user data for multiple Workflows, you have two options:

1. 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
```

2. 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 Workflows 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.:

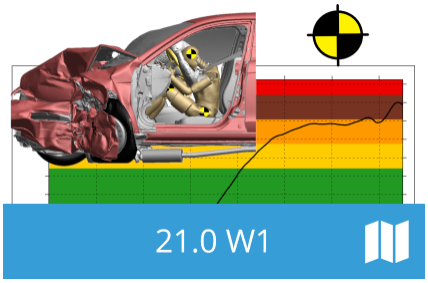
```
$
*END
*PRIMER_USER_DATA
WORKFLOW_START
{"workflows":[{"program":"PRIMER","major_version":21,"minor_version":0,"build
_ +
number":34854,"workflow_definition":{"filename":"$OA_WORKFLOW\\automotive_ass
e +
ssments.json"},"data":{"user_data_version":"21.0","regulations":["EuroNCAP"],
" +
crash_test":"FFB","version":"2017","drive_side":"LHD","occupants":[],"structu
r +
es":[],"b_pillar":null,"head_excursion":null,"head_intrusion":null},"model_un
i +
t_system":"U2"}]}
WORKFLOW_END
```

When you click **Save To Model** 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 to the keyword file.

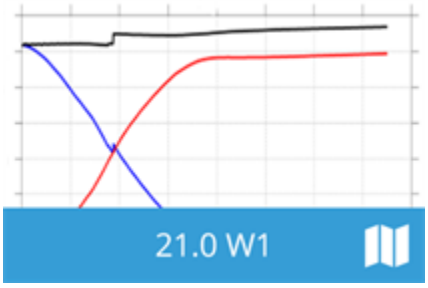
3. Workflow Tools

Workflow Tools

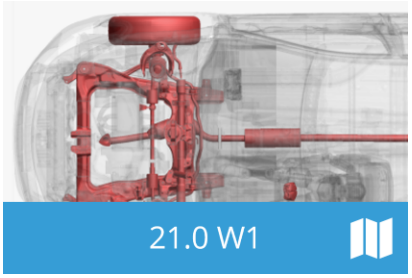
The Workflow Tools currently available are listed below (or use the topics navigator on the left). Please contact us if you have an idea for a tool and would like some help creating it.



Automotive Assessments
Process crash and occupant safety simulations according to legal regulations and consumer safety NCAP protocols

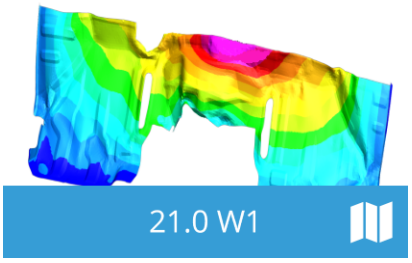


Energy Check
Plot total, kinetic, internal and hourglass energy, and perform various checks to highlight possible issues

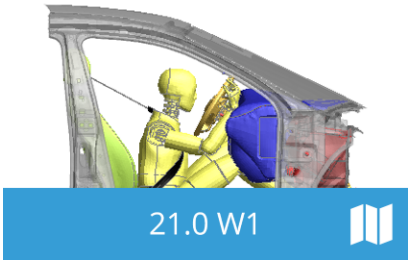


Entities of Interest
Perform actions on predefined groups of parts

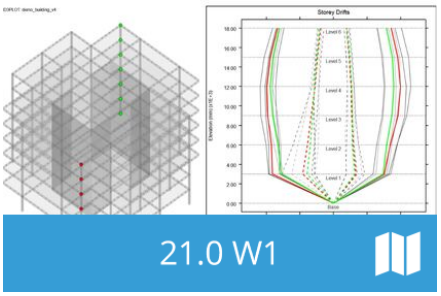
Eroded Elements
Visualise deleted elements at different states



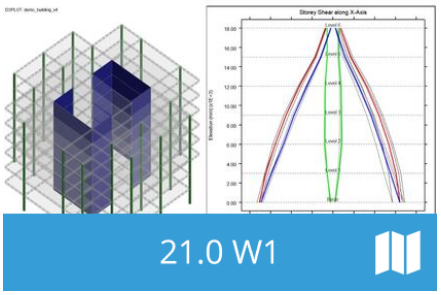
[Intrusion Contour Plot]
Produce a front intrusion contour plot (e.g. for vehicle cockpit intrusion) for selected parts



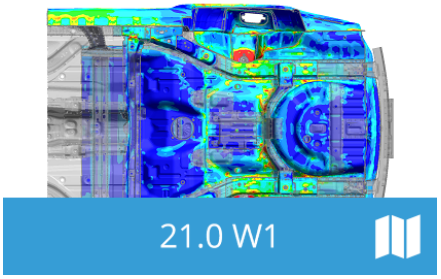
[Pulse Index Tool]
Perform a vehicle pulse index calculation for a virtual occupant



[Storey Drift]
Extract storey drift data from a seismic response time history analysis



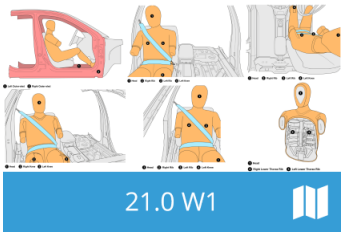
[Storey Force]
Extract storey force data from a seismic response time history analysis



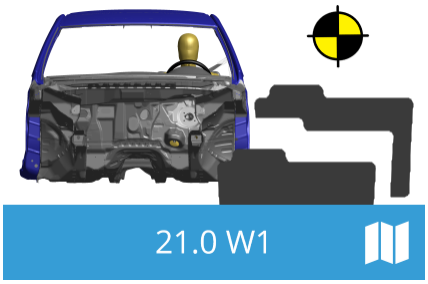
[Strength Check]
Identify and visualise yielding parts

Virtual Testing

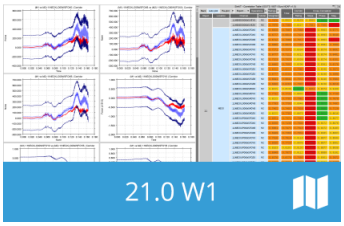
[Euro NCAP VTC Quality Criteria]
Quality Criteria check according to Section 6.1 of the Euro NCAP Virtual Far Side Simulation & Assessment Protocol



[Euro NCAP VTC Videos]
Create the videos specified in Section 5.2.1 of the Euro NCAP Virtual Far Side Simulation & Assessment Protocol



[LS-DYNA to ISO-MME]
Automatically export ISO-MME channels from LS-DYNA results



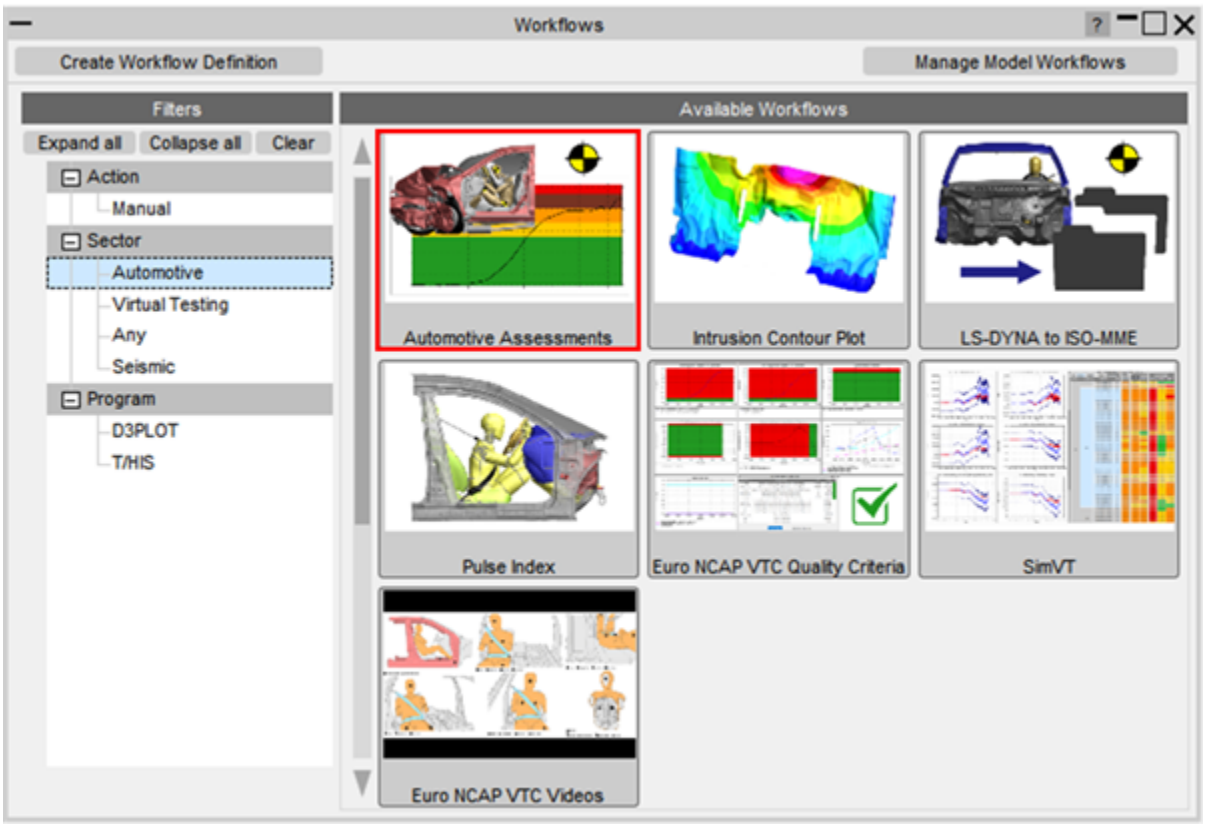
[SimVT]
Compare simulation versus test curves, or indeed any combination of: LS-DYNA models, ISO-MME data and CSV data.
Supports the ISO/TS 18571 rating method.

3.1. Automotive Assessments

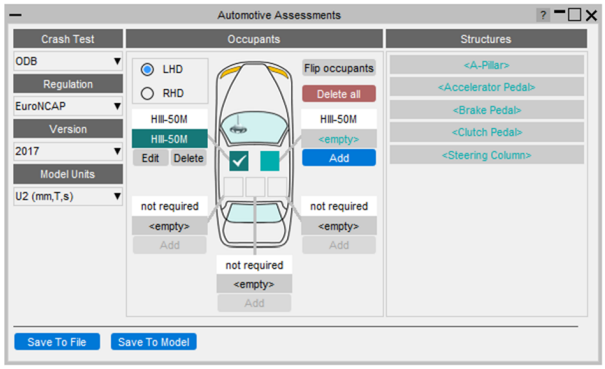
Automotive Assessments

Tools → Workflows → Automotive Assessments

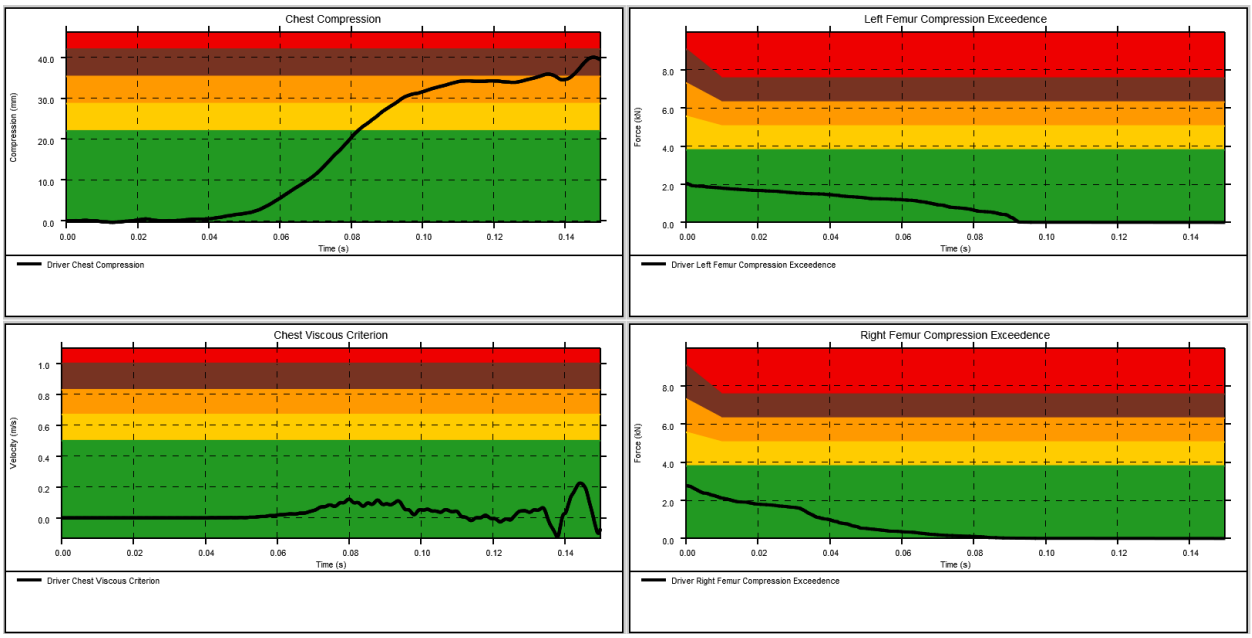
The Automotive Assessments workflow tool is used to post-process analyses according to various crash test regulations.

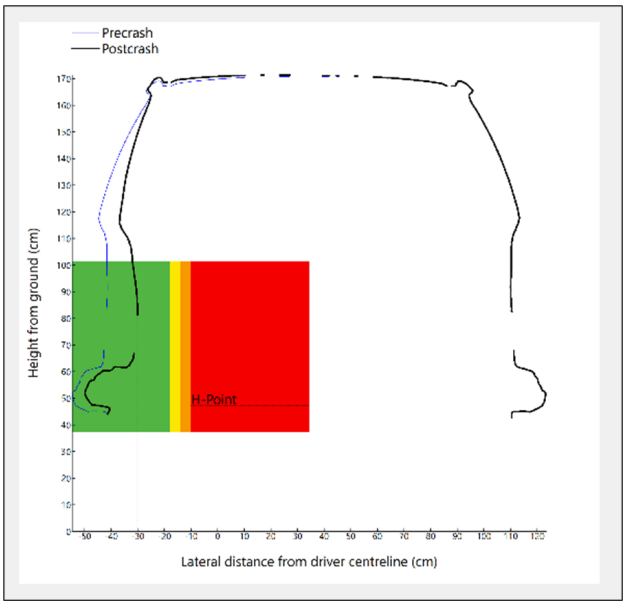


In PRIMER you select the crash test type and the occupants and structures to be assessed.



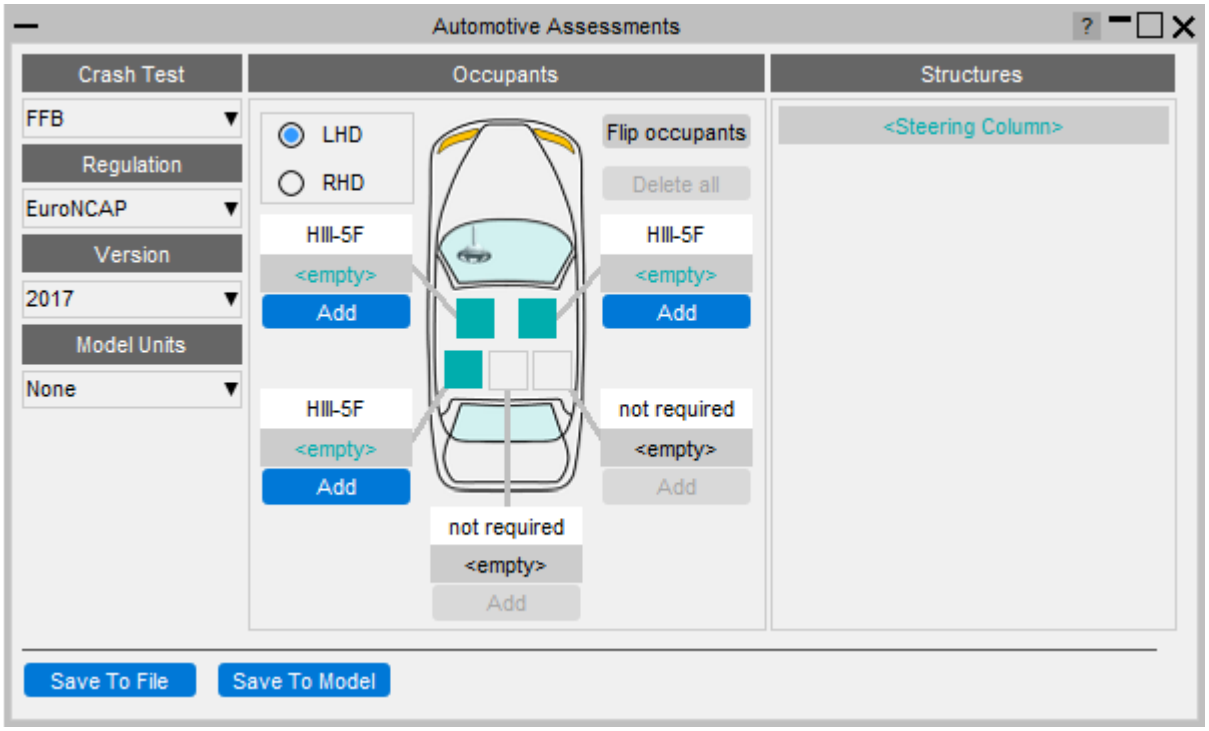
In D3PLOT, T/HIS and REPORTER this data is used to carry out assessments according to the crash test type and regulation.




Precrash
Postcrash
Height from ground (cm)
H-Point
Lateral distance from driver centreline (cm)

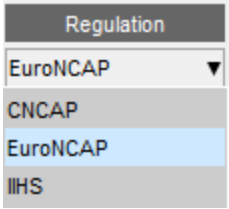
3.1.1. Automotive Assessments PRIMER

When Automotive Assessments 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:

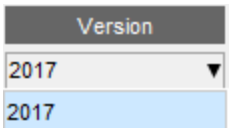


You first specify the crash test type of your model from the dropdown menu:

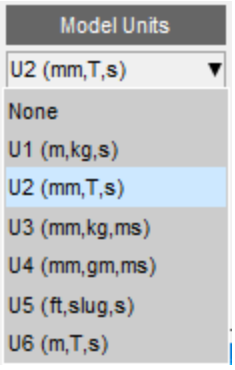
This will update the Regulation dropdown menu with regulations that are supported by the tool for the selected crash test type. Select the regulation you want to assess your model with:



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:

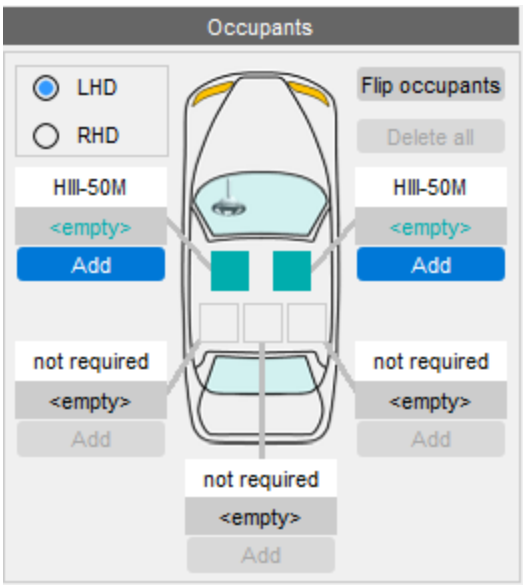


Then select the unit system of your model:

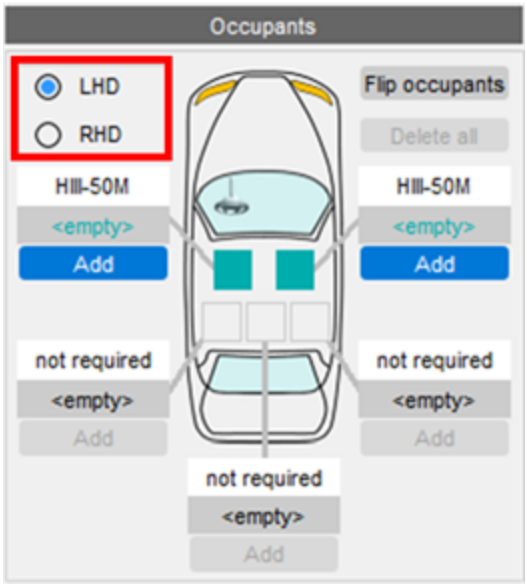


Occupants

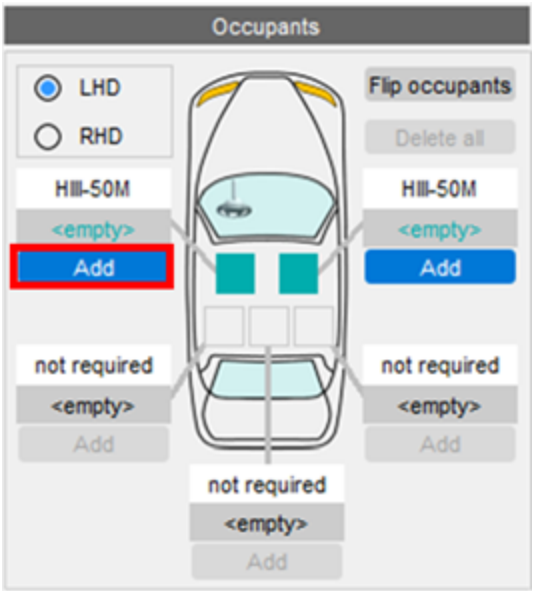
The Occupants section is used to select which occupants are in the vehicle, their positions in the vehicle and the IDs of the entities from which data can be read from. This section updates automatically to show the required occupants for the selected crash test type and regulation. In the image below 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 **Add**. 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.



Clicking **Add** opens 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

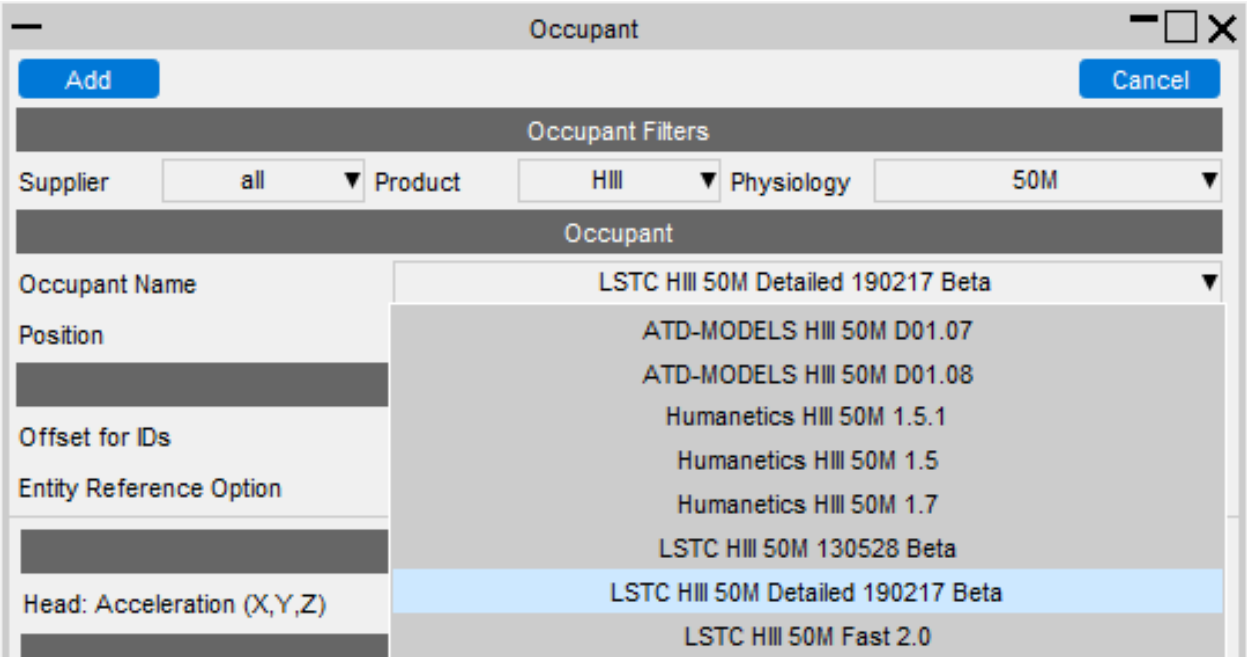
Offset for IDs	0	Get offset from include transform
Entity Reference Option	Use ID numbers + offset	Use Database History Titles First

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

The occupant type can be selected from the **Occupant Name** dropdown menu.

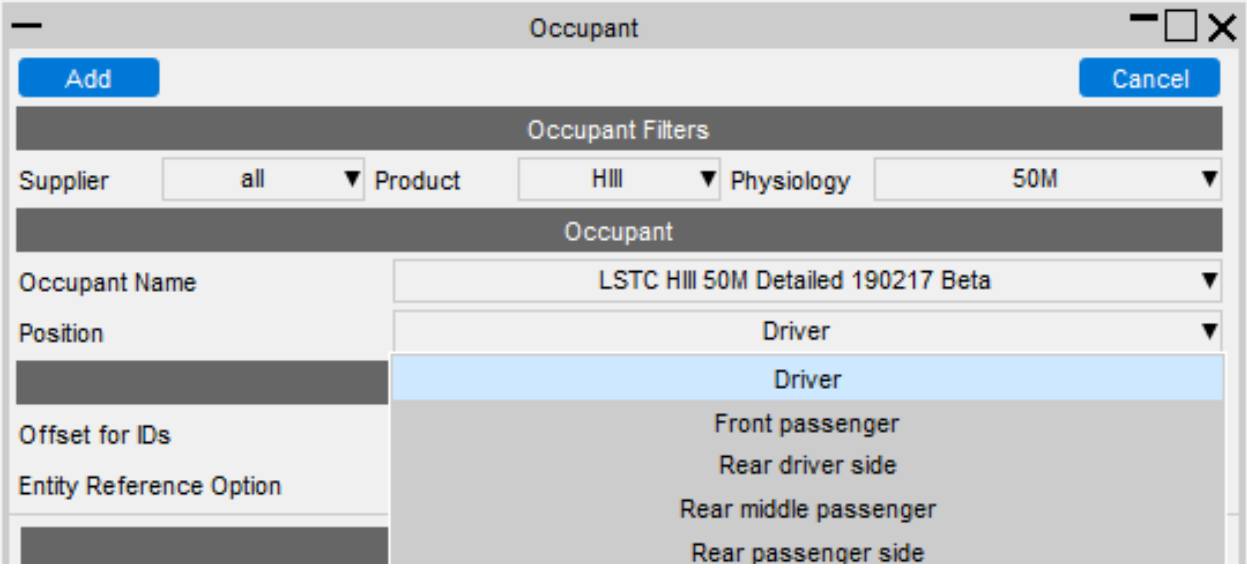
The options shown in this dropdown are filtered by the values in the **Supplier**, **Product** and **Physiology** dropdown menus. When the window is first opened these are automatically set so only occupant types that are relevant for the selected crash test type, regulation and occupant position are 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.



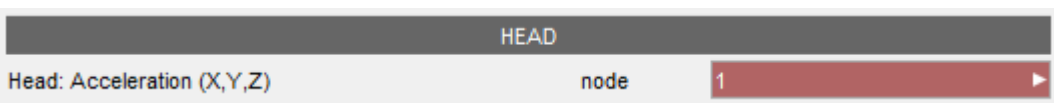
If for some reason you wish to select an occupant of a different type to the one expected, you can change the values of the filters to list other occupant types.

The position of the occupant in the vehicle will be set automatically, but you can change this with the **Position** dropdown menu if required:

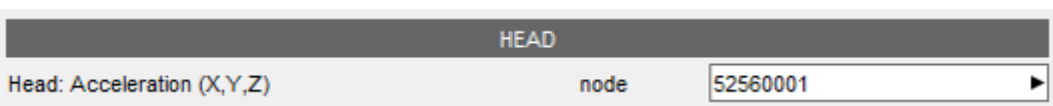


Entity IDs can be specified either by their numerical labels or DATABASE_HISTORY titles (for entities that have them defined) or DATABASE_CROSS_SECTION titles for X-Sections.

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.

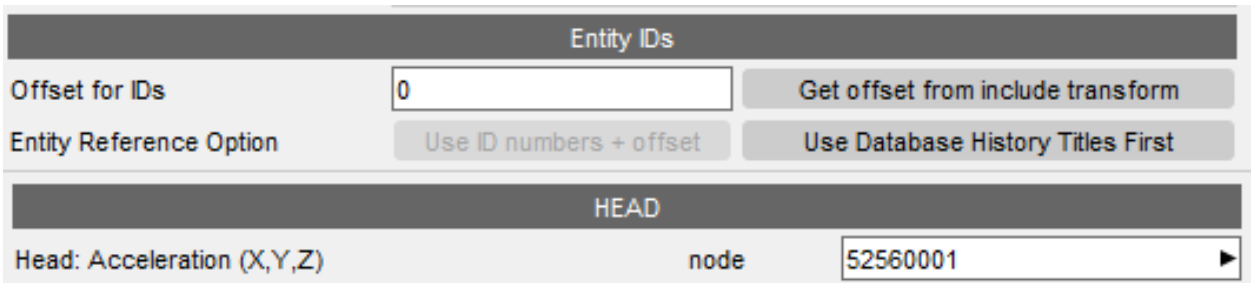


If they do exist the textboxes will change colour to indicate that (the colour will depend on the UI Theme), e.g.

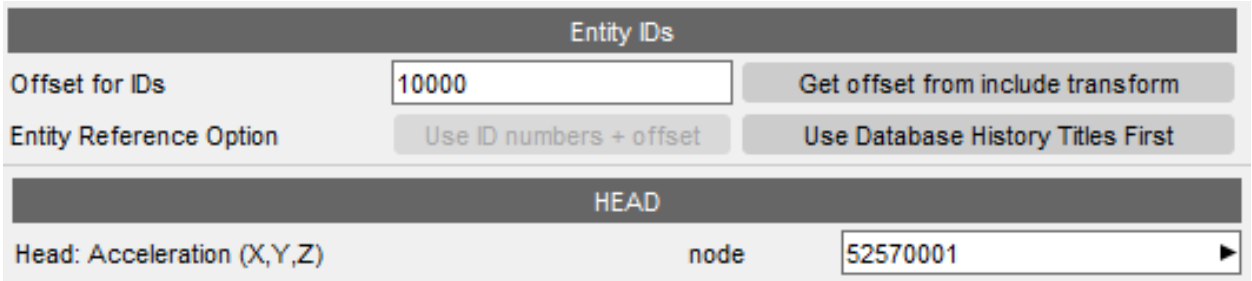


If the occupant has been renumbered so the labels are offset from the default ones, the Offset for IDs option can be used to apply the offset. This is useful when you have two or more occupants of the same type in the model as they both can't have the same entity labels.

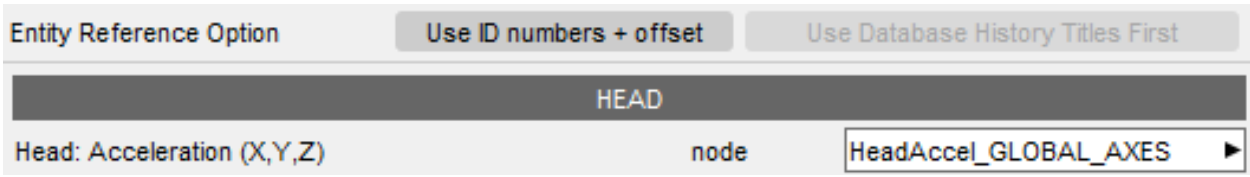
Offset = 0:



Offset = 10000:

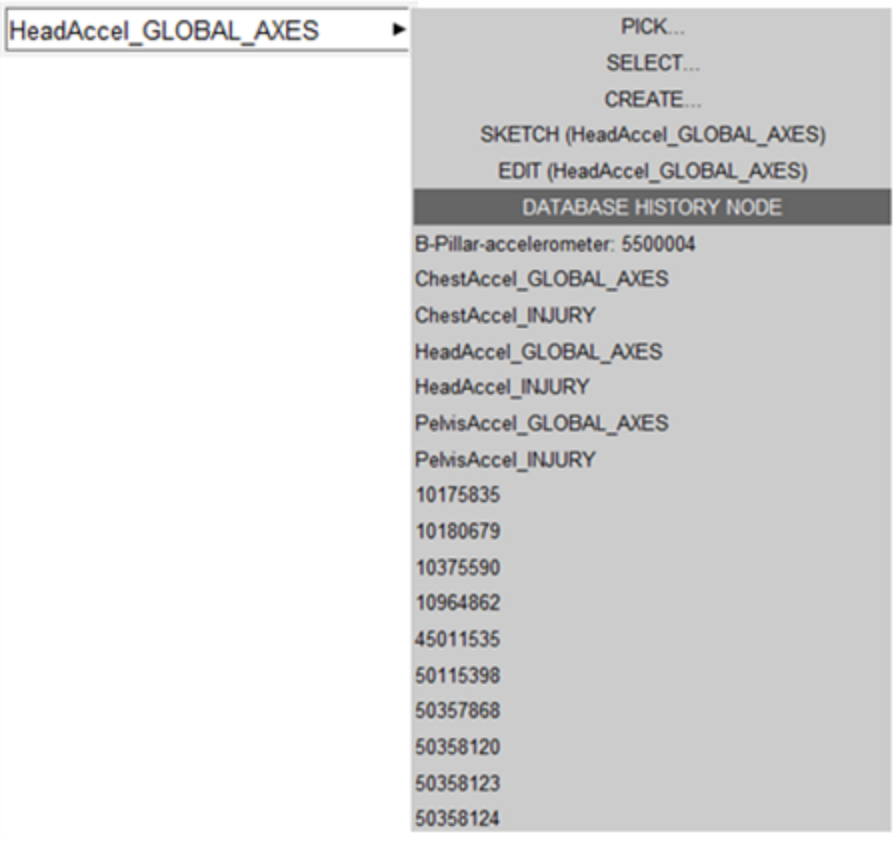


Alternatively, the entity IDs can be specified using DATABASE_HISTORY and DATABASE_CROSS_SECTION titles instead of their numerical labels. To automatically switch to use titles where they exist you can click on the Use Database History Titles button:

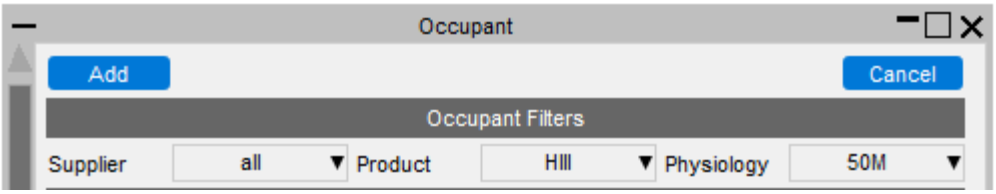


Note that a mix of defining some entities using numerical labels and others with titles is perfectly valid, they don't all have to be defined the same way.

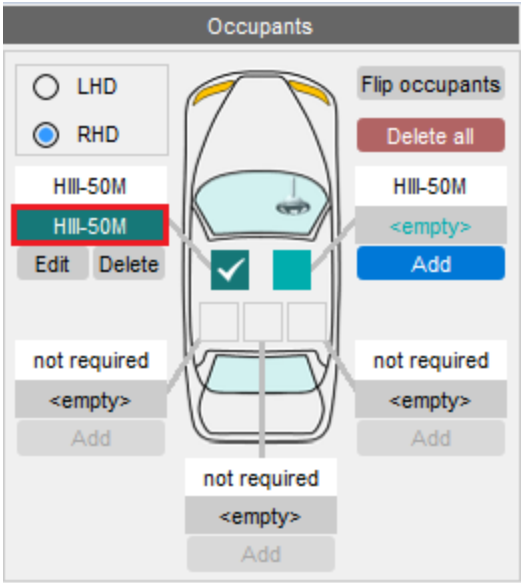
You can manually select entities by right clicking on a textbox. This opens a popup window which allows you to **Pick** or **Select** the entity interactively on the screen or select it from a list of DATABASE_HISTORY / DATABASE_CROSS_SECTION entities (ones with titles are listed first and ones without at the bottom).



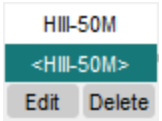
Once the entities have been defined you can press **Add** at the top of the window to add the occupant definition.



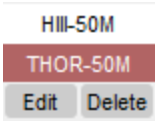
This will close the window and the Occupants section will update to show that an occupant has been defined in the selected position. If all the entity IDs are defined and valid the occupant will be shown like this:



If there are any undefined or invalid entity IDs it will look like this, i.e. enclosed in angular brackets "< >":



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

The following table lists all the occupants supported in the Automotive Assessment Workflows, along with the corresponding supplier documentation filename that was referenced in creating the respective occupant JSON files.

Supplier	Product	Physiology	Version	JSON	Manual

Supplier	Product	Physiology	Version	JSON	Manual
PDB	WSID	50M	v4.0 LHD	PDB WSID 50M v4.0 LHD	wsid50_pdb_v4.0_manual_v0.0.pdf
PDB	WSID	50M	v4.0 RHD	PDB WSID 50M v4.0 RHD	wsid50_pdb_v4.0_manual_v0.0.pdf
PDB	WSID	50M	v6.0 LHD	PDB WSID 50M v6.0 LHD	wsid50_pdb_v6.0_manual_v0.0.pdf
PDB	WSID	50M	v6.0 RHD	PDB WSID 50M v6.0 RHD	wsid50_pdb_v6.0_manual_v0.0.pdf
PDB	WSID	50M	v7.6 LHD	PDB WSID 50M v7.6 LHD	wsid50_pdb_v7.6.1_manual_v0.pdf
PDB	WSID	50M	v7.6 RHD	PDB WSID 50M v7.6 RHD	wsid50_pdb_v7.6.1_manual_v0.pdf
PDB	WSID	50M	v8.0 LHD	PDB WSID 50M v8.0 LHD	wsid50_pdb_v8.0_manual.pdf
PDB	WSI	50M	v8.0	PDB	wsid50_pdb_v8.0_manual.pdf

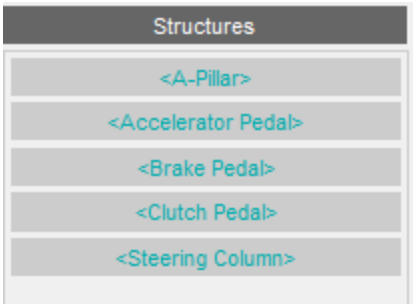
Supplier	Product	Physiology	Version	JSON	Manual
	D		RHD	WSID 50M v8.0 RHD	
PDB	WSID	50M	v8.1 LHD	PDB WSID 50M v8.1 LHD	wsid50_pdb_v8.1_manual.pdf
PDB	WSID	50M	v8.1 RHD	PDB WSID 50M v8.1 RHD	wsid50_pdb_v8.1_manual.pdf
DYNA MORE	ES-2re	50M	v6.0	DYNA MORE ES-2re 50M v6.0	es2_v_6.0_users_manual_v0.0.pdf
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	LSTC.H3_50TH_FAST.111130_V2.0_Documentation.pdf

Supplier	Product	Physiology	Version	JSON	Manual
				2.0	
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	HUMANETICS_HIII_50M_V1.5.1_HARMONIZED_LS_DYNA_TECHNICAL_REPORT_USER_MANUAL.pdf

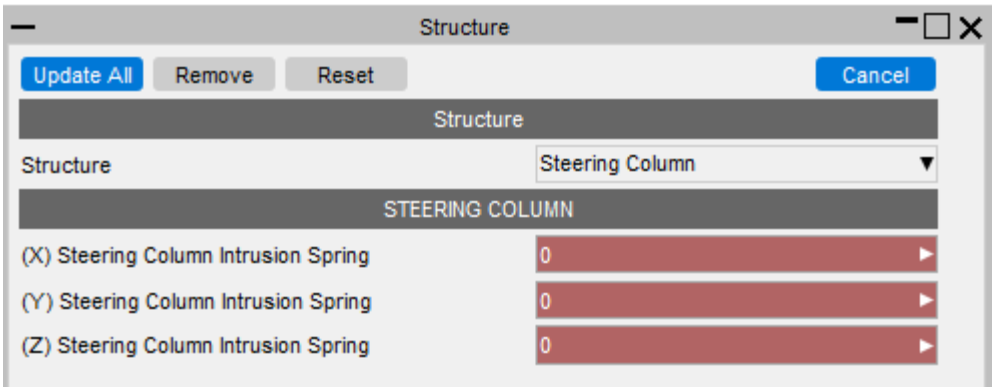
Supplier	Product	Physiology	Version	JSON	Manual
				HIII 50M v.1.5	
HUMANETICS	HIII	50M	v.1.5.1	HUMANETICS HIII 50M v.1.5.1	HUMANETICS_HIII_50M_V1.5.1_HARMONIZED_LS_DYNA_TECHNICAL_REPORT_USER_MANUAL.pdf
HUMANETICS	HIII	50M	v1.7	HUMANETICS HIII 50M v1.7	HUMANETICS_HIII_50M_V1.7_HARMONIZED_LS_DYNA_TECHNICAL_REPORT_USER_MANUAL.pdf
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.

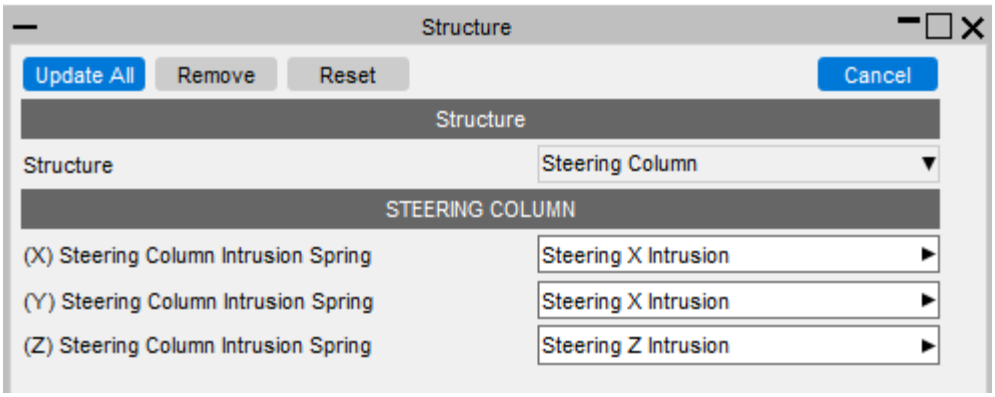


Click on one of the structures to open a window where you can select the entity IDs.

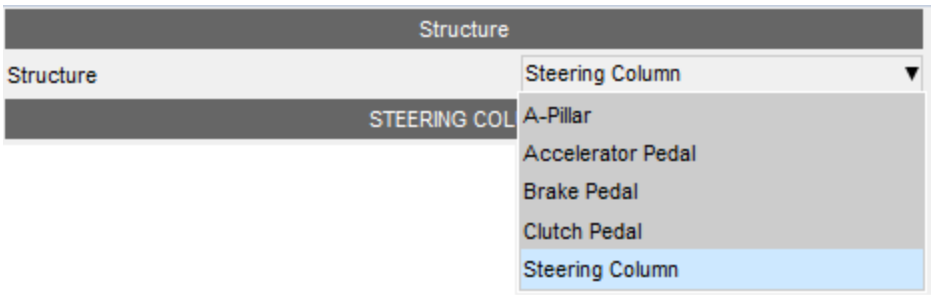


This works in the same way as the occupants window where IDs can be specified either by their numerical labels or DATABASE_HISTORY titles (for entities that have them defined).

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.

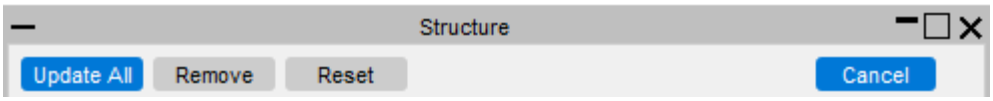


To select entity IDs for other structures you can use the Structure dropdown menu:



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.

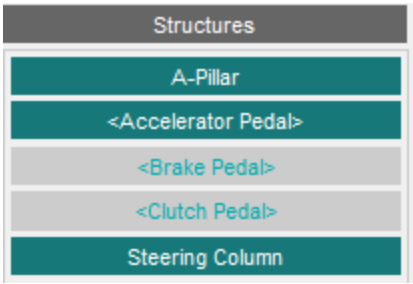


The Remove button sets all the entity IDs of the current structure to 0, effectively removing it from the assessment.

The Reset button sets the entity IDs back to what they were before any edits were made.

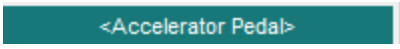
The Cancel button closes the window, without saving the selected entity IDs.

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:

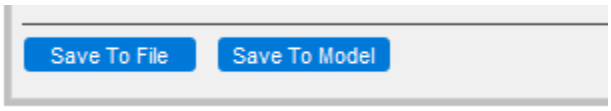
Structures that are coloured like this and enclosed in < >'s mean some of the required entity IDs are defined and exist in the model, but there are others that are either undefined or don't exist in the model:



Structures coloured like this and enclosed in < >'s mean none of the required entity IDs are defined or exist in the model:

Saving

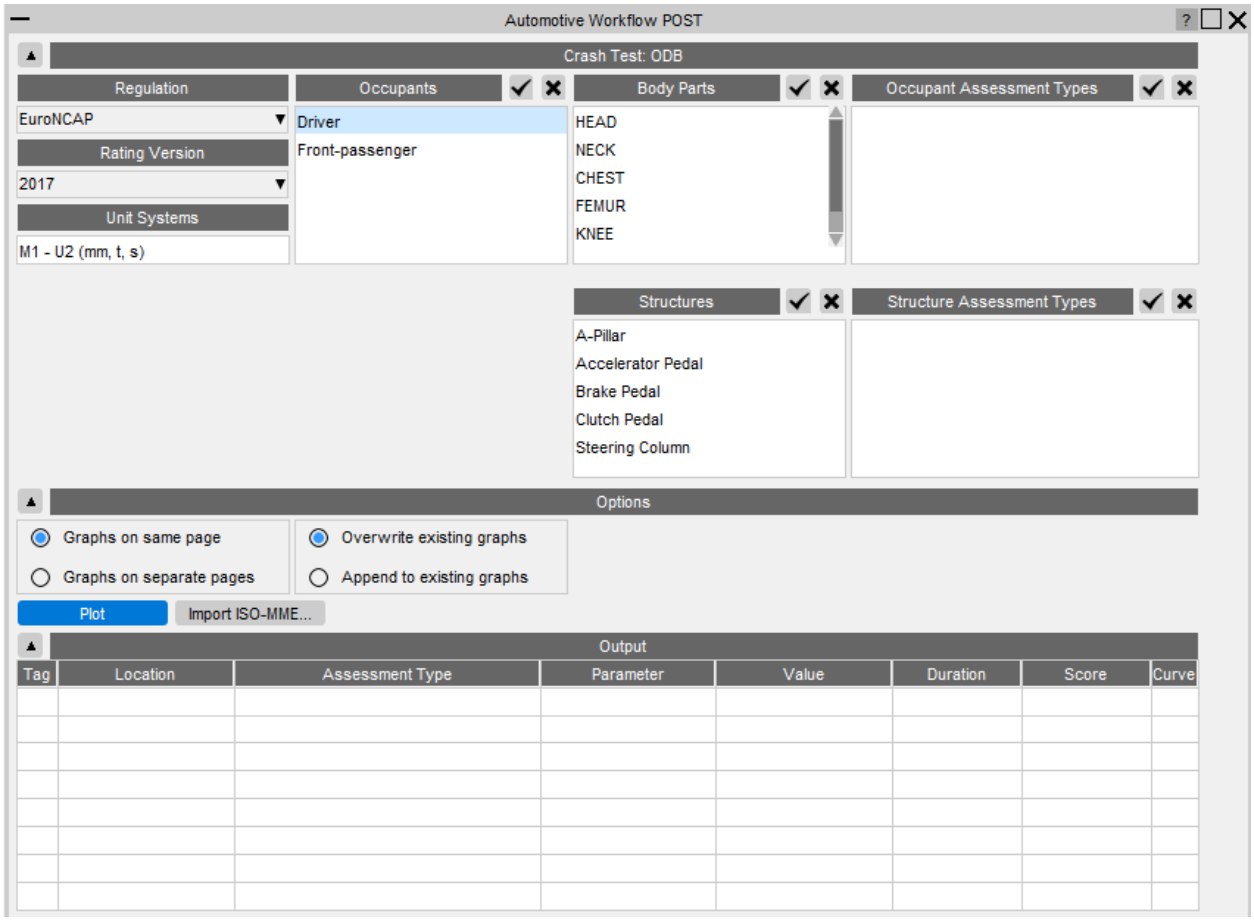
Users can choose to save the data to a .JSON file or directly to the model. The user data from the file will then be picked up when the workflow is selected in T/HIS or D3PLOT. Note if you choose to save the data to the model then you will need to save the model in PRIMER to ensure that the data is written to the master keyword file.



3.1.2. Automotive Assessments T/HIS

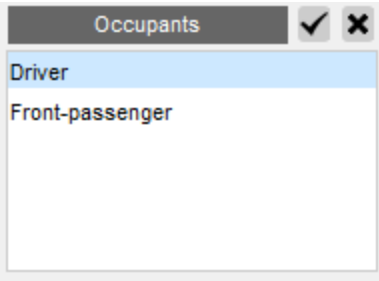
When the tool is launched in T/HIS you are presented with this window. This is where you select what assessments you want to carry out.

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.



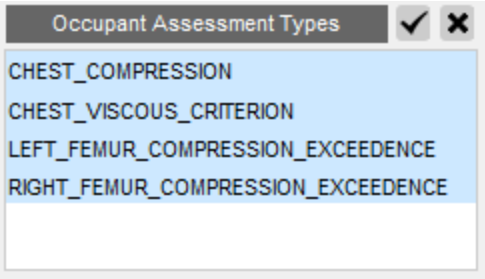
To select what assessments to carry out, you first need to select which occupant(s) you want to assess.

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.

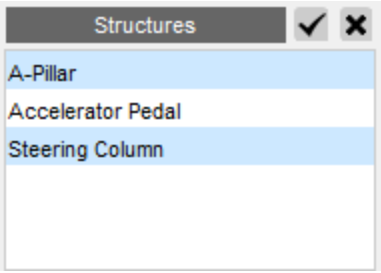


You can then select the body part(s) you want to assess.

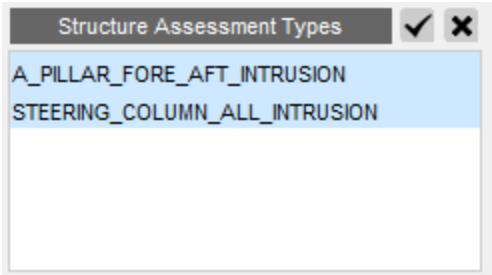
This will populate the Occupant Assessment Types list with the assessments that can be carried out for the selected body parts and occupants. By default they will all be selected, but you can chose to select only a subset of the list if you don't want to do them all.



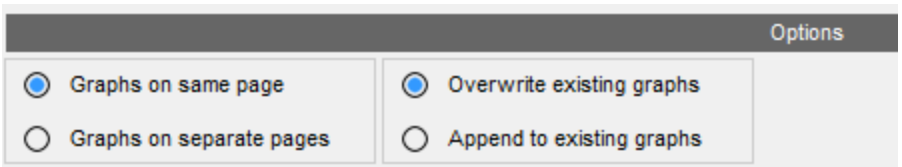
You can also select which structure(s) which you want to assess



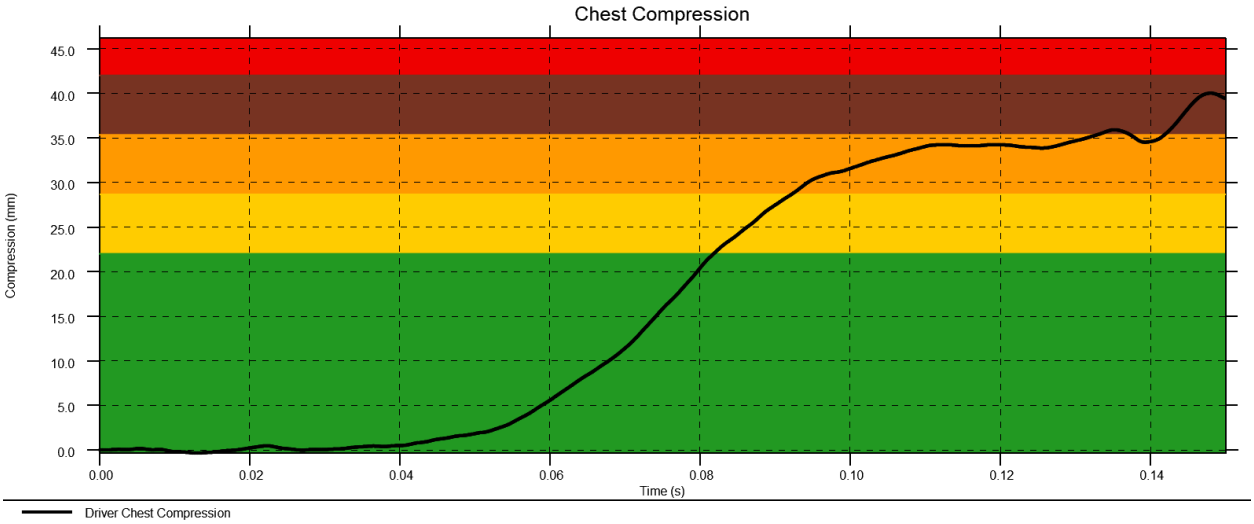
This will populate the Structure Assessment Types list with the assessments that can be carried out for the selected structures.



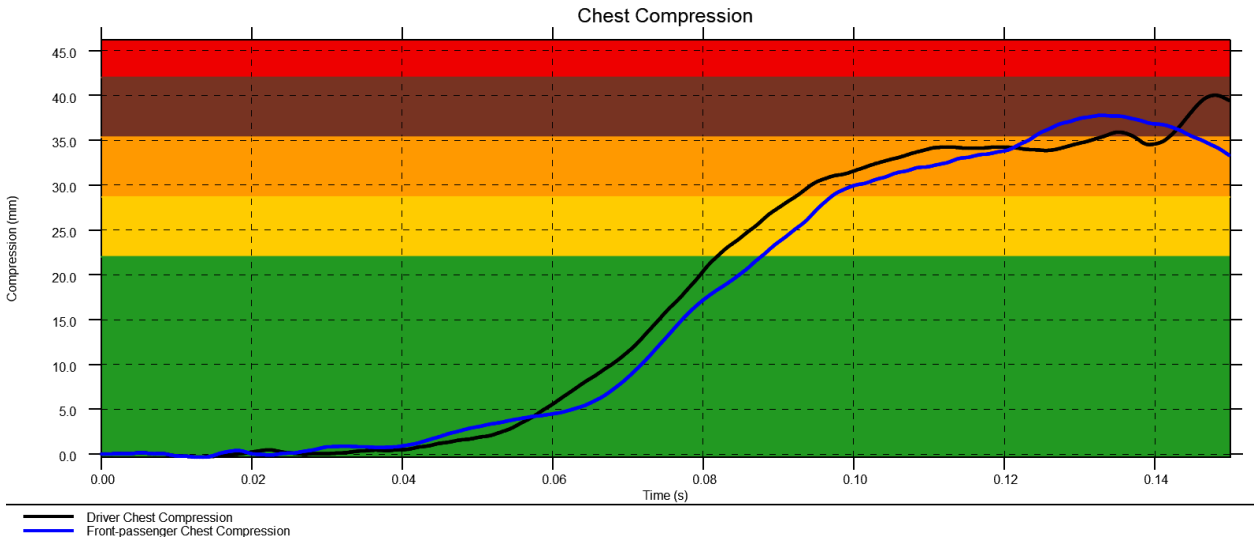
You can then chose how the graphs for each assessment should be displayed. By default they will all be put on one page and overwrite any existing graphs, but you can also chose to put each one on a separate page and append them to existing graphs.



Once you are happy with your choices, click **Plot** 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.



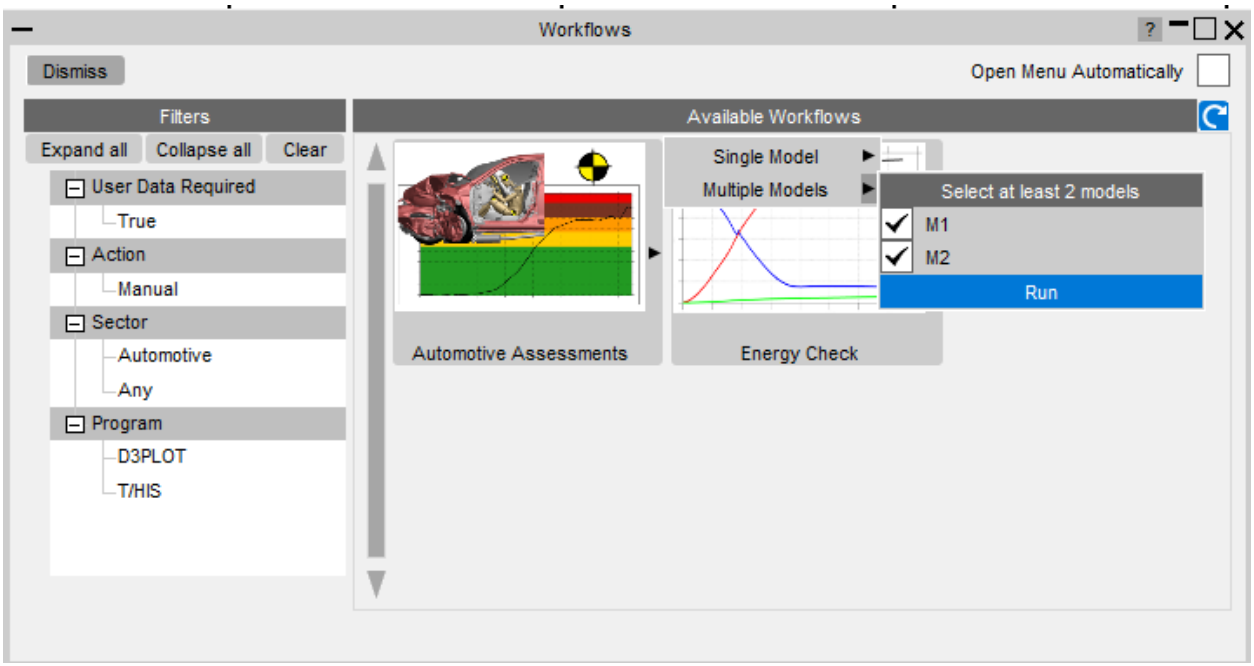
The output box at the bottom of the window lists the values and scores from the assessments carried out. Clicking on the '->' for each assessment will highlight the curve used for the assessment (and select the page if it's not on the current page) to make it easy to locate:

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	->

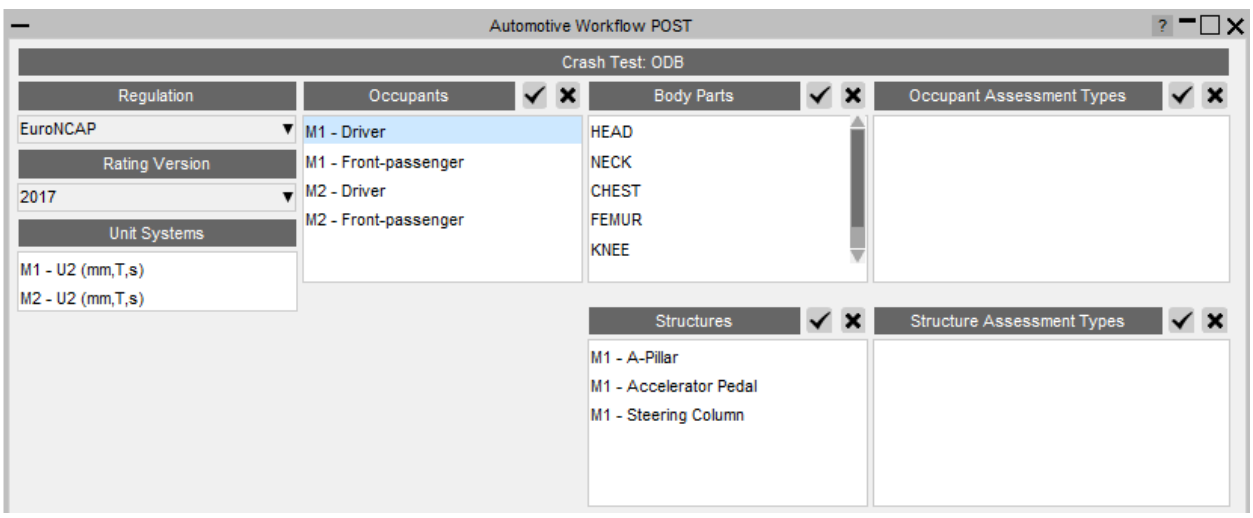
Multiple Models

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.

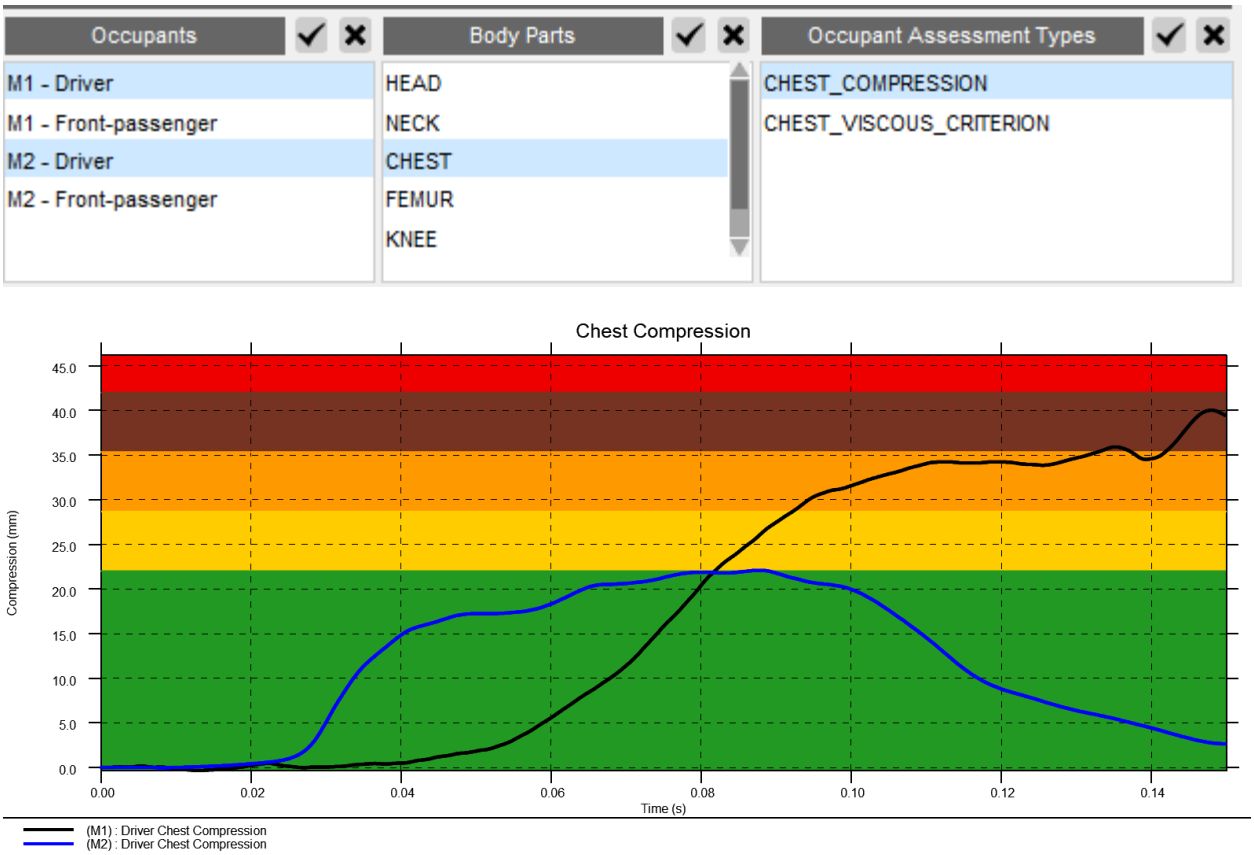
First you'll need to load the results from the models you want to compare into T/HIS and then on the workflow menu, select Multiple Models, pick the models you want to compare and press Run. Note that the models need to be of the same crash test type and regulation. If they're not the tool will refuse 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.

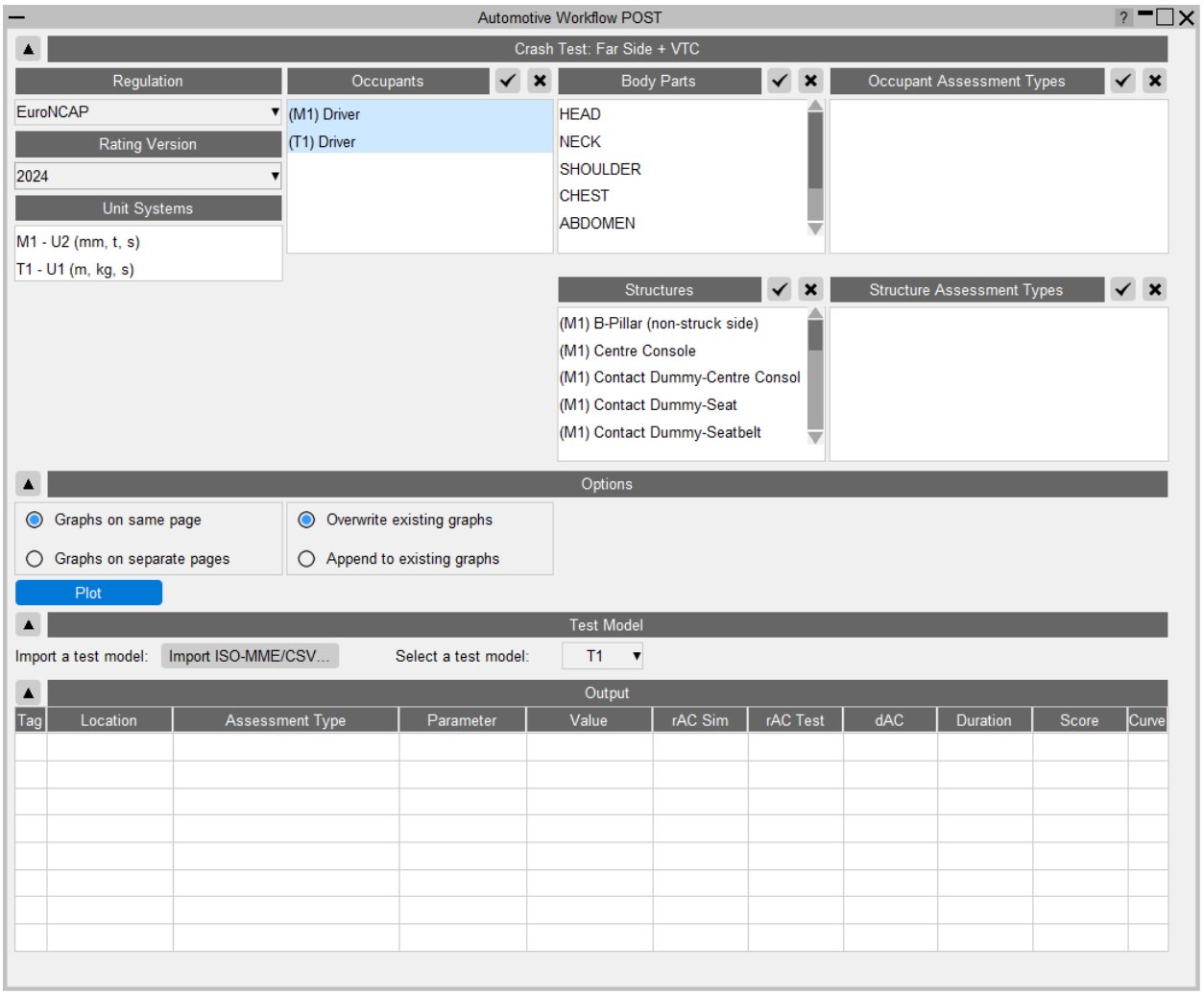


Euro NCAP Virtual Far Side Protocol Validation Criterion 2

Automotive Assessments can be used to check Validation Criterion 2, according to Section 6.3 of the Euro NCAP Virtual Far Side Simulation & Assessment Protocol. The Assessment Criteria ratios and differences are calculated and presented in the output table.

If you selected the Far Side + VTC crash test in PRIMER, then when you open Automotive Assessments in T/HIS, you will be presented with the window below. Here, you can import and select a test model and choose the assessments you want to carry out.

You can import a test model using the **Import ISO-MME/CSV...** button. Once imported, you can choose which model to use as a test model from the **Select a test model** combo box.



The output table at the bottom of the window lists the injury values, Validation Criteria 2 ratios and differences (\(r_{AC_{sim}}\), \(r_{AC_{test}}\) and \(d_{AC}\) from Equation 4 of the Euro NCAP VTC protocol) and scores for the assessments carried out.

The following example below explains how \(r_{AC_{sim}}\), \(r_{AC_{test}}\) and \(d_{AC}\) are represented in the output table.

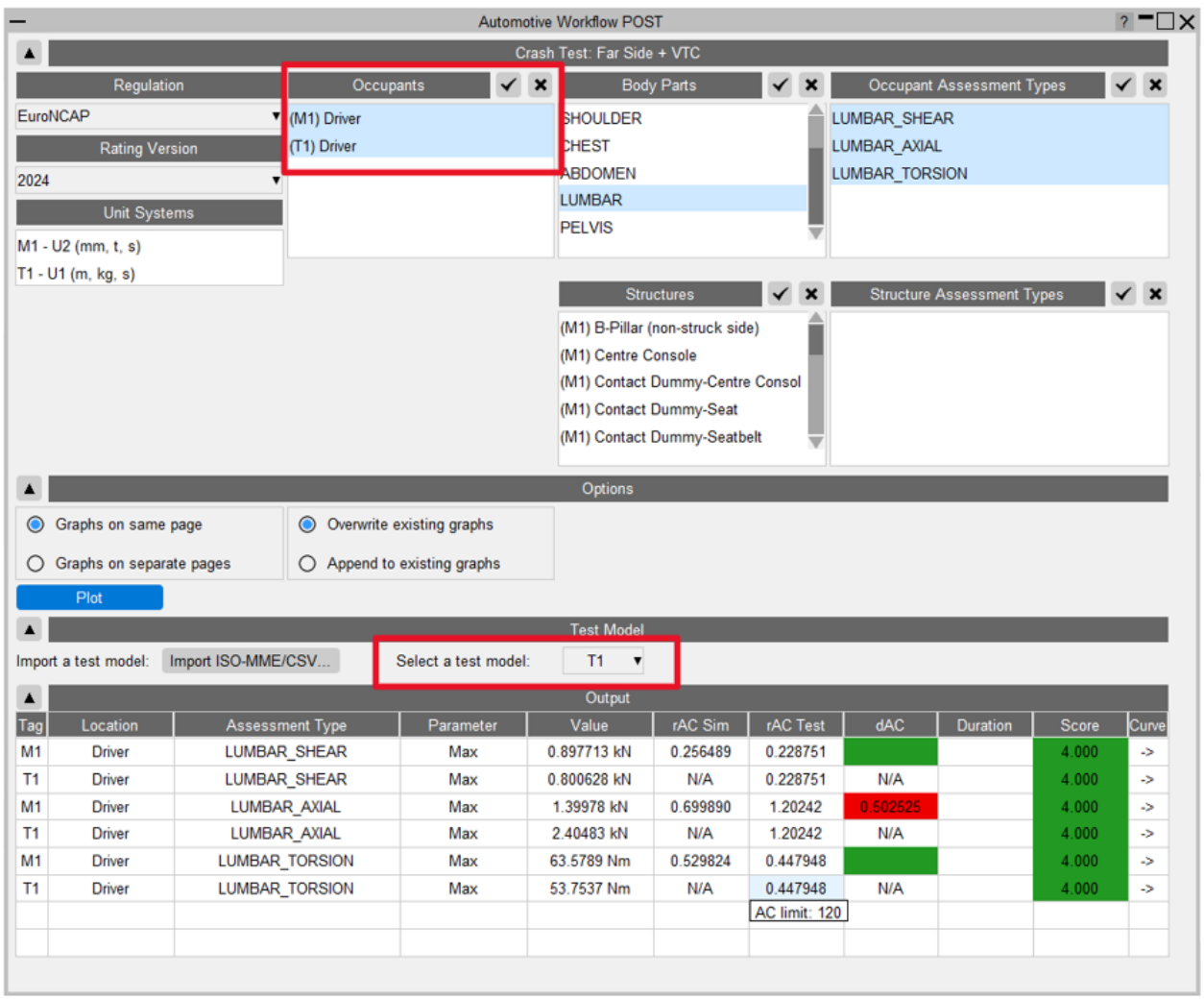
If you select simulation (M1) Driver and test data (T1) Driver in the Occupants selection and click **Plot**, assessments will be carried out for both M1 and T1, with T1 used as the test model (reference) for both.

- According to equation 4, the \(r_{AC_{sim}}\) value is calculated as \(r_{AC_{sim}} = \frac{AC_{sim}}{AC_{limit}}\), for all occupants selected, except for those where the assessment model is the same as the selected test model. In this example, it is only calculated for M1. Appropriate hover text is available for cells where \(r_{AC_{sim}}\) was not calculated, and 'N/A' is displayed for such cells.

- According to equation 4, the $r_{AC_{test}}$ value is calculated as $r_{AC_{test}} = \frac{AC_{test}}{AC_{limit}}$, for all occupants. The model selected in the Select test model combo box will be used as the test model. The $r_{AC_{test}}$ cells will have hover text displaying the Assessment Criteria Limit value for that particular assessment, as specified in Table 7 of the Euro NCAP VTC protocol.
- According to equation 5, the d_{AC} value is calculated as $d_{AC} = \left| r_{AC\text{ test}} - r_{AC\text{ sim}} \right|$. The d_{AC} values will only be calculated if the $r_{AC_{sim}}$ is calculated and $r_{AC\text{ test}} \geq 50\%$. If calculated, appropriate hover text will be added to the cell, and the cell will be colour-coded to indicate if that assessment meets Validation Criterion 2.

The deviations between the ratios must be < 30% for each considered assessment criteria to fulfill the validation criterion 2.

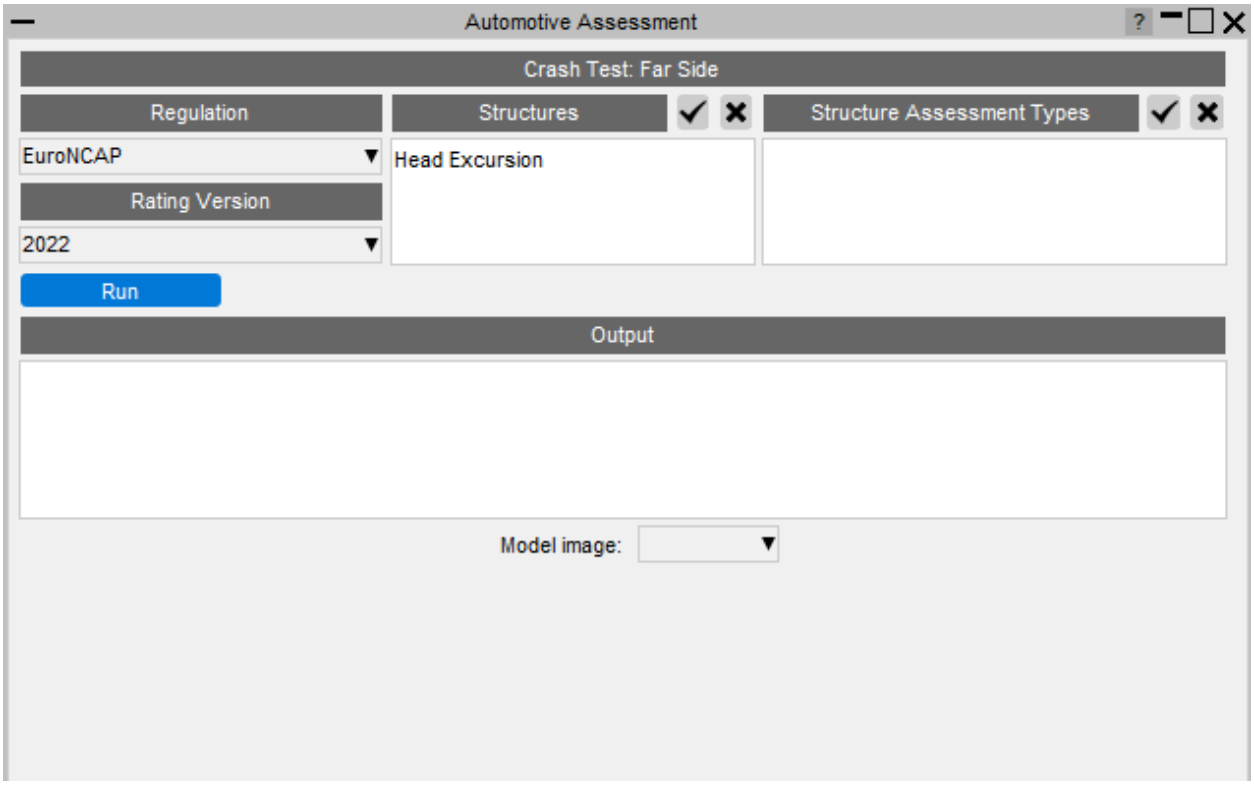
$$\text{Validation Criterion 2 (Assessment Criteria): } d_{AC} < 30\%$$



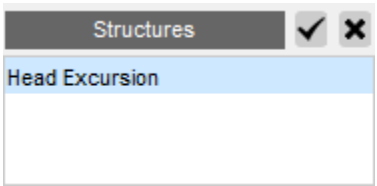
3.1.3. Automotive Assessments D3PLOT

Some structural assessments involve more than just plotting curves on graphs – for example, taking cut sections through the model to measure intrusion. These assessments are carried out in D3PLOT.

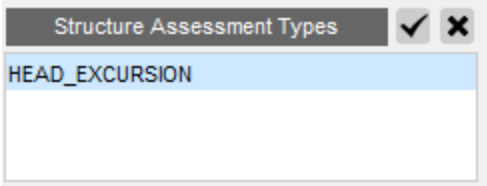
When the tool is launched in D3PLOT you are presented with this window. This is where you select what assessments you want to carry out.



Select the structure you want to assess

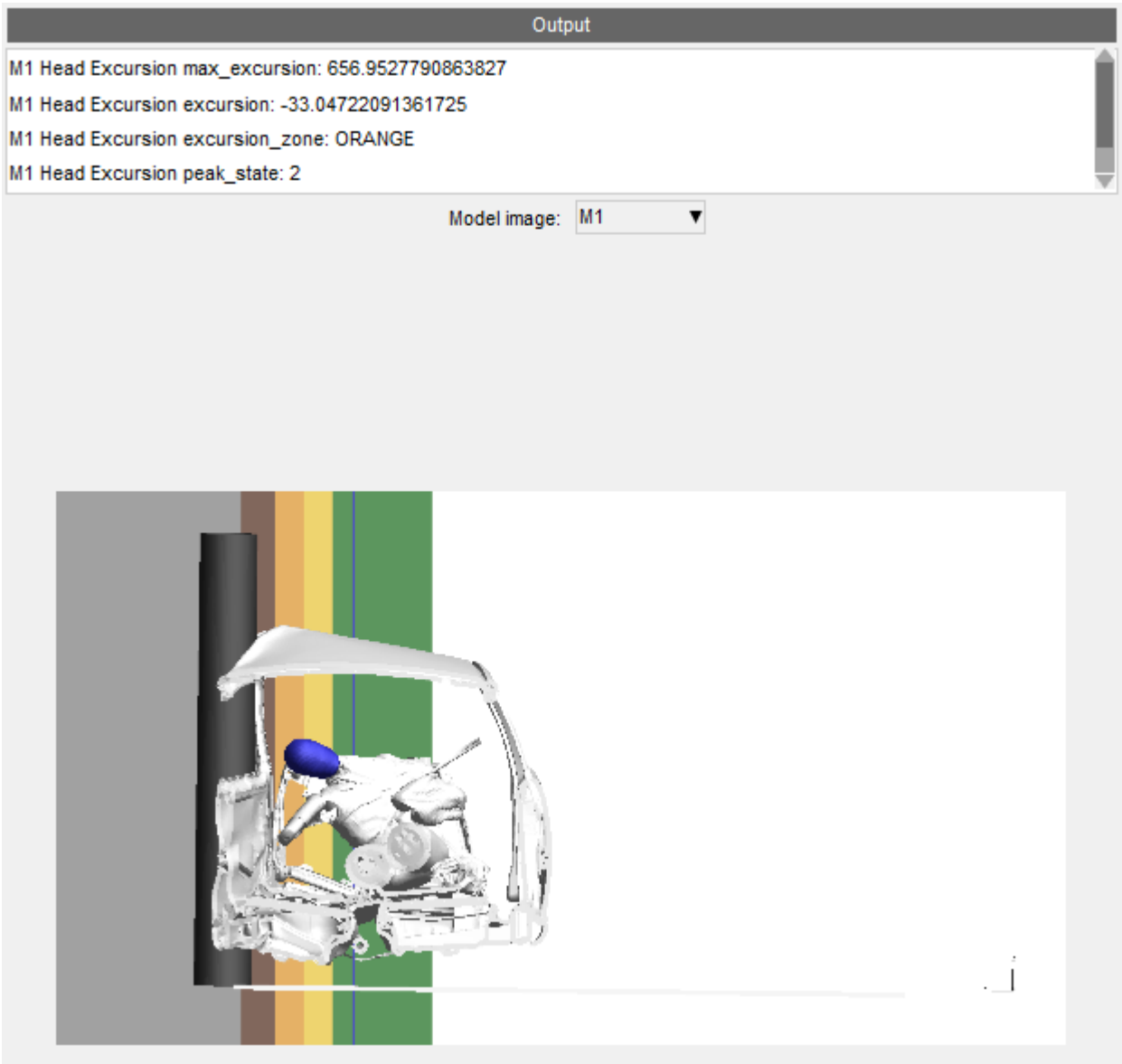


This will populate the Structure Assessment Types list with the assessments that can be carried out for the selected structures.



Click **Run** to carry out the assessment. D3PLOT will extract the data required for the assessment and process it according to the rules set out in the regulation. Depending on the assessment, this may involve starting other programs like PRIMER or REPORTER to carry out parts of the assessment.

It will eventually produce an image which it will display in the window and a list of output values in the Output window:



3.1.4. 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

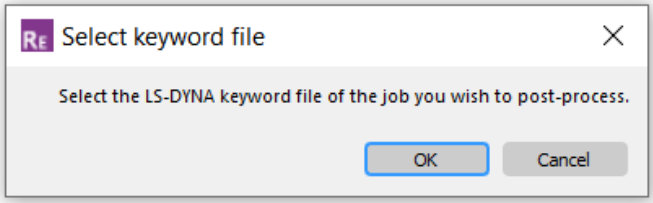
Running the templates interactively

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:

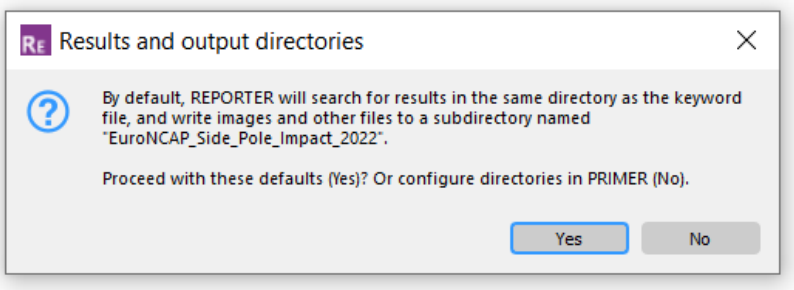
- In PRIMER specify and save the required data using the Automotive Assessments workflow
- In REPORTER use **File → Open Library Template** to select the relevant template.

  Templates that use workflow data are indicated by the icon:

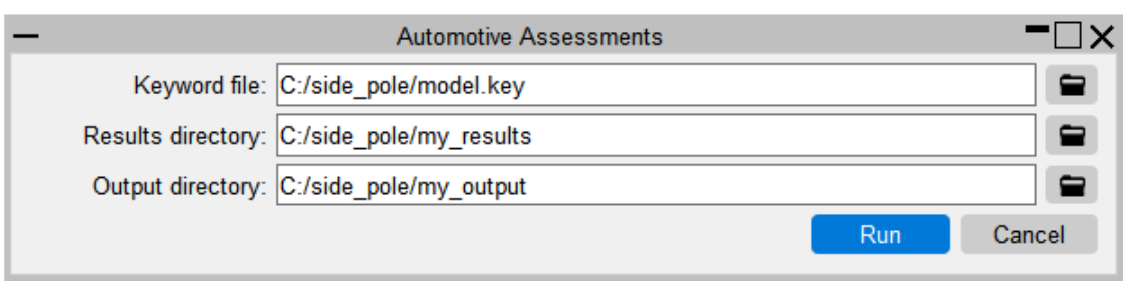
- On opening the template you will be prompted to select the keyword file of the job you want to post-process



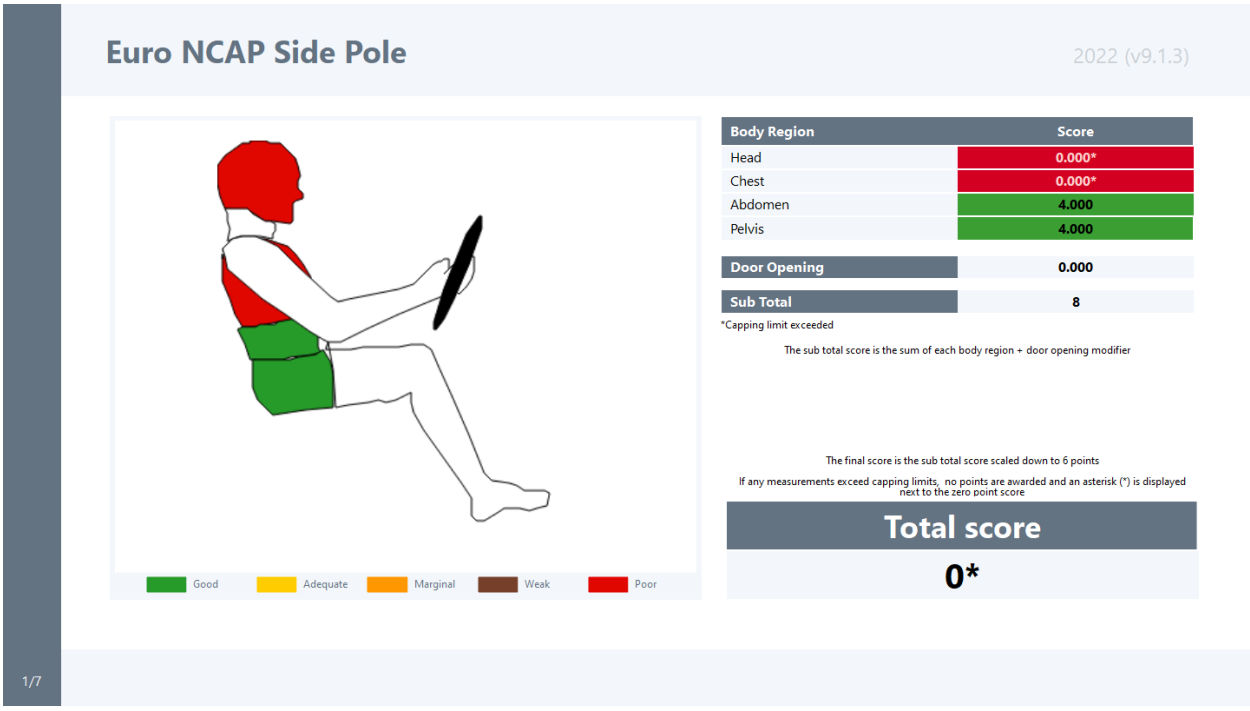
- After pressing **OK** a file selector is mapped for you to select the keyword file.
- After selecting the keyword file a prompt will ask if you want to proceed with some default directories to search for model results and for writing images and other files. The defaults assume:
    1. The results are 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***

reporter_exe	The full path and filename to the REPORTER executable
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

3.2. Energy Check

Energy Check

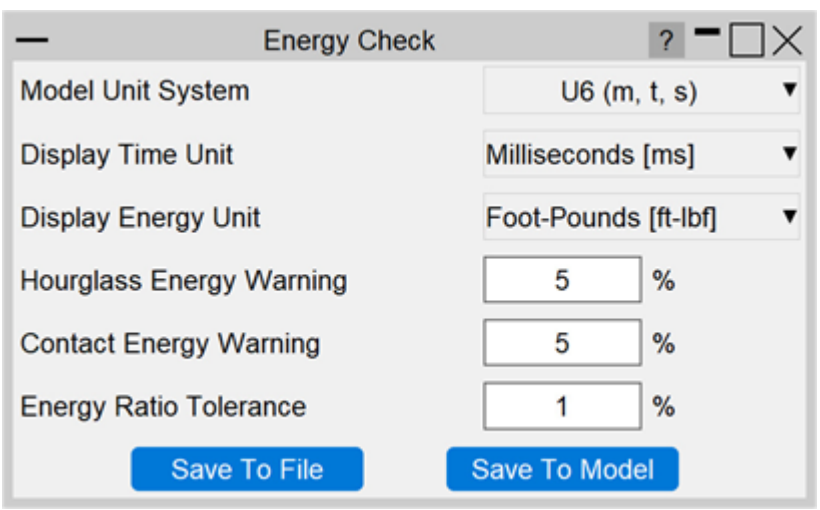
Tools → Workflows → Energy Check

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

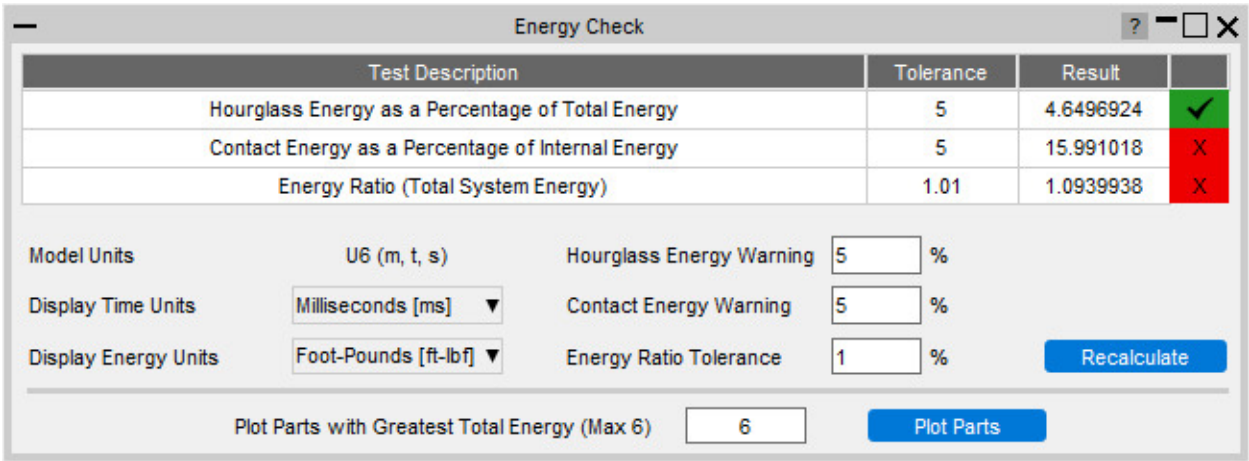
You can change the warning value of which Energy Ratio (Total Energy minus External Work or Total System Energy) will be flagged at if the curve has exceeded the tolerance. Total System Energy should remain constant (Energy Ratio should stay at 1.0), although this is not realistic therefore the default 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.

3.3. Entities of Interest

Entities of Interest

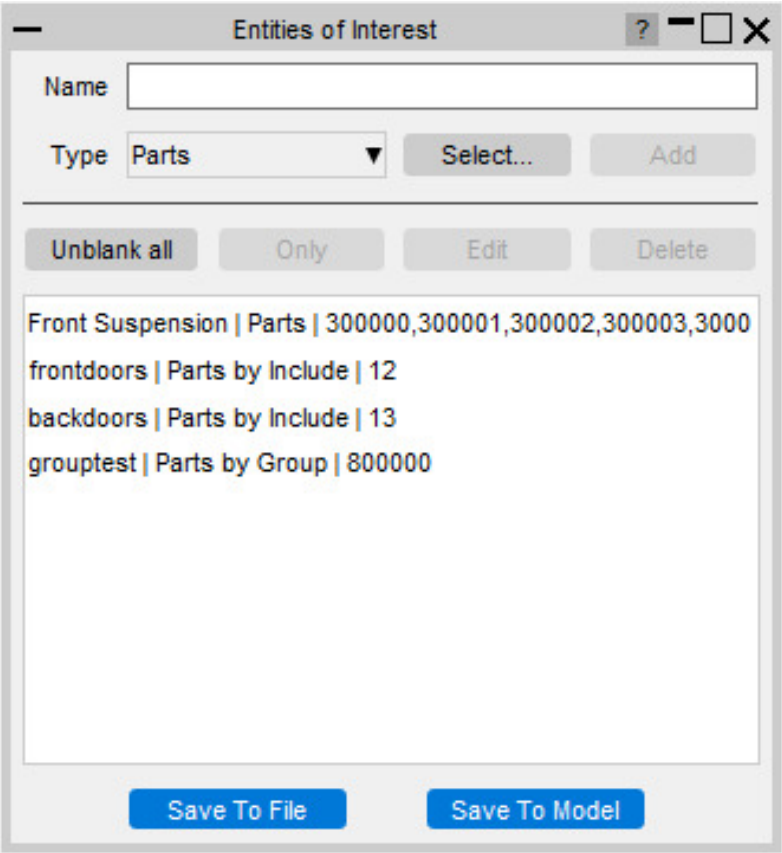
Tools → Workflows → Entities of Interest

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

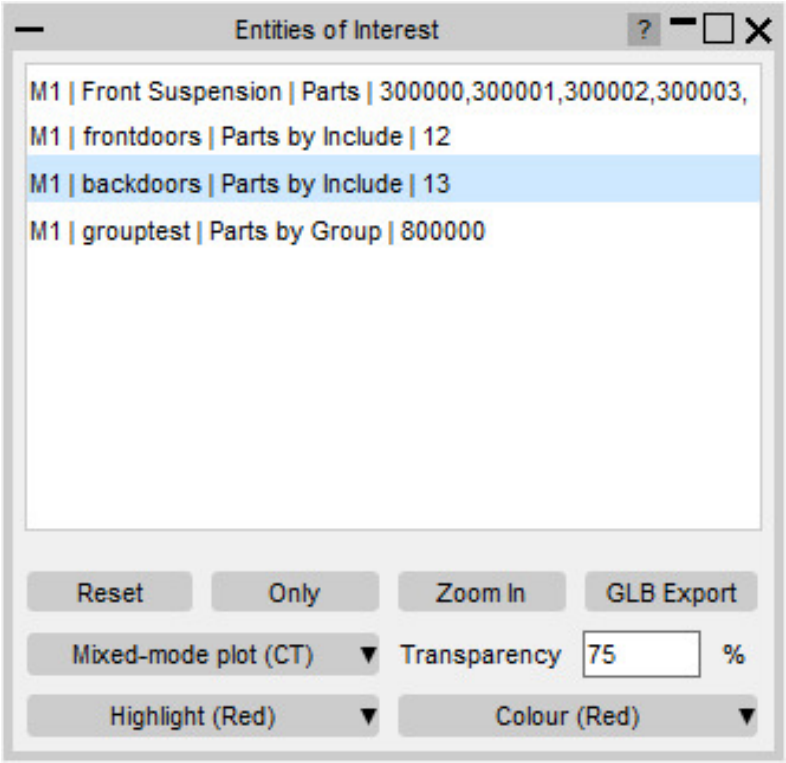
Select one or more entries from the list and click **Delete** to delete them.

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.

3.4. Eroded Elements

Eroded Elements

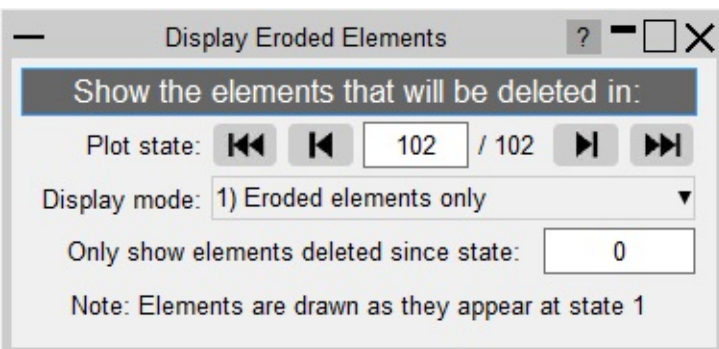
Tools → Workflows → Eroded Elements

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

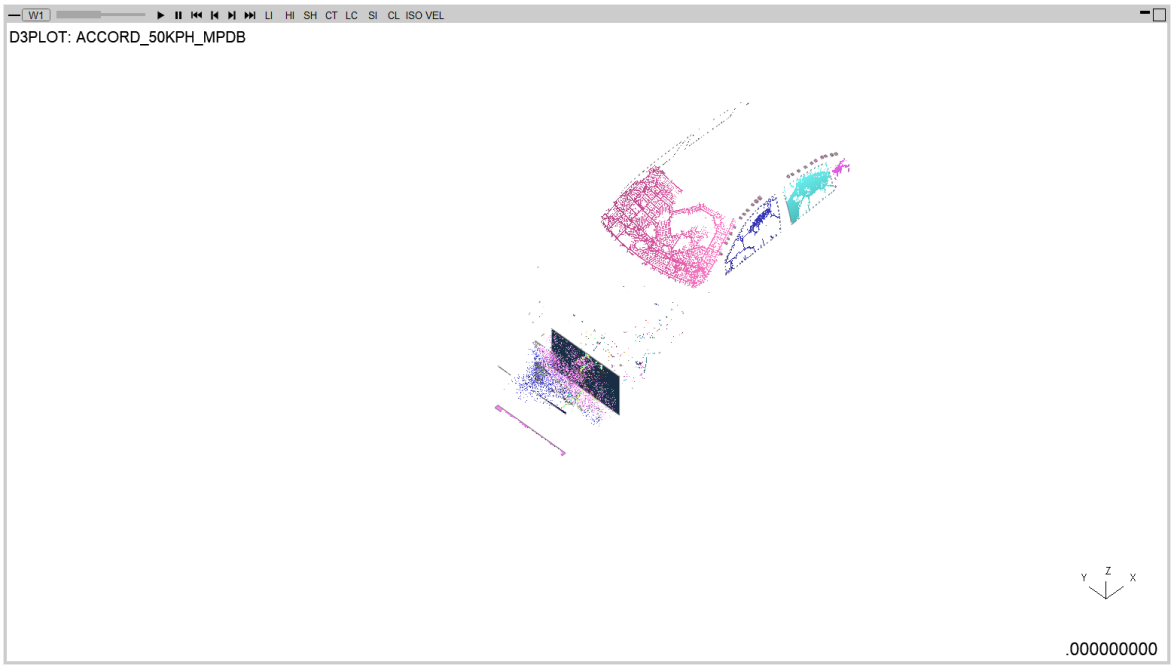
You can choose to display the elements deleted at any plot state. Use the controls in the menu to change plot state, rather than D3PLOT's main controls. Only elements deleted between the comparison state and the plot state will be displayed. Note that the plot state cannot be before the comparison state.

Display mode

There are three display modes:

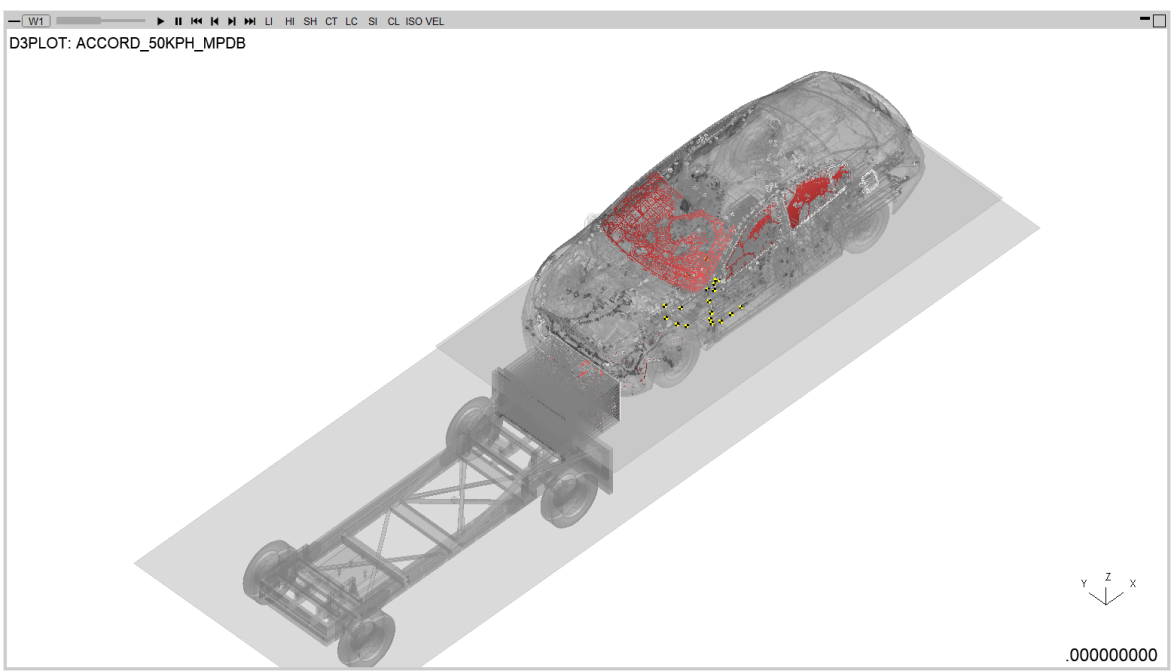
1. Eroded elements only (default)

Only the elements deleted between the comparison state and the plot state are shown (all other elements are blanked)



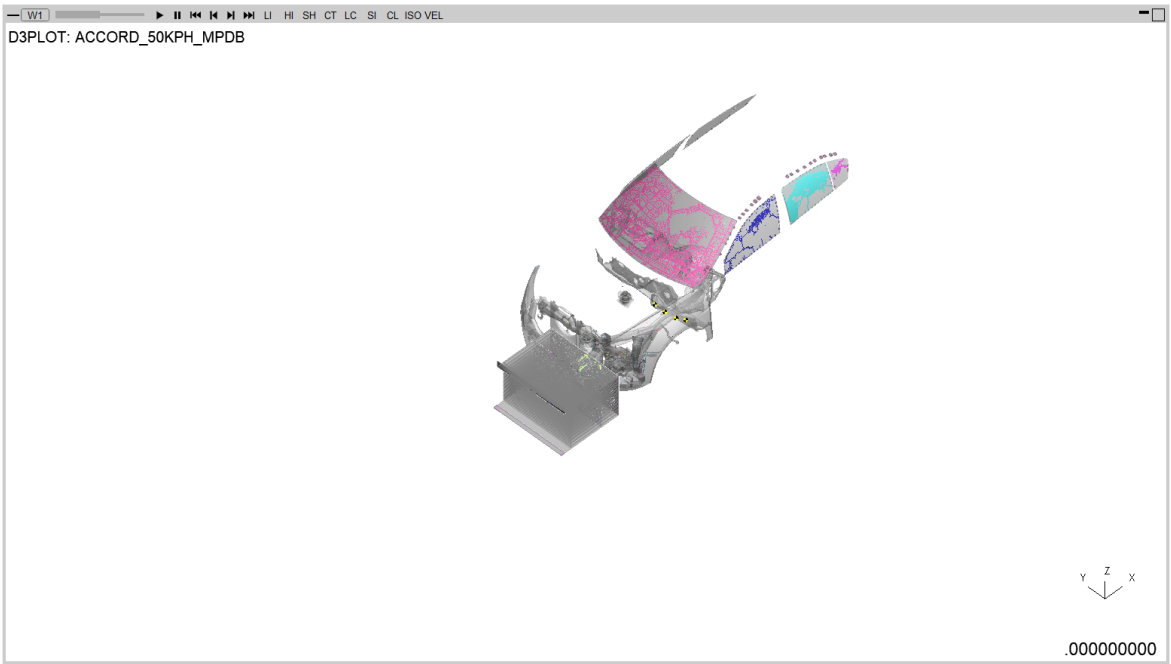
2. Eroded elements in red

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

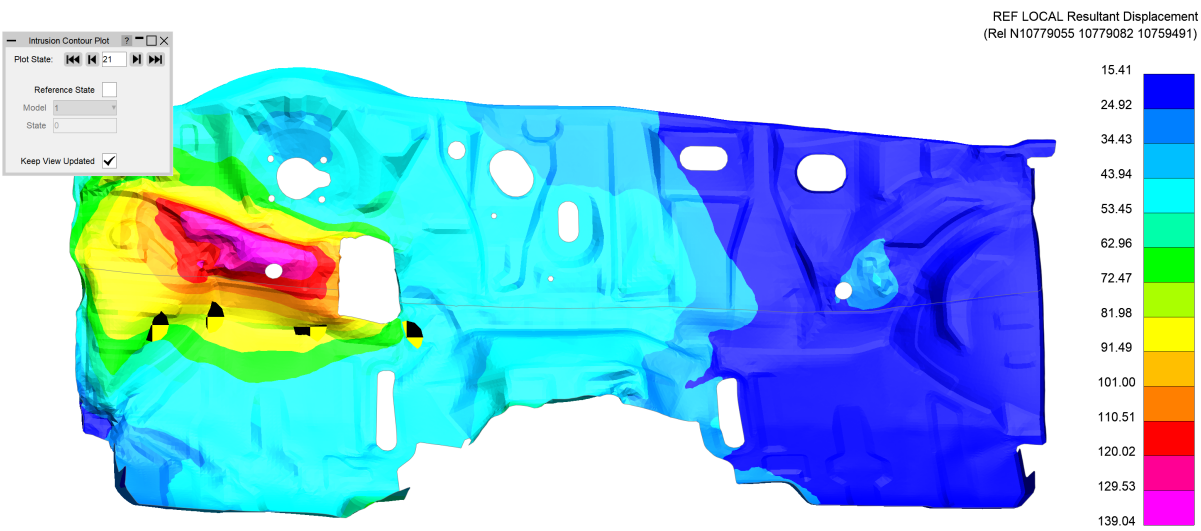
When Eroded Elements is opened, a temporary properties file is saved. When you exit the tool, you can choose to restore the model properties (view, blanking, colours, etc.) to their appearance before you opened the tool.

3.5. Intrusion Contour Plot

Intrusion Contour Plot

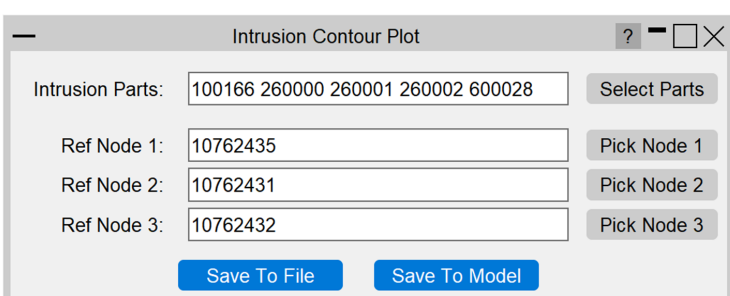
Tools → Workflows → Intrusion Contour Plot

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.

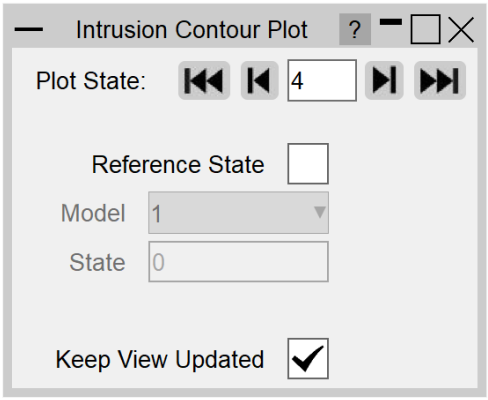
Use in D3PLOT

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

3.6. 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.

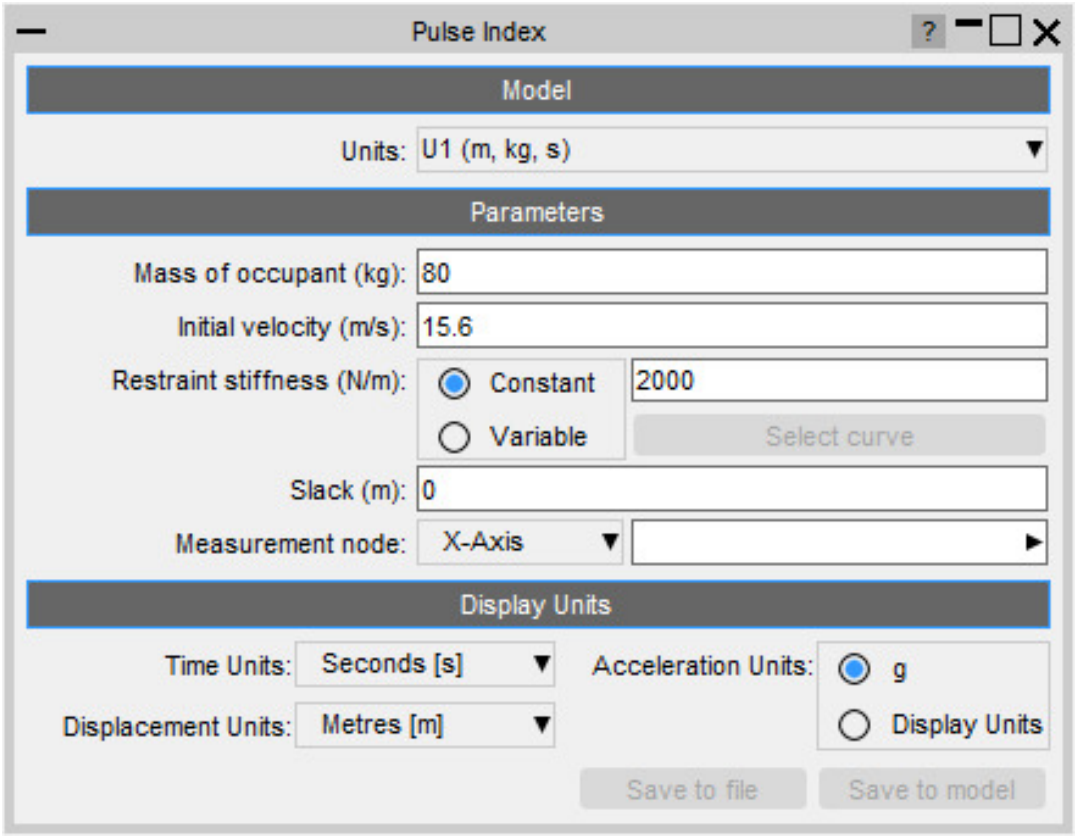
The tool assumes a virtual single-degree-of-freedom mass-spring system in which the occupant is represented by a **mass** and the seatbelt is represented by a **spring**. See below for more details of the pulse index calculation.

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.

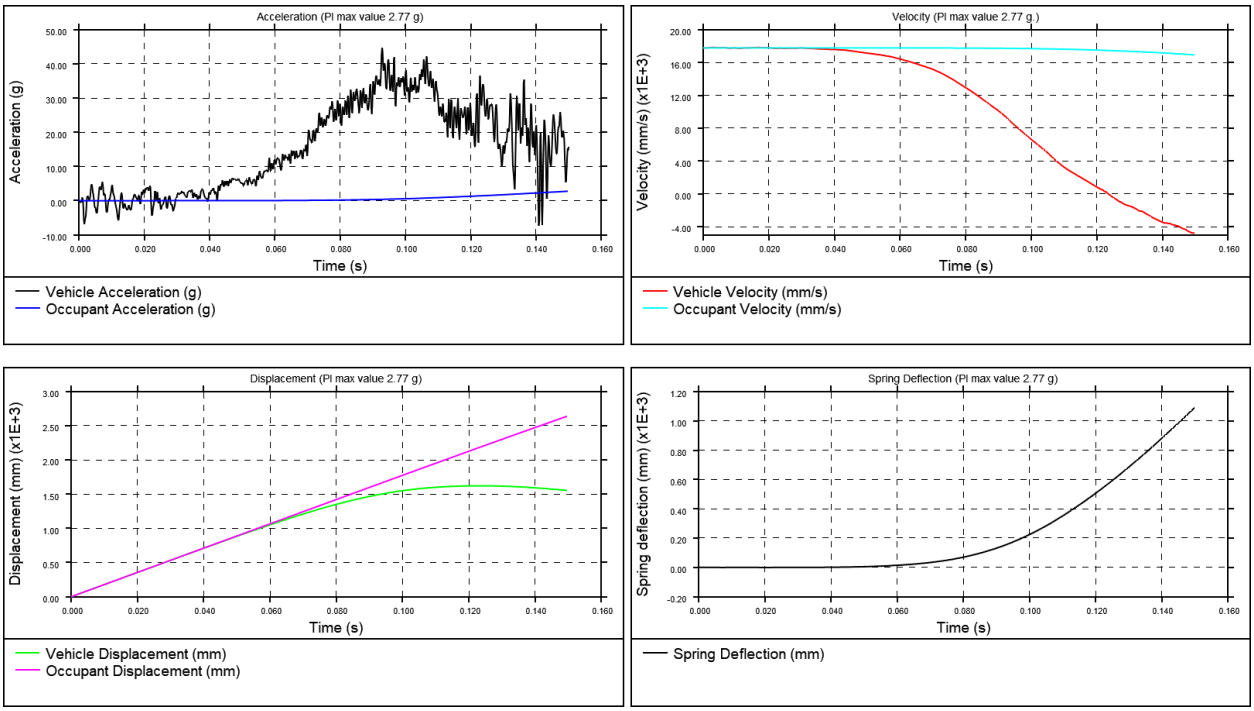
How to use the Pulse Index tool in T/HIS

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:

$$v_{t+1}^{occ} = v_t^{occ} + a_t^{occ} * dt$$

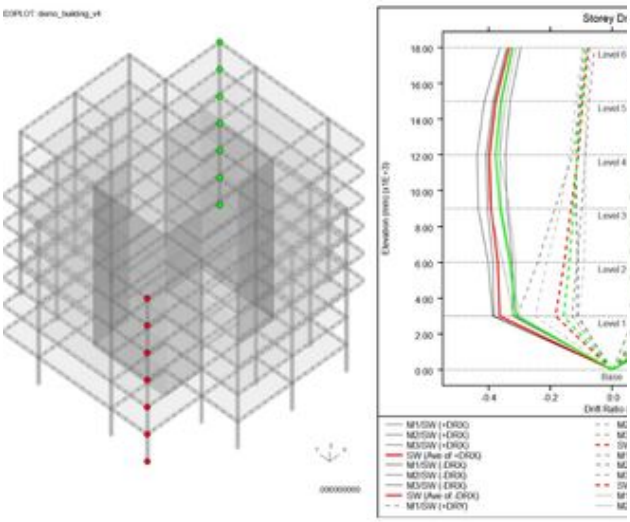
$$s_{t+1}^{occ} = s_t^{occ} + v_t^{occ} * dt$$

3.7. Seismic

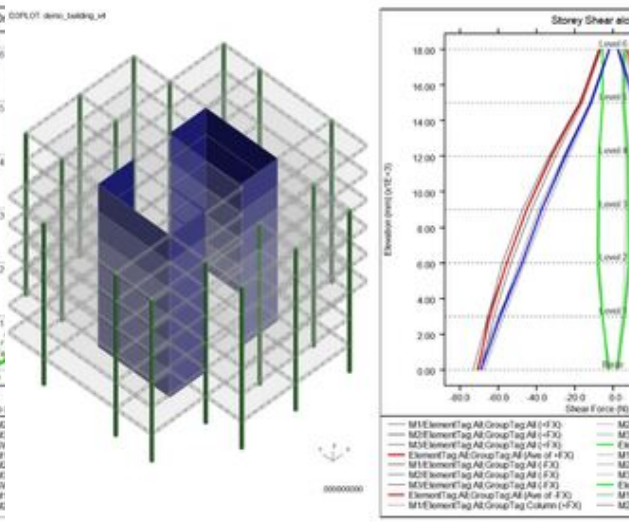
Seismic Workflows

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



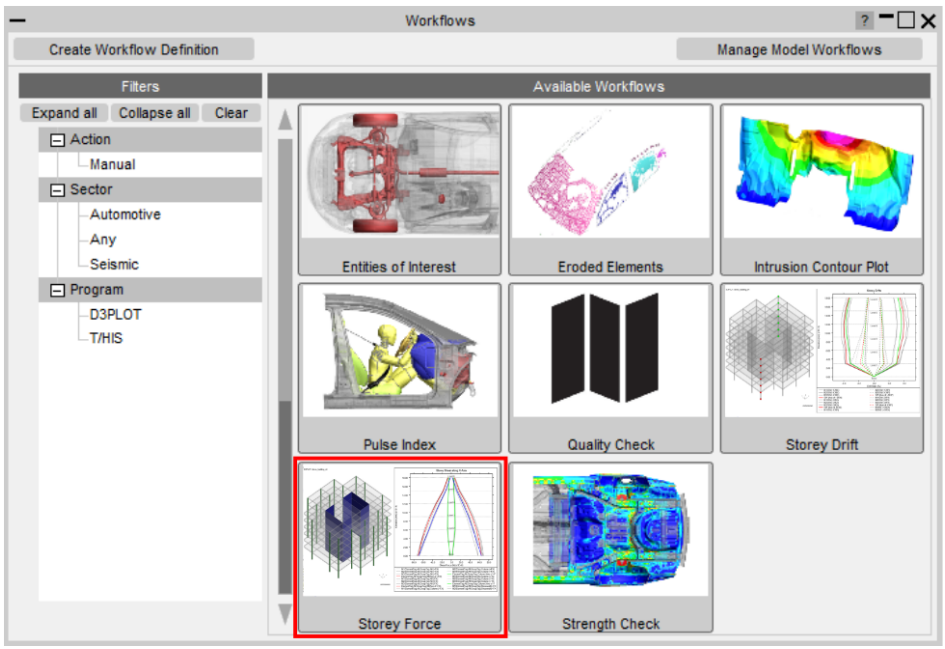
Storey Force

3.7.1. Storey Force

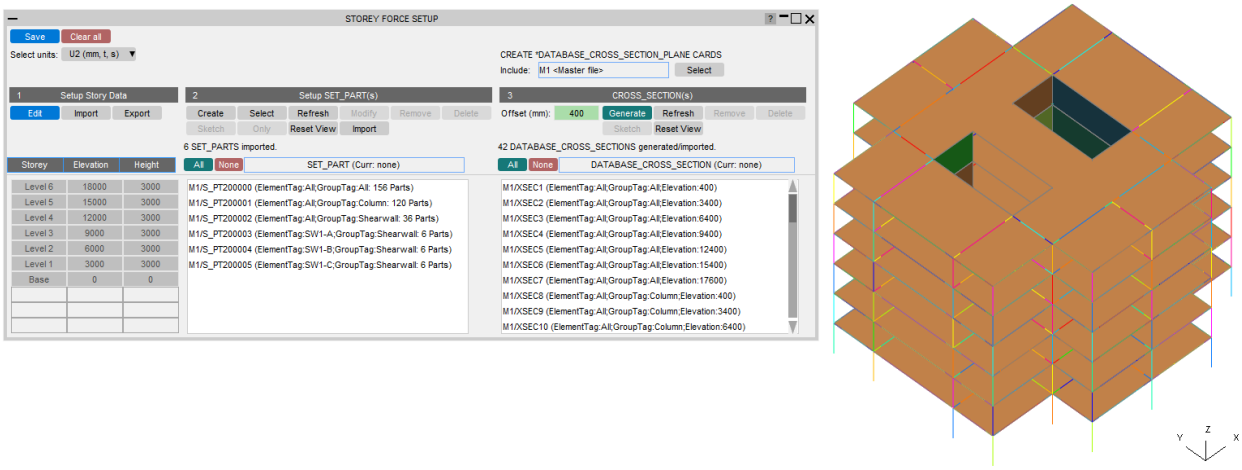
Storey Force

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.

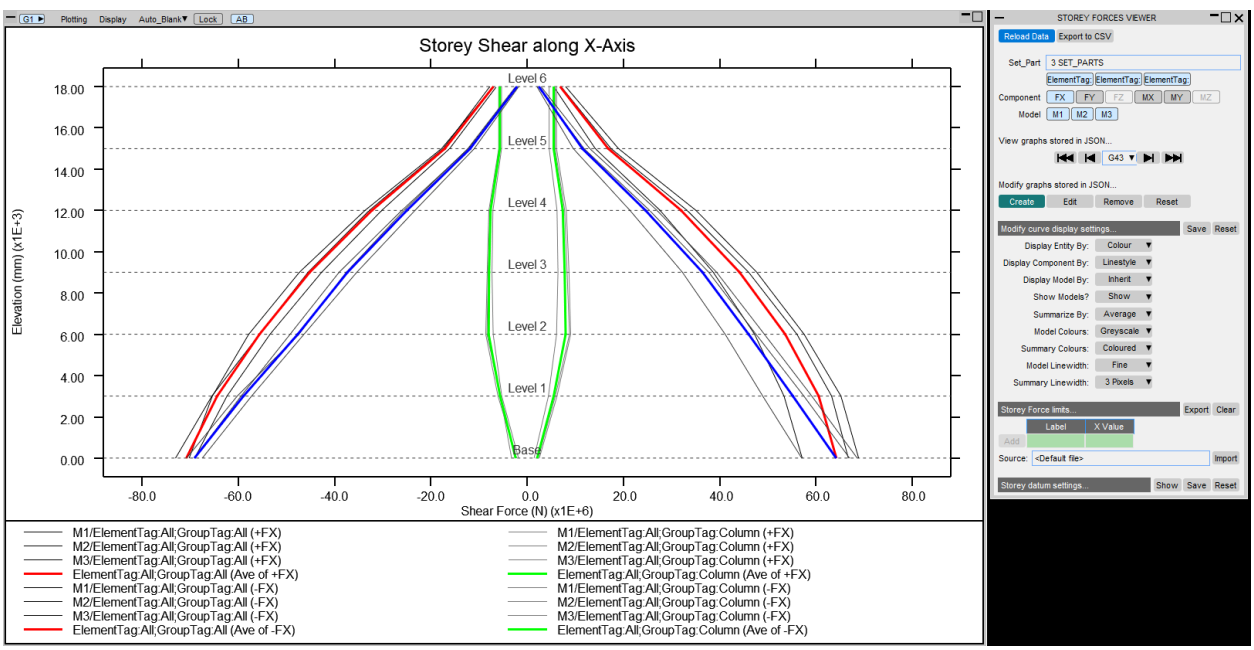


In PRIMER, you can define cross sections for each SET_PART, for every storey of the building.



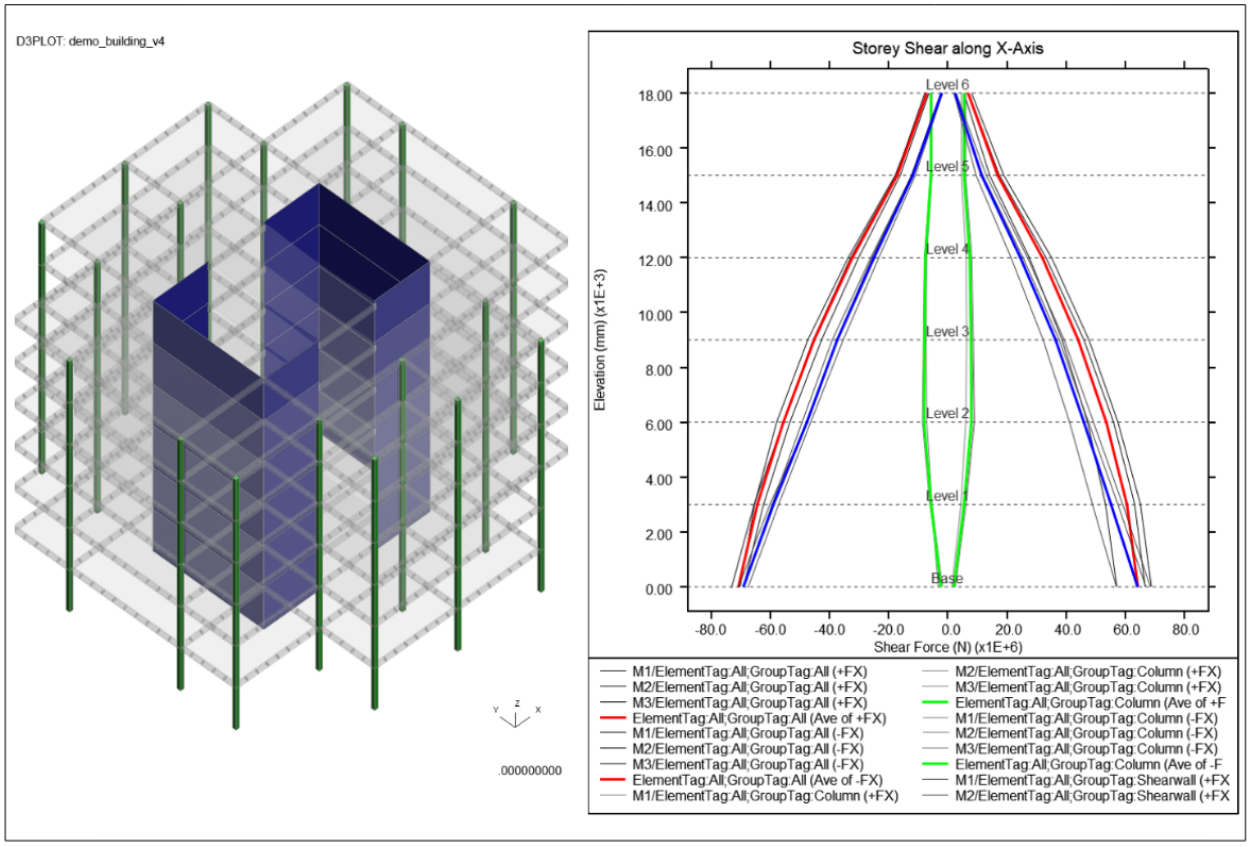
In T/HIS, storey forces are extracted for each of the cross sections you defined in PRIMER and then storey curves are generated – plotted on graphs.

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
Storey Shear along X-Axis
Elevation (mm) (x1E+3)
Shear Force (N) (x1E+6)
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
-80.0
-60.0
-40.0
-20.0
0.0
20.0
40.0
60.0
80.0
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
.000000000

Storey Force Report

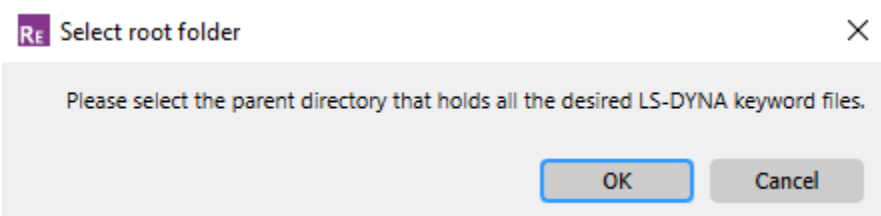
This workflow provides you with REPORTER templates to automatically generate report documents. The templates compile all T/HIS graphs you have set in PRIMER and T/HIS along with a model view from D3PLOT to show you the SET_PART(s) you have specified on each graph.

There are currently two templates with different report layouts available:

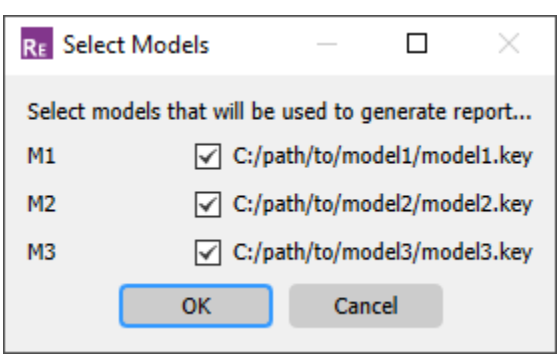
- **1x1** layout showing one T/HIS-graph/D3PLOT-model-view pair per page, split vertically.
- **2x1** layout showing two T/HIS-graph/D3PLOT-model-view pairs per page.

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.

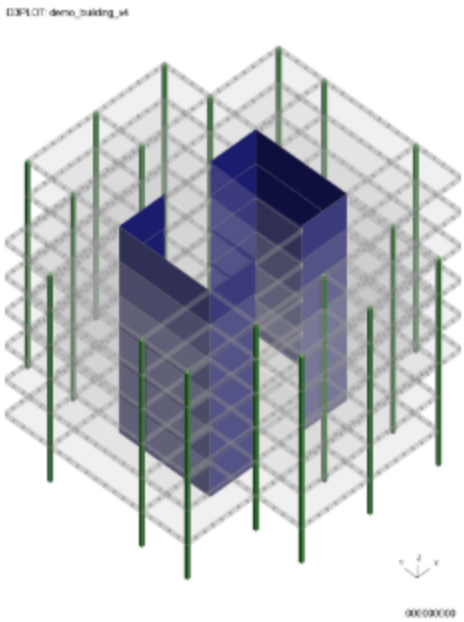
The REPORTER variables hold a record of the paths of models you have chosen to run. This can serve as a way to validate that you have run the models you intended.

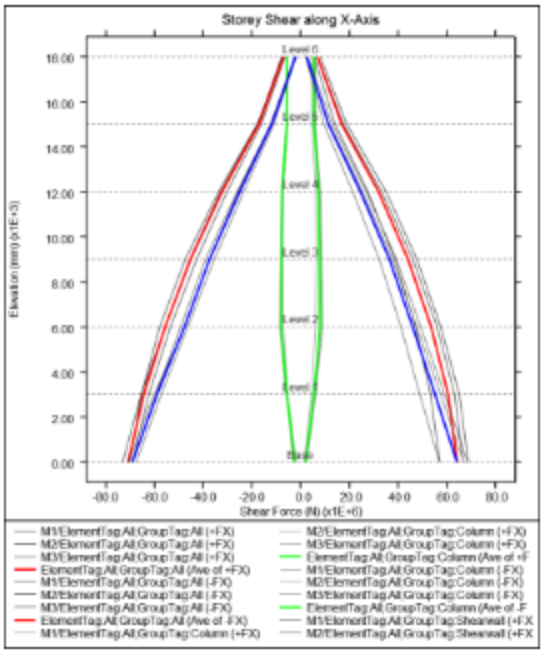
Storey Force

Seismic Analysis

3 SET_PARTS

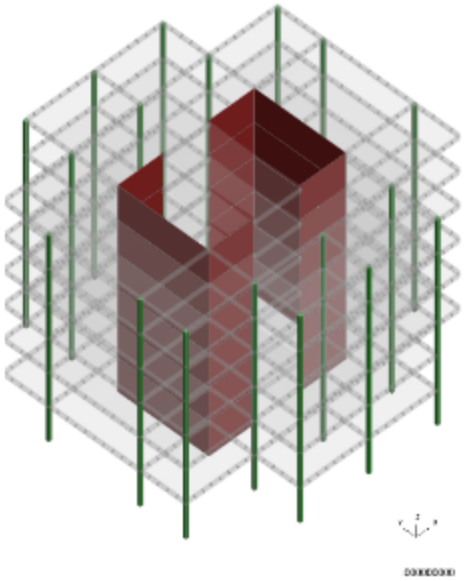
Shear along X

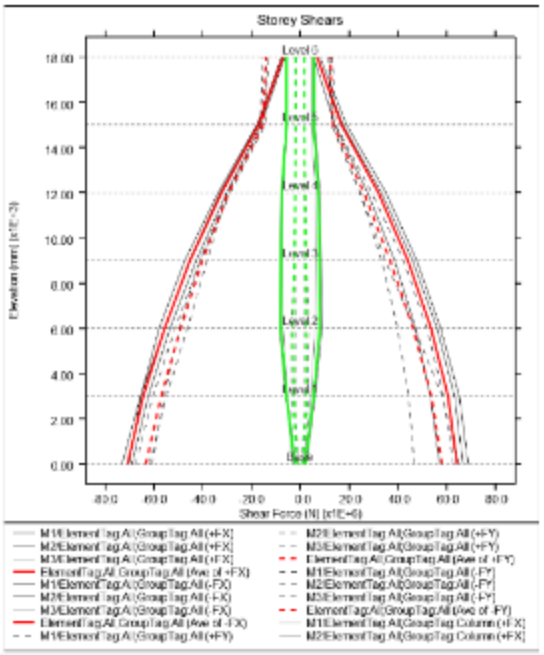




2 SET_PARTS

Shear along X, Y

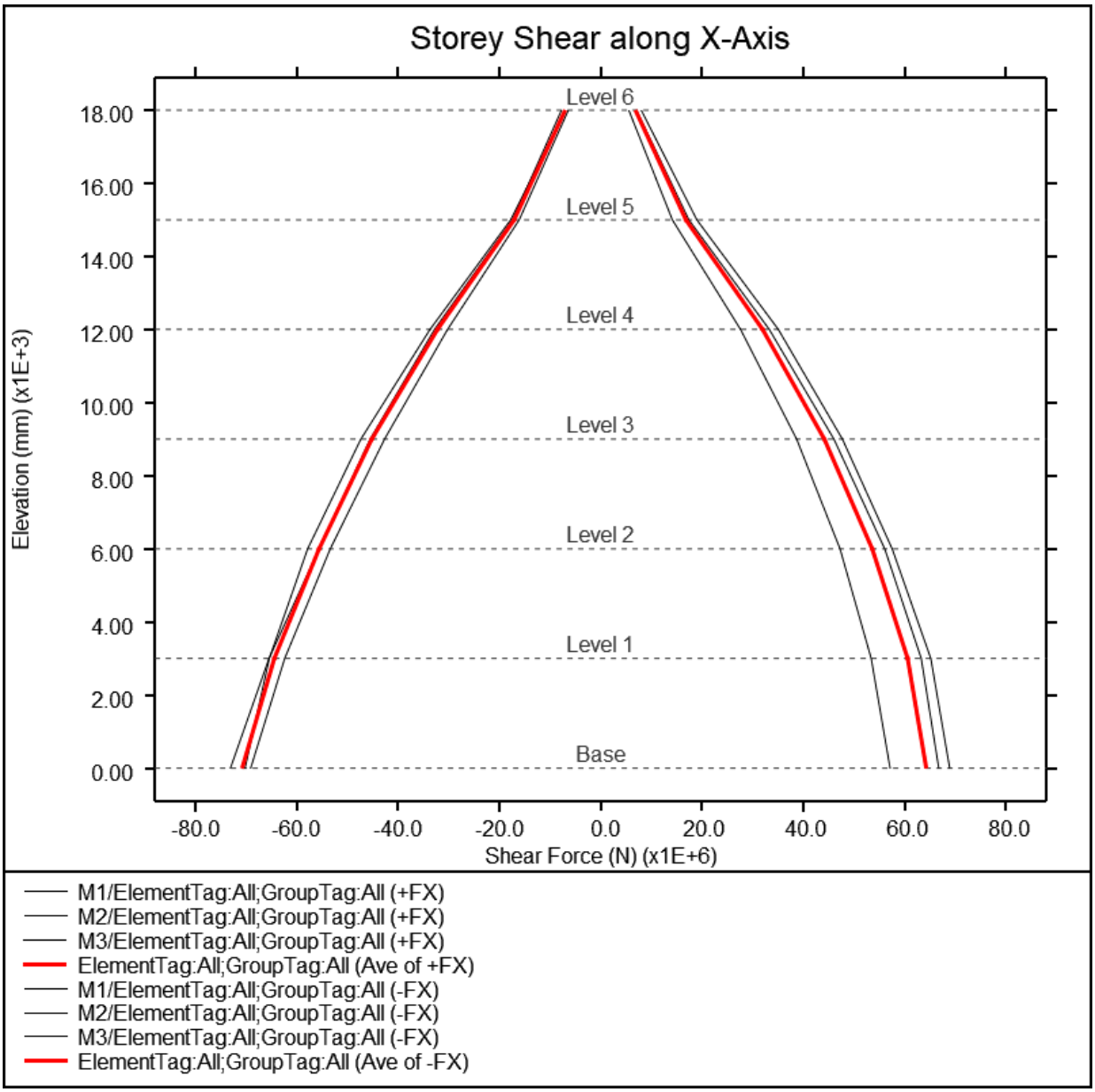


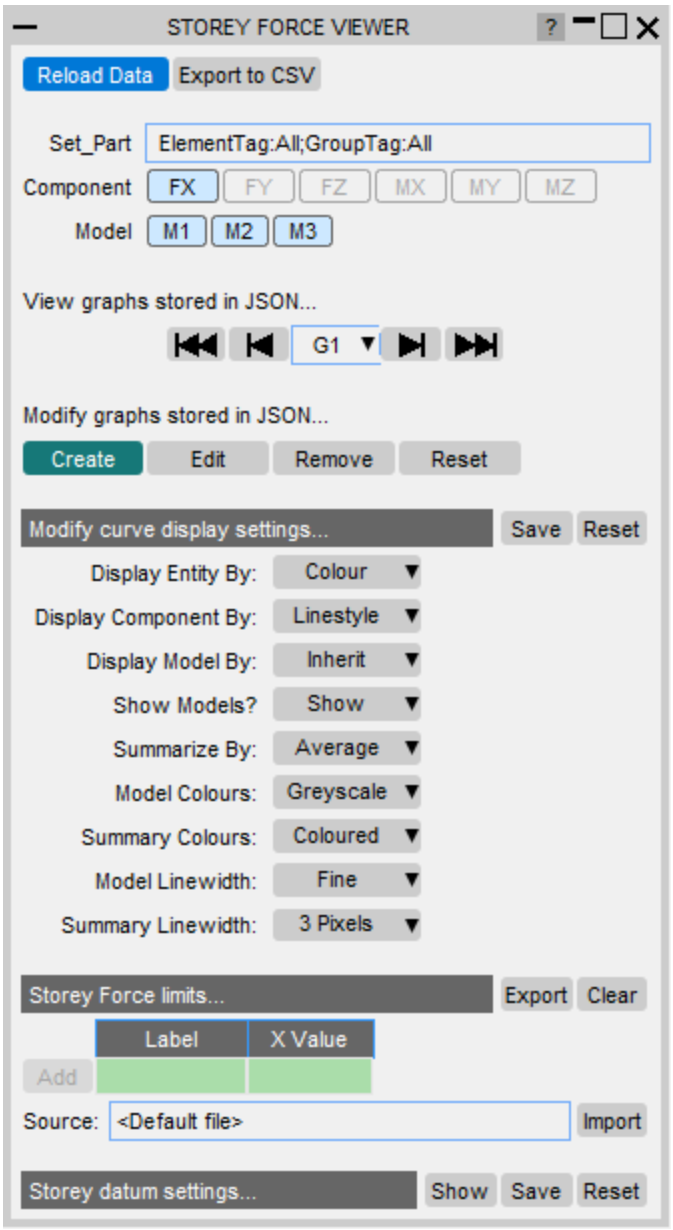


Storey Force Viewer

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
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)



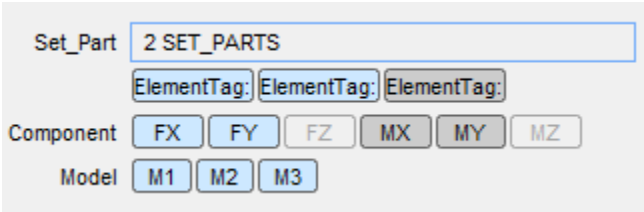
The Viewer GUI is generally split into four sections listed below:

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.

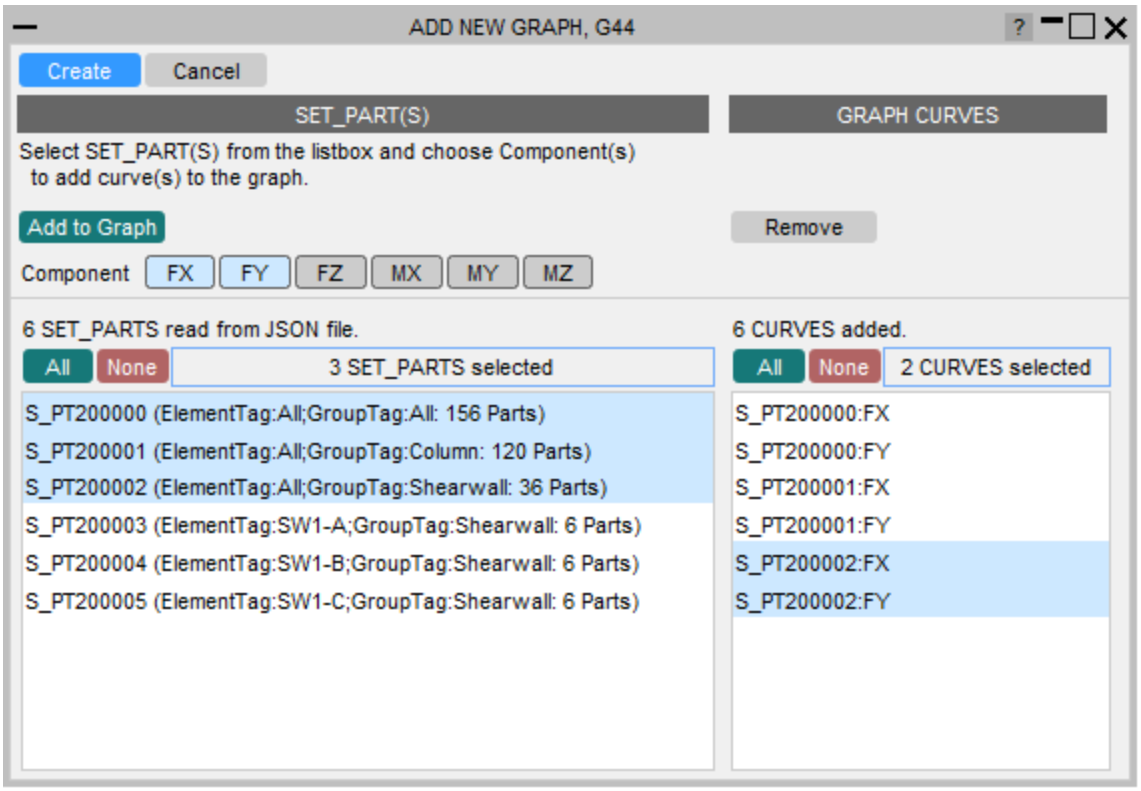
You will be provided with three toggles: **SET_PARTs**, **force components**, and **model**. The toggle for SET_PARTs will only be shown if more than one SET_PART is included in the current graph setup. All force component toggles will be shown, but only those included will be active. Finally, the model 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.

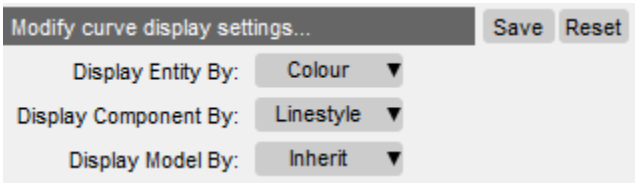
You may also wish to export the current T/HIS curves to an external file. You can do this by clicking **Export to CSV**.

Curve display settings

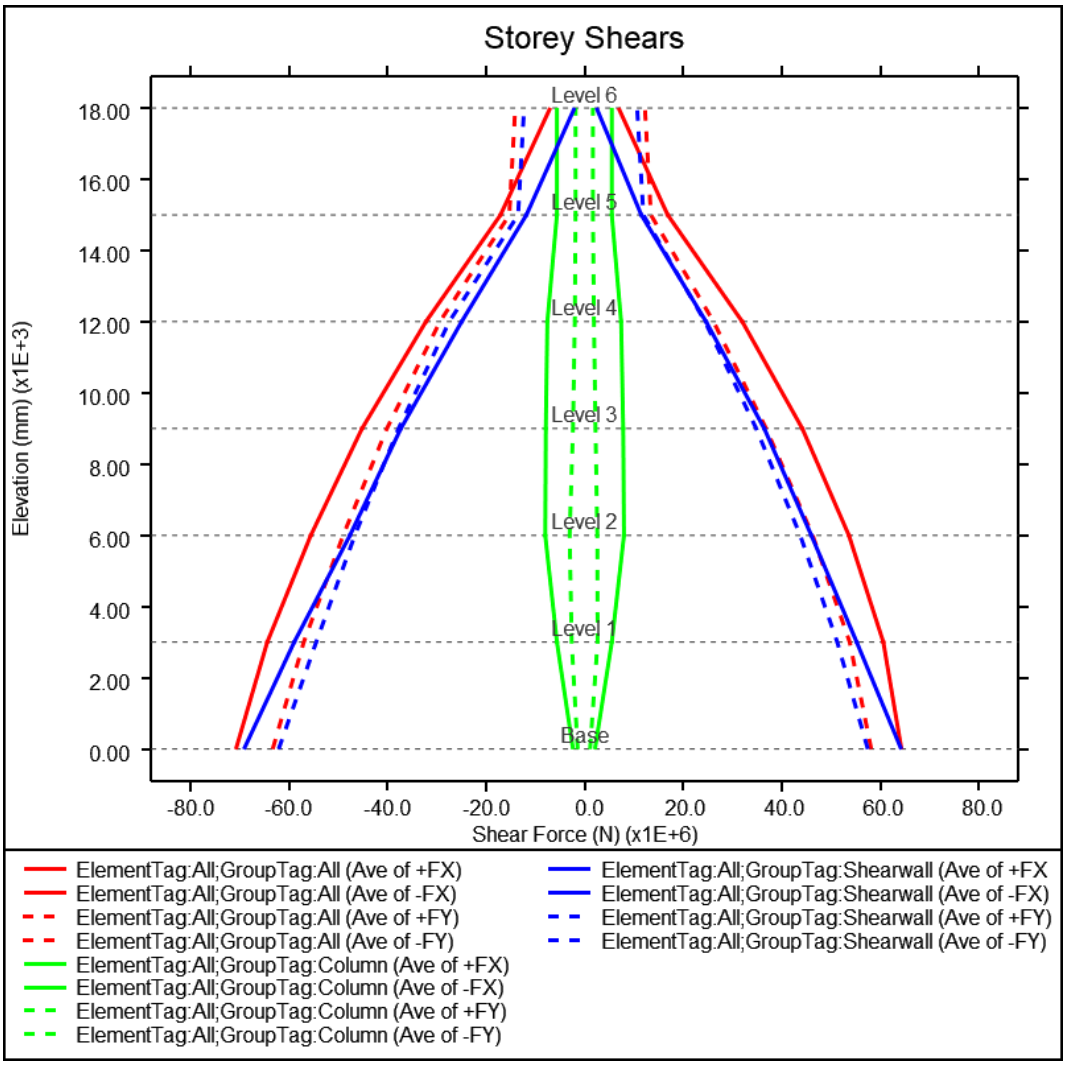
This panel allows you to define the formatting of the curves in the T/HIS graph. These settings will be applied to all graph setups stored in your Workflow file. Later when you generate the report, REPORTER will read these settings and apply the styling you have defined.

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:

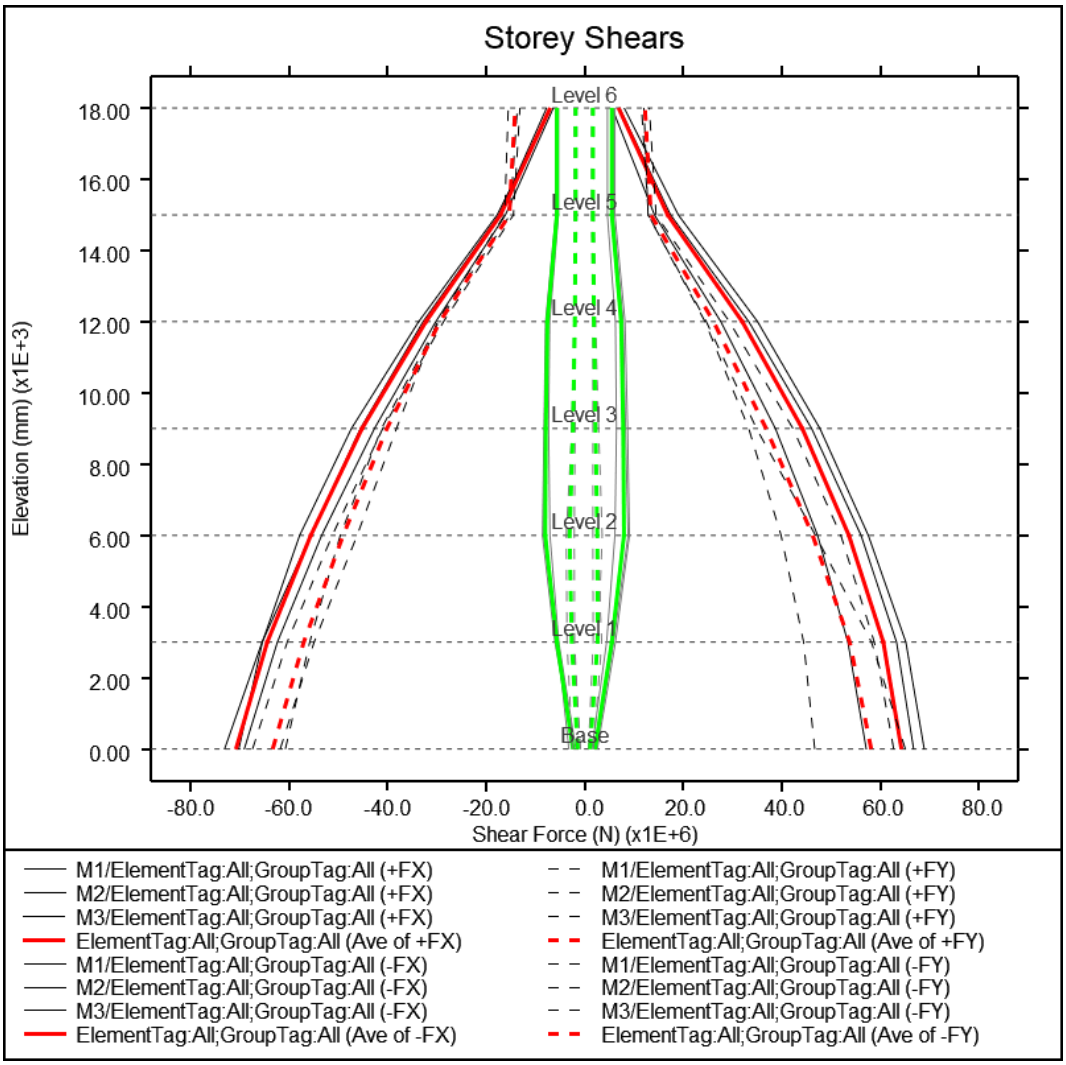


You can categorise the SET_PARTs and force components by **Colour** or **Line style.** For instance, if you display the SET_PARTs by colour and the force components by line style, the tool will then assign one colour for all curves under a SET_PART and will assign one line style for all curves under 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.

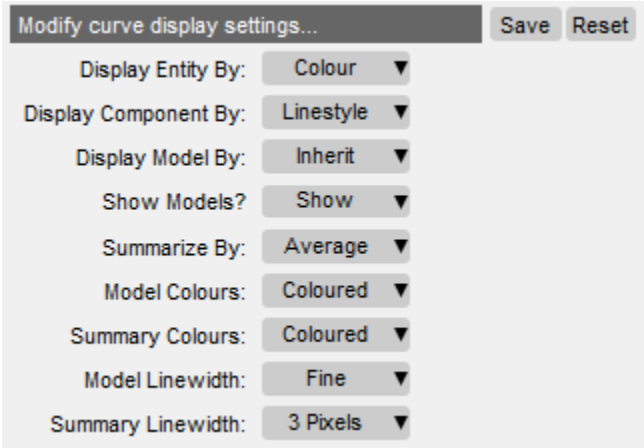


This current implementation of curve categorisation may not work for all scenarios, and could be improved in future. Please contact us with any feedback.

The other curve settings available to you are as follows:

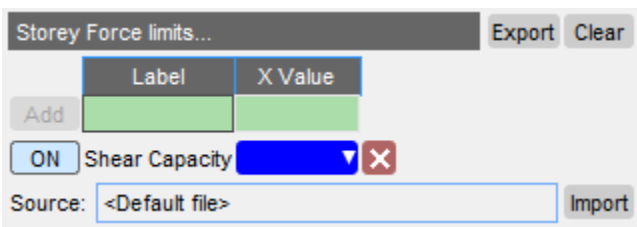
1. **Show Models** – set whether the model curves are shown or hidden in the plot. This is only relevant for **multiple model mode**.
2. **Summarise by** – choose which aggregate curve is shown. You have the following options: **None, Average, Envelope**.
3. **Model Colours** – choose whether the model 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:

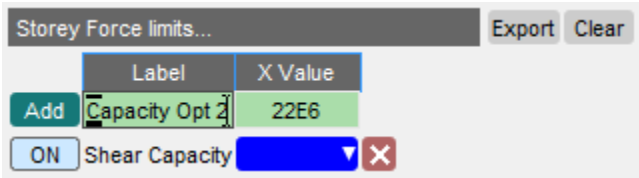
1. Constant curve limit along the structure elevation
2. Stepped curve limit, where the desired limit per storey extent varies

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.

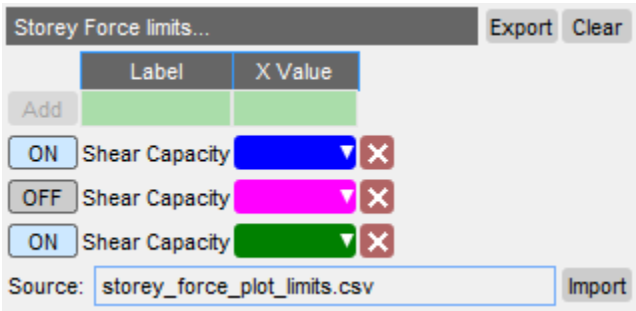
To define a constant curve limit, you need to define a label and the X-axis value in the textboxes provided. Then, click **Add**.

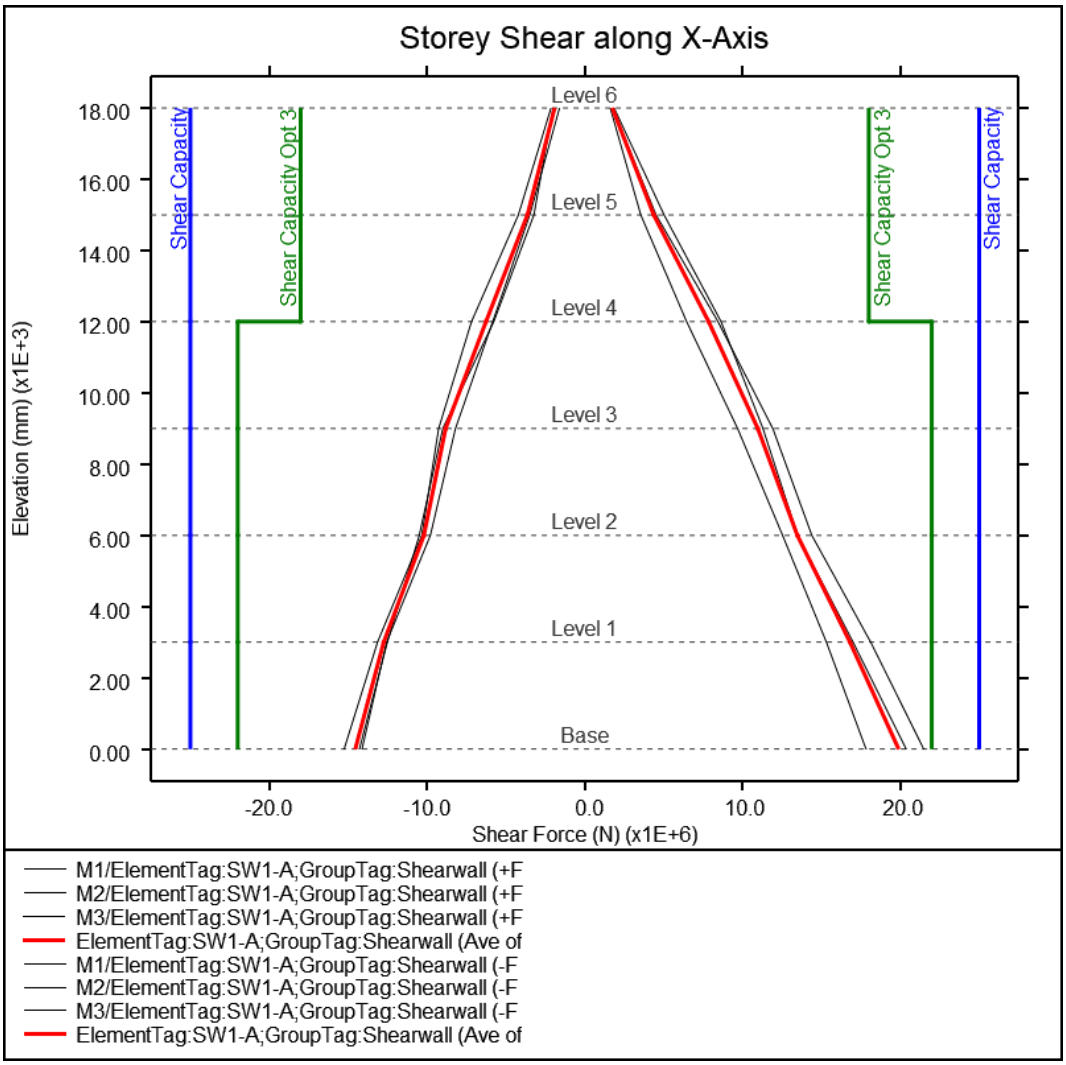


To define a stepped curve limit, create a CSV file following the data format of the exported sample file, as described above. Click **Import** to add the data to the plot.

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:

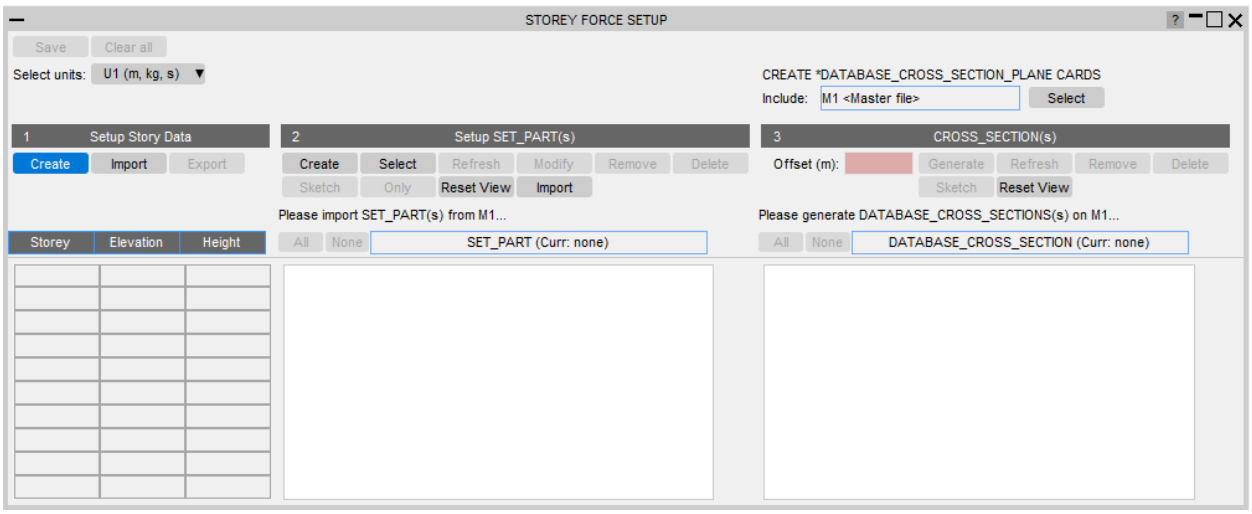
1. **Colour** – choose the colour of the storey datums
2. **Line Style** – choose the line style of the storey datums
3. **Line Width** – choose the line width of the storey datums
4. **Label Font** – choose the font of the storey datum labels
5. **Label Size** – choose the font size of the storey datum 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.

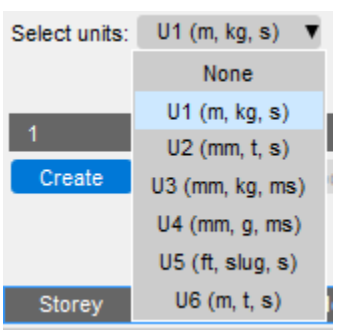
3.7.1.3. Storey Force PRIMER

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:

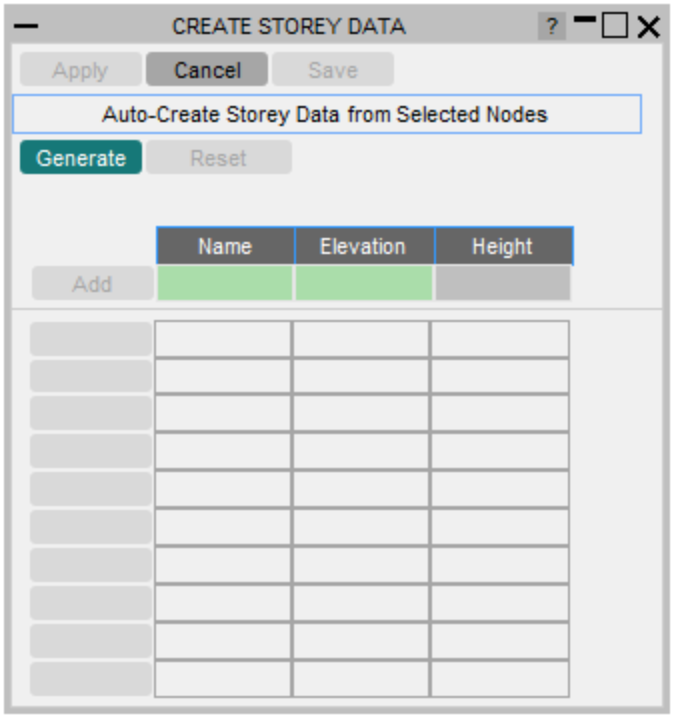


First, you need to choose the appropriate unit system from the dropdown menu:



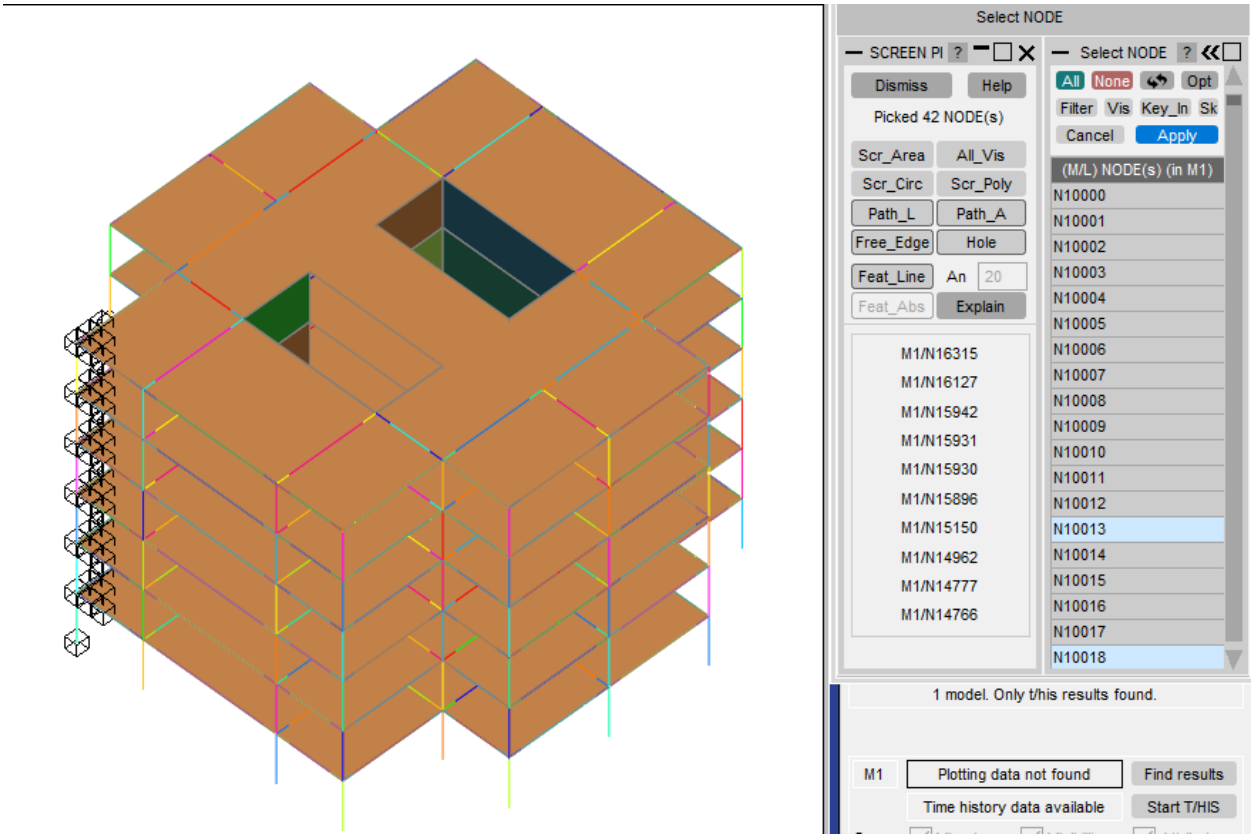
Defining Storey Data

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:



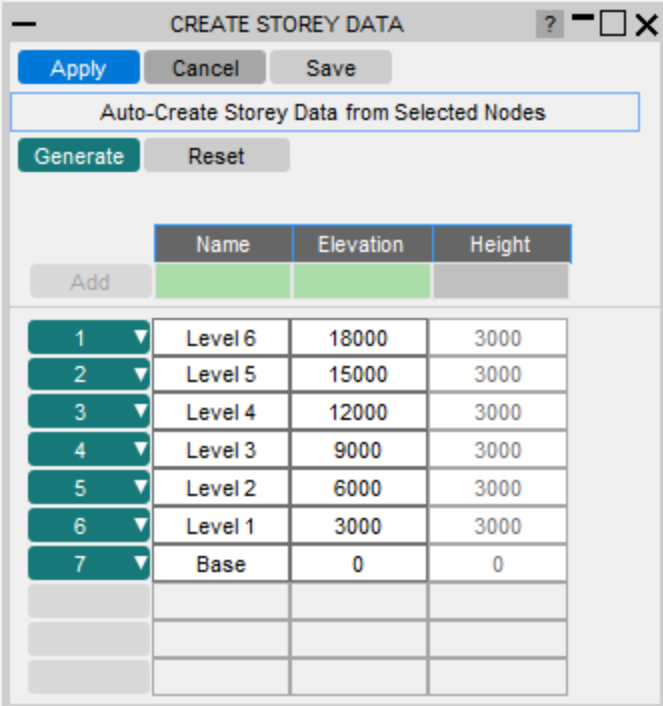
Each storey can be defined manually by populating **Name** and **Elevation** textboxes and then clicking the **Add** button which will be activated if the inputs are valid.

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.

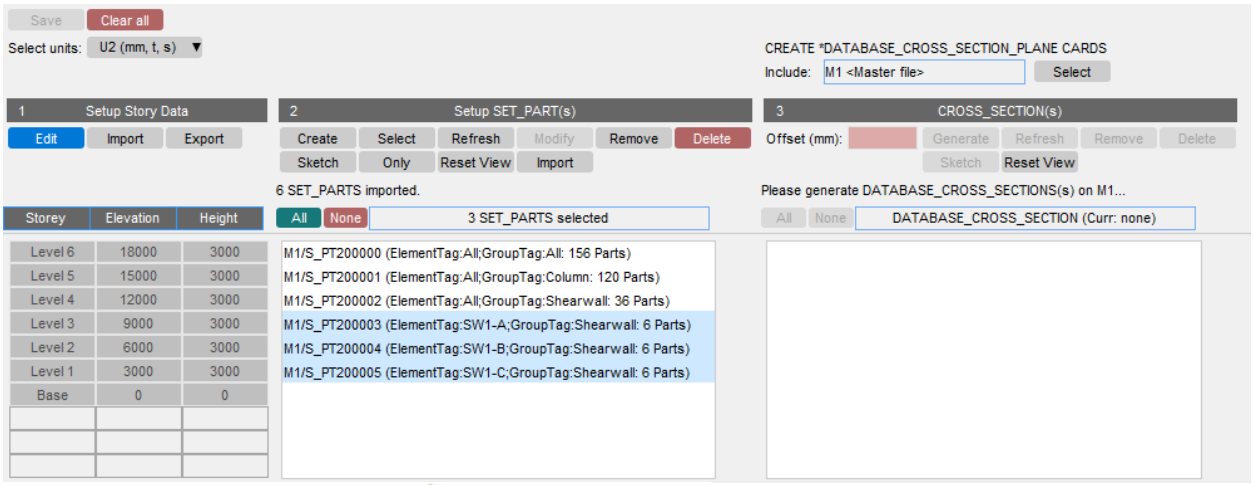
You can optionally save this data by clicking **Save**. This will write it to a separate JSON file, which you can **Import** when you are starting a new setup. Normally, storey data would be applicable to multiple Seismic workflows, so saving this data will be useful to those other workflows too.

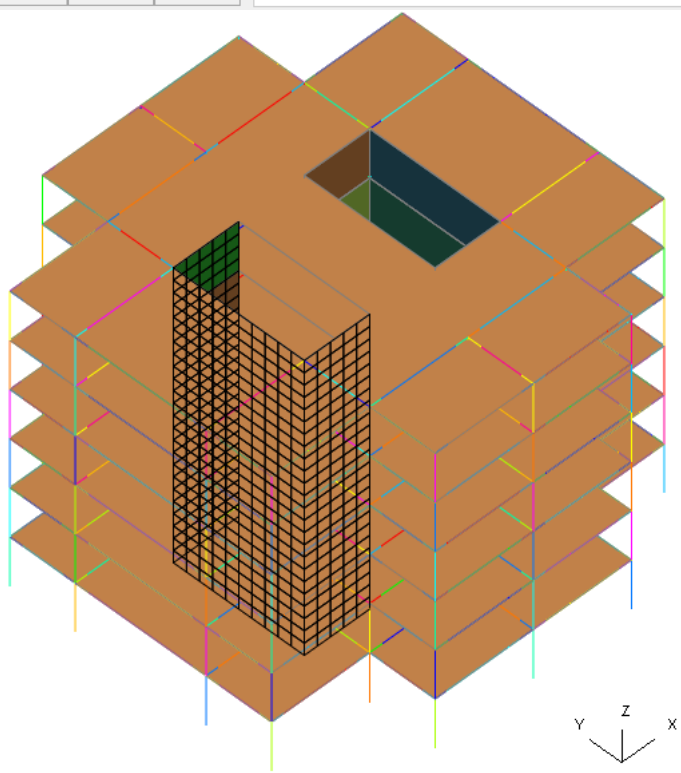


Defining SET_PARTs

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.





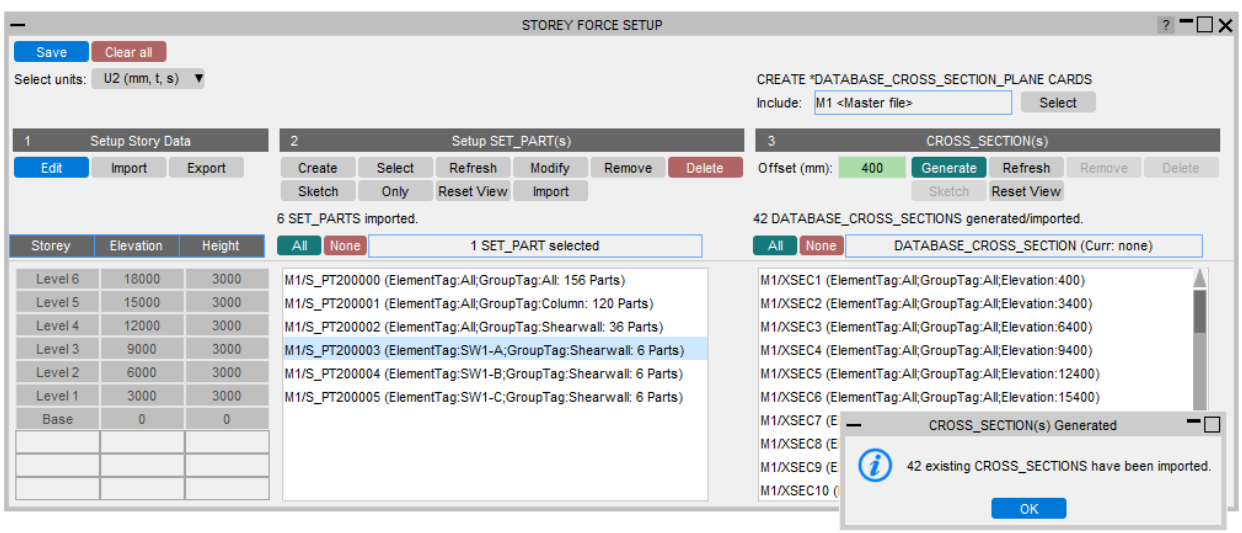
Some section controls will be active when you select SET_PARTs in the list box as shown above. You will have access to SET_PART commands that will allow you to modify a SET_PART or update your list.

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.

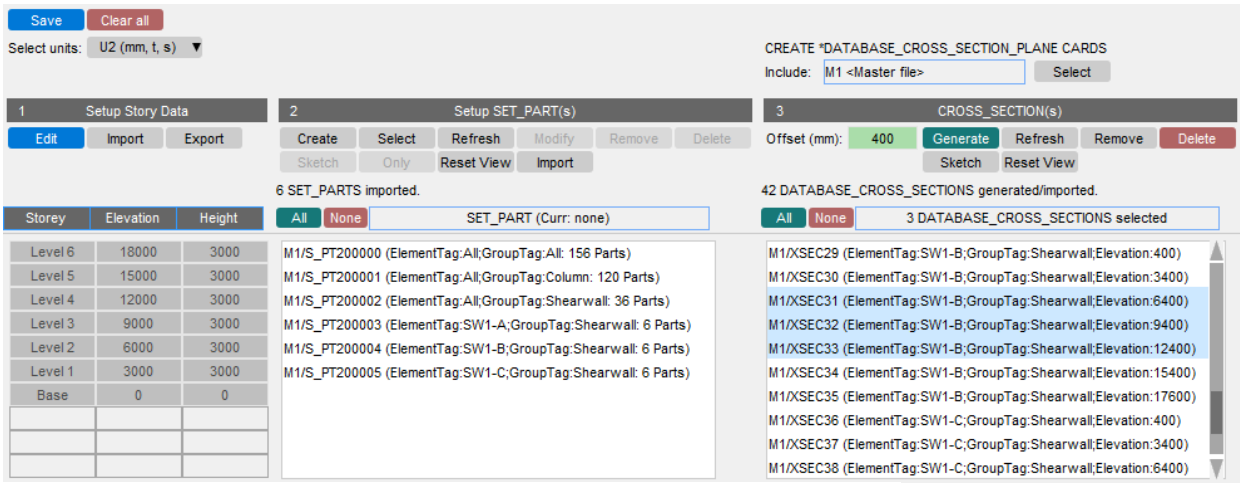
3. **Delete** – remove existing SET_PARTs from the list and delete them from the model. The deletion is **not recursive** so the PARTs along with their components will still remain.
4. **Sketch** – highlight the contents of one or more SET_PARTs in the model.
5. **Only** – isolate the components of one 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.

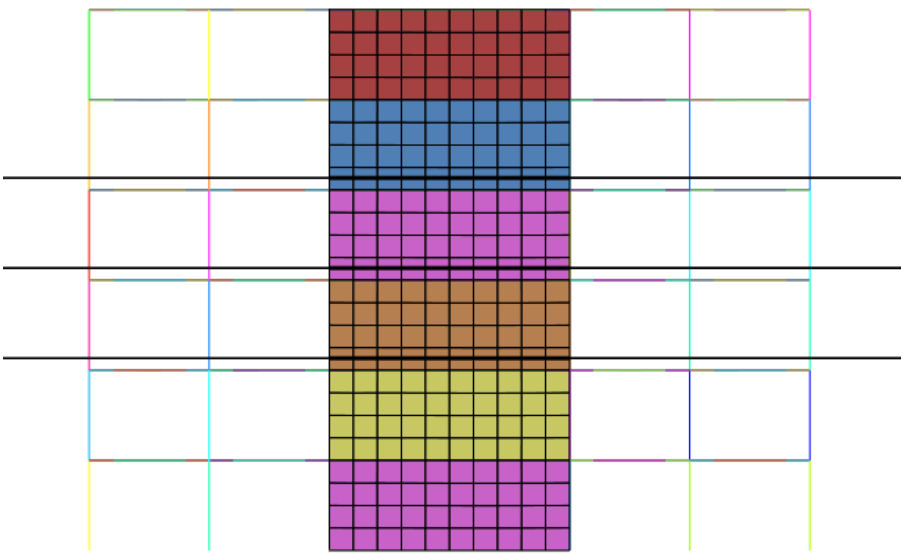
Defining Storey DATABASE_CROSS_SECTIONs

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.

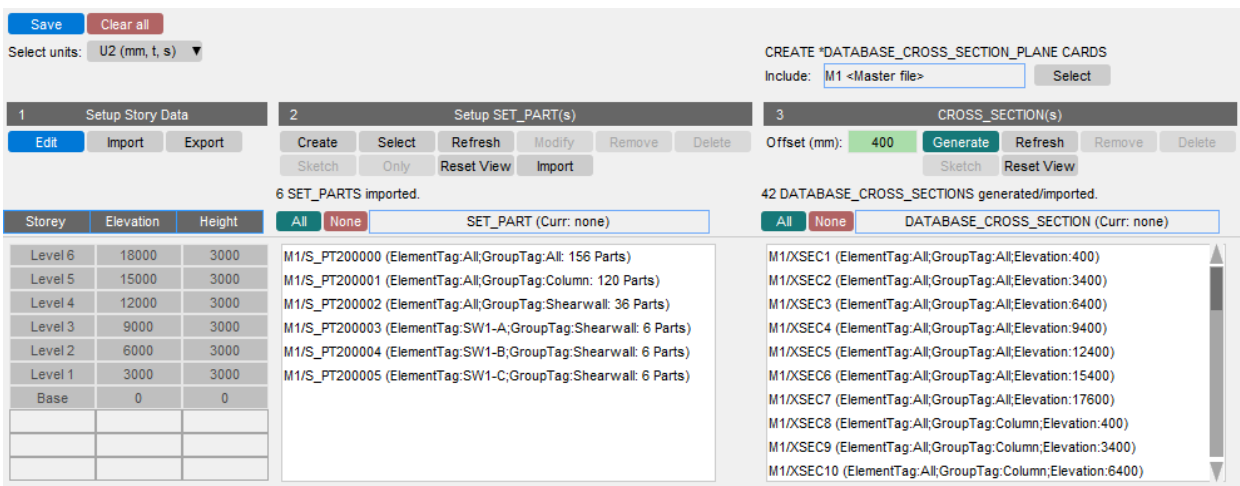
1. **Remove** – remove existing DATABASE_CROSS_SECTIONs from the list. This will not delete them from the model.
2. **Delete** – remove existing DATABASE_CROSS_SECTIONs from the list and delete them from the model.
3. **Sketch** – highlight the DATABASE_CROSS_SECTIONs you selected, along with the SET_PARTs 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.

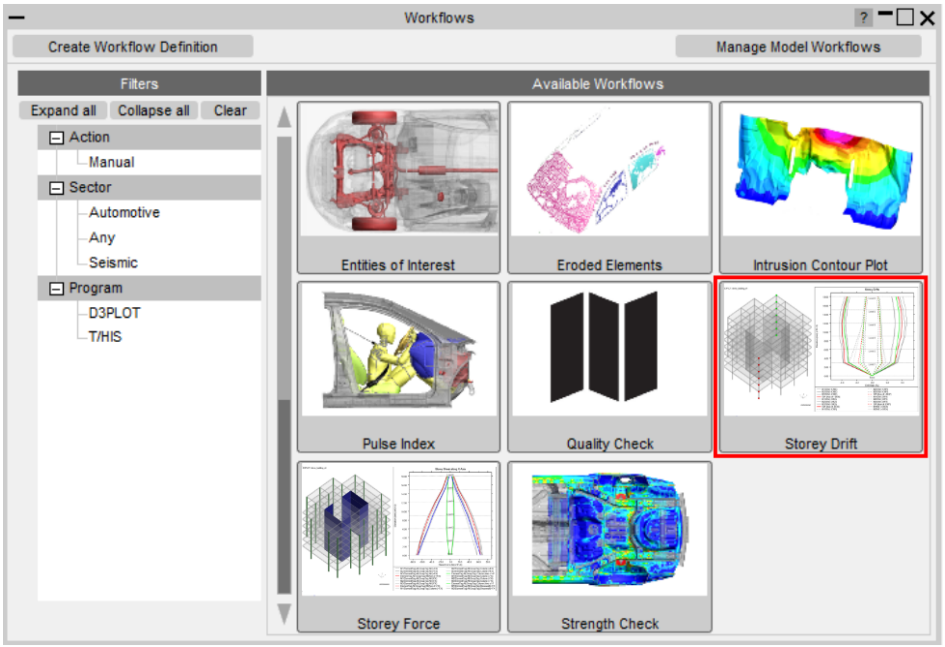
STOREY DRIFT SETUP
Save
Clear all
Select units:
U2 (mm, t, s)
Create *DATABASE_HISTORY_NODE cards
Include:
M1 <Master file>
Select
1
Setup Story Data
Create
Import
Export
Storey
Elevation
Height
2
Setup Drift Nodes
Add Nodes
Delete All
Import
Reset View
Add Location
Import Storey Data
Do you wish to get the storey data from the existing workflow file?
Yes
No

3.7.2. Storey Drift

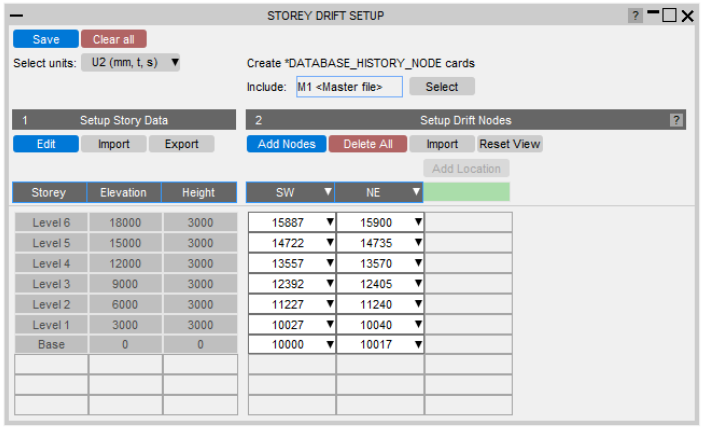
Storey Drift

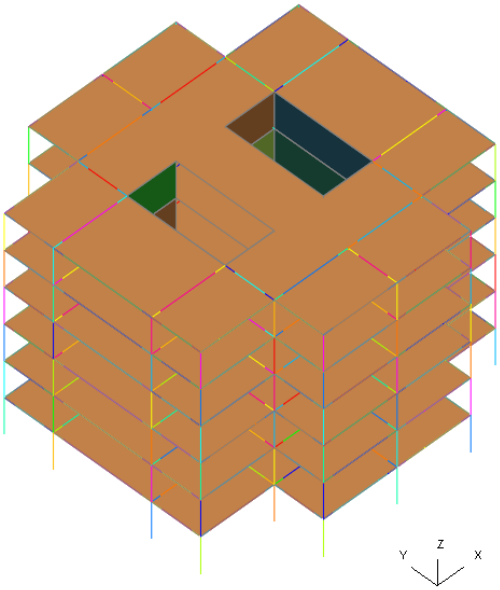
Tools → Workflows → Storey Drift

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.



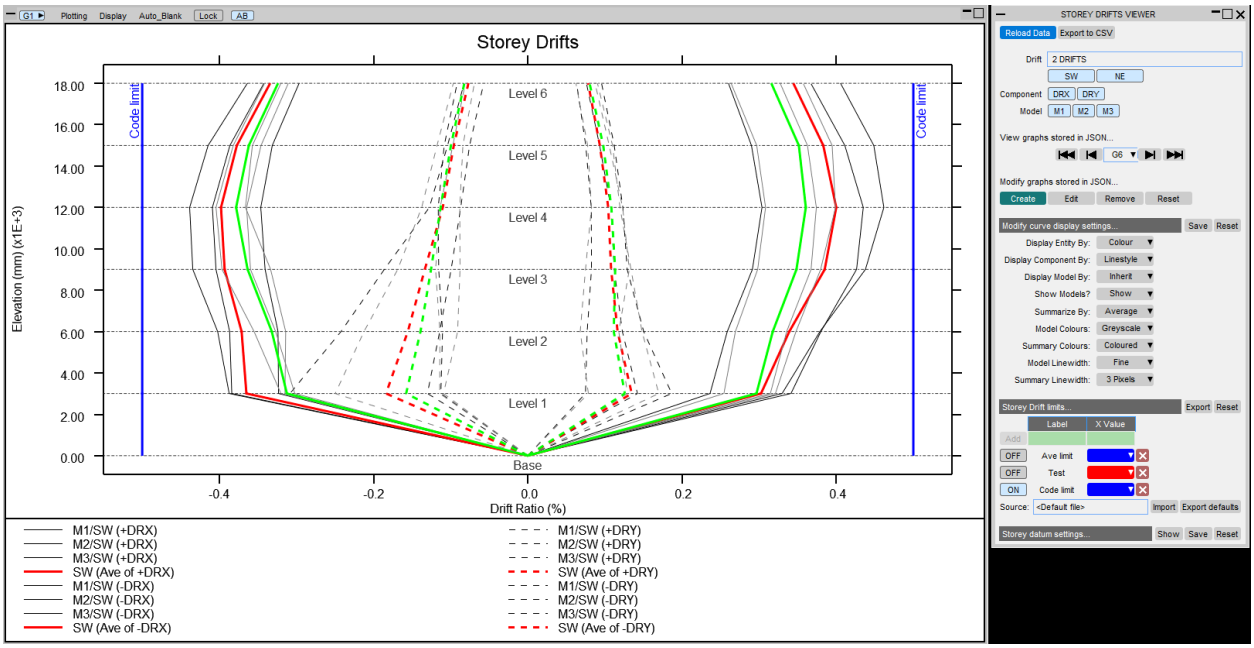
In PRIMER, you can setup drift locations, defining nodes for each storey of the building.





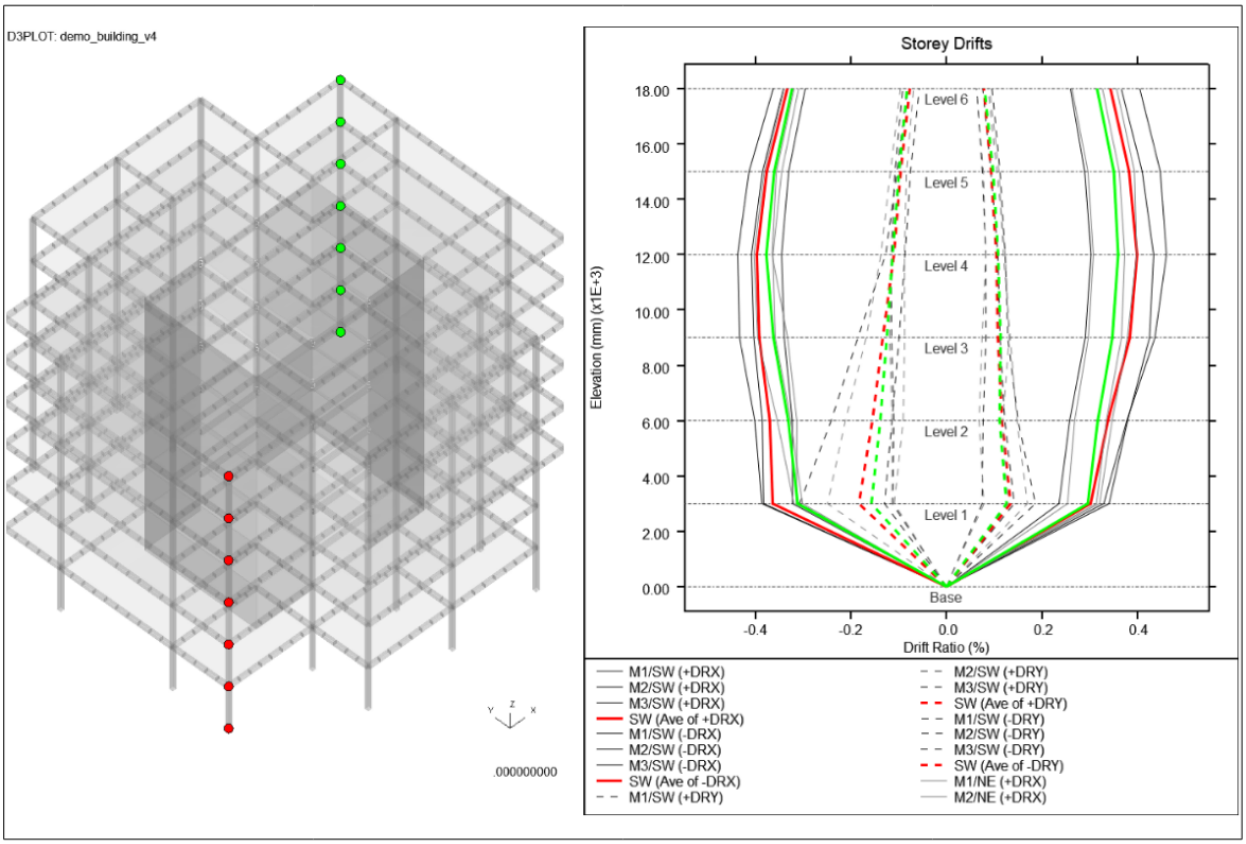
In T/HIS, storey drifts are calculated for each of the locations you defined in PRIMER and then storey curves are generated – plotted on graphs.

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.



Storey Drift Report

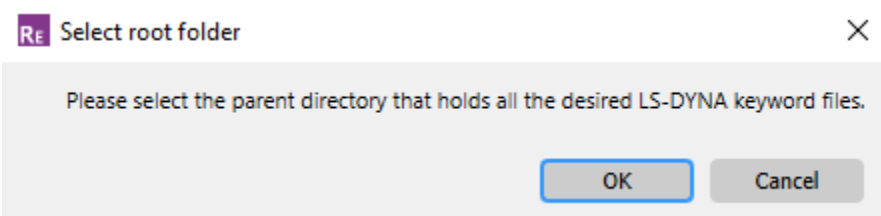
This workflow provides you with REPORTER templates to automatically generate report documents. The template compiles all T/HIS graphs you have set in PRIMER and T/HIS along with a model view from D3PLOT to show you the locations of drifts you have specified on each graph.

There are currently two templates with different report layouts available.

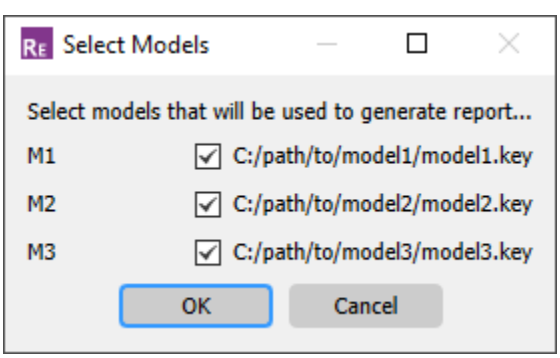
- **1x1** layout showing one T/HIS-graph/D3PLOT-model-view pair per page, split vertically.
- **2x1** layout showing two T/HIS-graph/D3PLOT-model-view pairs per page.

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.

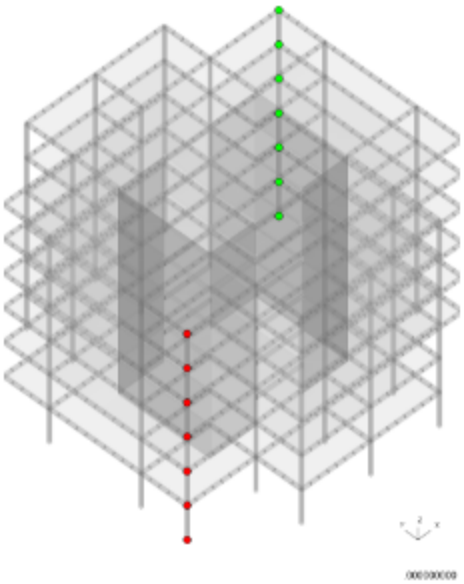
Storey Drift

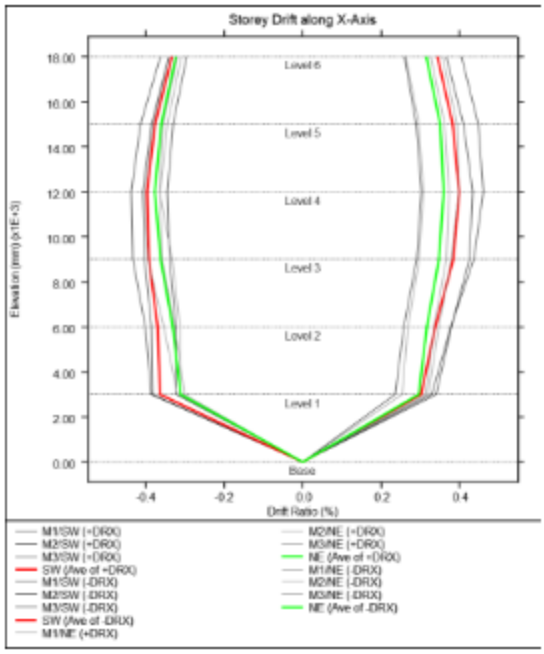
Seismic Analysis

2 DRIFTS

Drift along X

D3PLOT: demo_building_v1

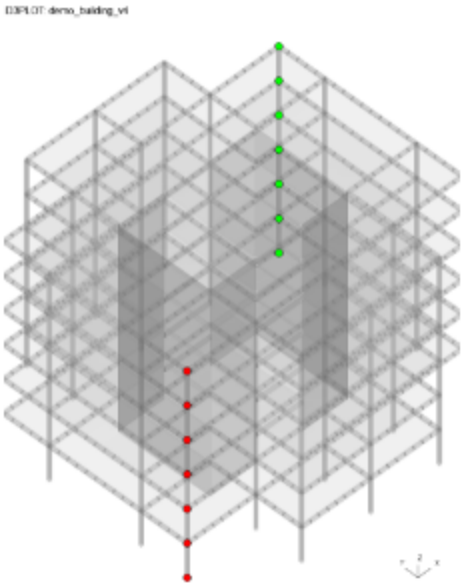


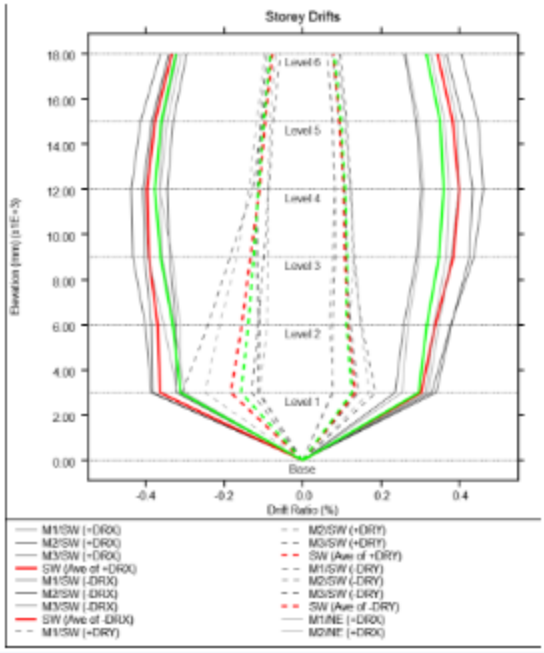


2 DRIFTS

Drift along X, Y

D3PLOT: demo_building_v1



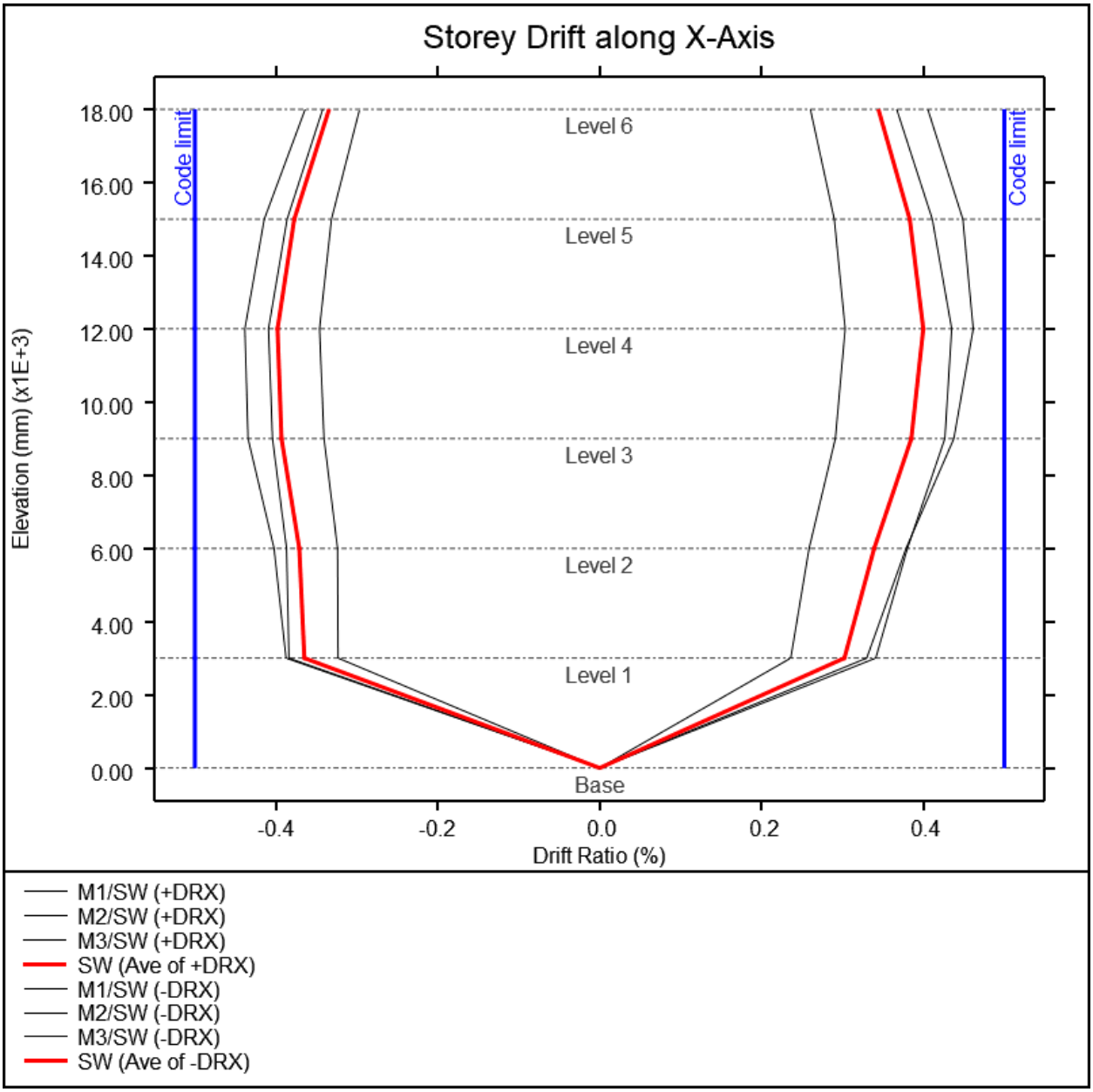


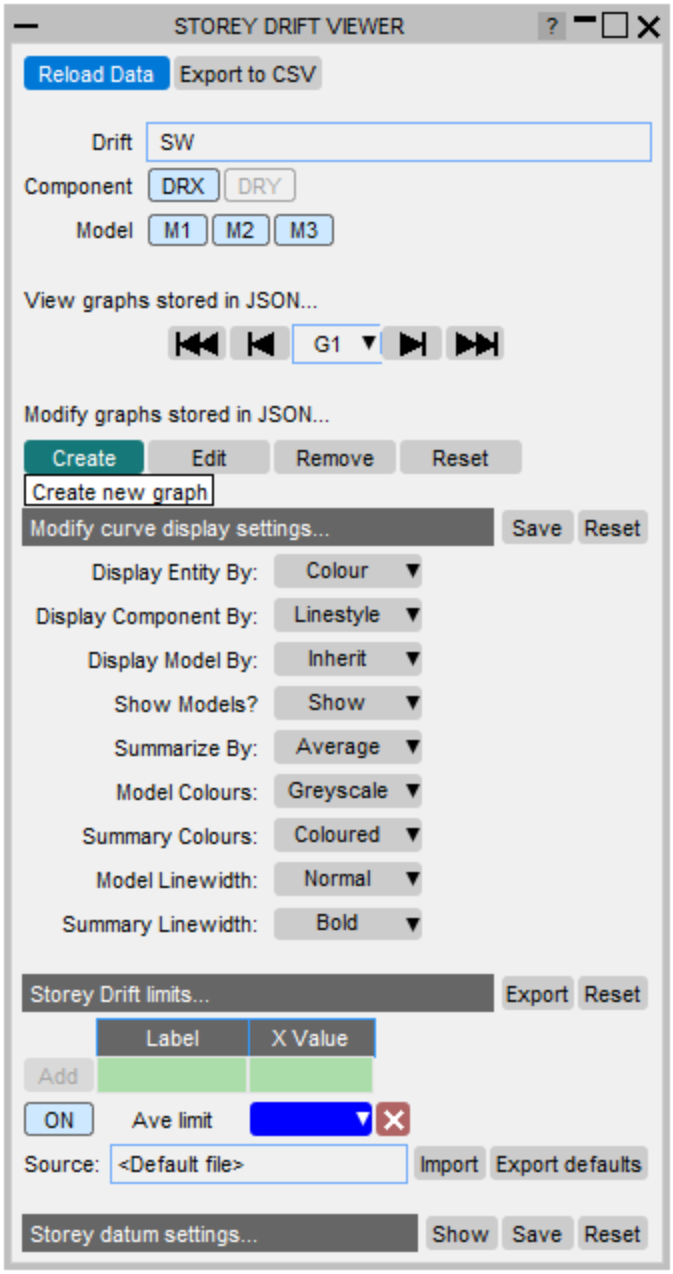
Model Directory | C:/Users/Ian-Keith.Relucio/OneDrive - Arup/Arup UK/LS-Dyna_Dev/DEMO_MDL/01_MULTI_MDL_TEST

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:


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)

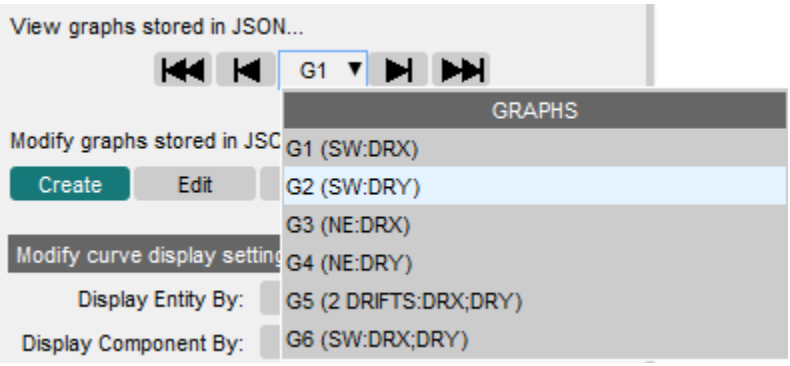


The Viewer GUI is generally split into four sections listed below:

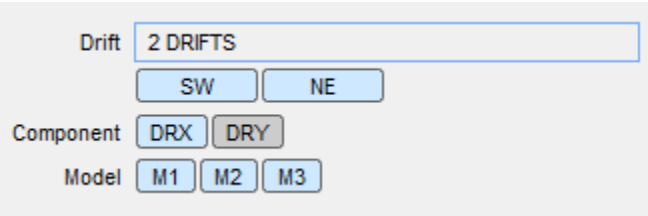
1. Graph selection/creation panel
2. Curve display settings
3. Storey drift limits definition
4. Storey datum settings

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.



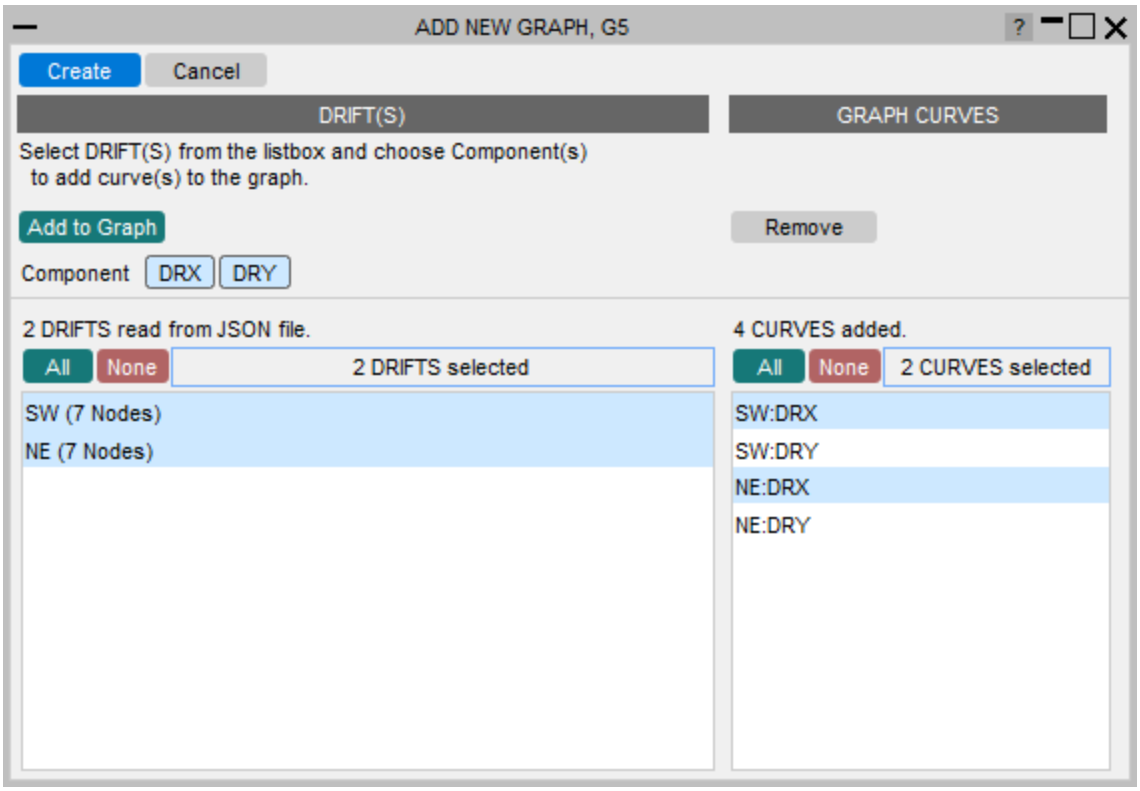
You will be provided with three toggles: **drift locations**, **direction components**, and **model**. The toggle for drift locations will only be shown if more than one drift is included in the current graph setup. All direction component toggles will be shown, but only those included will be active. 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.

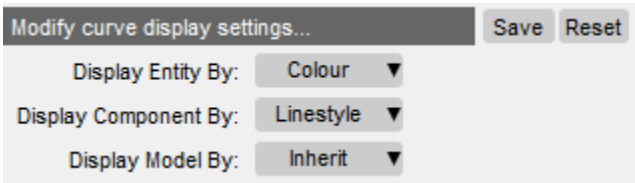
You can also export the current T/HIS curves to an external file. You can do this by clicking **Export to CSV**.

Curve display settings

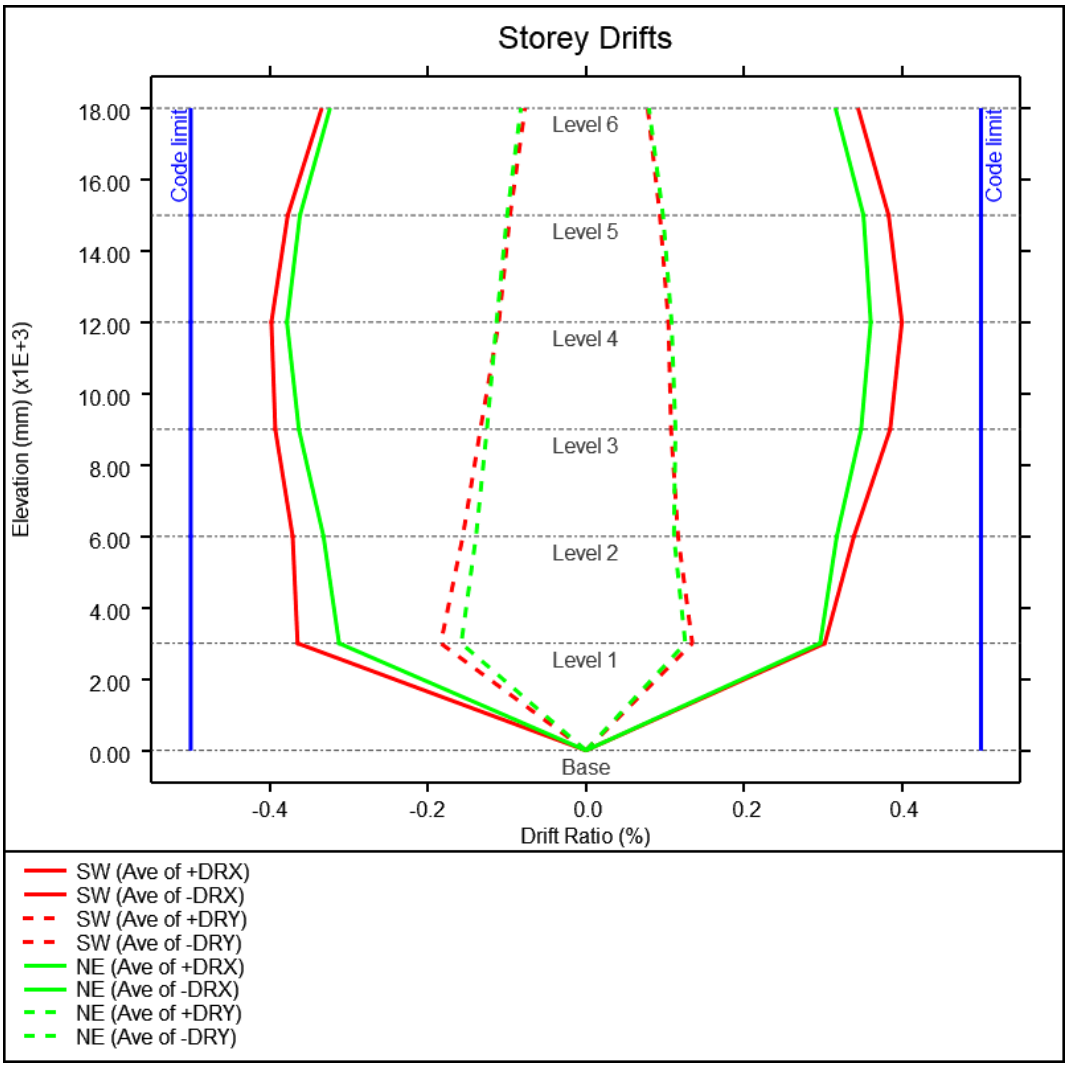
This panel allows you to define the formatting of the curves in the T/HIS graph. These settings will be applied to all graph setups stored in your Workflow file. Later, when you generate the report, REPORTER will read these settings and apply the styling you have defined.

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:

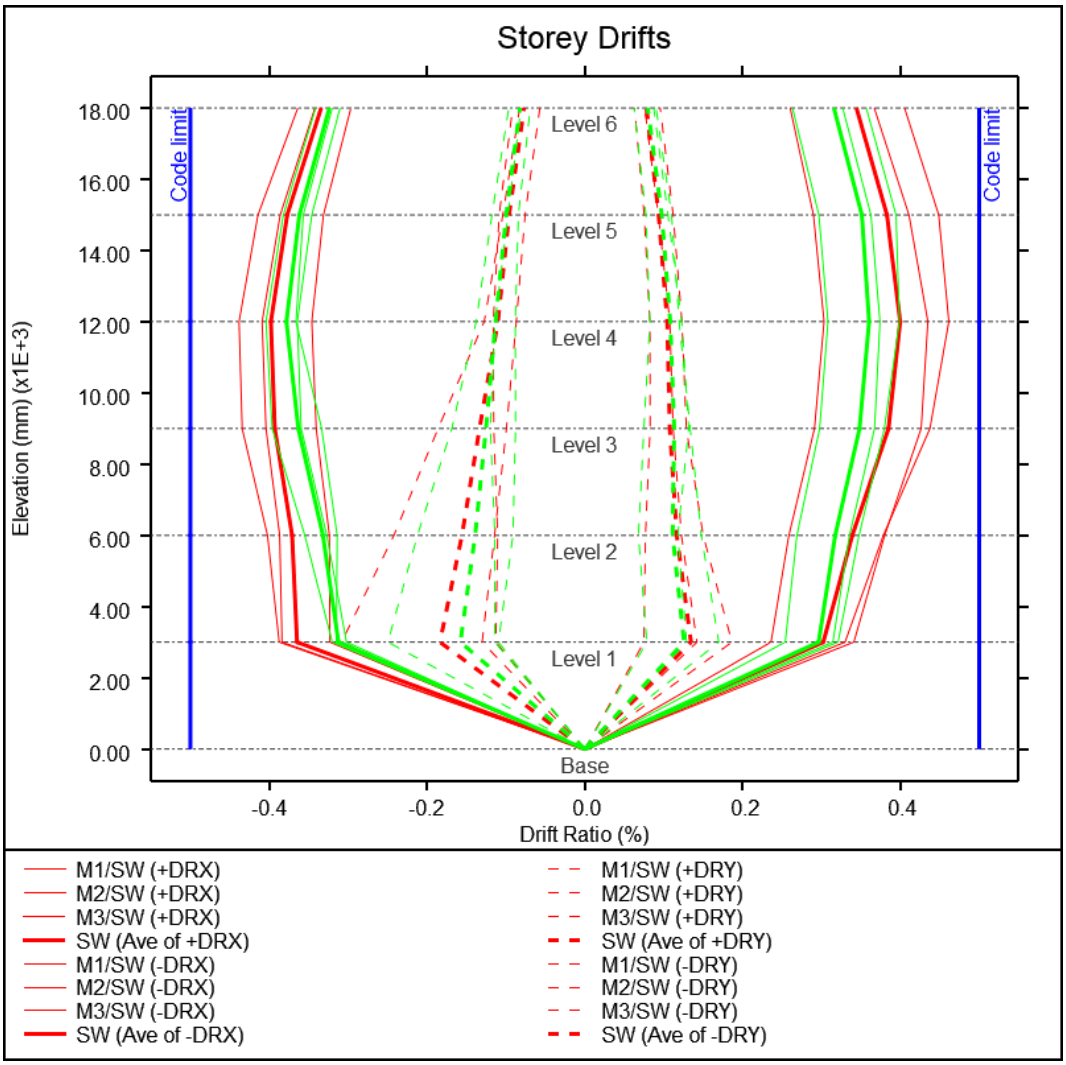


You can categorise the drift locations and direction components by **Colour** or **Line style.** For example, if you display the drift locations by colour and the direction components by line style, the tool will then assign one colour for all curves under a drift location and will assign one line style 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:

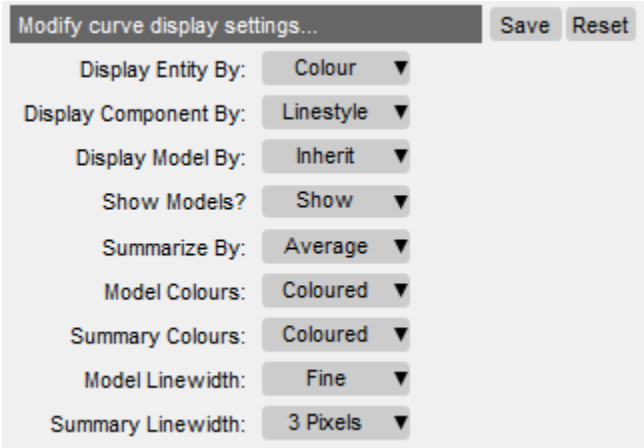


This current implementation of curve categorisation may not work for all scenarios, and could be improved further in future. Please contact us with your feedback.

The other curve settings available to you are described below:

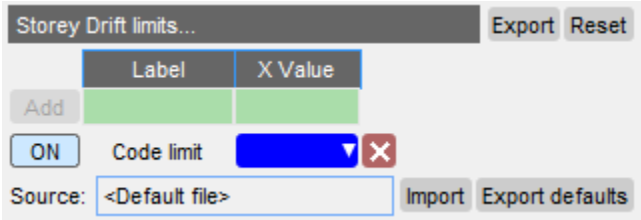
1. **Show Models** allows you to set if the model curves are shown or hidden in the plot. This is only relevant for **multiple model mode**.
2. **Summarize by** allows you to choose which aggregate curve is shown. You have the following options: **None, Average, Envelope**.
3. **Model Colours** allows 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.

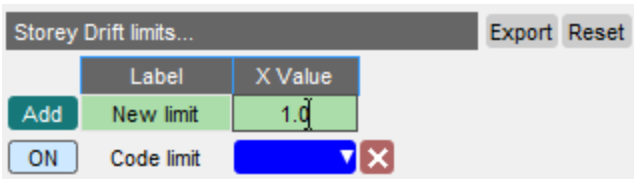
There are two types of vertical storey curve limits that you can define:

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.

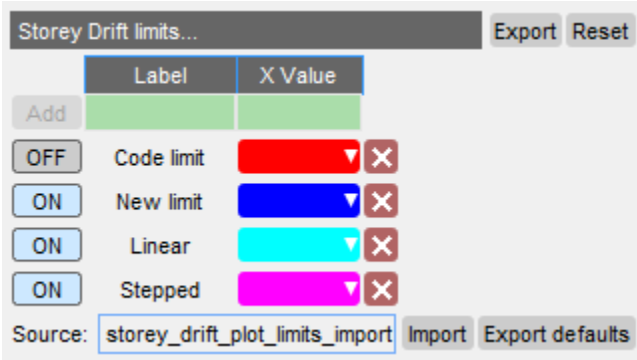
To define a constant curve limit, you would need to define a label and the X-axis value on the text boxes provided. Then, click **Add**.

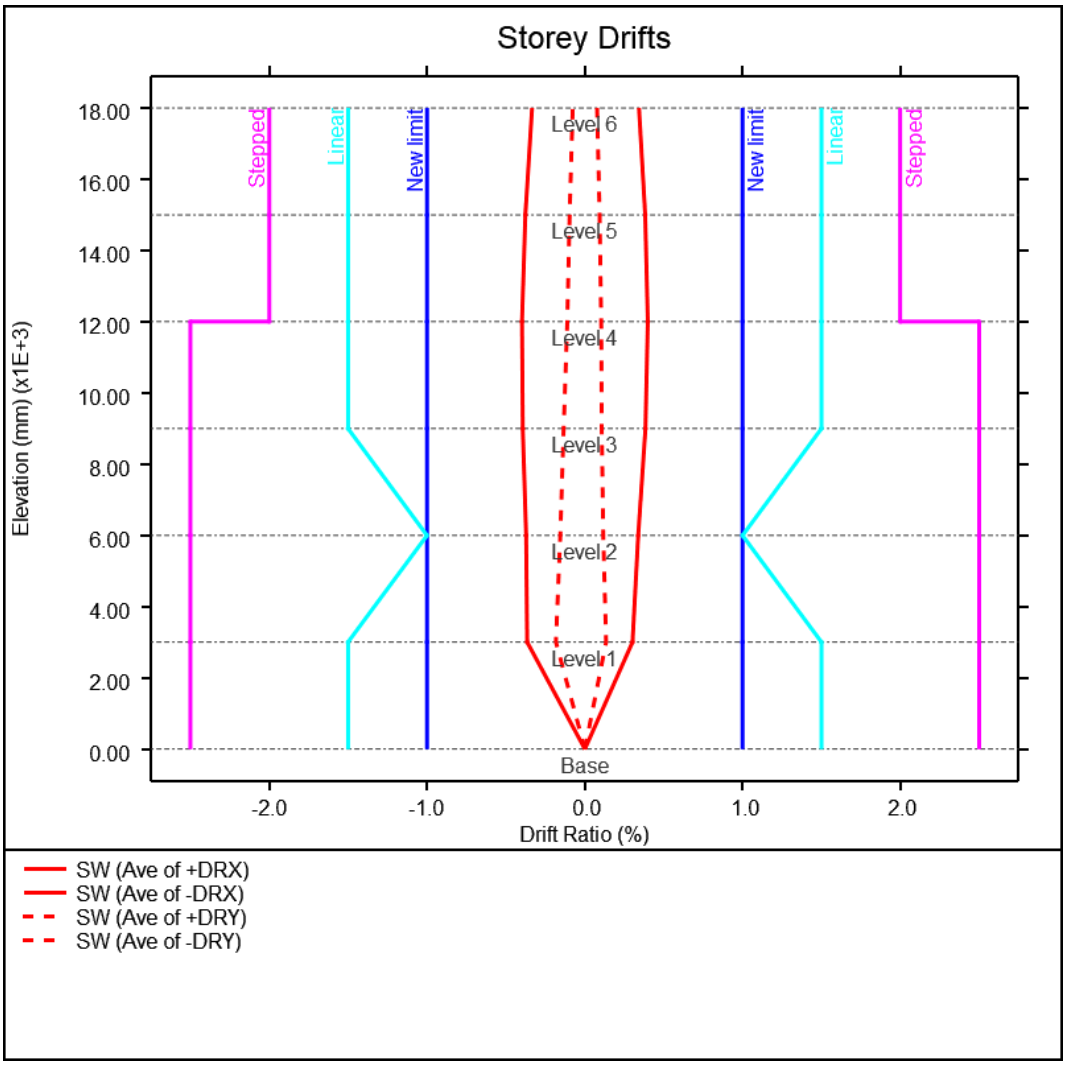


To define a stepped curve limit, create a CSV file following the data format of the exported default file. Click **Import** to add the data to the plot.

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.

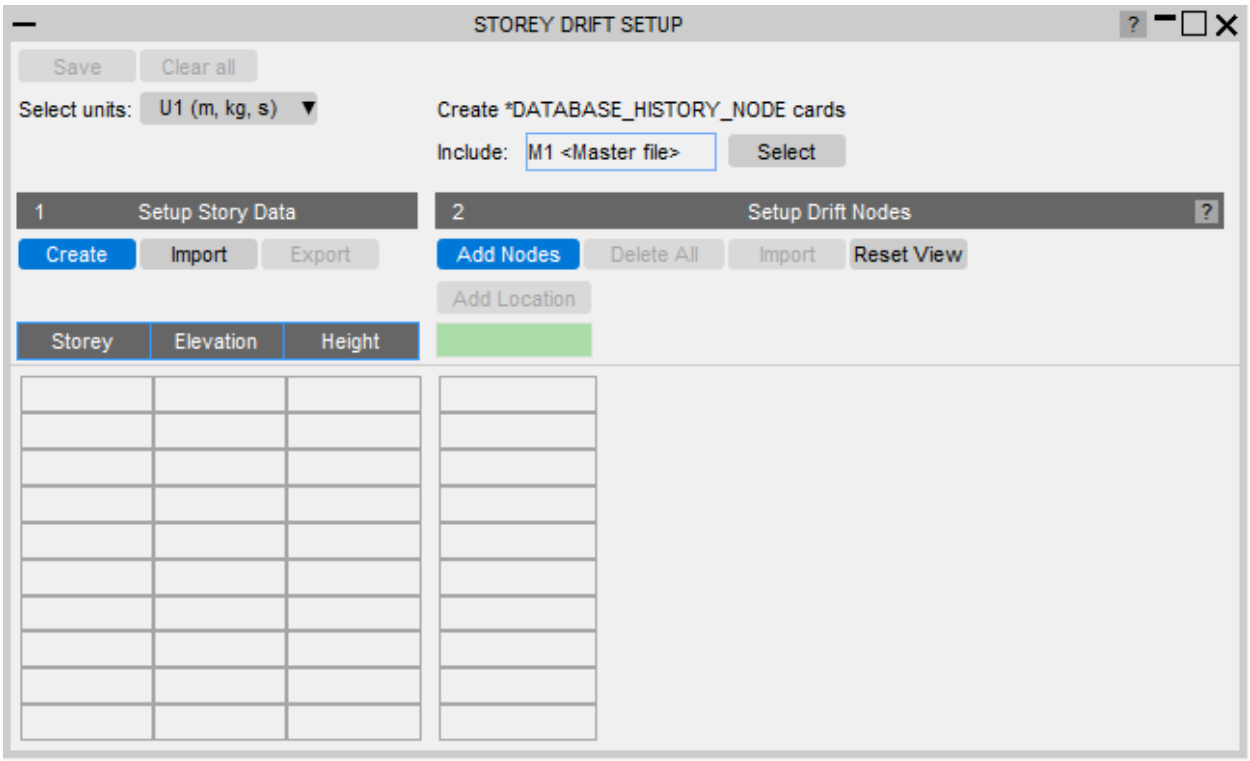
The settings available to you are as follows:

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

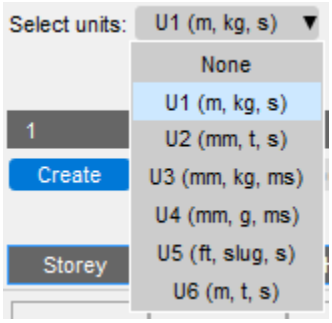
Any modifications made on this settings panel will not be automatically saved to the Workflow file. Click **Save** to write these settings to the Workflow file. You may also revert back to default settings by clicking **Reset**, which will simultaneously update these settings on the Workflow file.

Storey Drift Setup

When the tool is launched in PRIMER, a window appears for you to set up the drift definitions you wish to process:



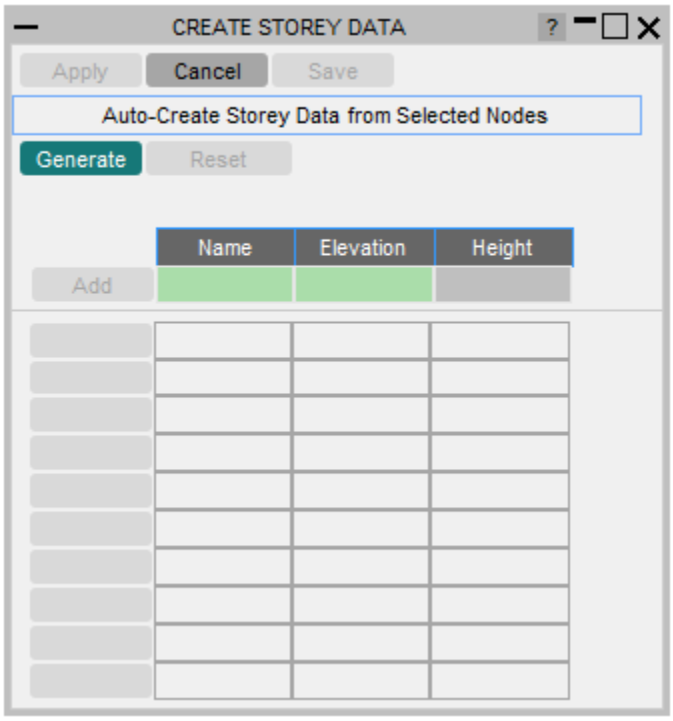
First, you need to choose the appropriate unit system from the dropdown menu:



Defining Storey Data

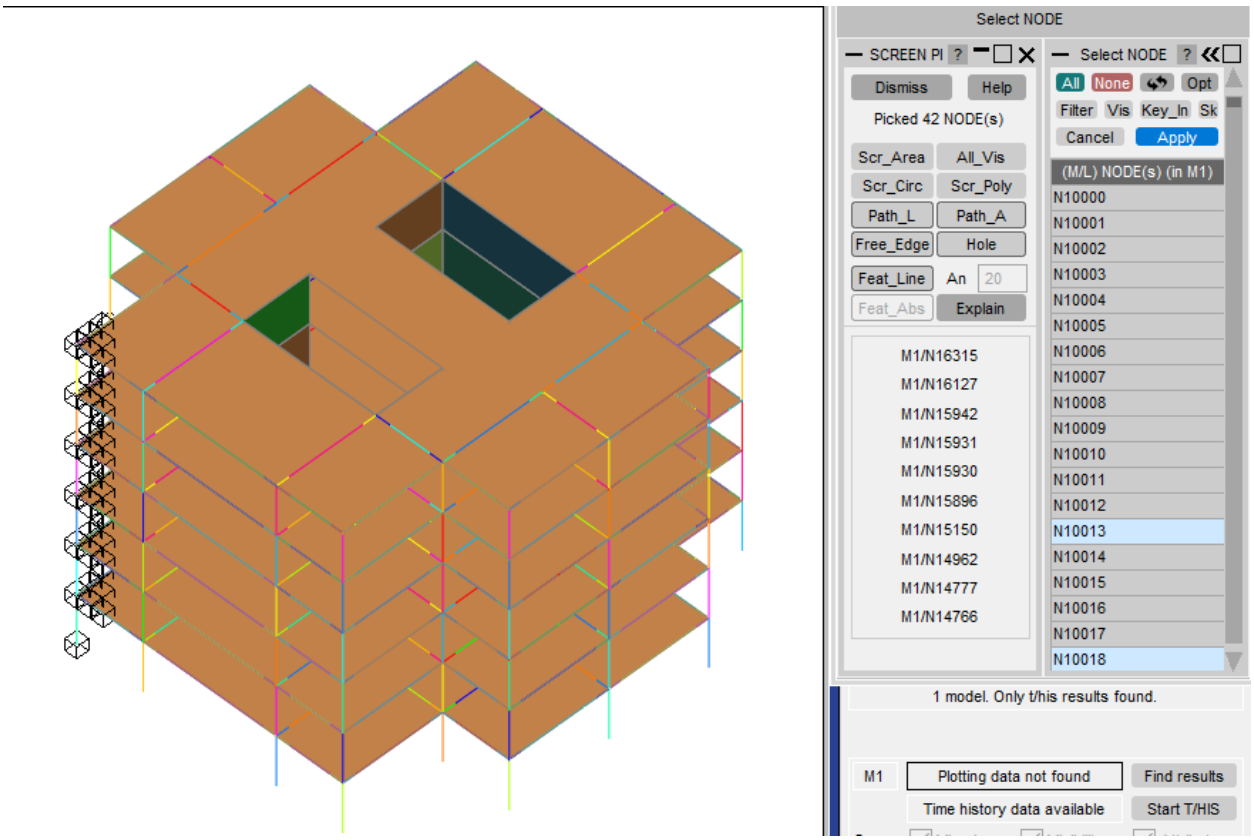
You can define the storey data for the structure either by clicking the **Create** button or the **Import** button under the Setup Storey Data section. **Import** allows you to

import previously saved storey definitions (e.g. those created for the Storey Force workflow). When you click **Create**, the Storey Data window appears:



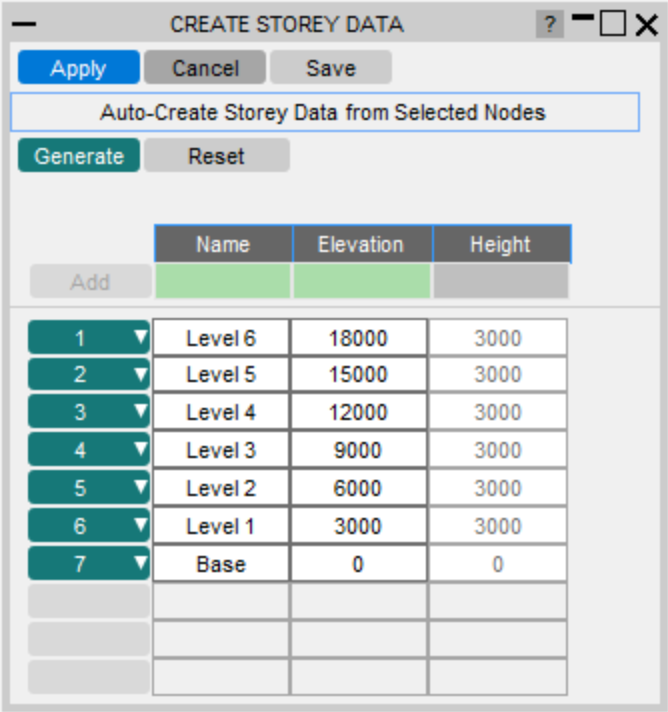
Each storey can be defined manually by populating **Name** and **Elevation** textboxes and then clicking the **Add** button which will be activated if the inputs are valid.

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.

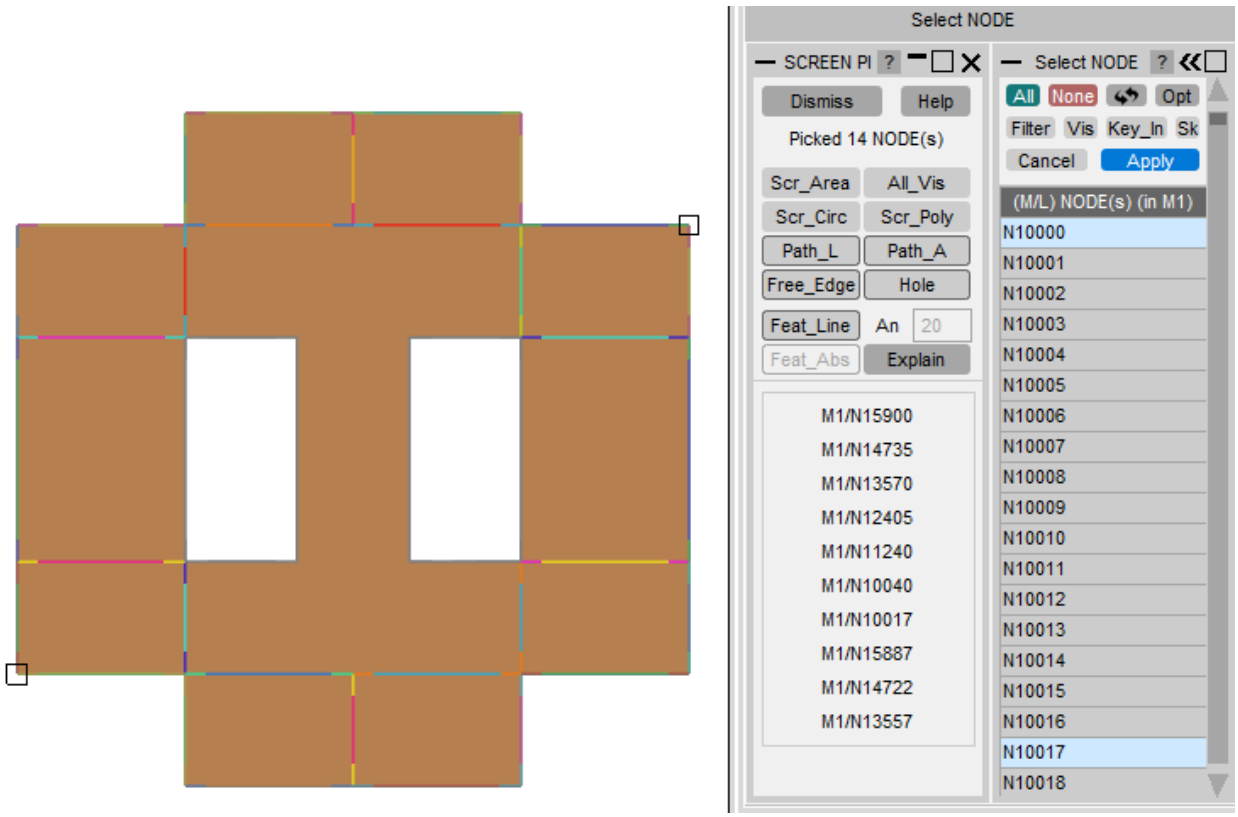


Defining Drift Locations

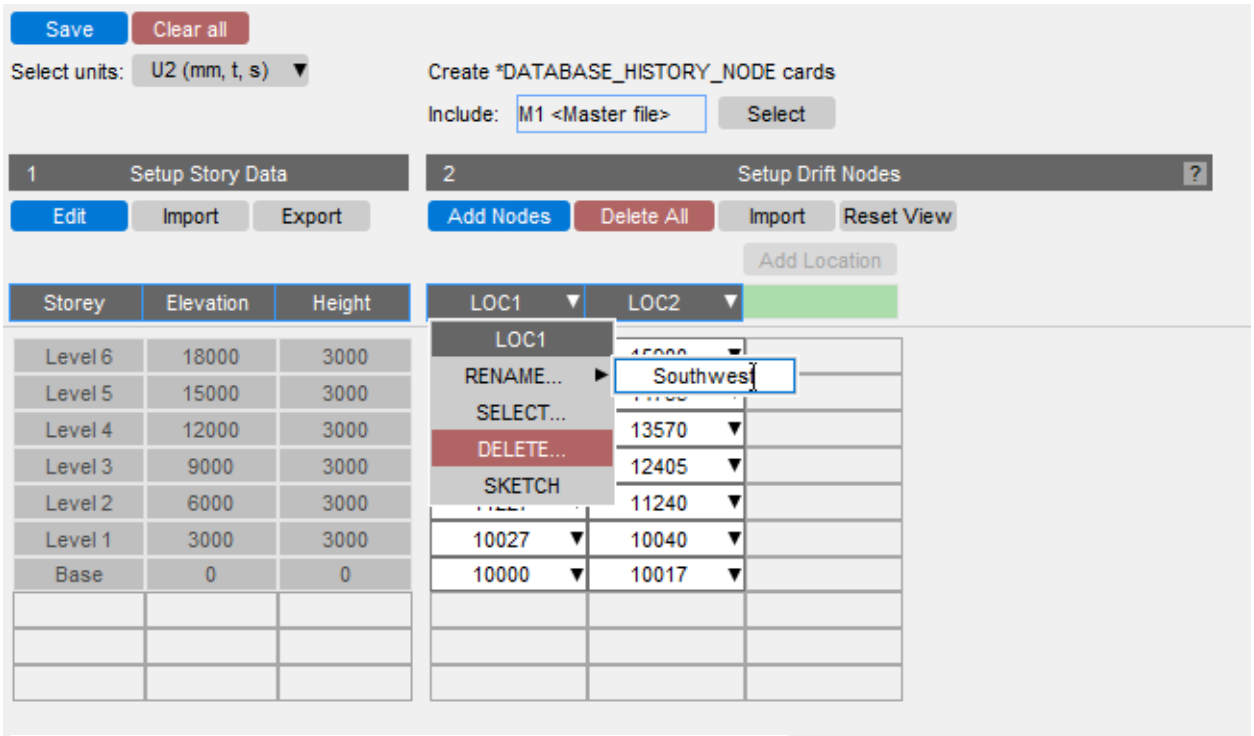
There are two ways to define drift locations.

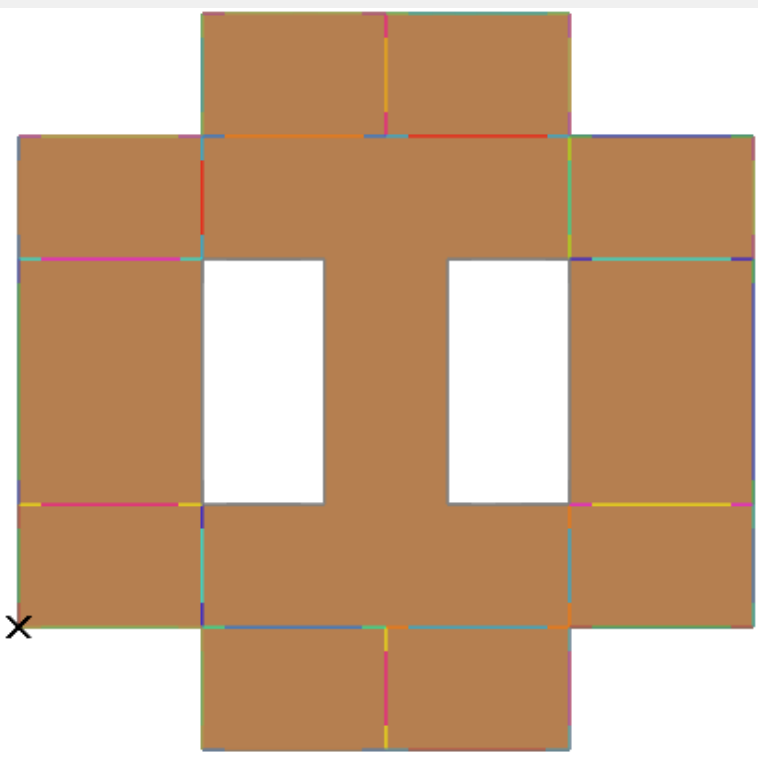
1. Create several at once using **Add Nodes**.
2. Create one at a time by defining the drift label in the column header text box and then clicking **Add Location**.

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:

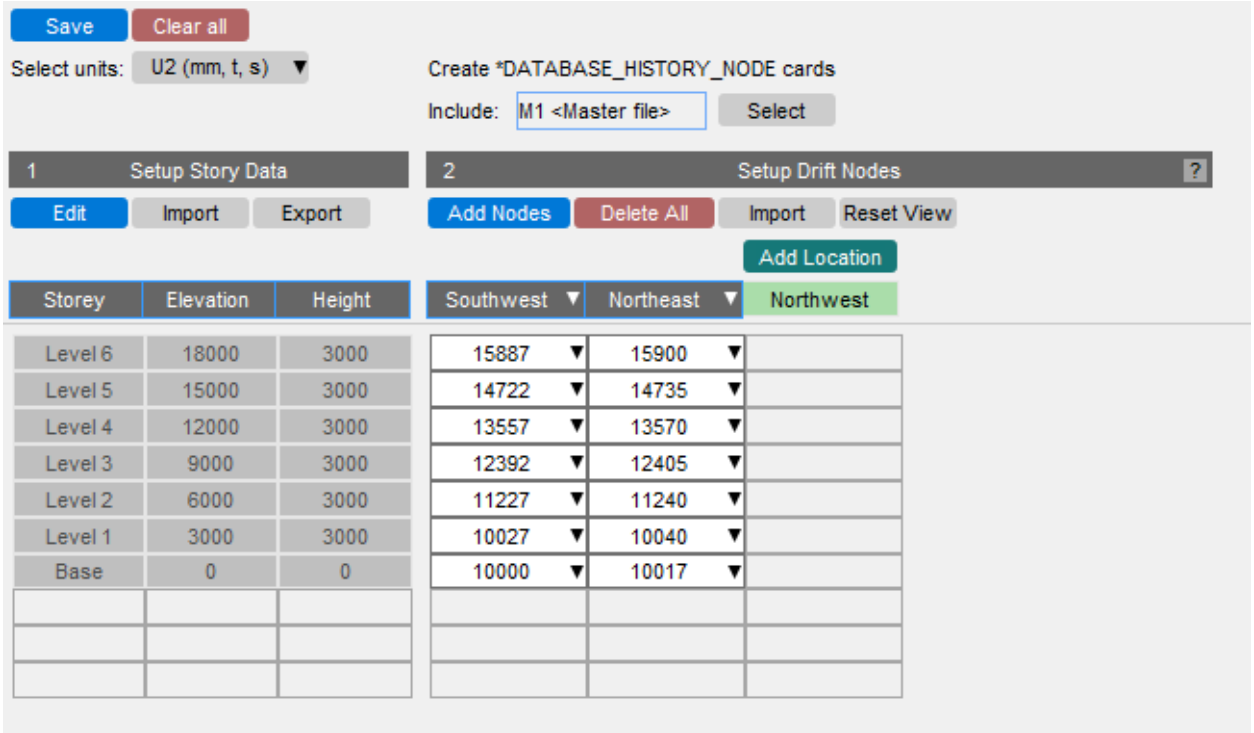


The drifts will be assigned with default labels. By right-clicking the drift header, you can rename the drift with a more informative label, as shown below. You can click **Sketch** in this popup menu to locate the drift nodes in the model, helping you to define an appropriate drift label. You can also redefine new drift nodes for an existing drift via **Select**, and even delete a current drift via **Delete**.



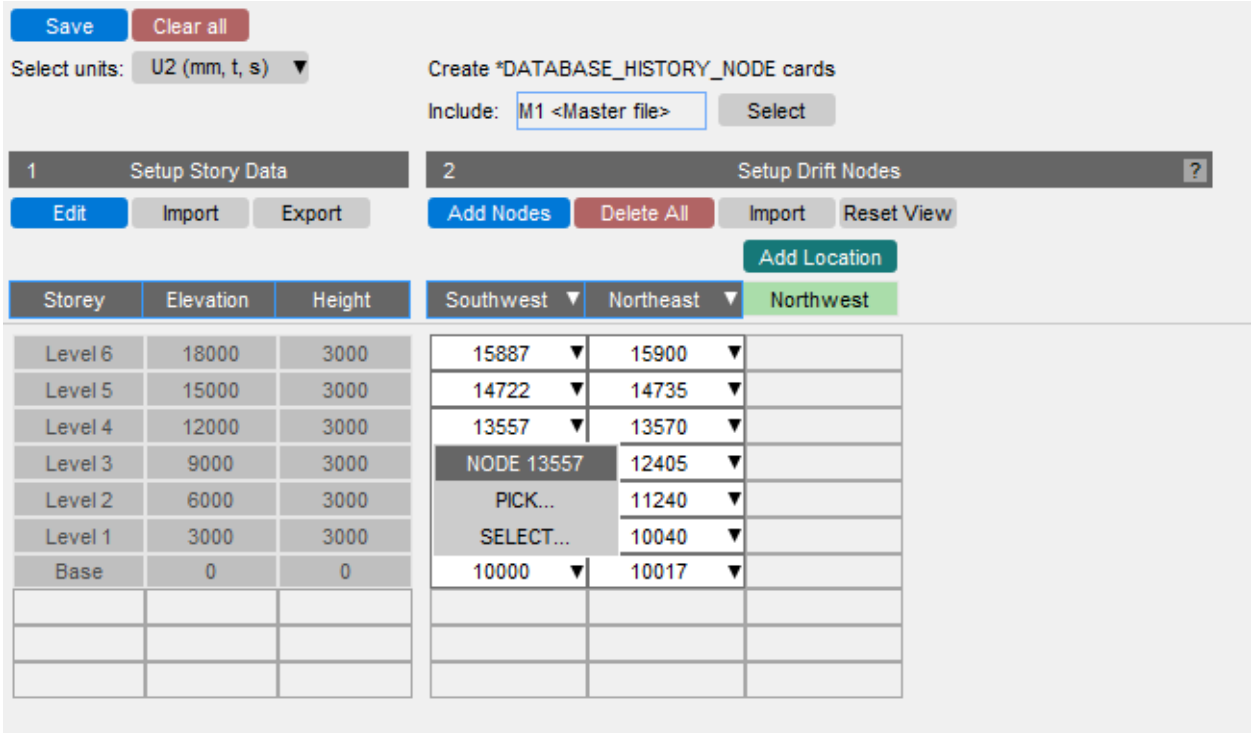


To add drift locations individually, define the drift label in the column header text box and click **Add Location**. A new blank drift column will be added to the table. You can then add nodes by right-clicking the drift header and then clicking **Select**.



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.

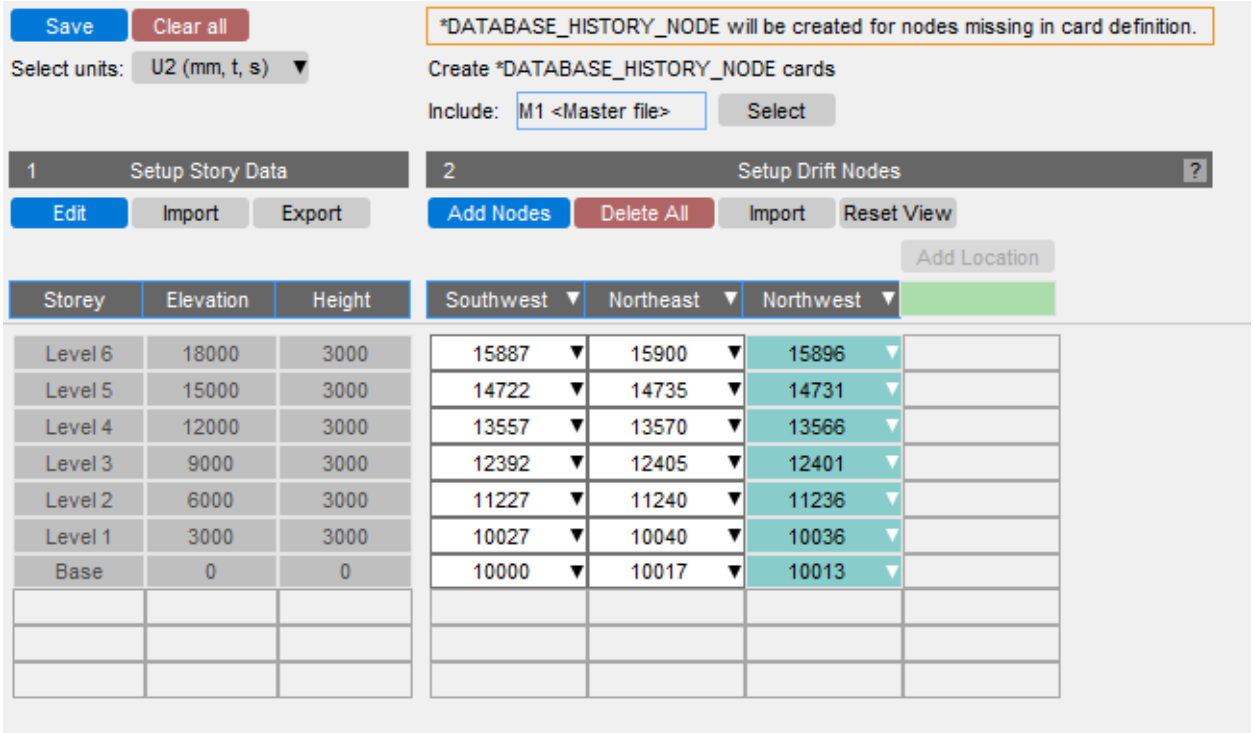
To delete a node for a particular storey in the drift, just delete the contents of the cell in the table.



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.

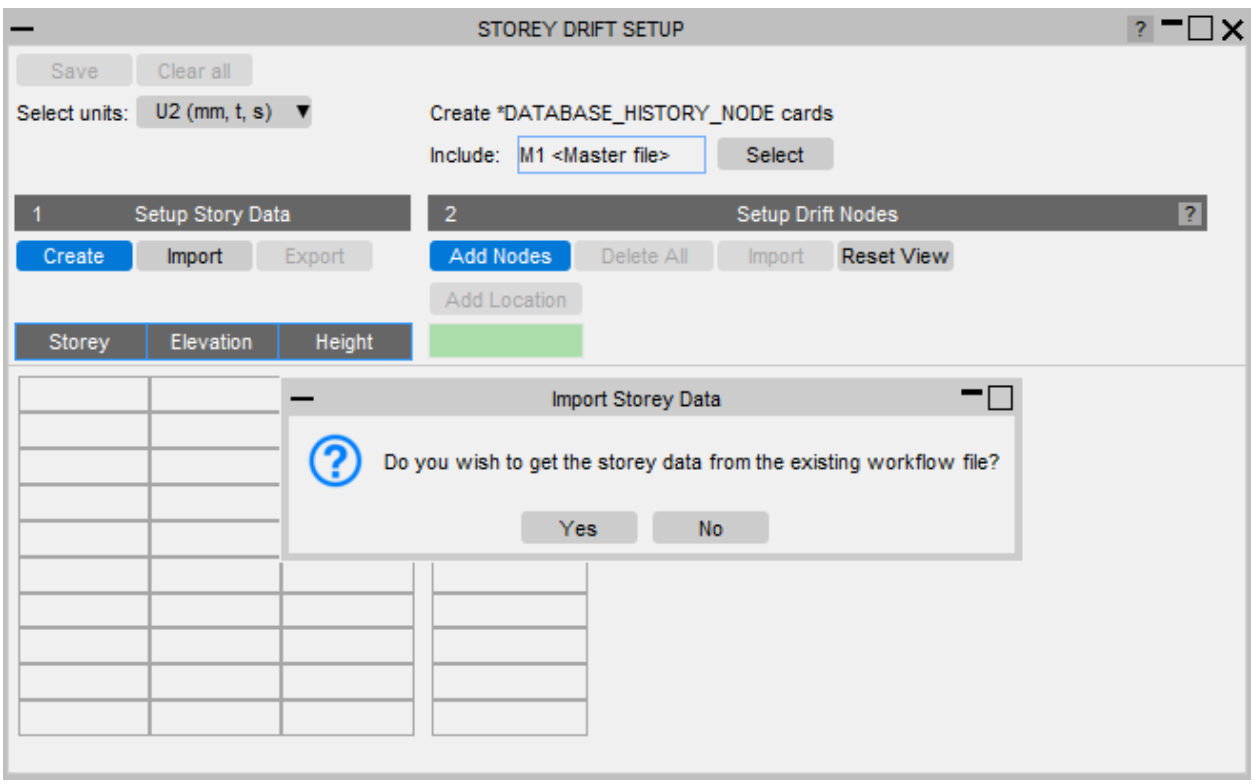
Importing existing Workflow Data

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.

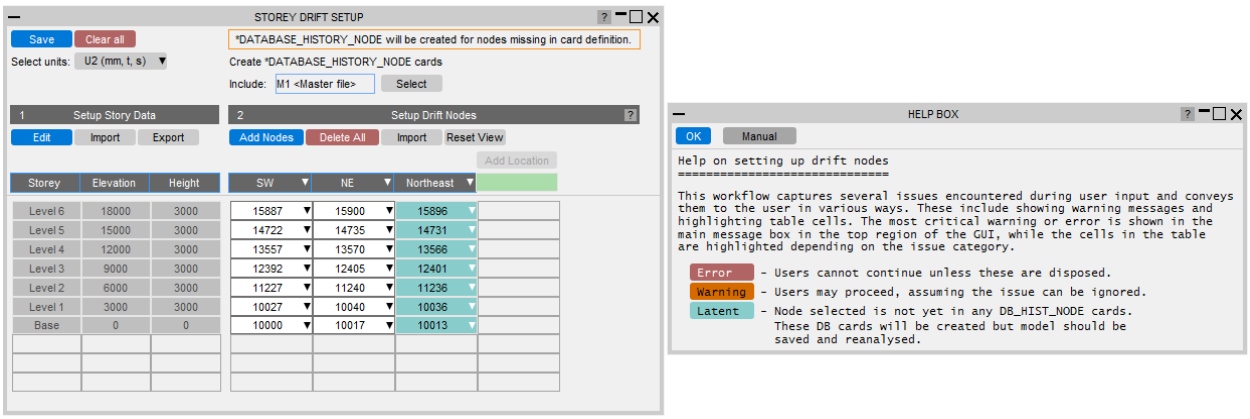


Dealing with input Errors/Warnings

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.

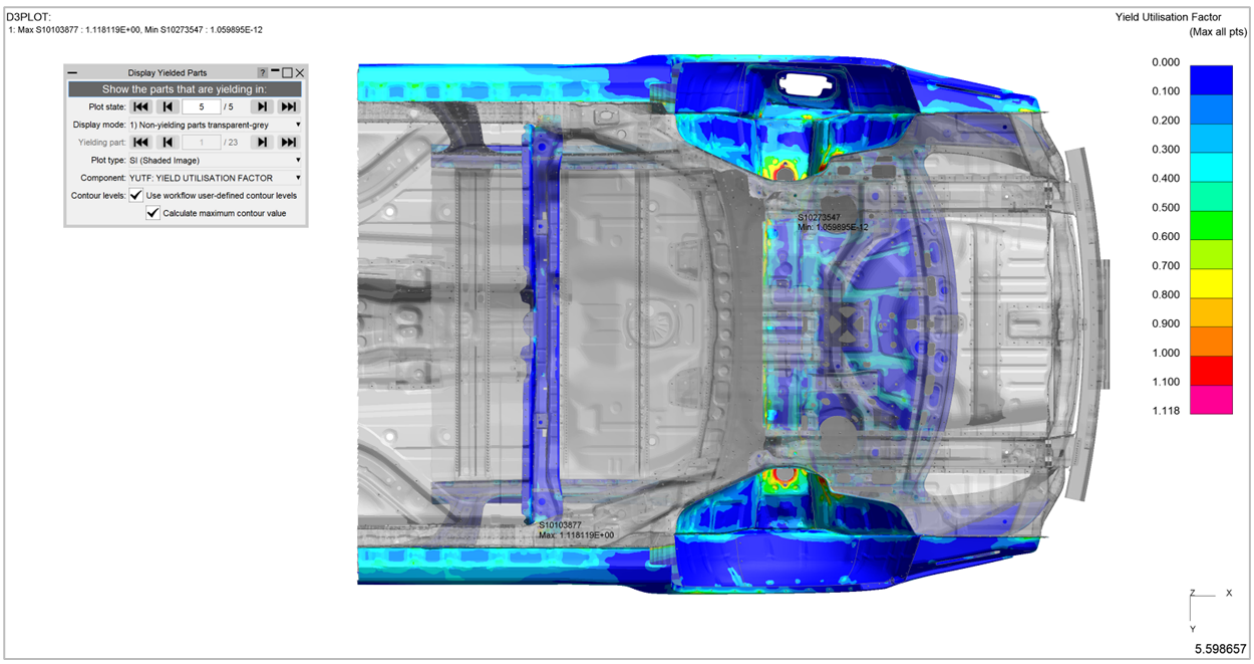


3.8. Strength Check

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%).



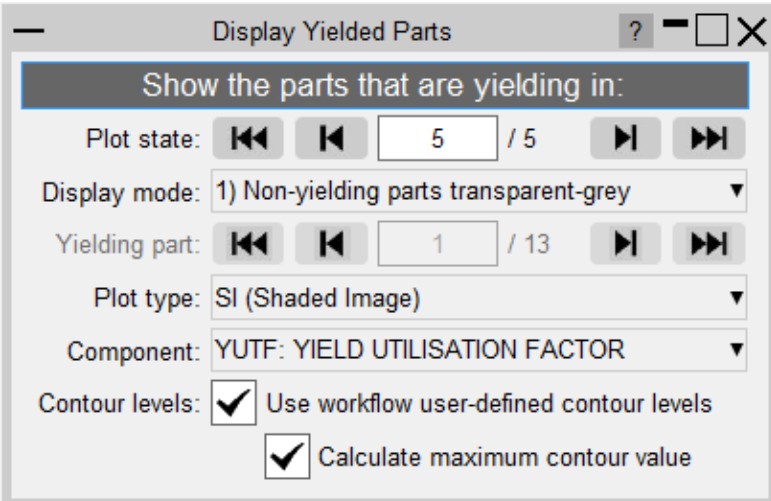
Setup in PRIMER

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

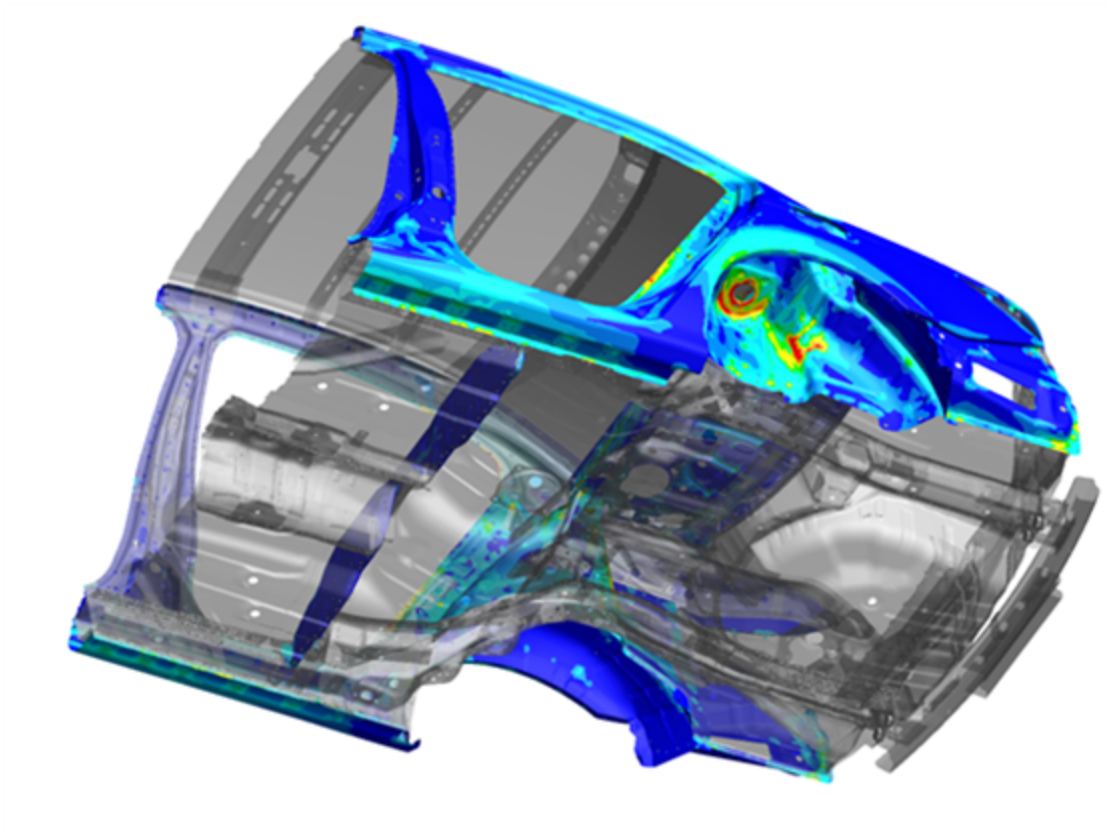
You can display the intrusion contour plot at any state. Use the controls in the menu to change plot state, rather than D3PLOT's main controls.

Display mode

This tool has three display modes:

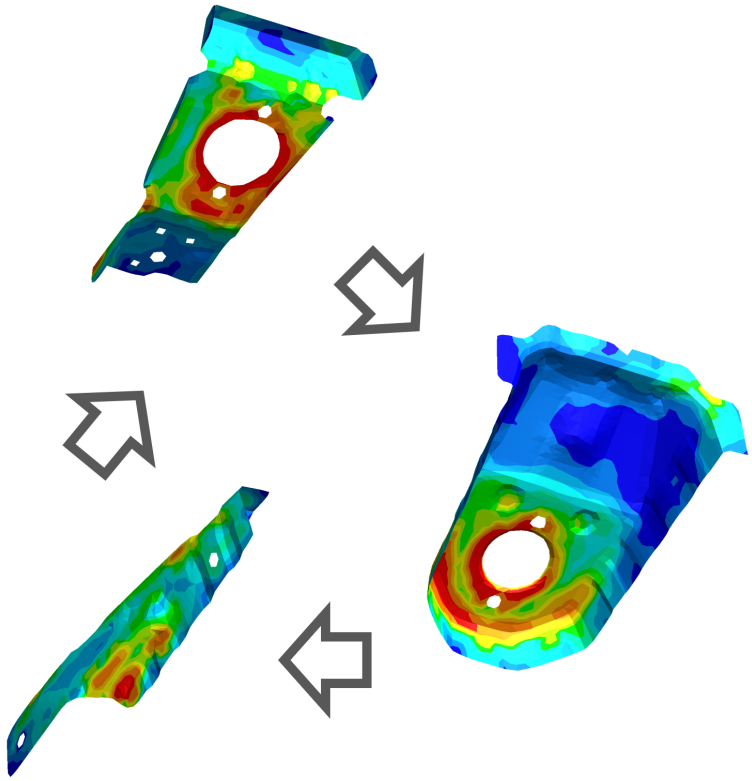
1. Non-yielding parts transparent-grey (default)

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:



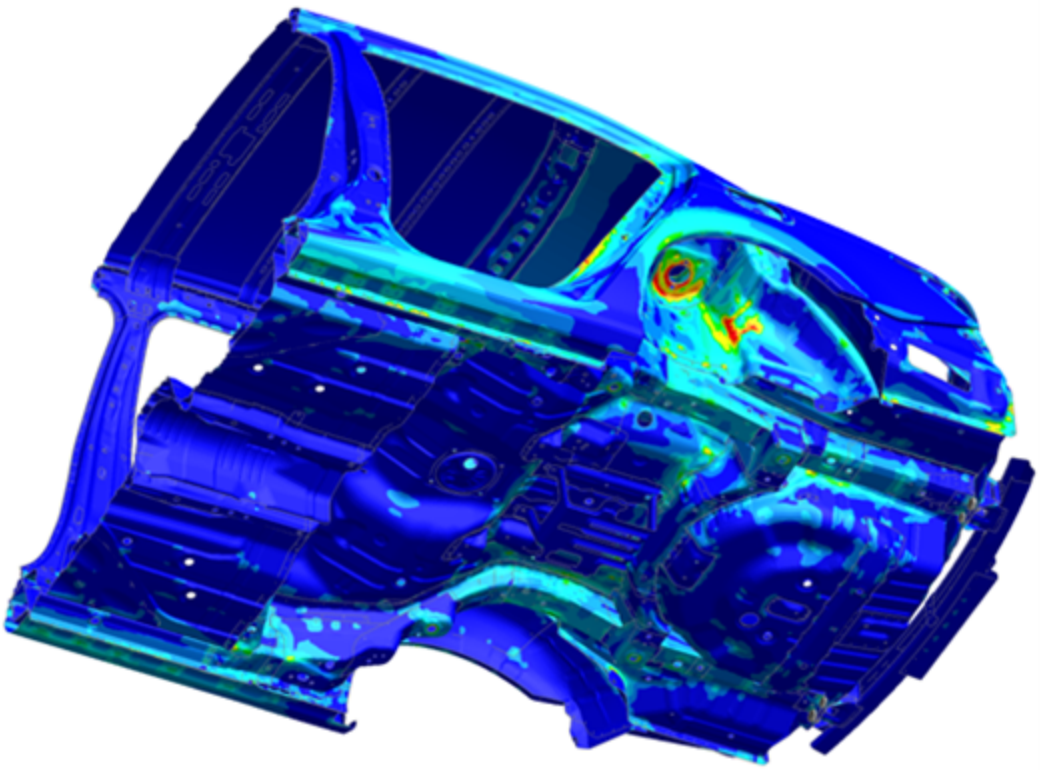
2. Cycle through yielding parts

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.

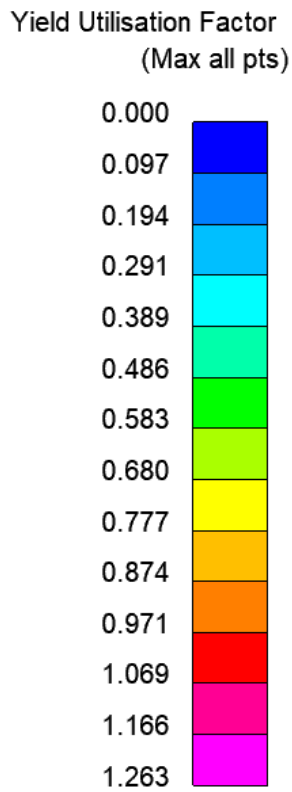
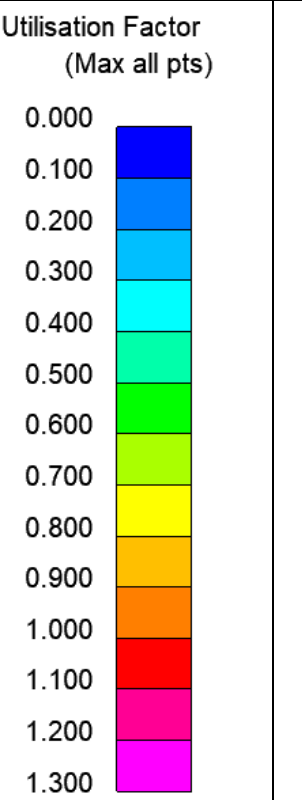
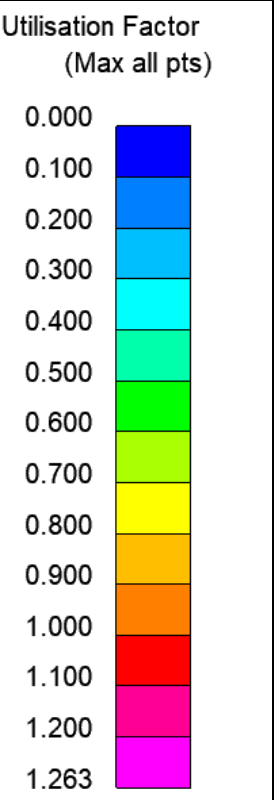
Contour Levels

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.

3.9. Virtual Testing

What is Virtual Testing?

Historically, vehicle crashworthiness regulations and NCAP safety ratings have been based on physical testing alone. This is changing with the advent of virtual testing crashworthiness protocols. Virtual testing protocols aim to improve overall vehicle safety by testing the sensitivity of vehicle designs to a wider range of conditions, while minimising the cost of physical testing.

In virtual testing, a simulation model is validated against a physical test and then – if the validation is deemed acceptable – variations of the test are rated based on simulation results called “virtual loadcases”. Initially, these virtual loadcases will consider variations in impact angle or occupant seating position. In time, they may accommodate the introduction of human body models to augment the traditional use of anthropomorphic test devices. Virtual loadcases aim to improve the safety of vehicles in a greater number of representative real-world scenarios, and for a more diverse range of human physiologies.

Virtual Testing in the Oasys LS-DYNA Environment

For the last thirty years, the Oasys LS-DYNA Environment has enabled OEMs and their suppliers to develop vehicle designs that meet legal crashworthiness regulations and achieve excellent NCAP safety ratings. Automotive CAE has always been about ensuring that the vehicle performs on test day, so that it can make it to production, and beyond. Now, automotive CAE is itself part of the test.

We recognise the important role that Oasys LS-DYNA can play, so that you can excel at Virtual Testing.

Oasys 21.0 W1 (Workflows Update 1)

At Oasys Ltd., we are working on software features to support the upcoming Virtual Testing protocols. The first protocol to be introduced is the Euro NCAP Virtual Far Side Simulation & Assessment Protocol, with C-NCAP and others to follow soon.

Oasys 21.0 W1 contains a set of integrated and complementary Workflow tools to power your Virtual Testing CAE workflows:

1. **Automotive Assessments**
   The Automotive Assessments Workflow now includes support for the Euro NCAP Virtual Far Side Protocol, including the injury metrics that feed into Validation Criterion 2. Automation available via the Euro NCAP Virtual Far Side 2024 VC2 (Assessment Criteria) REPORTER template.

2. **LS-DYNA to ISO-MME**
   A new tool to convert LS-DYNA results into the ISO-MME format as specified by the Euro NCAP Virtual Far Side Protocol, ready to upload to the VTC Server. The added REPORTER automation enables you to generate ISO-MME output after each LS-DYNA job.

3. **SimVT**
   A new, powerful tool for correlating simulation versus test curves, or indeed any combination of: LS-DYNA models, ISO-MME data, and CSV data. Supports the ISO/TS 18571:2024 rating method and calculates the ISO Scores used in Validation Criterion 1 of the Euro NCAP Virtual Far Side Protocol. Enables detailed analysis of correlation results, so you can improve the validity of your CAE models. Automation available via the Euro NCAP Virtual Far Side 2024 VC1 (ISO Scores) REPORTER template.

4. **Euro NCAP VTC Quality Criteria**
   A new, convenient tool for assessing the quality criteria specified in section 6.1 of the Euro NCAP Virtual Far Side Protocol. The added REPORTER automation enables you to check the validity of your CAE models after each LS-DYNA job.

5. **Euro NCAP VTC Videos**
   A new, convenient tool for creating the videos specified in section 5.2.1 of the Euro NCAP Virtual Far Side Protocol. The tool assists with camera positioning and can be automated to generate videos with your preferred settings.

Further Development

Development does not end with Oasys 21.0 W1. We continue to work on enhancements to the Virtual Testing Workflows, including:

- Improved usability and automation features
- Support for the Far Side Virtual Testing Protocol published with the C-NCAP Management Regulation 2024 Edition
- Support for other upcoming protocols

If you are interested in early access to the latest features, or if you would like more information or a demonstration of Virtual Testing with the Oasys LS-DYNA Environment, please contact us.

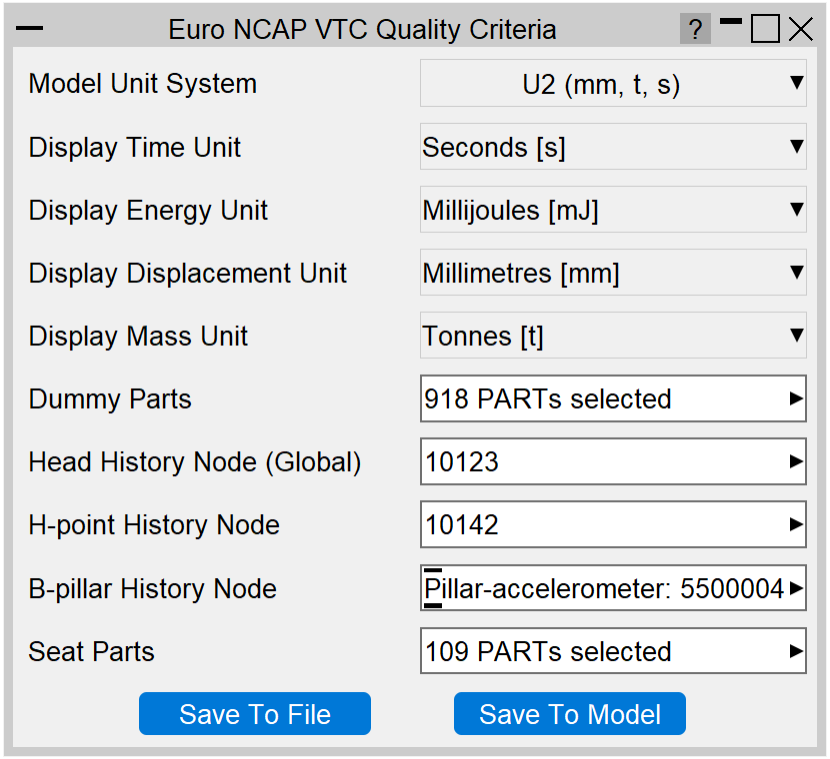
3.9.1. Euro NCAP VTC Quality Criteria

Tools → Workflows → Euro NCAP VTC Quality Criteria

The Euro NCAP VTC Quality Criteria workflow tool allows you to perform the quality checks outlined in Section 6.1 of the Euro NCAP Virtual Far Side Simulation & Assessment Protocol.

How to use the Workflow Tool in PRIMER

When this tool is initially launched, the tool will attempt to collect all previously saved data from the Automotive Assessments Workflow.
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.

Dummy Parts

Select the include file containing the WorldSID Dummy by pressing the right arrow for multiple selection and picking options or manually typing in the textbox.

Head History Node (Global)

Select the DATABASE_HISTORY_NODE matching the Global Head Node of the WorldSID Dummy by pressing the right arrow for multiple selection and picking options or manually typing in the textbox. The default is 10123.

H-point History Node

Select the DATABASE_HISTORY_NODE matching the H-point Node of the WorldSID Dummy by pressing the right arrow for multiple selection and picking options or manually typing in the textbox. The default is 10501.

B-pillar History Node

Select the DATABASE_HISTORY_NODE matching the B-pillar Node of the Vehicle by pressing the right arrow for multiple selection and picking options or manually typing in the textbox.

Seat Parts

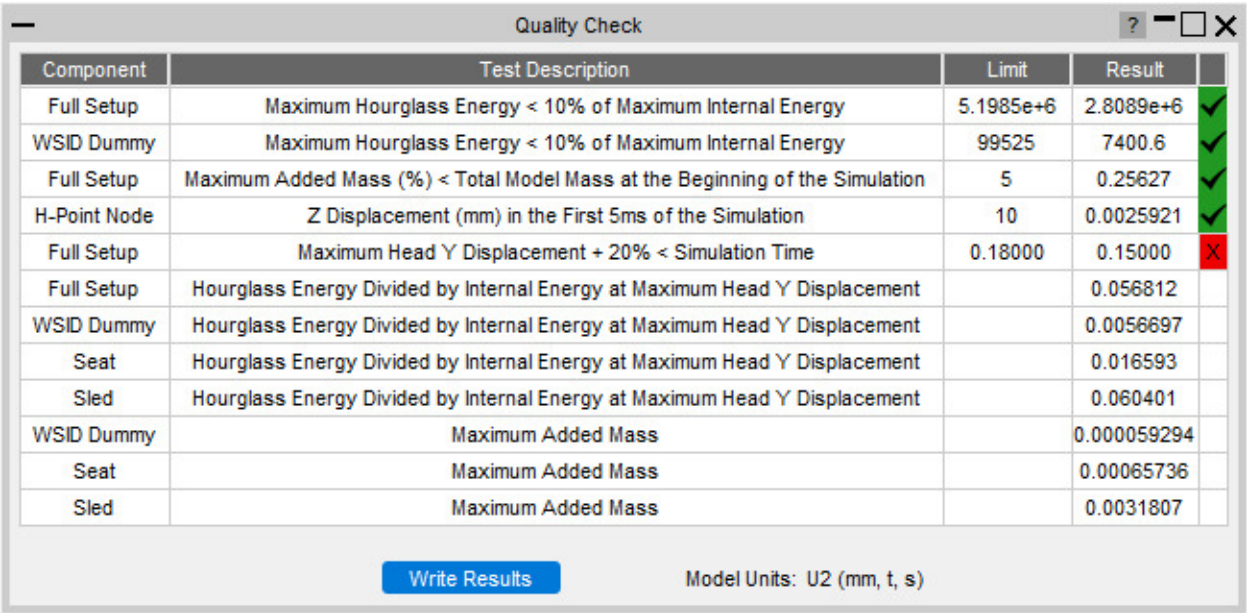
Select the include file containing the Seat of the model by pressing the right arrow for multiple selection and picking options or manually typing in the textbox.

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.



Quality Check

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	✗
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

Model Units: U2 (mm, t, s)

Write Results

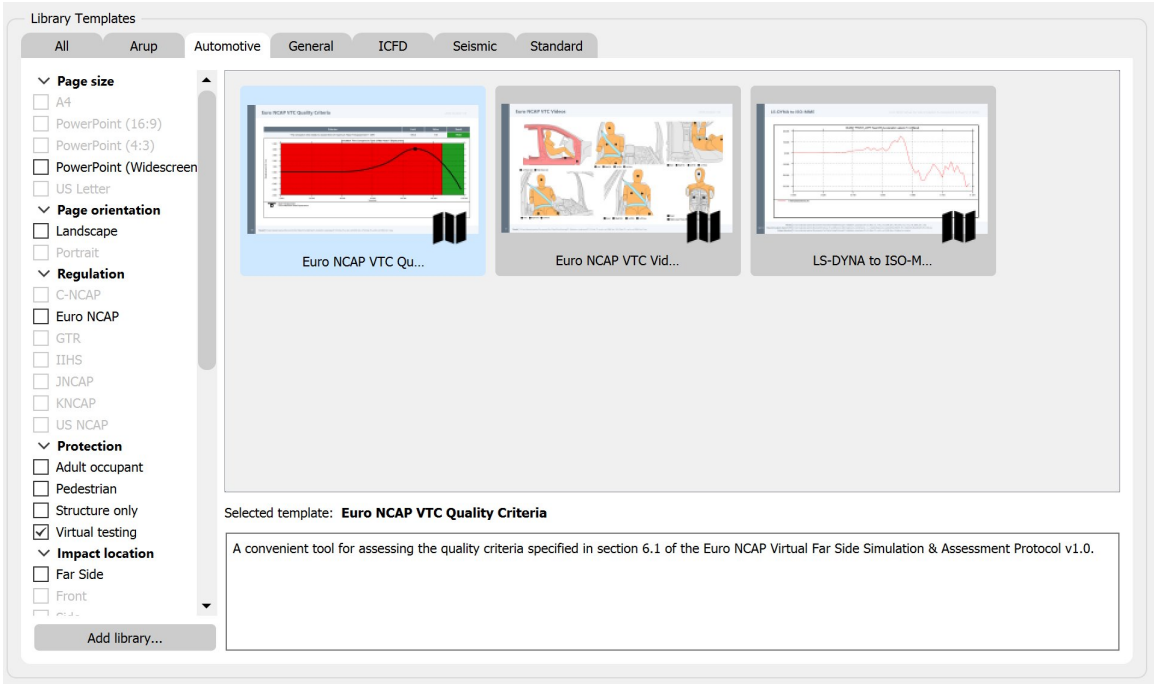
Writes the results out as displayed in the table in CSV format.

Model Unit System

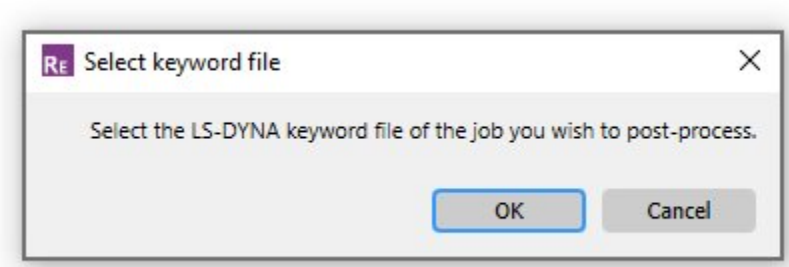
Displays the unit system that has been selected in PRIMER for this model.

How to use the Workflow Tool in REPORTER

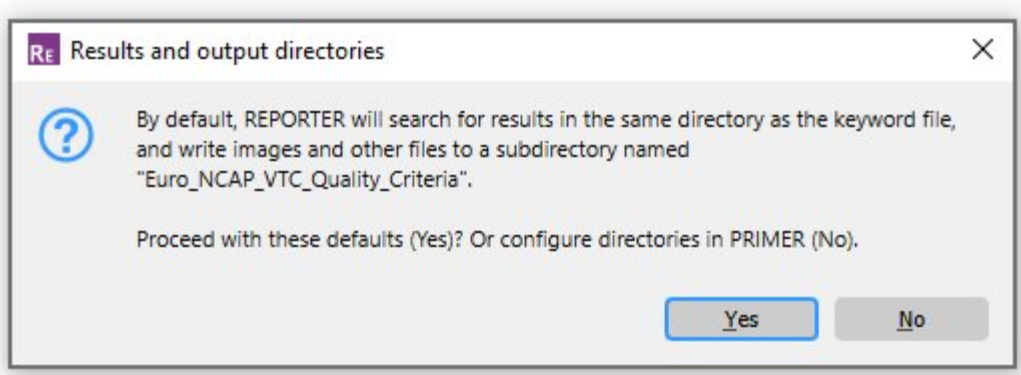
1. Within the Automotive tab in REPORTER, select the Euro NCAP VTC Quality Criteria template. It can be found by filtering for 'Virtual Testing'.



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

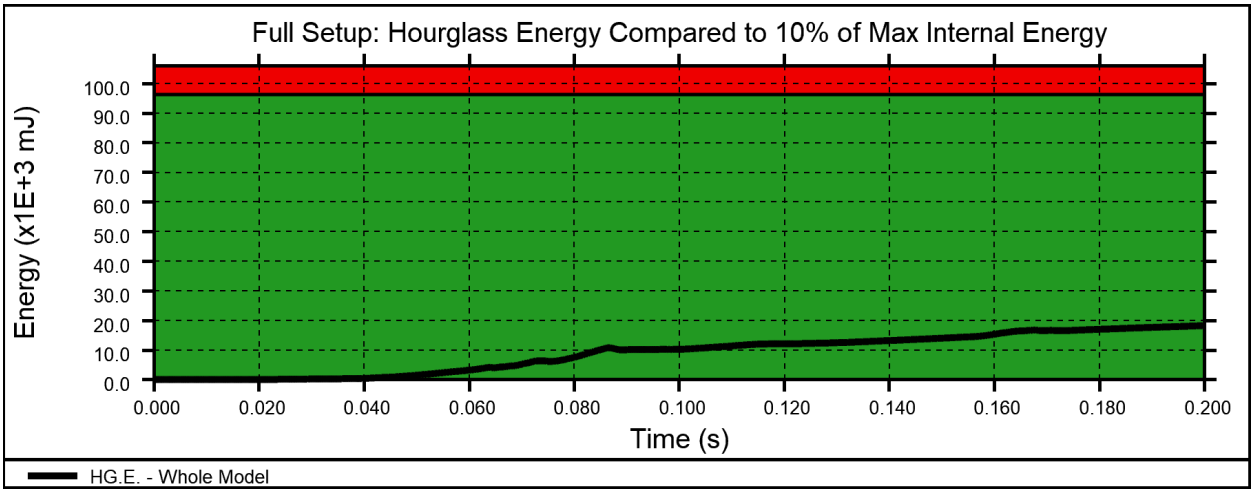
Model | C:/Users/harry.graham/Documents/Work/QualityCheck/far_side_Pole_2022.key

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.

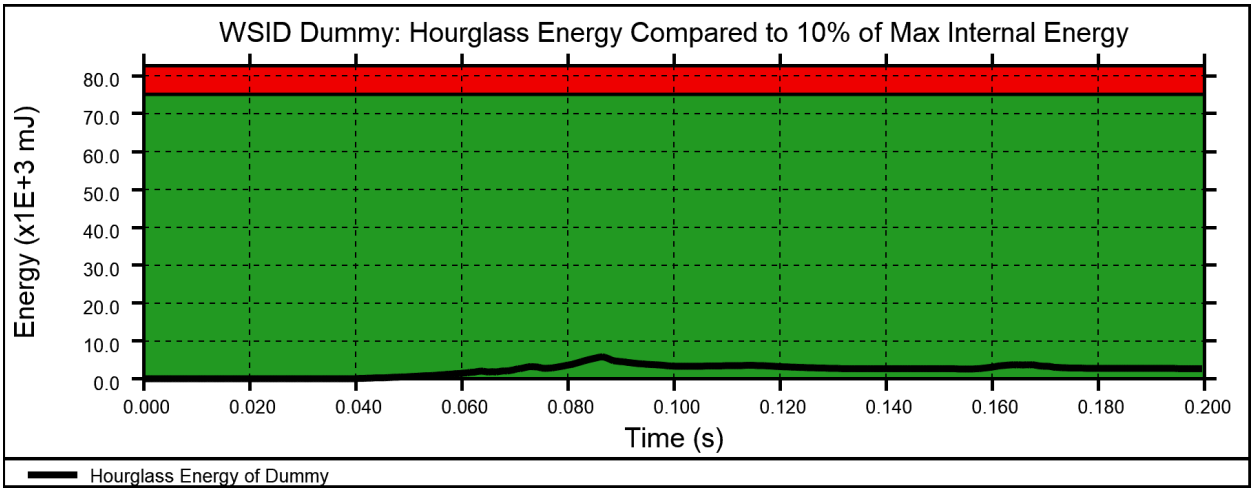
In blue colour, the internal energy of the full setup is displayed (only if the test is a fail).
The datum line is drawn at 10% of the maximum internal energy.
In foreground colour, the hourglass energy of the full setup is displayed.
For this check to pass, the peak of the hourglass energy curve must 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

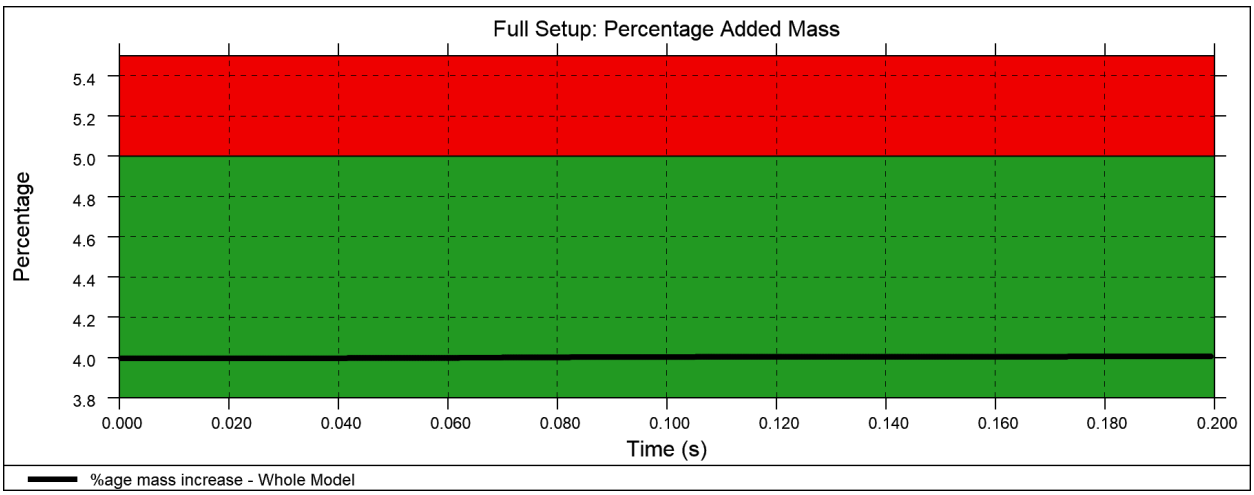
In blue colour, the internal energy of the WorldSID Dummy is displayed (only if the test is a fail).
The datum line is drawn at 10% of the maximum internal energy.
In foreground colour, the hourglass energy of the WorldSID Dummy is displayed.
For this check to pass, the peak of the hourglass energy curve 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.

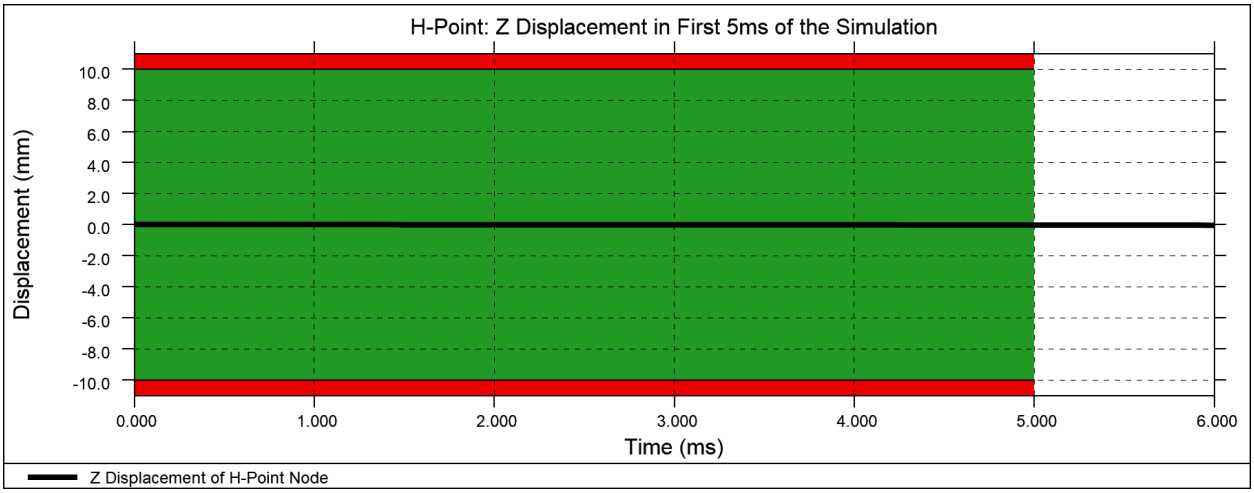


H-Point Node: Z Displacement (mm) in the First 5ms of the Simulation

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.

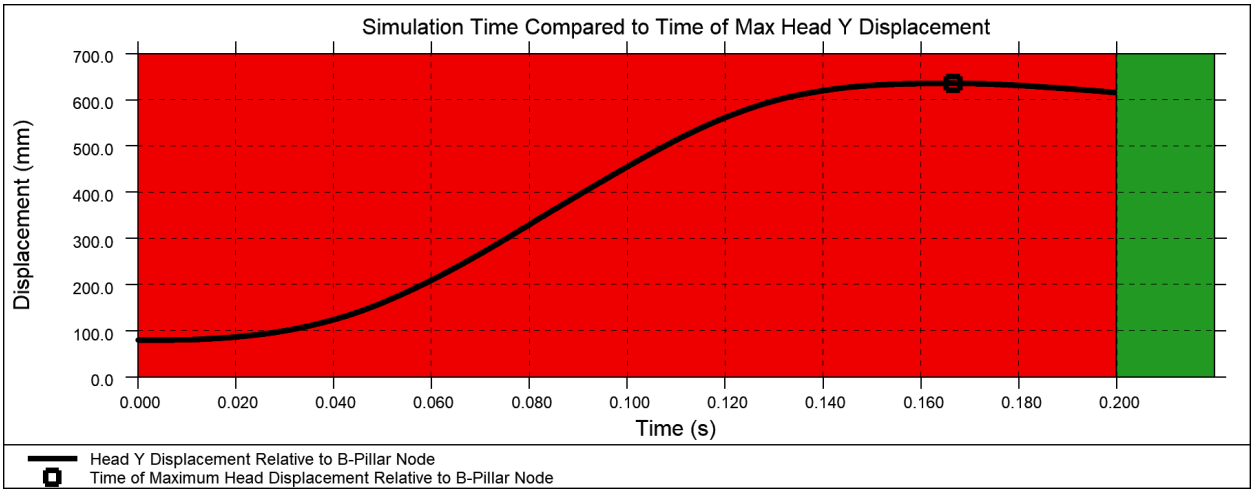
For this check to pass, the peak of the Z displacement curve must be within the green zone within the first 5ms.
The limit and result are displayed in the table.



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.

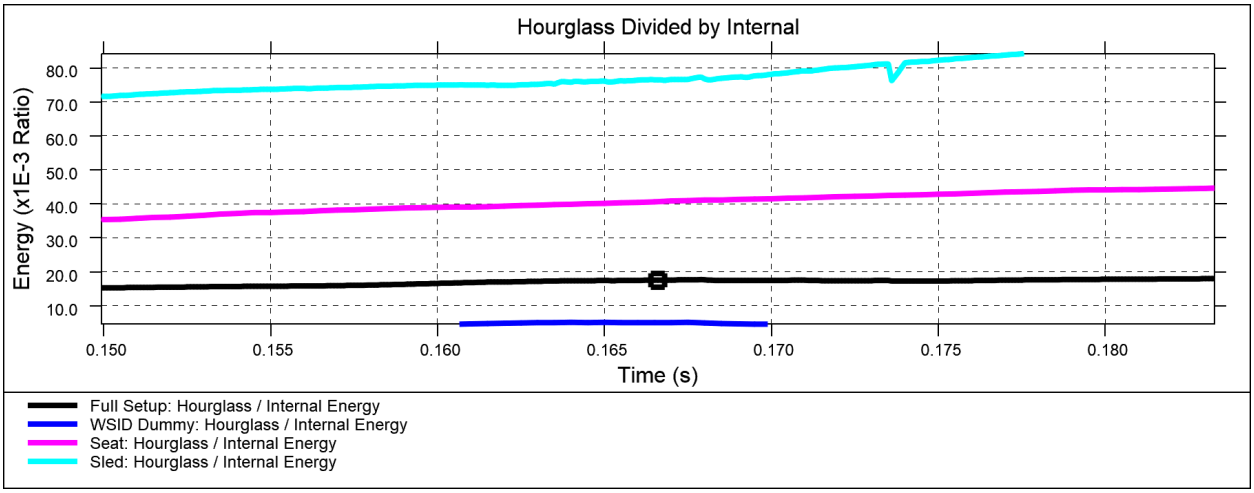
In foreground colour, the Head Y Displacement is displayed.
The datum line is drawn at Maximum Head Y Displacement Relative to B-Pillar Node Time + 20%.
For this check to pass, the Head Y Displacement curve should finish in the green zone.
The limit and result are displayed in the table.



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.

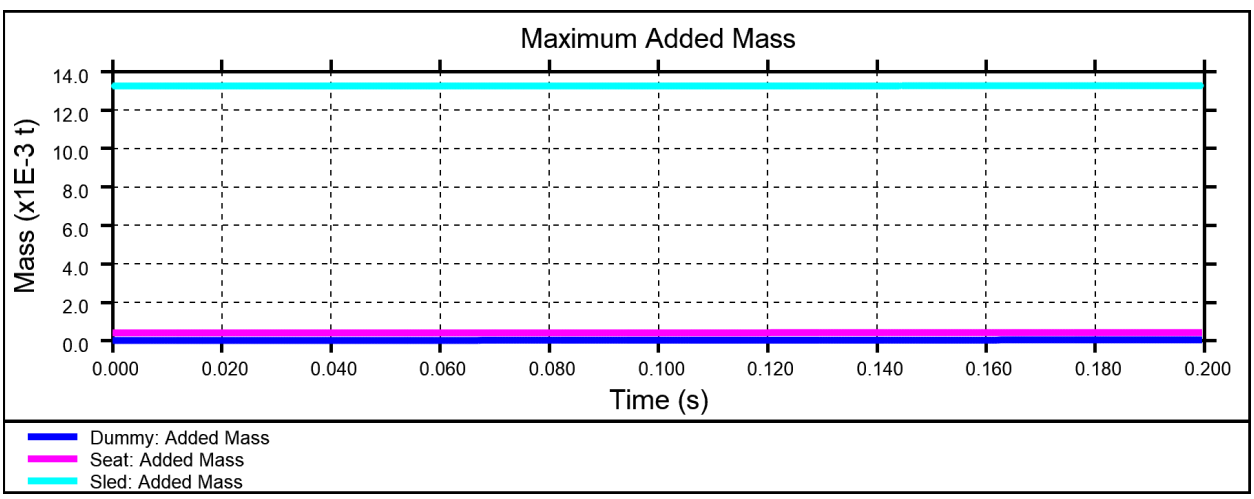
In foreground colour, the Hourglass divided by Internal Energy of the full setup is displayed.
In blue colour, the Hourglass divided by Internal Energy of the WorldSID Dummy is displayed.
In magenta colour, the Hourglass divided by Internal Energy of the Seat is displayed.
In cyan colour, the Hourglass 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.



3.9.2. Euro NCAP VTC Videos

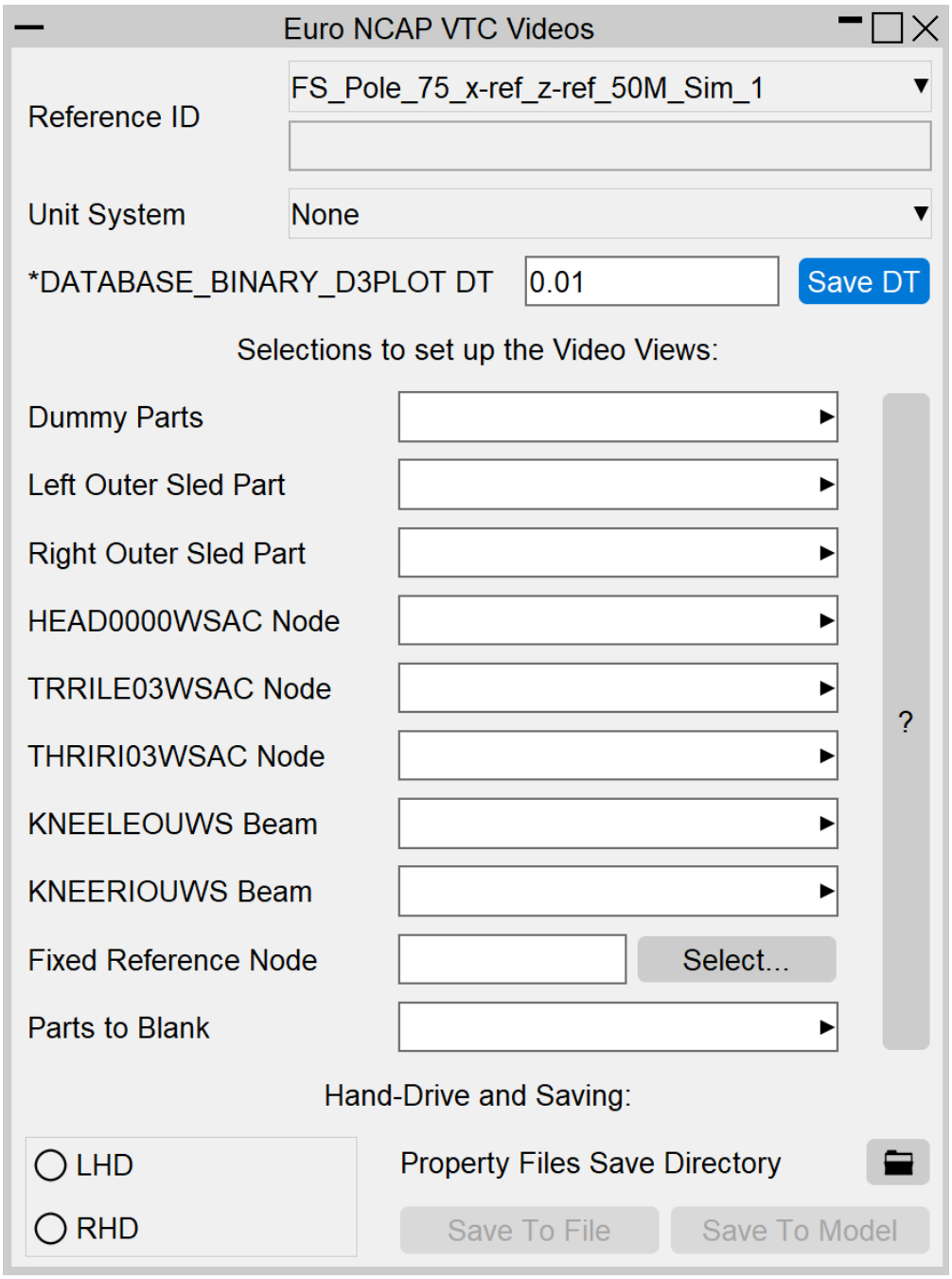
Tools → Workflows → Euro NCAP VTC Videos

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.

*DATABASE_BINARY_D3PLOT_DT

Animations need to be analysed with an output interval of 2ms (0.002s) or less. Changing the output interval will change the *DATABASE_BINARY_D3PLOT DT keyword field in the model and thus you will need to save the model and re-run in LS-DYNA to get the updated output files.

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

Press the right arrow for multiple selection or picking options or manually type in the textbox for the Dummy.

Left Outer Sled Part

Press the right arrow for multiple selection or picking options or type the Part ID into the textbox. It should contain the door frame.

Right Outer Sled Part

Press the right arrow for multiple selection or picking options or type the Part ID into the textbox. It should contain the door frame.

HEAD0000WSAC Node

Press the right arrow for multiple selection or picking options or type the *DATABASE_HISTORY_NODE for the Head Node into the textbox. For the WSID Dummy this is likely to be 10001.

TRRILE03WSAC Node

Press the right arrow for multiple selection or picking options or type the *DATABASE_HISTORY_NODE for the Left Lower Thorax Rib into the textbox. For the WSID Dummy this is likely to be 10013.

THRIRI03WSAC Node

Press the right arrow for multiple selection or picking options or type in the *DATABASE_HISTORY_NODE for the Right Lower Thorax Rib into the textbox. For the WSID Dummy this is likely to be 10023.

KNEELEOUWS Beam

Press the right arrow for multiple selection or picking options or type in the *DATABASE_HISTORY_BEAM for the Left Outboard Knee Load Cell. For the WSID Dummy this is likely to be 10012.

KNEERIOUWS Beam

Press the right arrow for multiple selection or picking options or type in the *DATABASE_HISTORY_BEAM for the Right Outboard Knee Load Cell. For the WSID Dummy this is likely to be 10013.

Fixed Reference Node

Press 'Select...' to Select or Pick a *NODE for the fixed reference node. The purpose of this is to hold the sled in position during the videos.

Parts to Blank

There might be some parts you wish to blank during all the videos, for example the windscreen. Press the right arrow for multiple selection or picking options or manually type the parts in the textbox.

LHD/RHD

Using the Radio buttons, select LHD or RHD for Left Hand Drive or Right Hand Drive Occupant.

Property Files Save Directory

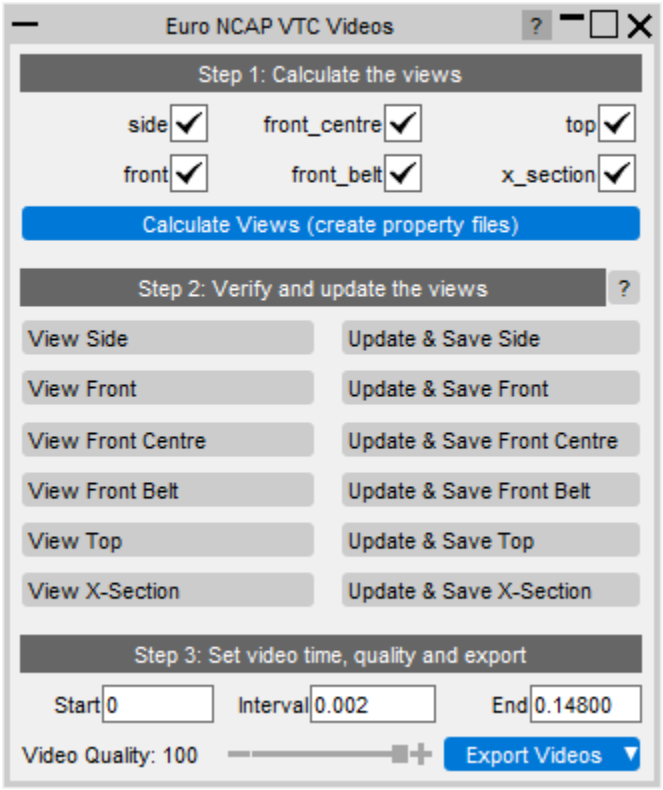
Select the Directory by pressing on the Directory icon to save the view files (Property (.prp) and Cut-Section (.cut)).

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

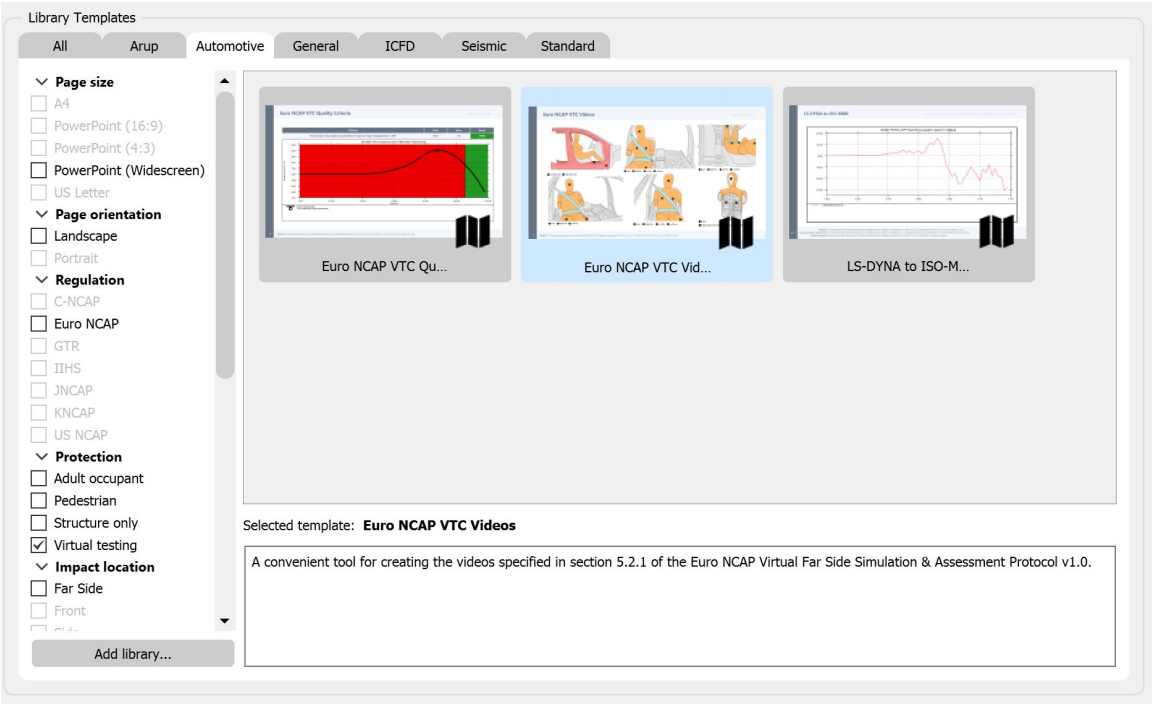
Once you have verified your views, set the start, interval and end time required for your videos:

- The default start time is set to zero
- The default interval time is set to either 2 or 0.002 dependant on your unit system as animations need to be analysed with an output interval of 2ms (0.002s) or less
- The end time needs to be set to the time of maximum head excursion x 1.2

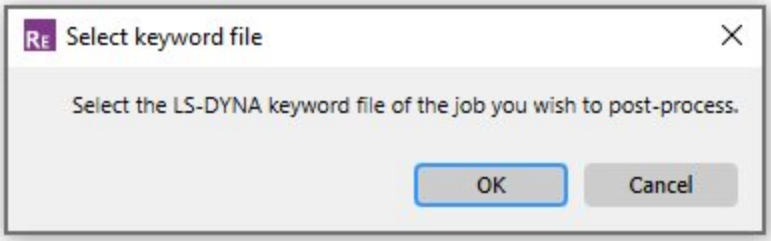
Then set the video quality using the slider between 10 and 100. According to the Euro NCAP specification, the videos should be 1-10 MB in size.
"Export Videos" will export all six videos by default to the directory you defined in PRIMER – you can change the views to be exported via the dropdown.

How to use the Workflow Tool in REPORTER

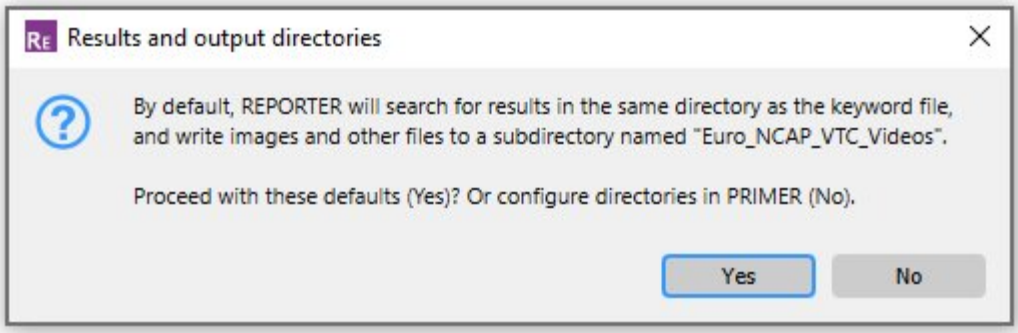
1. Within the Automotive tab in REPORTER, select the Euro NCAP VTC Videos template. It can be found by filtering for 'Virtual Testing'.



2. Once open you will be prompted to select the LS-DYNA keyword file of the job you wish to post-process.

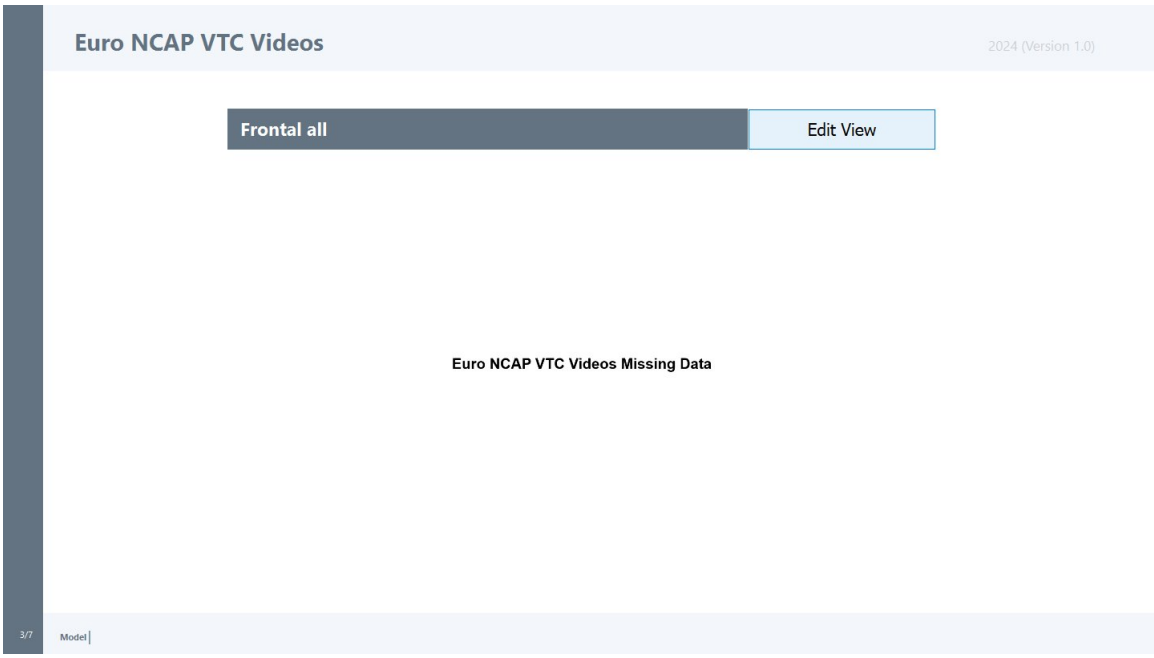


3. You will then be asked whether you want to continue with the default results and output directories or configure them in PRIMER.

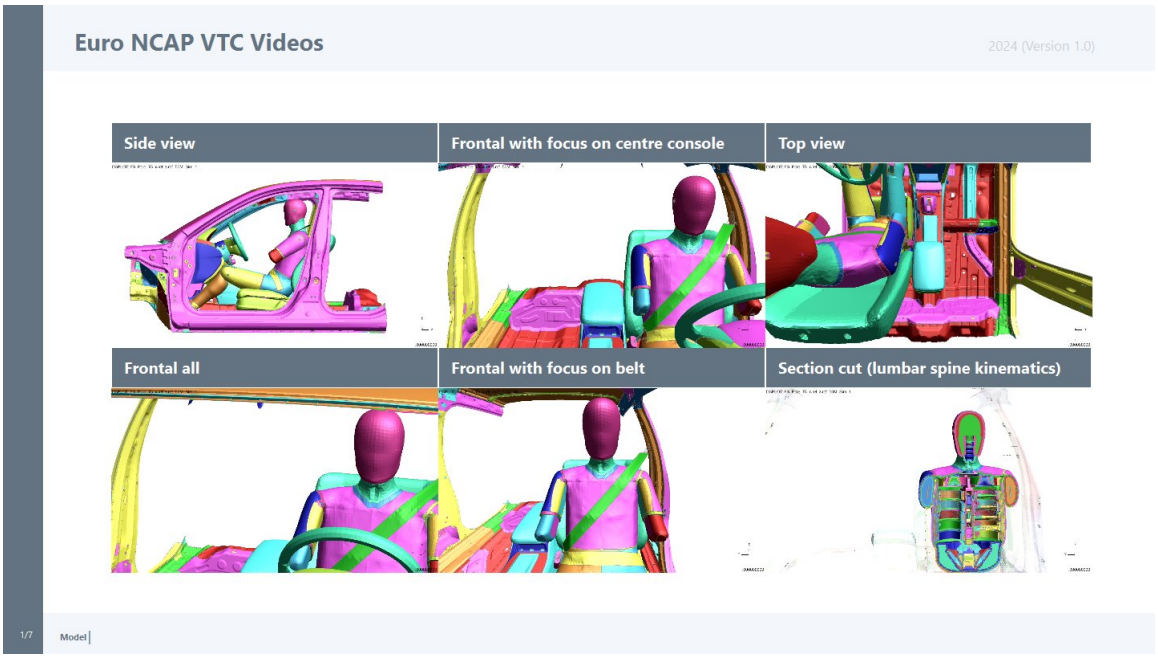


4. REPORTER will then proceed to generate the report. If there is any missing data, PRIMER will be launched for you to edit the setup. From then on, REPORTER will attempt to generate the report with the data it has available.

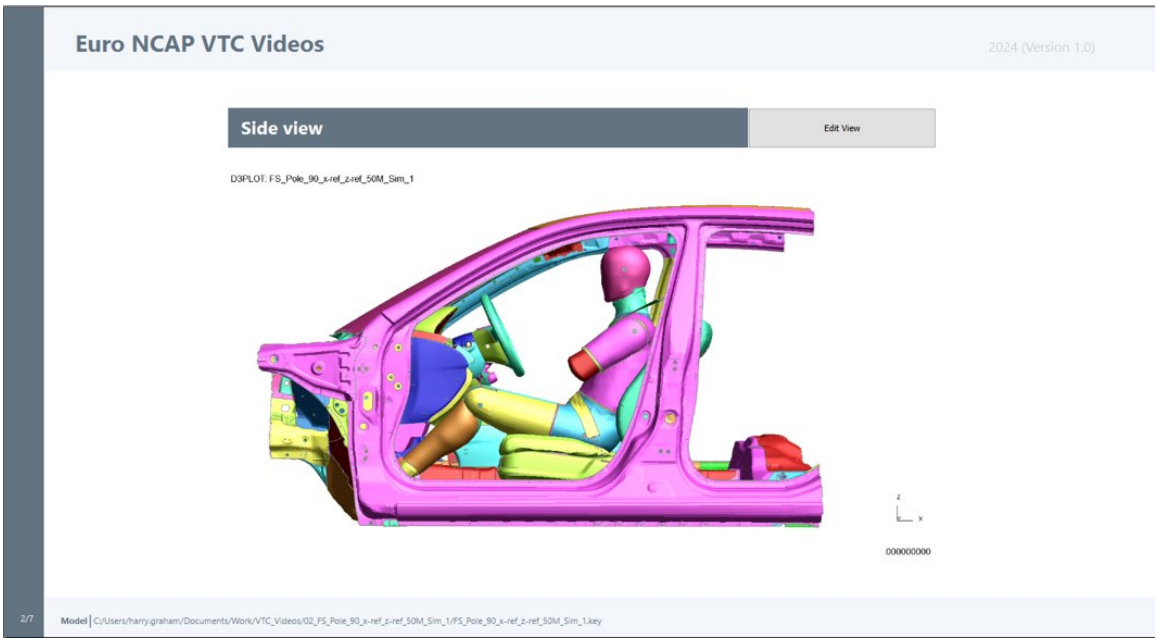
5. Where videos cannot be produced, a “missing data” image will be displayed.



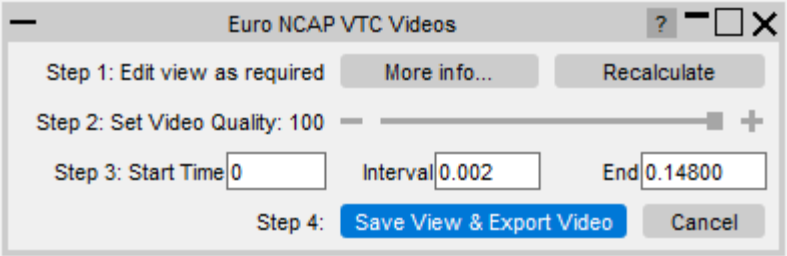
6. REPORTER automatically calculates the views and exports the videos. If you have previously adjusted any of the views, REPORTER will use the saved views. On the first page of the report, an overview of the results is presented.



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.


Euro NCAP VTC Videos
Step 1: Edit view as required
More info...
Recalculate
Step 2: Set Video Quality: 100
Step 3: Start Time
0
Interval
0.002
End
0.14800
Step 4:
Save View & Export Video
Cancel

3.9.3. LS-DYNA to ISO-MME

Tools → Workflows → LS-DYNA to ISO-MME

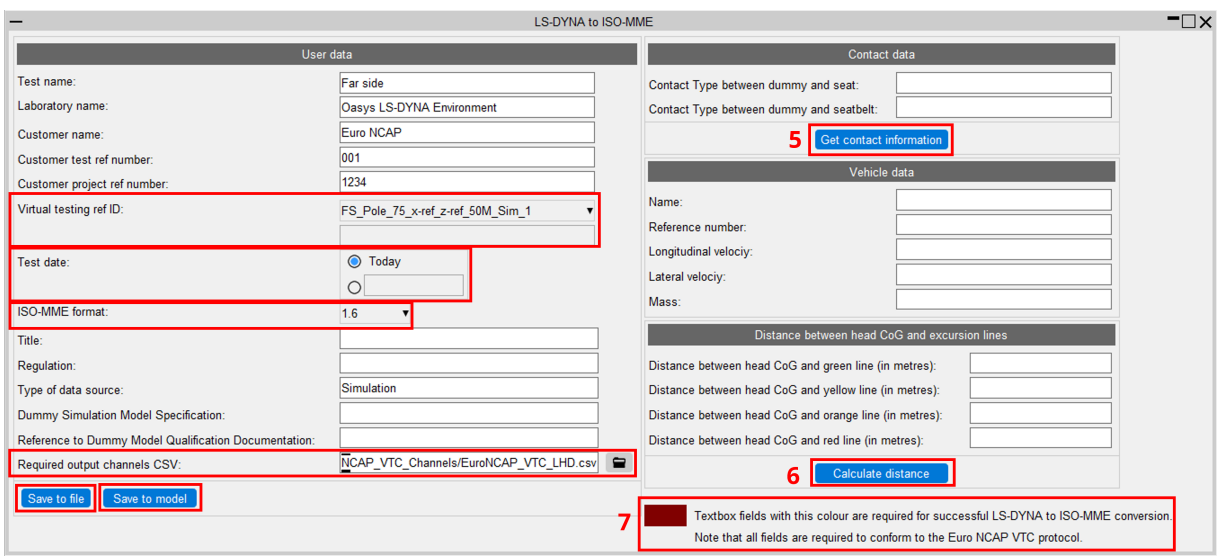
The LS-DYNA to ISO-MME workflow tool is one of the Virtual Testing tools. It helps you automatically export LS-DYNA results data to ISO-MME format as specified by the Euro NCAP Virtual Far Side Simulation & Assessment Protocol v1.0, ready for upload to the VTC Server. Automotive Assessments workflow 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

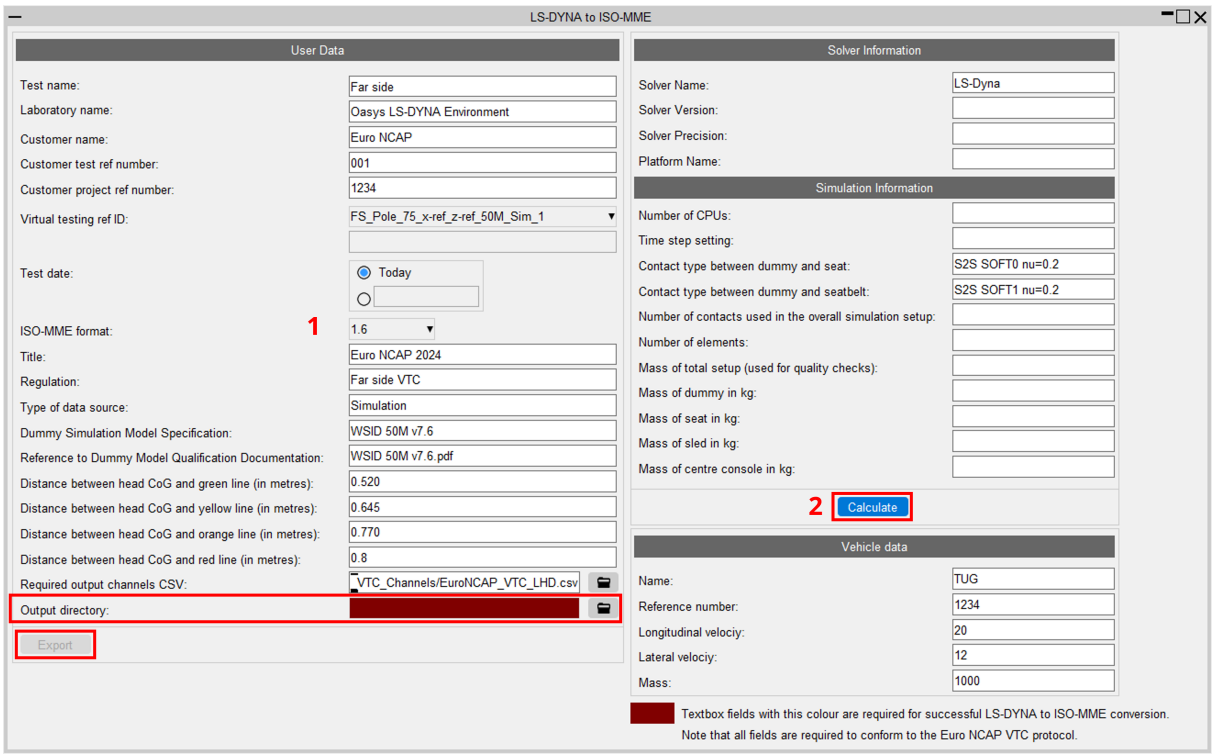
Only “Test name” and “Required output channels CSV” are required for exporting channels. However, note that technically, all inputs are required to conform to the Euro NCAP Far Side VTC protocol.

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

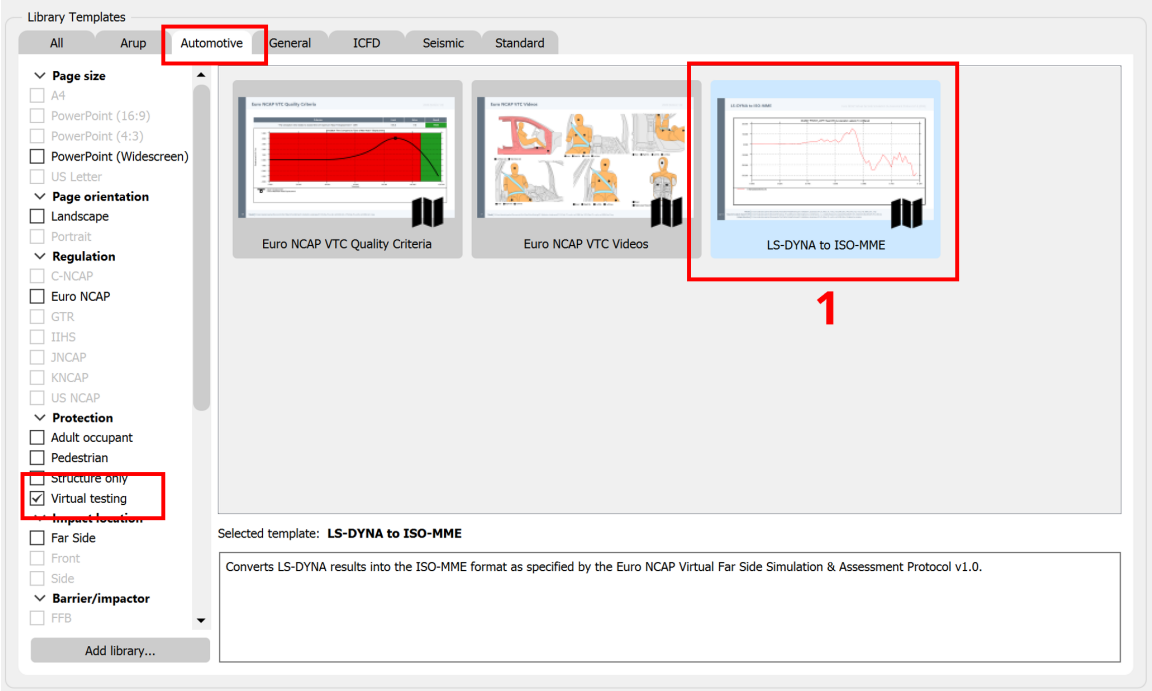
Select the output directory where you want to export channels in ISO-MME format.

4. Export

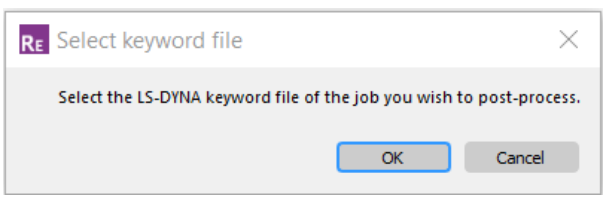
Once the output directory is selected, the **Export** button will be enabled. To perform the export, the LS-DYNA to ISO-MME workflow tool generates a configuration file from all the data and runs a separate T/HIS session in batch mode to export ISO-MME channels in the selected output directory.

Automation in REPORTER

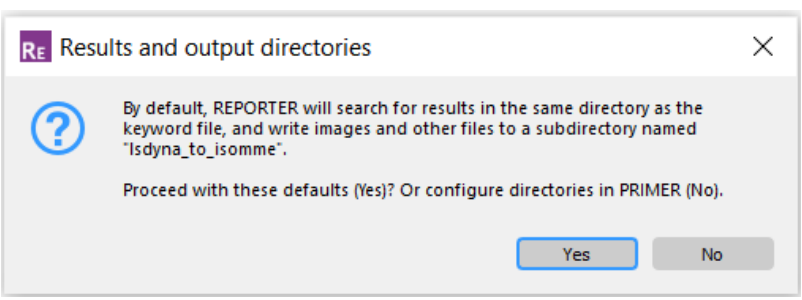
1. Within the Automotive tab in REPORTER, you will be able to select the LS-DYNA to ISO-MME Template. Filter by 'Virtual Testing' to easily find it.



2. Once open you will be prompted to select the LS-DYNA keyword file of the job you wish to post-process.

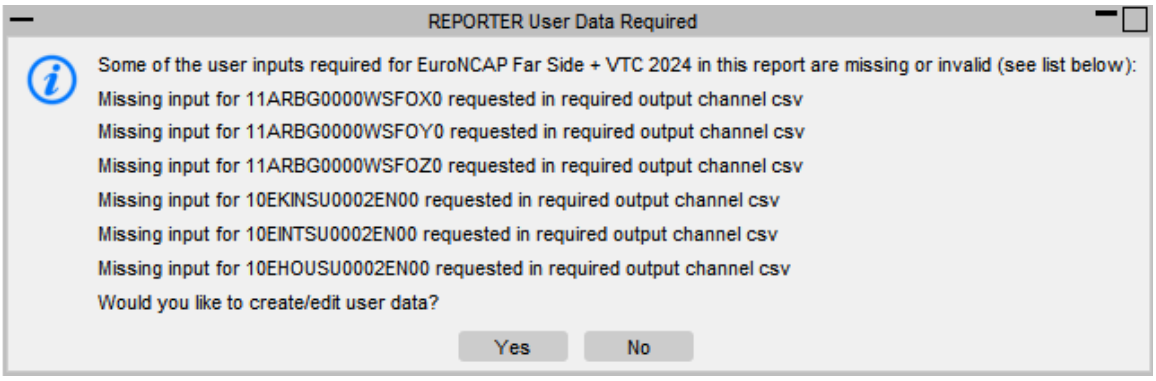


3. After you have selected your keyword file, you will then be asked if you want to continue with the default options of REPORTER searching for results in the same directory of the keyword file and writing images and outputs into a subdirectory called "lsdyna_to_isomme". If **No** is selected then PRIMER will be launched and a GUI will be displayed to configure the options. If **Yes** is selected the default options will be used.



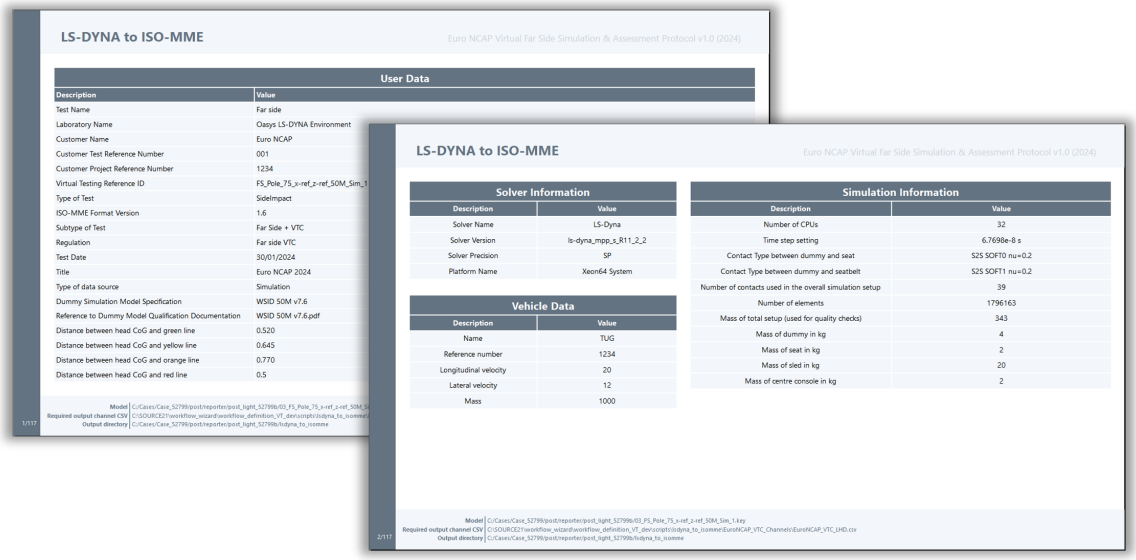
4. T/HIS will then launch automatically to produce the output files for the report.

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.

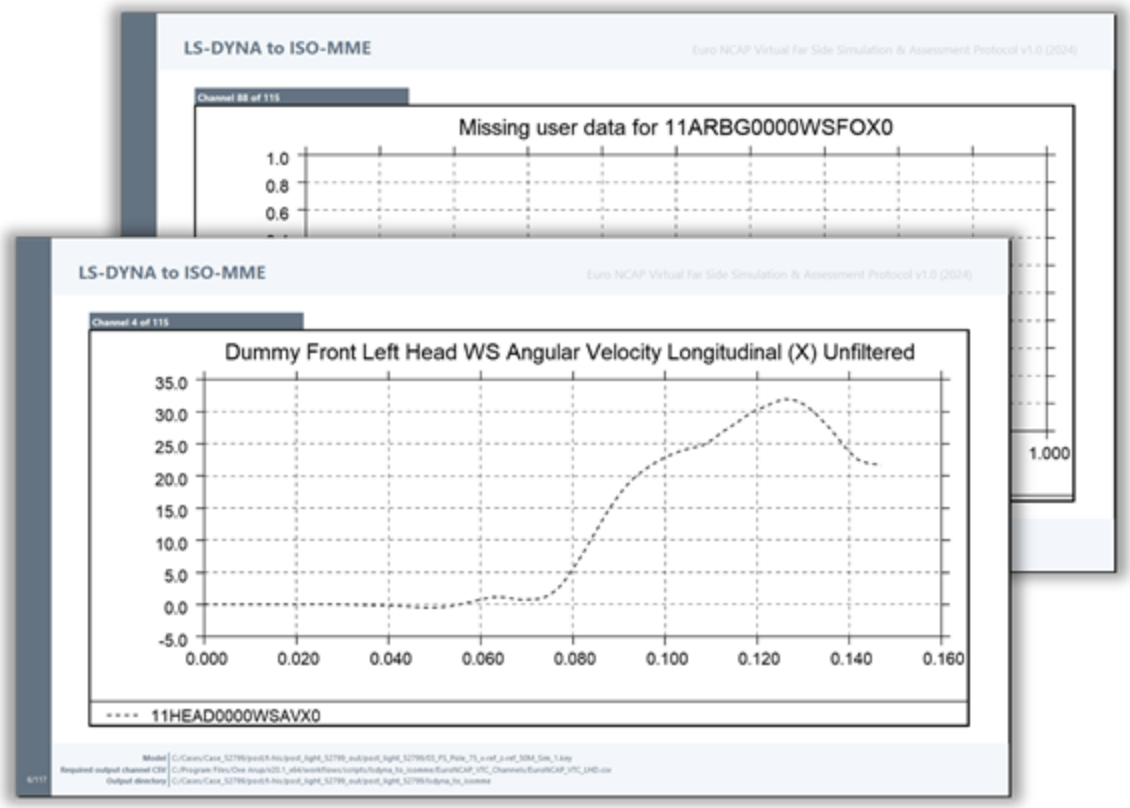


6. Once it has completed, T/HIS will close and return back to REPORTER.

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:


LS-DYNA to ISO-MME
Euro NCAP Virtual Far Side Simulation & Assessment Protocol v1.0 (2024)
Channel 88 of 115
Missing user data for 11ARBG0000WSFOX0
1.0
0.8
0.6
1.000
LS-DYNA to ISO-MME
Euro NCAP Virtual Far Side Simulation & Assessment Protocol v1.0 (2024)
Channel 4 of 115
Dummy Front Left Head WS Angular Velocity Longitudinal (X) Unfiltered
35.0
30.0
25.0
20.0
15.0
10.0
5.0
0.0
-5.0
0.000
0.020
0.040
0.060
0.080
0.100
0.120
0.140
0.160
11HEAD0000WSAVX0

3.9.4. 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.

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 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 Validation Criterion 1 (ISO Scores)

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 Virtual Far Side 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 18571:2024"
    b. The curves will be automatically regularised to10 kHz 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 "Calculate Head Excursion" button will become active, and can be clicked to set the maximum evaluation window time to the time of the maximum head excursion + 20%.
7. Click Calculate Head Excursion 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 contain 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.

Automation in REPORTER

If you save your SimVT settings in step 14 above, you can then use the Euro NCAP Virtual Far Side 2024 VC1 (ISO Scores) REPORTER template to automate the correlation process and the ISO Scores calculation for Validation Criterion 1.

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.

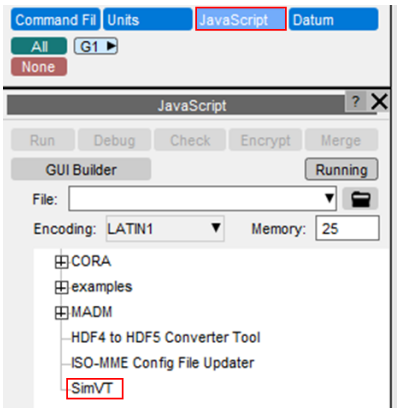
Steps in T/HIS

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 Without LS-DYNA Results

The SimVT tool can can be run directly from the JavaScript menu in T/HIS (**Tools → JavaScript → SimVT**). 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.

3.9.4.2. SimVT Windows

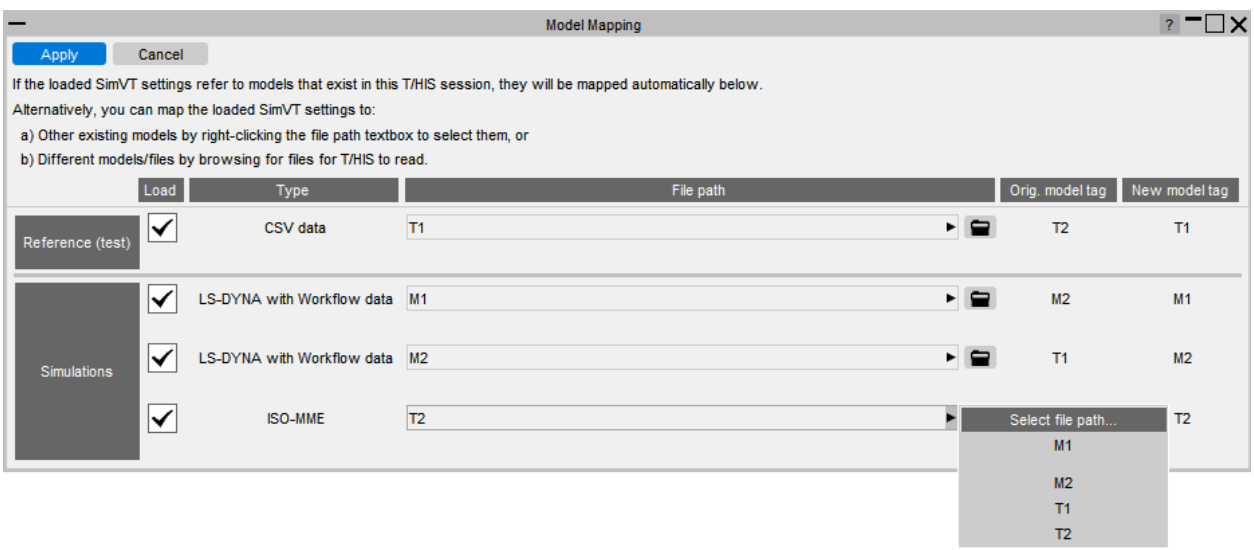
Model Mapping Window

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.

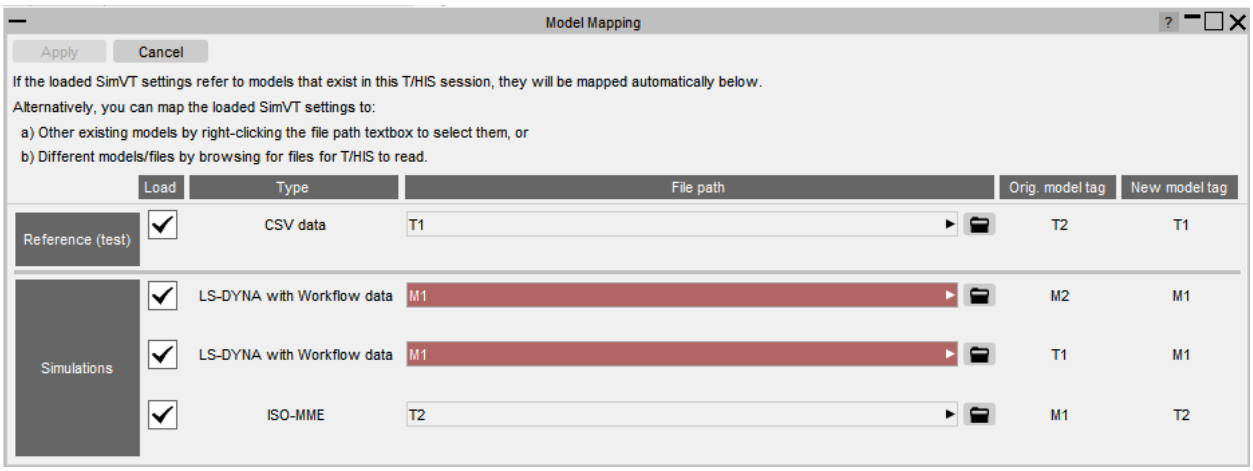
Mapping to Existing Data

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 T2 was the reference test in the settings file, but it has been mapped to T1 which will become the reference test when **Apply** is clicked.

Note: You can use the checkboxes under load column to select which models to load, by default all models are selected for loading.

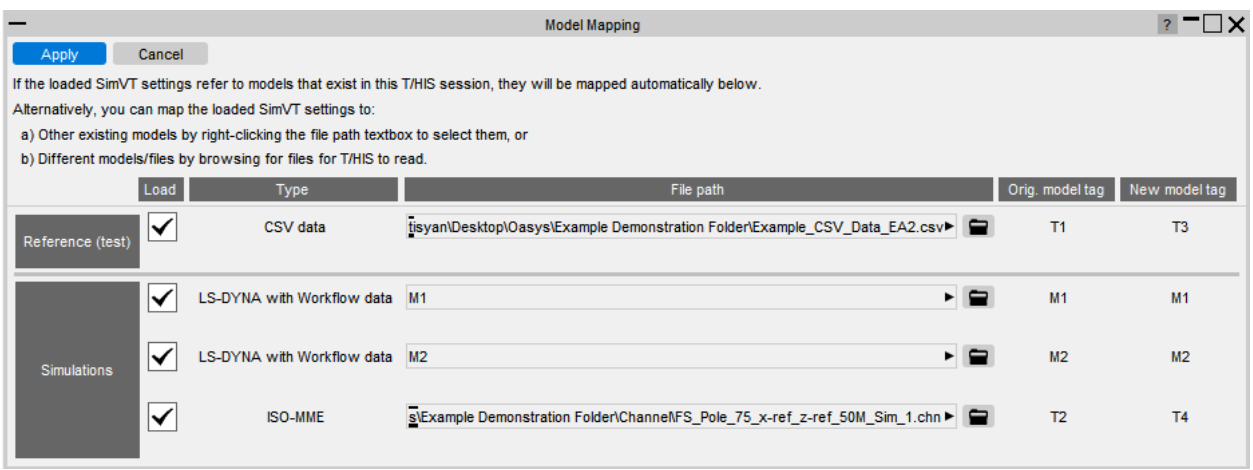


You cannot map the same data source multiple times and the file path textboxes will turn red to indicate the issue (e.g. M1 and M2 in the settings file cannot both map to the existing M1 model).



Mapping to New Data

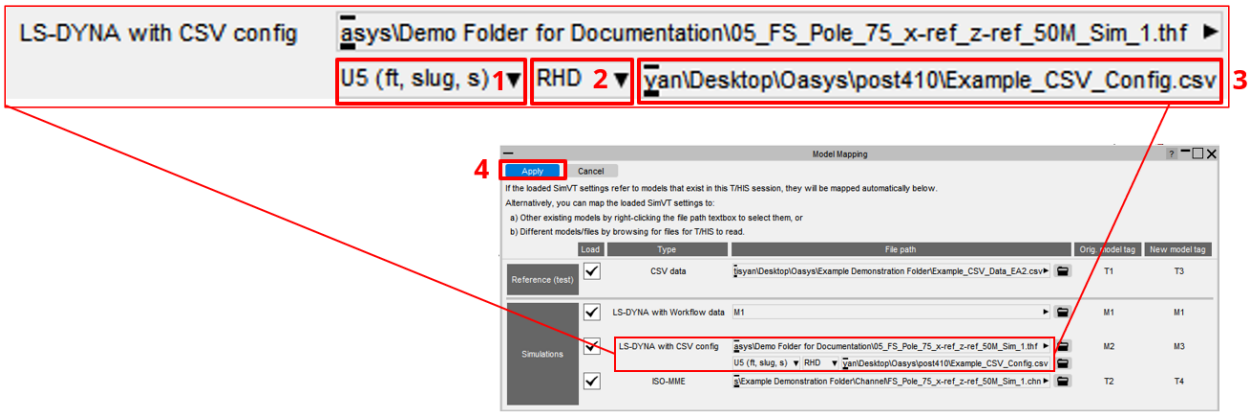
It is possible to import ISO-MME data and CSV data directly from the model mapping window. You can paste the path to the data into the file path textbox or you can click on the file selector and open it from there. In the example below T1 has been remapped to T3 which is a new CSV data source and T2 has been remapped to T4 which is a new ISO-MME 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 from the Model Mapping window. The LS-DYNA models will not be imported until **Apply** is clicked. If your model does not have Automotive Assessments Workflow Data (AAWD) you will 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].



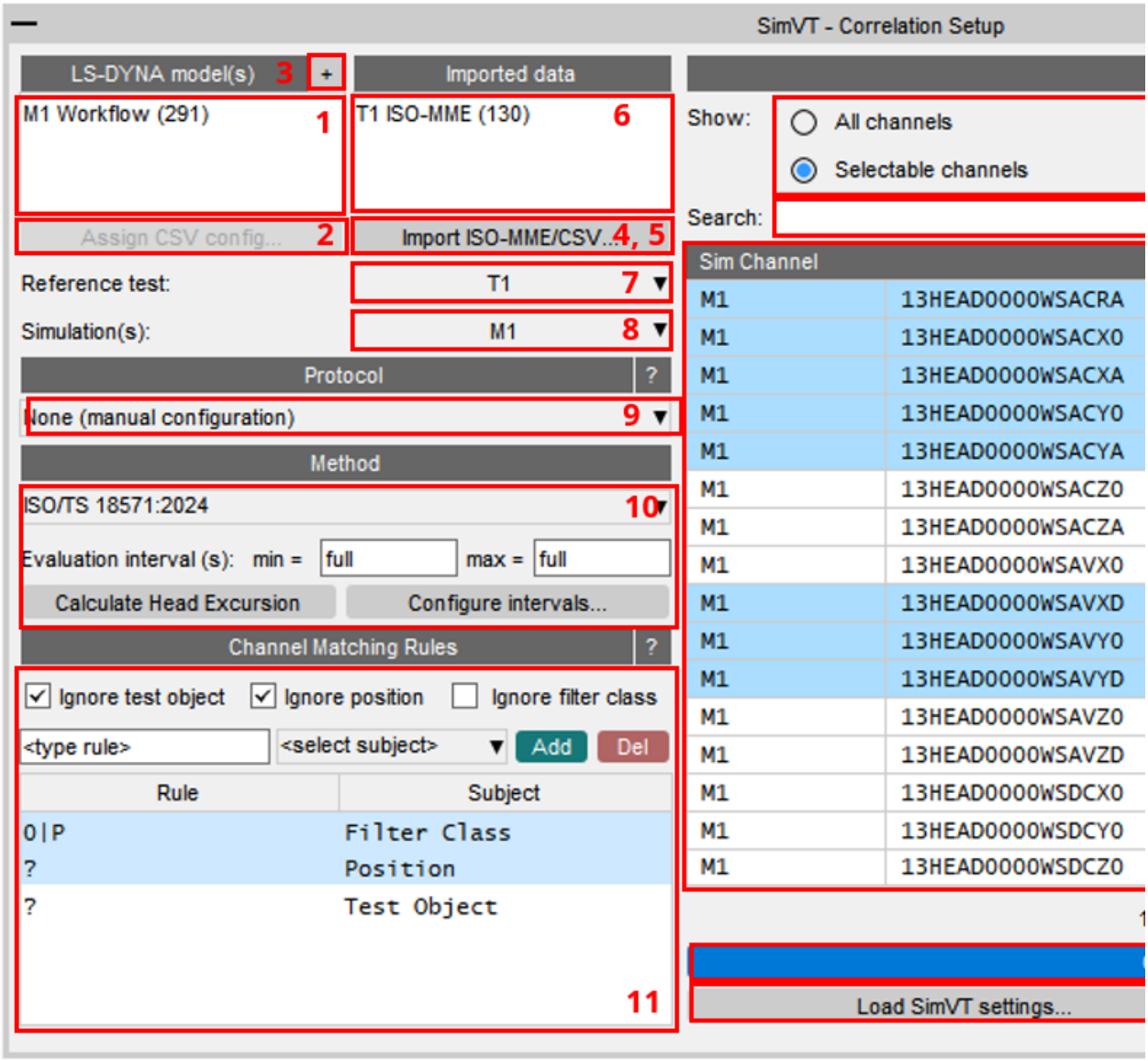
Correlation Setup Window

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 an LS-DYNA model
4. Import ISO-MME data
5. Import CSV data
6. View a list of the imported data
7. Select which model or data source will be used as the reference (this is typically physical test data from a laboratory)
8. Select which model(s) or data source(s) will be compared against the reference test (this is typically LS-DYNA simulation data)
9. Select the virtual testing protocol
10. Select the correlation method, the evaluation intervals and calculate head excursion
11. Apply channel matching rules
12. Inspect the available and selectable channels
13. Search for channels using regular expressions
14. Select the channels to be correlated
15. Perform “Simulation(s)” versus “Reference test” correlations on the selected channels

16. Save SimVT settings files

17. Load SimVT settings files



1. Model List

The models that are present in T/HIS when SimVT is loaded will be listed in 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. Click **Assign CSV config...** to open a window where the CSV config can be loaded and checked before it is assigned to the selected model.

3. Import an LS-DYNA Model

LS-DYNA models can be imported into SimVT for correlating. Click the **+** button to open a file selector window. Navigate to your model file and open it to load the data into SimVT.

4. Import ISO-MME data

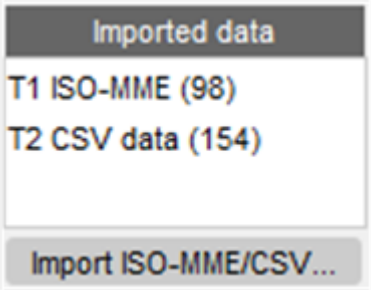
ISO-MME test data (v1.6 and v2.0) can be imported into SimVT for correlating. Click **Import ISO-MME or CSV...** to open a file selector window. Navigate to the ".chn" or ".mmi" file in your ISO-MME data and open it to load the data into SimVT.

5. Import CSV data

CSV channel data can be imported into SimVT for correlating. Click **Import ISO-MME or CSV...** to open a file selector window. Navigate to the ".csv" file with your channel data and open it to load the data into SimVT.

6. 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.

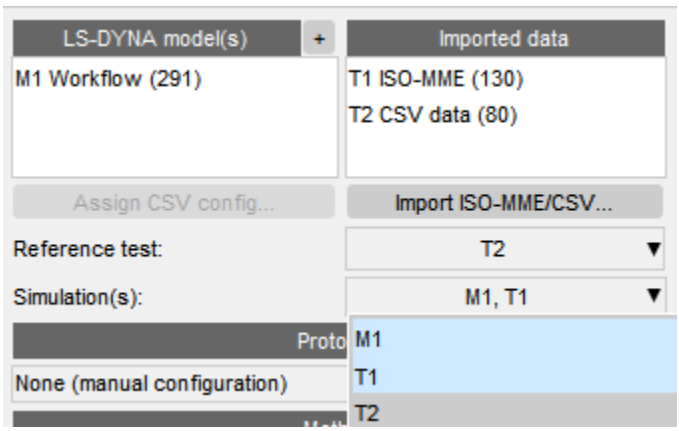


7. Selecting Reference (test) Source

The **Reference test** 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.

8. Select the Simulation(s)

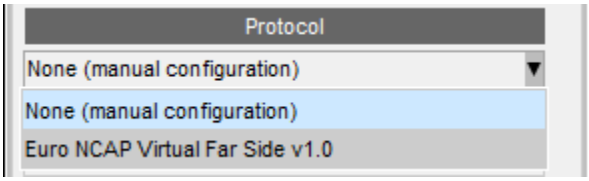
SimVT supports correlating multiple simulations against reference/test data. The simulations that are to be compared against the reference test data can be selected using the **Simulation(s)** drop-down (hold down Shift to select multiple). Note that the reference test data cannot be selected.



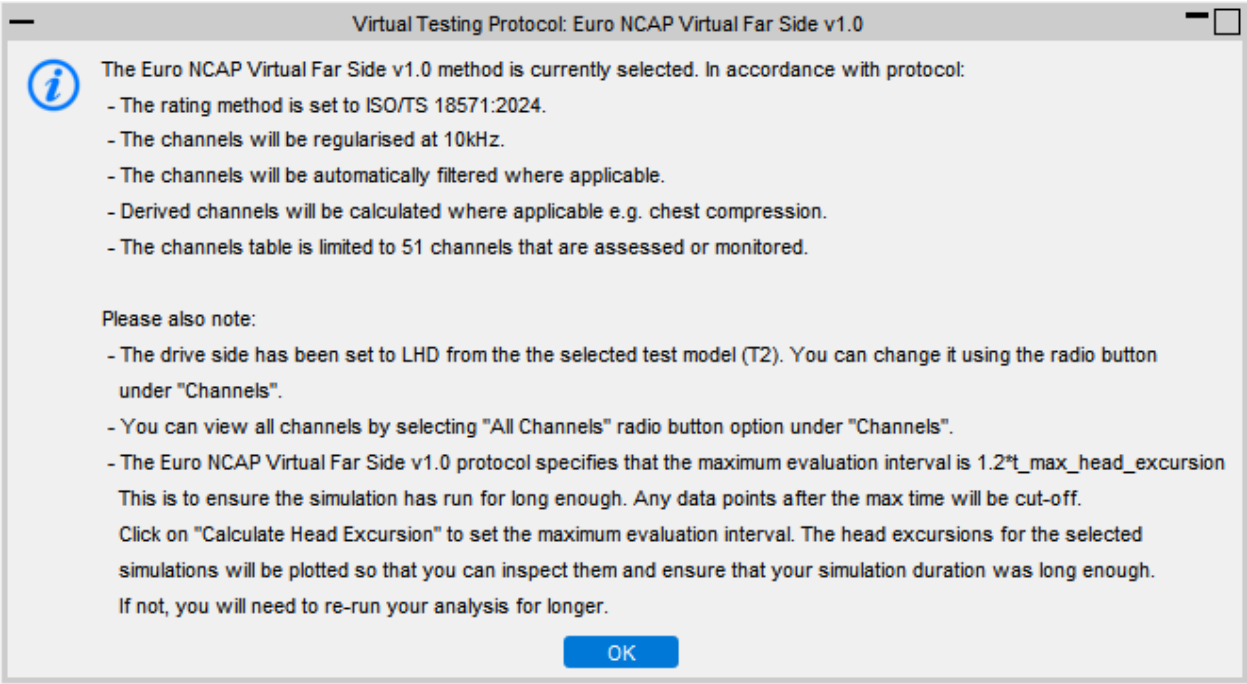
9. Select the Virtual Testing Protocol

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. Support for additional virtual testing protocols will be added in future releases.

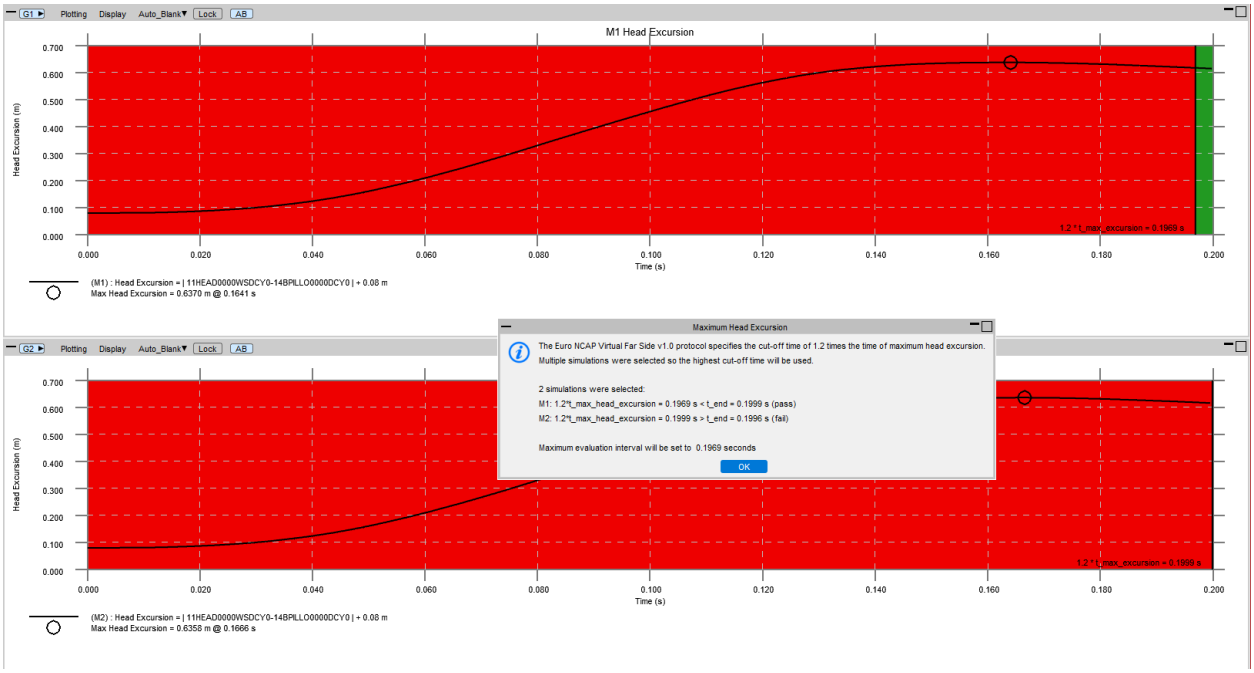


When a protocol is selected an information window will appear to explain the protocol specific configurations that have been applied.



Clicking **Calculate Head Excursion** under Method section will attempt* to set the maximum evaluation interval based on the time of maximum head excursion + 20% (according to Equation 1 of 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.



10. Select the Correlation Method

SimVT supports the same correlation methods that CORA uses. The default correlation method is ISO/TS 18571:2024, 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 T/HIS CORA tool.

Evaluation intervals can also be set in this section by inputting into 'min' and 'max' to set the values globally or by clicking 'Configure intervals...' to open a window where values can be set individually for each sim vs test model pair.

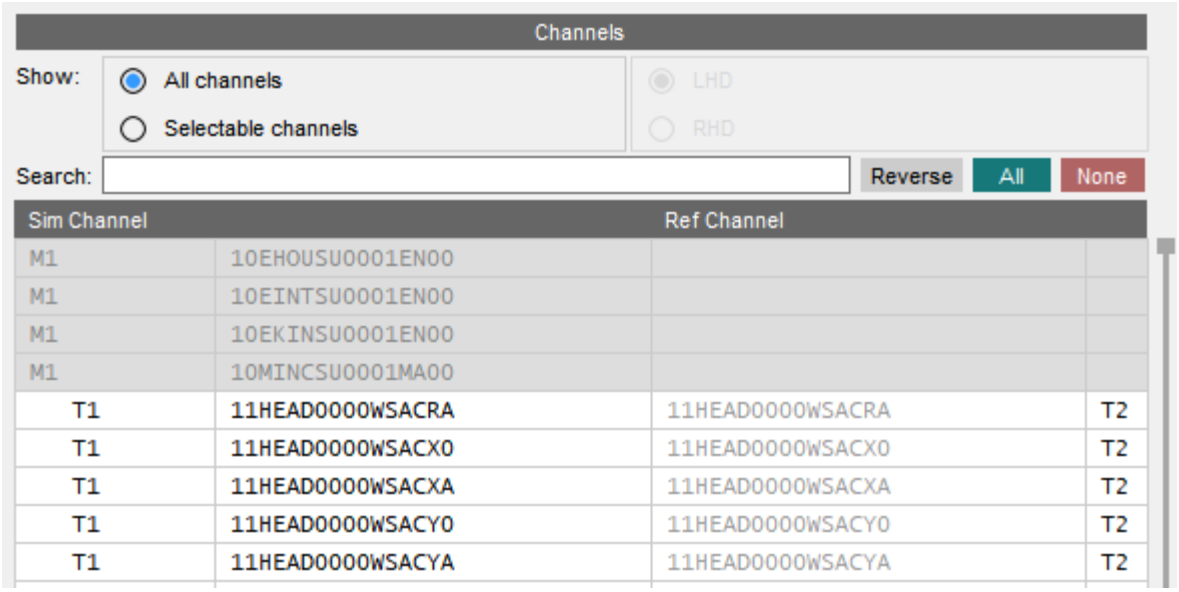
11. Channel Matching Rules

Channel matching rules can be used to help match channels from different data sources which may differ from the standard ISO-MME naming convention.

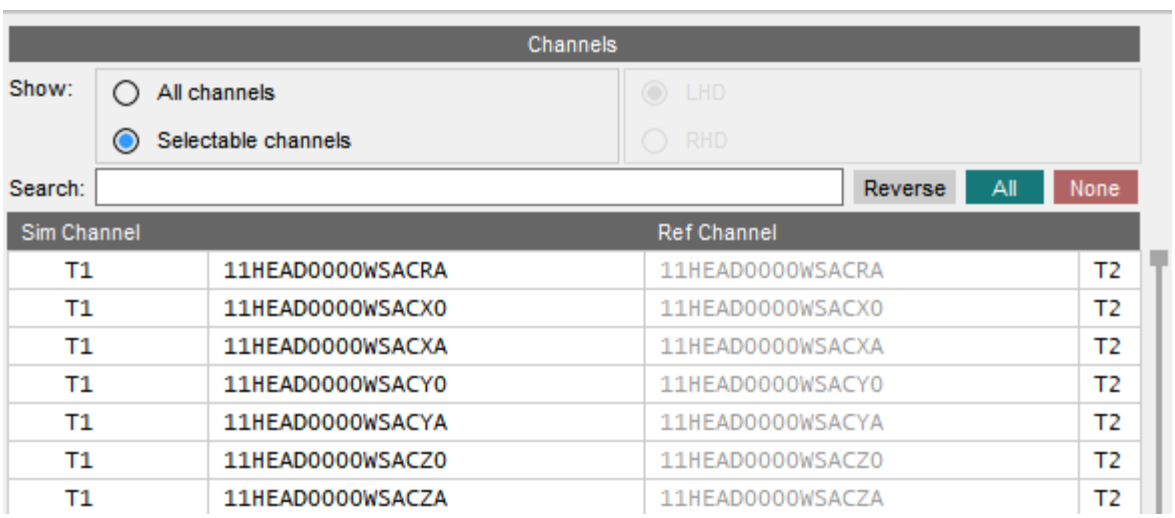
12. Inspect the available and selectable channels

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.

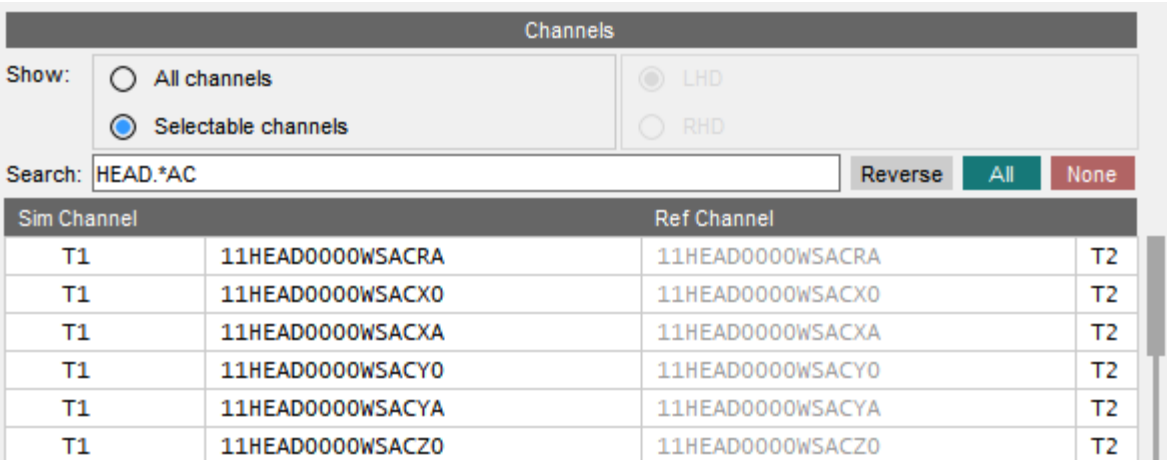


Selecting **Selectable channels** will update the list to only show channels which are selectable.

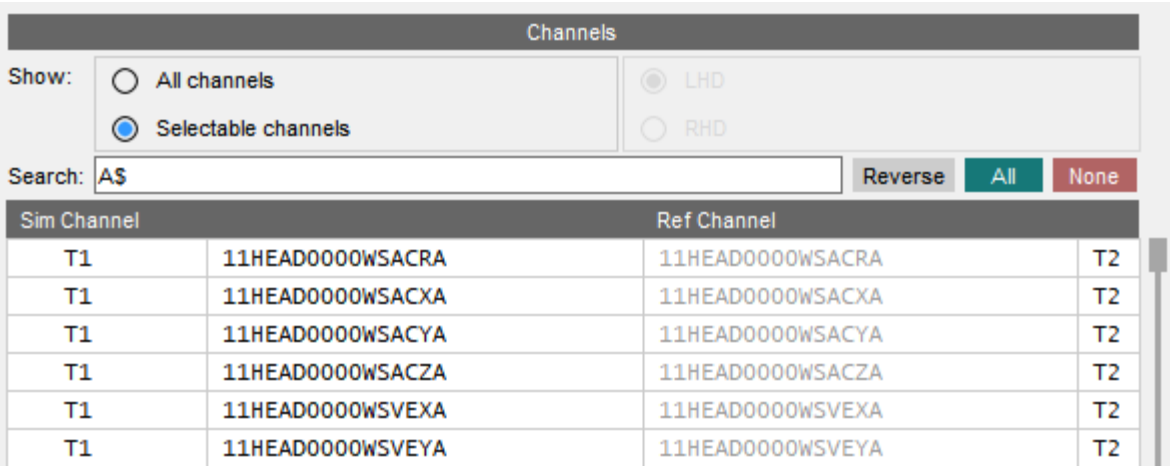


13. Search for channels using regular expressions

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.

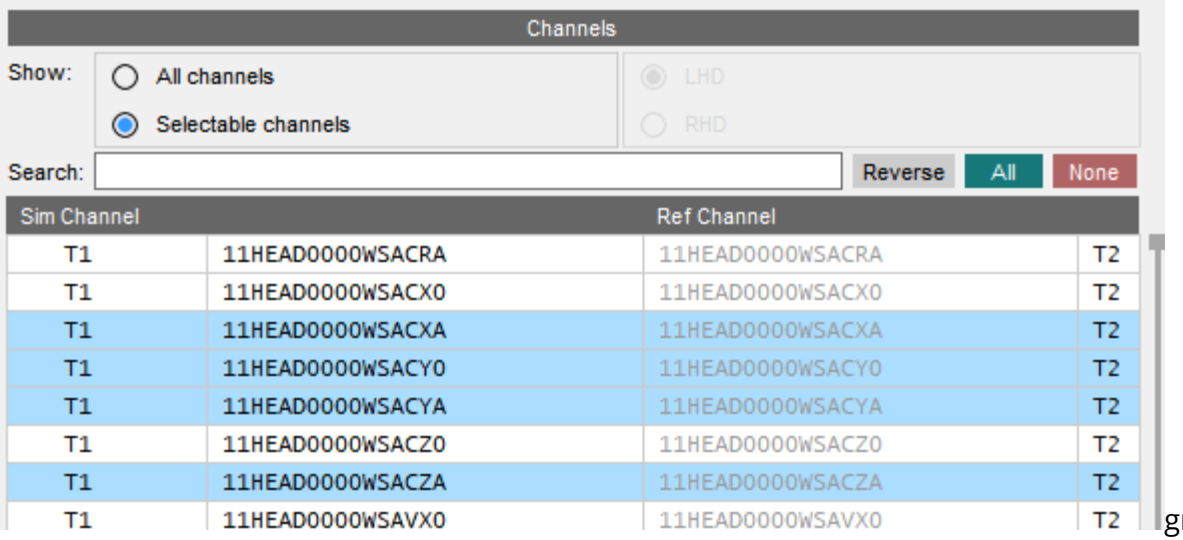


Or searching for "A$" will shown only the channels where the last character is "A" (i.e. ISO-MME channels that have been filtered with CFC1000).



14. 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.



gma

15. Perform “Simulation(s)” versus “Reference test” correlations on the selected channels

The **Correlate** button will become active when 1 or more channels are selected from the channel list. Click **Correlate** to commence the correlation.

16. Saving SimVT settings files

A SimVT settings file (.simvt) can be saved from the correlation setup window by clicking **Save SimVT settings...**. The settings file saves all the information required to restore the session (e.g. data sources, correlation method, selected and plotted channels). Note that prior to saving a settings file, a channel selection must be made.

17. Loading SimVT settings files

A SimVT settings file (.simvt) can be loaded from the Correlation Setup window by clicking **Load SimVT settings...**. 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.

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:

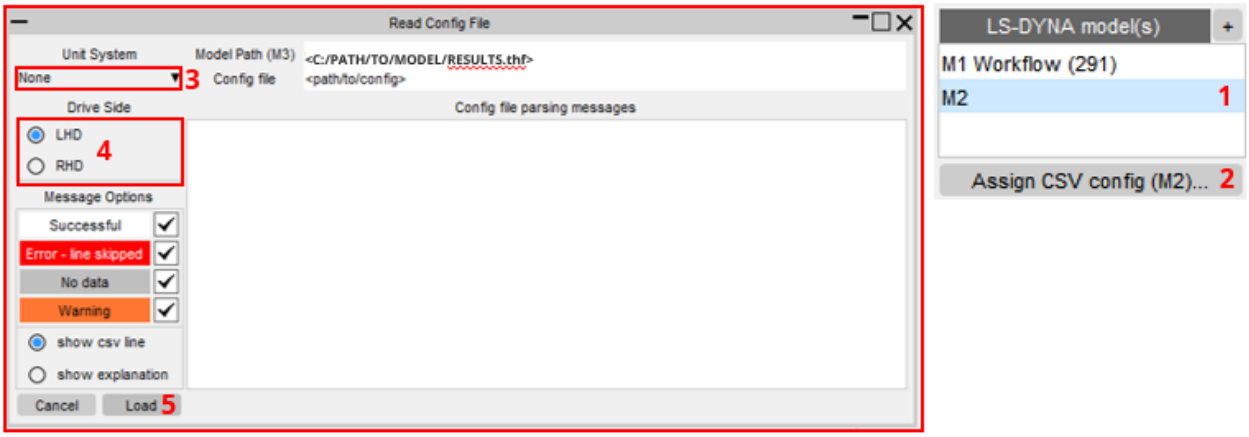
1. Select an LS-DYNA model from the model list that has no Automotive Assessment Workflow Data (AAWD) or CSV config data assigned (i.e. only the model tag will be listed (e.g. "M2"))

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)

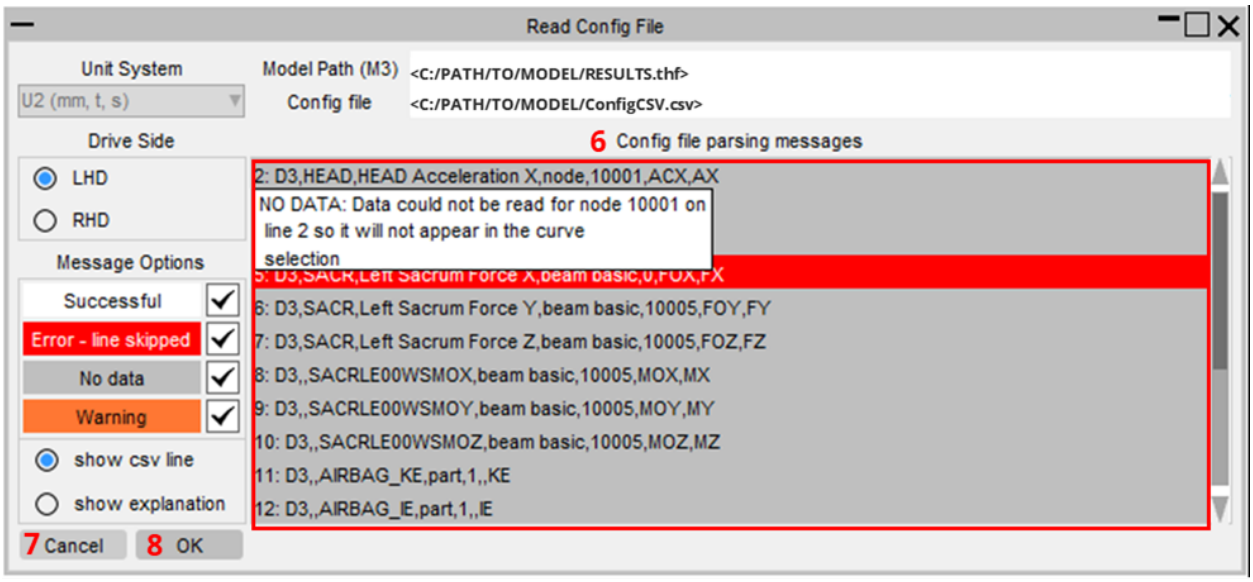
5. Click "Load" and open a CSV file with the correct configuration file format. The **Load** button will change to say **OK** when a config CSV file has been loaded.



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 clicking **Cancel** at any stage.

8. If you happy to proceed with assigning the CSV config, click **OK**.



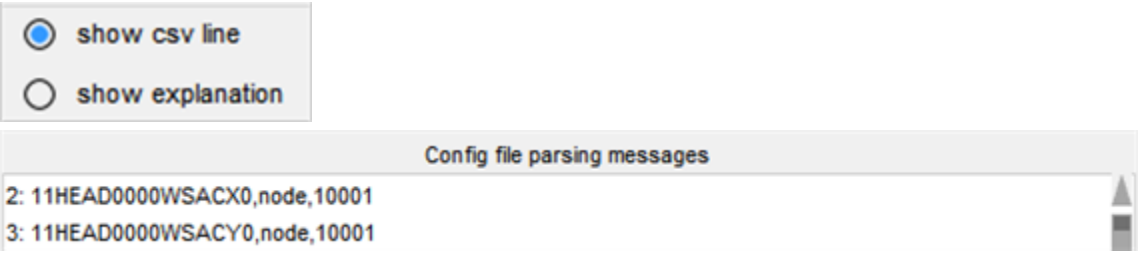
Checking Parsing Messages

When the configuration file is loaded, messages will appear in the window. Each row is coloured according to the parsing status:

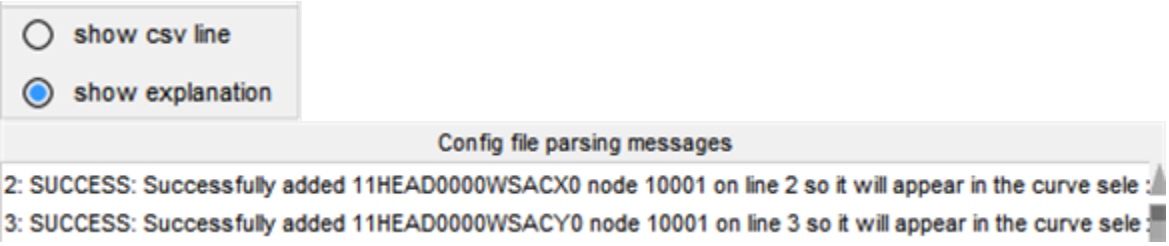
- White means that all the inputs were valid and that the model contains the entity and ID specified
- Red means there was an error with one or more inputs on the line
- Grey means that the model did not contain data for the entity type and ID (e.g., node 1001 below).
- Orange means that there was something 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.

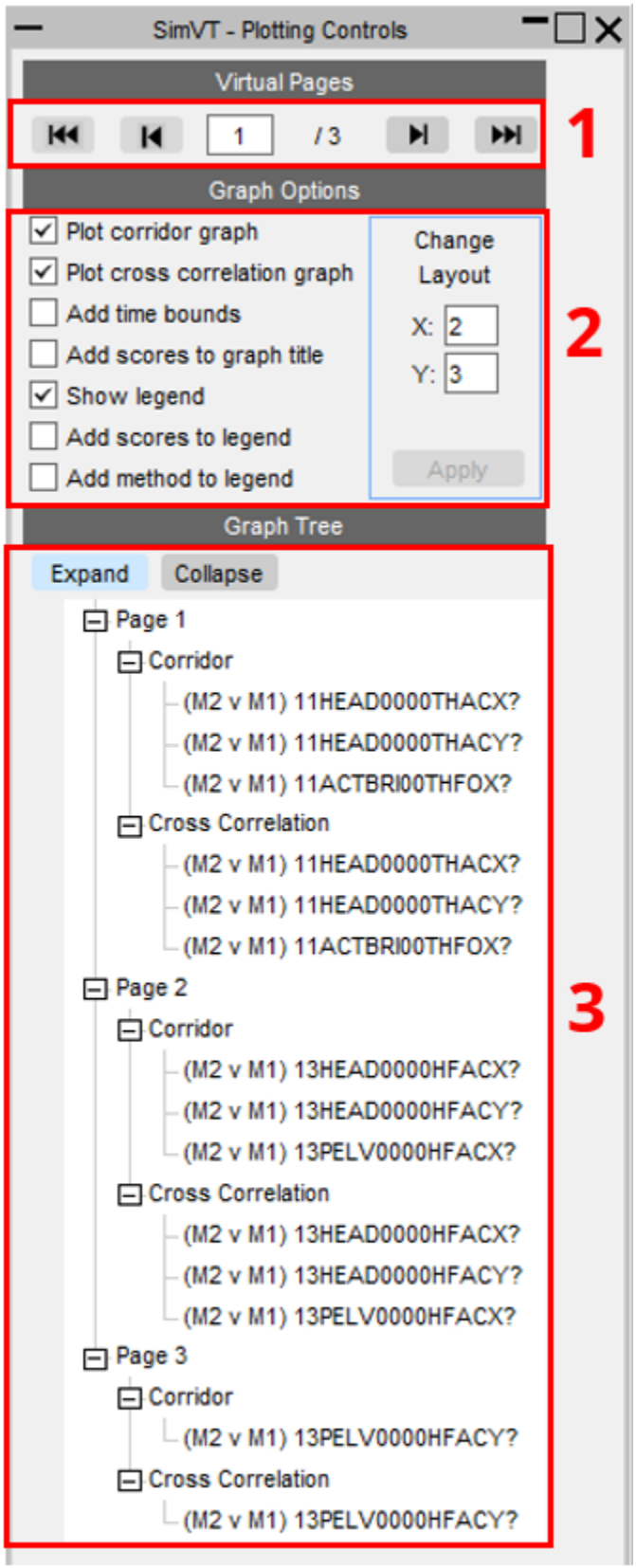
If **show CSV line** is selected then the messages will show each row of the CSV file.



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.



Plotting Controls Window


SimVT - Plotting Controls
Virtual Pages
1
/ 3
1
Graph Options
Plot corridor graph
Plot cross correlation graph
Add time bounds
Add scores to graph title
Show legend
Add scores to legend
Add method to legend
Change Layout
X: 2
Y: 3
Apply
2
Graph Tree
Expand
Collapse
Page 1
Corridor
(M2 v M1) 11HEAD0000THACX?
(M2 v M1) 11HEAD0000THACY?
(M2 v M1) 11ACTBRI00THFOX?
Cross Correlation
(M2 v M1) 11HEAD0000THACX?
(M2 v M1) 11HEAD0000THACY?
(M2 v M1) 11ACTBRI00THFOX?
Page 2
Corridor
(M2 v M1) 13HEAD0000HFACX?
(M2 v M1) 13HEAD0000HFACY?
(M2 v M1) 13PELV0000HFACX?
Cross Correlation
(M2 v M1) 13HEAD0000HFACX?
(M2 v M1) 13HEAD0000HFACY?
(M2 v M1) 13PELV0000HFACX?
Page 3
Corridor
(M2 v M1) 13PELV0000HFACY?
Cross Correlation
(M2 v M1) 13PELV0000HFACY?
3

The Plotting Controls Window in SimVT is used to control which 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

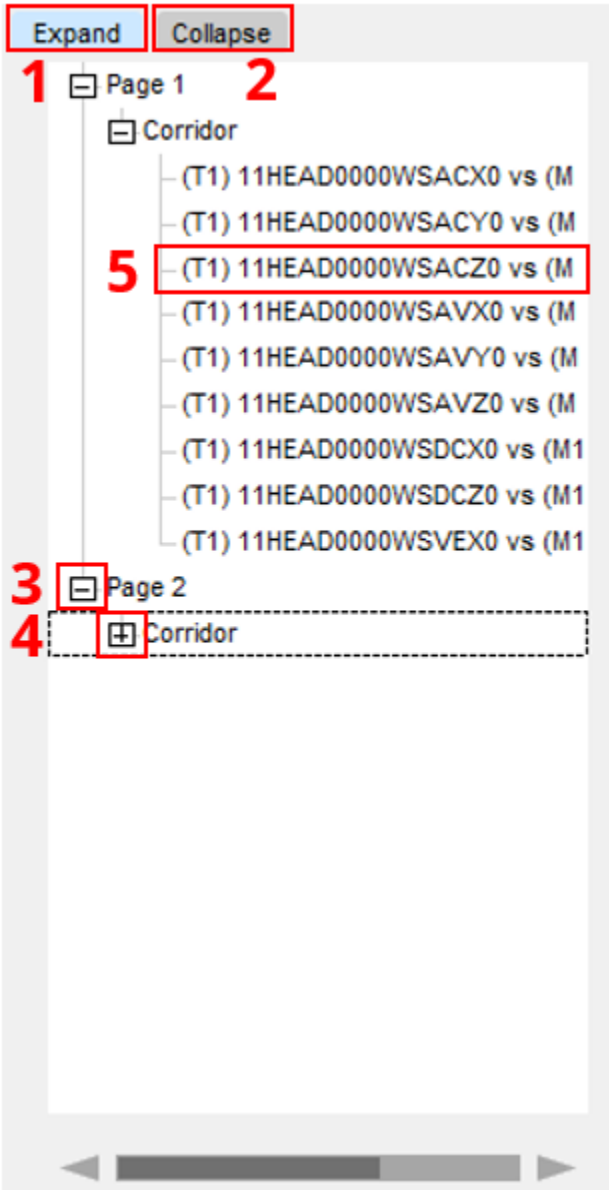
T/HIS currently has a limit of 32 graphs per session. Virtual pages are employed to work around this limitation, allowing SimVT to plot as many graphs as desired. When there is insufficient space to show the graphs for all the selected channels on the current page, additional graphs are added to Virtual 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

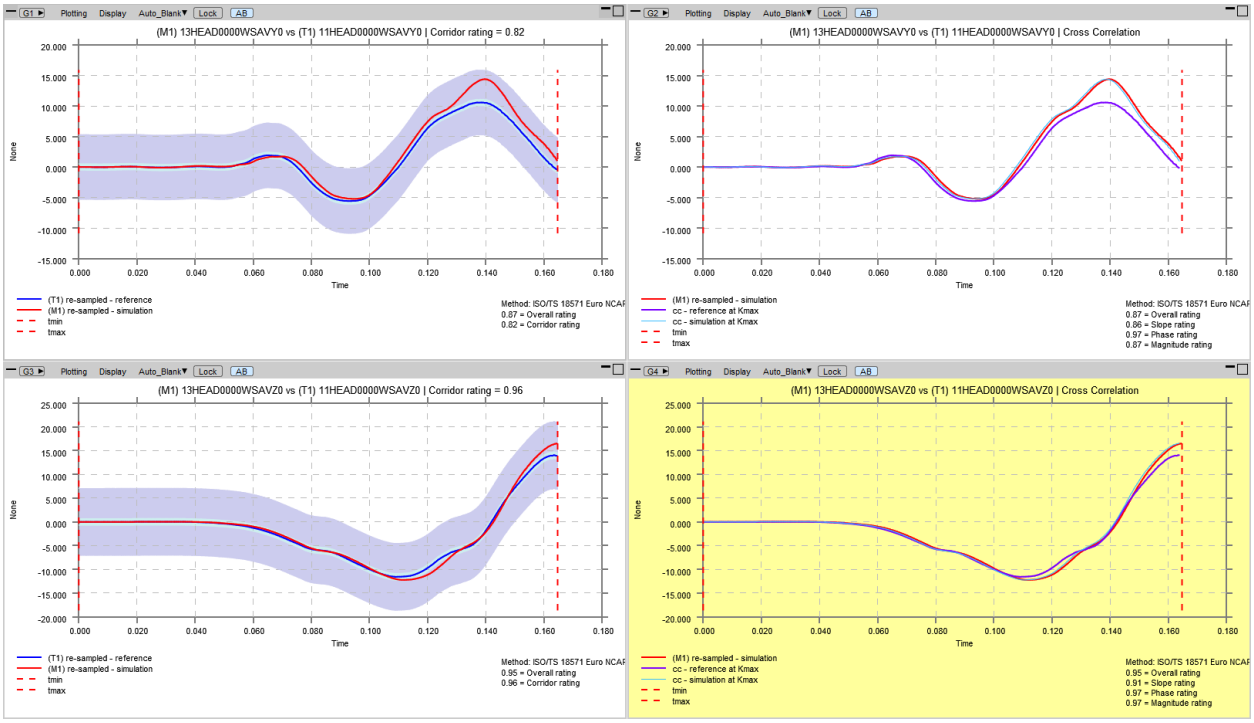
Under the Graph Options section of the Plotting Controls Window the main graph plotting options are located. These options are described in detail in Graph Options.

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:

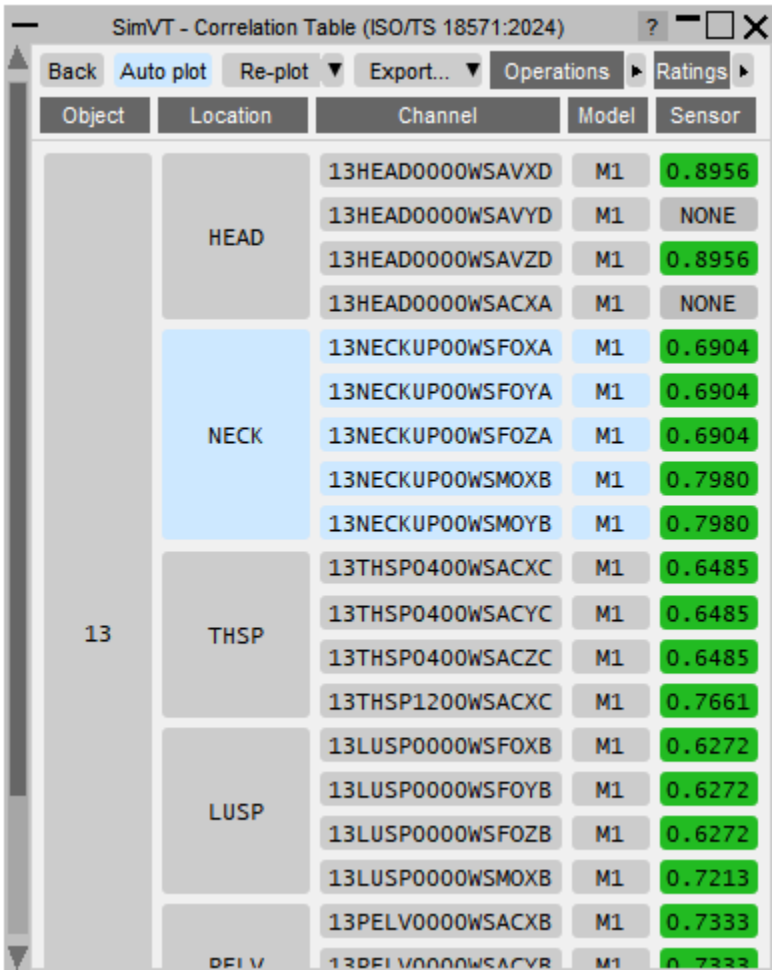
- Expanding the whole graph tree view **[1]**
- Collapsing the whole graph tree view **[2]**
- Expanding an individual section of the graph tree **[3]**
- Collapsing an individual section of the graph tree **[4]**
- Highlighting individual graphs as shown below by clicking graph title in the graph tree view **[5]**


G1
Plotting
Display
Auto_Blank
Lock
AB
(M1) 13HEAD0000WSAVY0 vs (T1) 11HEAD0000WSAVY0 | Corridor rating = 0.82
None
Time
(T1) re-sampled - reference
(M1) re-sampled - simulation
tmin
tmax
Method: ISO/TS 18571 Euro NCAP
0.87 = Overall rating
0.82 = Corridor rating
G2
(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
G3
(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
G4
(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

3.9.4.2.5. Correlation Table Window

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 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, inverted 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 18571:2024)

Back | Auto plot | Re-plot | Export... | Operations | Ratings

Object	Location	Channel	Model	Sensor	Weight	ISO	Corridor	Cross Correlation		
								Slope	Phase	Mag.
	HEAD	13HEAD00VEWSACXA	M1	0.7871	0.2762	0.7090	0.7959	0.3473	0.8544	0.7515
		13HEAD00VEWSACYA	M1	0.7871	0.7238	0.8170	0.8819	0.5003	0.9211	0.8995
		13HEAD0000WSAVZD	M1	0.9530	1.0000	0.9530	0.9608	0.9057	0.9727	0.9652
			M2	0.9530	1.0000	0.9530	0.9608	0.9057	0.9727	0.9652
		13HEAD0000WSACXA	M1	0.6775	0.1301	0.6969	0.7873	0.3015	0.8938	0.7147
			M2	0.6775	0.1301	0.6969	0.7873	0.3015	0.8938	0.7147
		13HEAD0000WSACYA	M1	0.6775	0.3452	0.8068	0.8825	0.4647	0.9211	0.8833
			M2	0.6775	0.3452	0.8068	0.8825	0.4647	0.9211	0.8833
		13HEAD0000WSACZA	M1	0.6775	0.5247	0.5876	0.6272	0.3289	0.8301	0.5244
			M2	0.6775	0.5247	0.5876	0.6272	0.3289	0.8301	0.5244
	NECK	13NECKLO00WSFOYA	M1	0.6306	0.4661	0.6608	0.7624	0.3194	0.8574	0.6023
			M2	0.6306	0.4661	0.6608	0.7624	0.3194	0.8574	0.6023
		13NECKLO00WSFOZA	M1	0.6306	0.5339	0.6042	0.6230	0.4374	0.8210	0.5167
			M2	0.6306	0.5339	0.6042	0.6230	0.4374	0.8210	0.5167
		13NECKLO00WSMOXB	M1	0.8039	0.6710	0.8377	0.9237	0.5865	0.8513	0.9035
			M2	0.8039	0.6710	0.8377	0.9237	0.5865	0.8513	0.9035
		13NECKLO00WSMOYB	M1	0.8039	0.2734	0.7172	0.7114	0.4462	1.0000	0.7170
			M2	0.8039	0.2734	0.7172	0.7114	0.4462	1.0000	0.7170

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 to toggle plots on and off. Clicking an un-toggled (grey) button in one these columns will automatically toggle all the buttons to the right of the button that was clicked. 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 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 which will cause the corresponding 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.

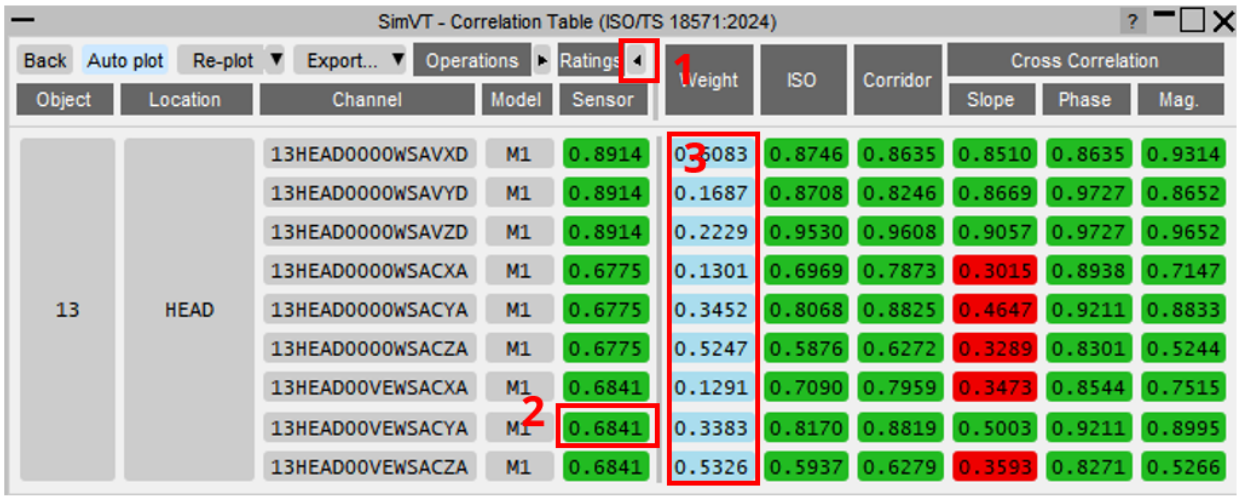
Clicking a button in a column and then holding down shift and clicking another button in the same column will change the selection of all the in-between buttons to match the first one clicked.

Clicking one of the ratings under Weighted, Total, Corridor and Cross Correlation columns highlights the plot if it is already plotted.

3.9.4.3. SimVT Features

3.9.4.3.1. Ratings and Weightings

Ratings and Weightings



SimVT provides the flexibility to calculate ratings using different methods available in the rating method dropdown in the Correlation Setup Window. The resulting rating scores are color-coded differently depending on the protocol that is selected in the protocol dropdown in the Correlation Setup Window. It's important to note that when you select a protocol, SimVT automatically applies the corresponding rating method associated with that protocol. However, this default selection can be overridden if you wish to use a different rating method. This ensures that while protocols streamline the rating process, you still have the control to adjust the methodology as needed.

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 protocol.

For Euro NCAP Virtual Far Side v1.0:

- [0.50, 1.00] - green
- [0.00, 0.50) - red

For anything else:

- [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 2 in Section 6.3.3 of the Euro NCAP VTC Simulation and Assessment Protocol v1.0:

$$w_i = \frac{max(|Channel_{test_i}|)}{max(|Channel_{test_x}|) + max(|Channel_{test_y}|) + max(|Channel_{test_z}|)} \quad \text{with} \quad i = X, Y, Z$$

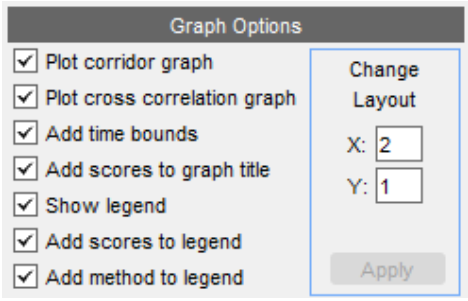
The weighting factors are presented in [**3**] and used to summarise the individual ISO scores for each axis of one sensor to one score per sensor $_{sensor}$ according to Equation 3 in Section 6.3.4 of the Euro NCAP VTC Simulation and Assessment Protocol v1.0:

$$S_{Sensor} = \displaystyle\sum_{i} w_i * S_i \quad \text{with} \quad i = X, Y, Z$$

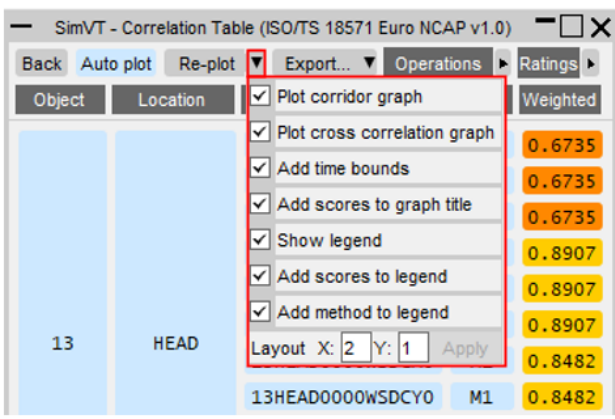
For channels without X, Y, Z siblings, the weighted rating is equal to the total rating. To find out how a single weighted rating was calculated hover over the weighted rating score [**2**] or Export Results.

3.9.4.3.2. Graph Options

Graph Options



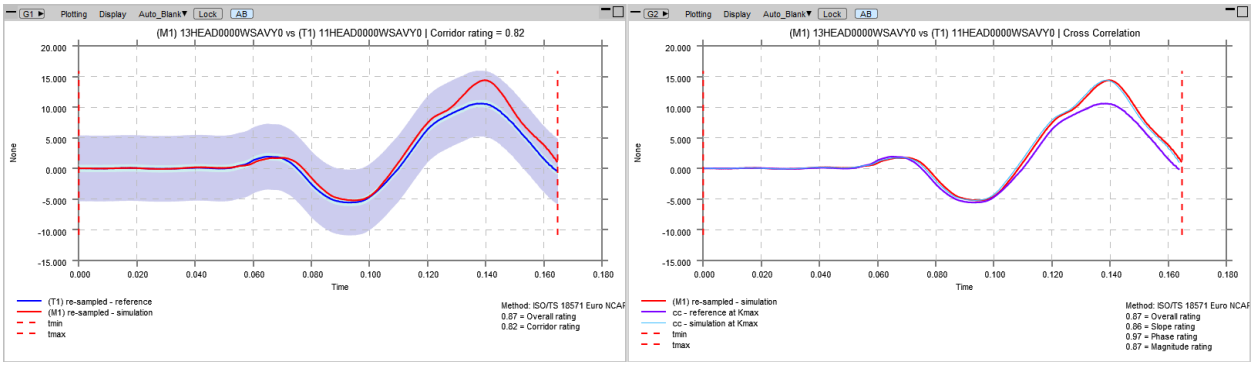
OR



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.



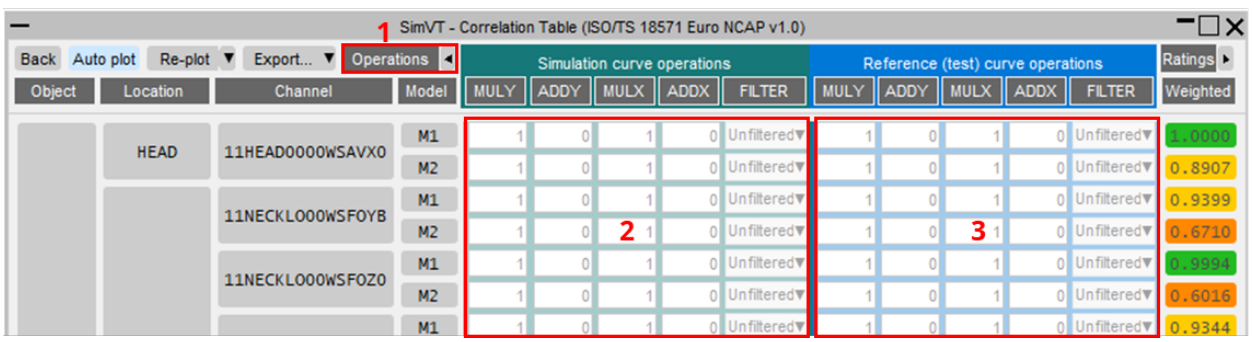
3.9.4.3.3. Operations

Operations

Corrective operations can be applied to both the simulation and test curves before the correlation is performed. When running with a Protocol option, operations, including filtering, are automatically applied according to the protocol and will not appear in the Operations table. The Operations table is available for performing additional operations, such as correcting for differences in sign or units, allowing you to fine-tune the process as needed.

Applying Operations

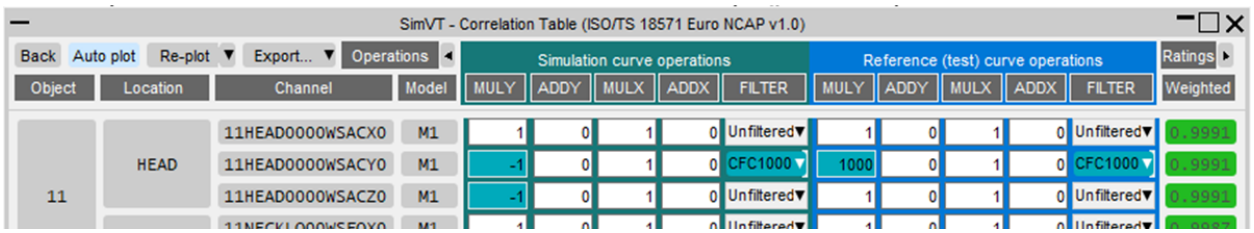
Operations are accessed by clicking **Operations** [1] in the Correlation Table Window which will expanding the window to show the operations that have been applied. This view is minimised by default to reduce the amount of space that the Correlation Table Window occupies.



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)

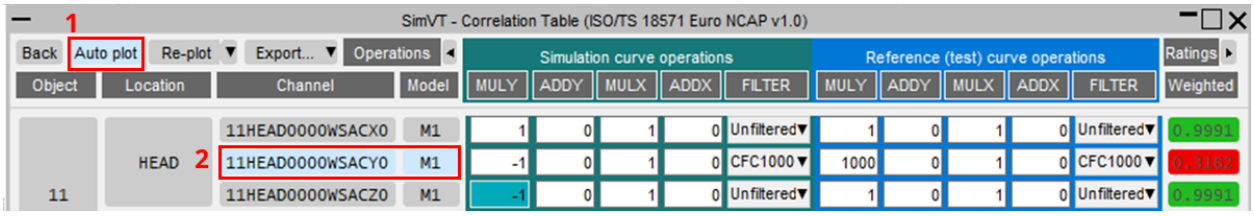
- **MULY** scales the Y coordinates by the value entered (default = 1, no scaling). For example, if you want to invert the sign of a curve, enter "-1".
- **ADDY** offsets the Y coordinates by the value entered (default = 0, no offset).
- **MULX** scales the X coordinates by the value input (default = 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. Note that the filter operation is in addition any filtering that may already have been applied. For example, in the image above, 11NECKLO00WSFOYB has 'B' filter class which indicates that CFC600 filtering has already been applied. No additional filter operations have been applied so they show "Unfiltered" as the default.)

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.

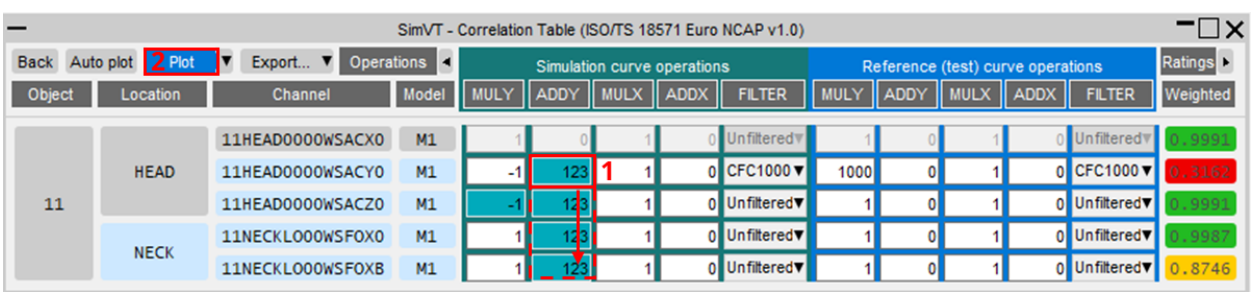
When **Auto plot** is active [1] and a channel is selected [2], operations are executed automatically. Note that the latent coloured textboxes change back to white to indicate that the operations have been applied and ratings will be recomputed.



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.

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 categories of rules that can be created:

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:

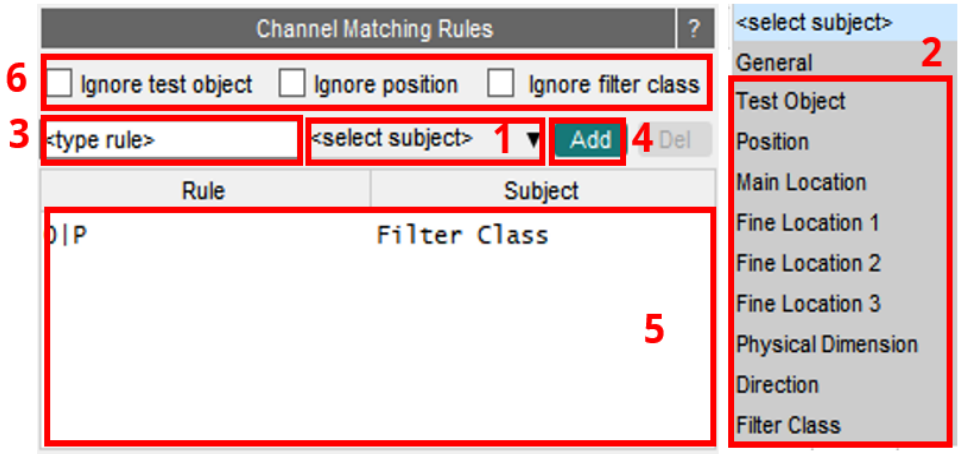
1. Ignore rule – allows you to ignore a part of a string, e.g. the main location (this rule is only applicable in ISO mode)
2. Equivalence rule – allows you to establish equivalency between two or more substrings (options).
    a. options are separated by the "|" e.g. <option1>|<option2>|<option3>|...
    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 an **ISO 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

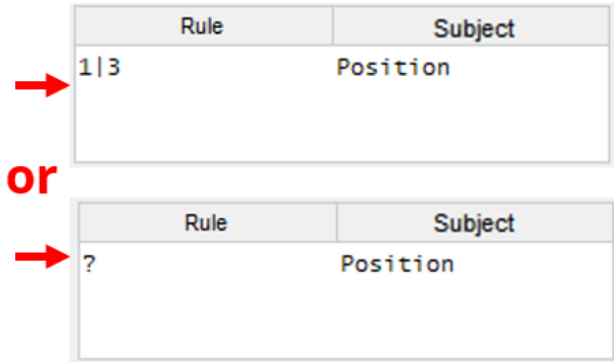


Note: you can add multiple rules for the same ISO subject.

Examples

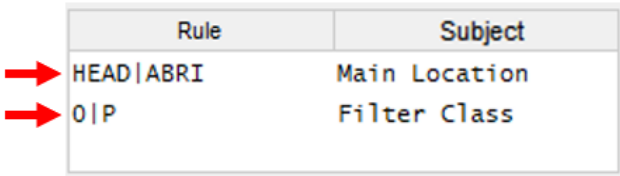
To match channels 13HEAD000000ACX0 with 11HEAD000000ACX0:

- 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 13HEAD000000ACX0 with 13ABRI000000ACXP:

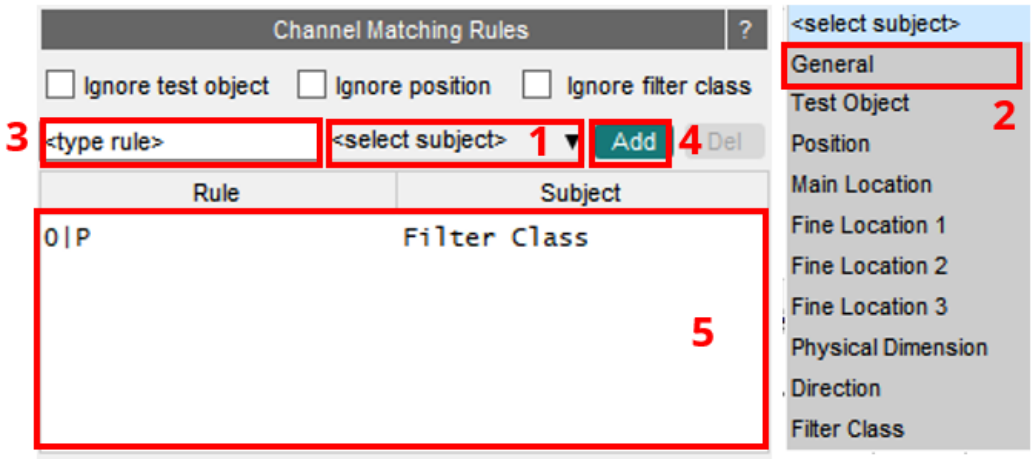
- Add an equivalence rule by typing in **HEAD|ABRI** and selecting **Main Location** as the subject.
- And add another equivalence rule by typing **0|P** and selecting **Filter Class**.



Adding a General Rule

To add a General rule, follow a similar process, but not that only equivalence rules are supported.

-
- Select the General s**ubject** by clicking [**1**] and picking [**2**].
- Enter a rule in [**3**] in the format: <option1>|<option2>|<option3>...
- Press **Add** [**4**]**.** If the rule is successfully validated it will appear in the list [**5**].

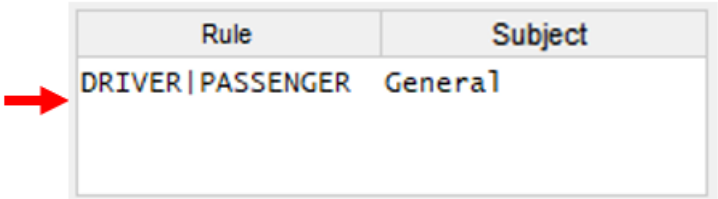


Note: you can add multiple General subject rules.

Examples

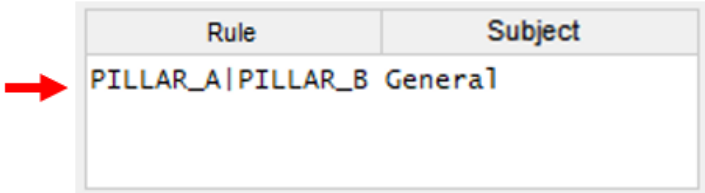
To match channels Driver_Airbag with Passenger_Airbag:

- Add an equivalence rule by selecting **General** in Mode and typing in **DRIVER|PASSENGER**. (Note: matching is case insensitive)



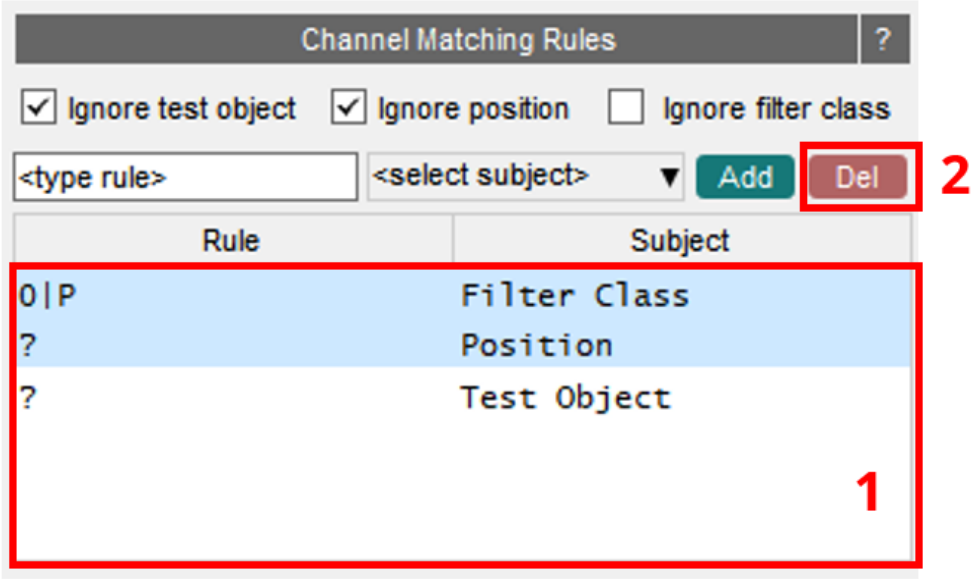
Similarly, to match channels Accel_X_Pillar_A with Accel_X_Pillar_B:

- 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.

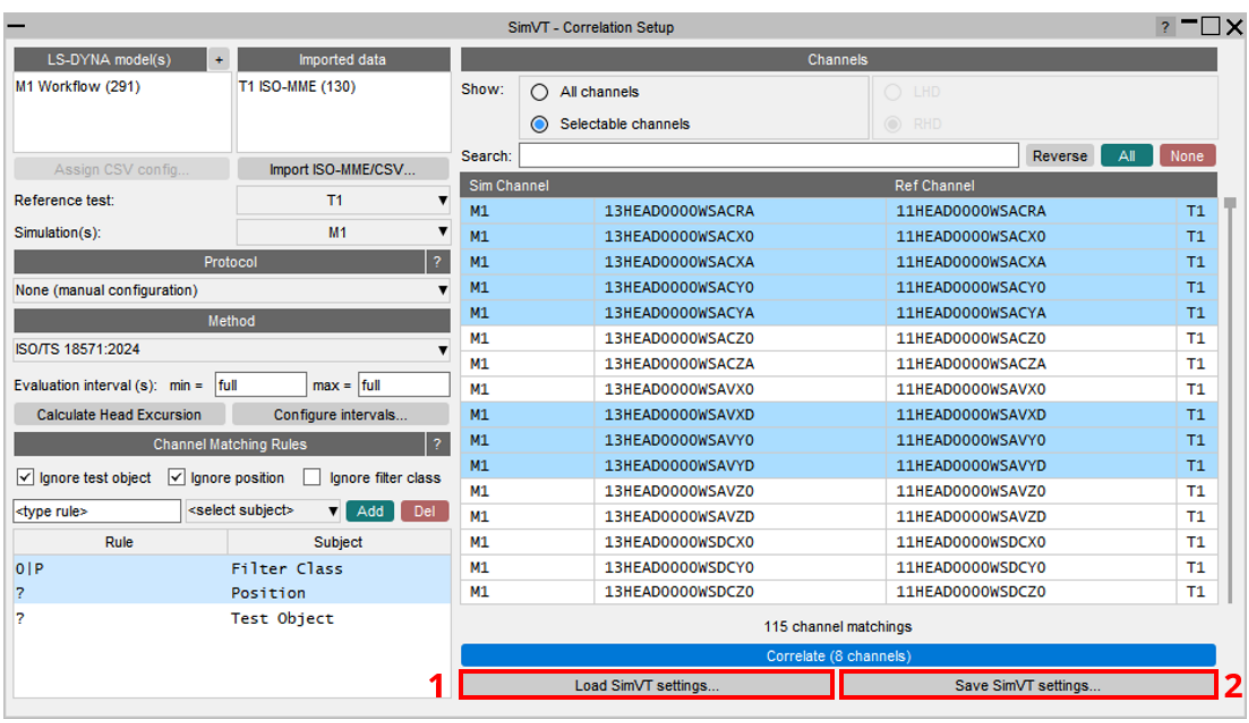
3.9.4.3.5. SimVT Settings File

Settings File

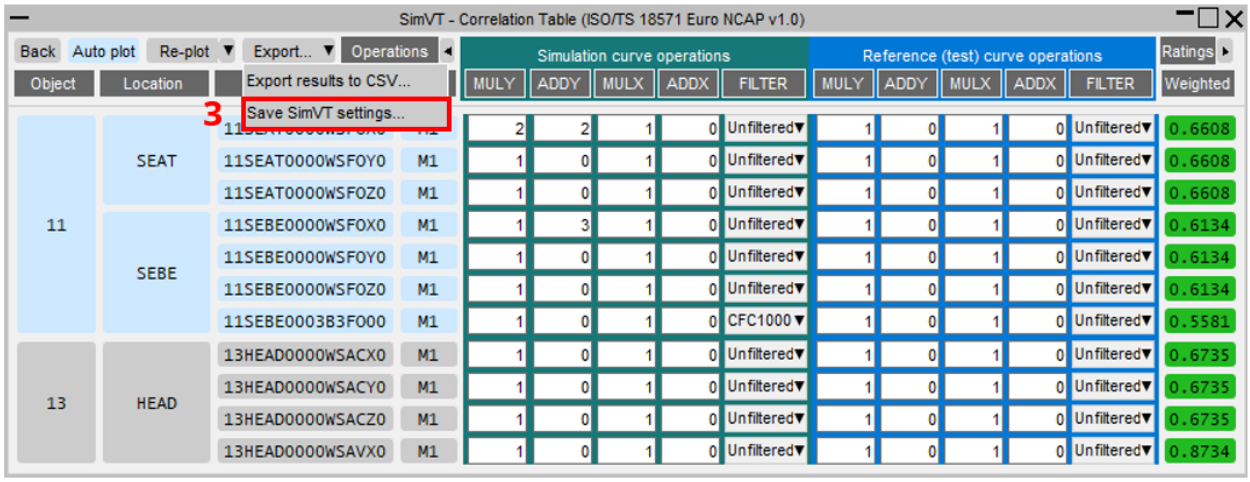
SimVT settings files are a new way for you to load and save your correlation settings from a particular SimVT session. You can load [1] and save [2] SimVT settings file from the Correlation Setup Window. A SimVT settings file can also be saved from the Correlation Table Window by going to **Export... → 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.

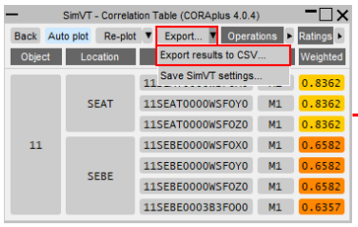


OR


SimVT - Correlation Table (ISO/TS 18571 Euro NCAP v1.0)
Back
Auto plot
Re-plot
Export...
Operations
Export results to CSV...
3
Save SimVT settings...
Simulation curve operations
Reference (test) curve operations
Ratings
Object
Location
MULY
ADDY
MULX
ADDX
FILTER
Weighted
11
SEAT
11SEAT0000WSF0Y0
M1
11SEAT0000WSF0Z0
SEBE
11SEBE0000WSF0X0
11SEBE0000WSF0Y0
11SEBE0000WSF0Z0
11SEBE0003B3F000
13
HEAD
13HEAD0000WSACX0
13HEAD0000WSACY0
13HEAD0000WSACZ0
13HEAD0000WSAVX0
Unfiltered
CFC1000
0.6608
0.6608
0.6608
0.6134
0.6134
0.6134
0.5581
0.6735
0.6735
0.6735
0.8734

3.9.4.3.6. Exporting 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 you to export your correlation results from to a CSV file. To export results, select **Export.. → Export results to csv** from the Correlation Table Window and save in a desired location. The following fields will be written:

- RUN_ID - Analysis run identifier
- 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

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 invert the sign of my simulation or test curve so that they are consistent?

You can invert 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.

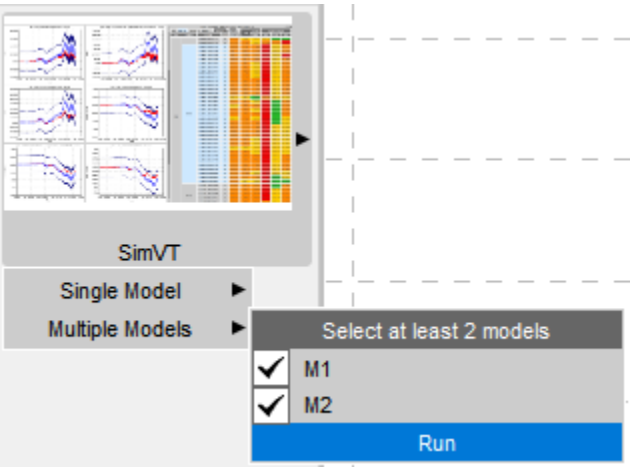
Data Sources

SimVT offers flexibility by supporting channel data from a range of different data sources:

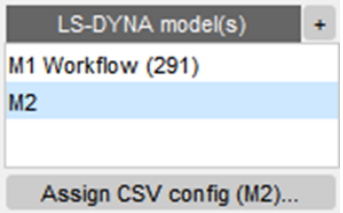
- LS-DYNA with Automotive Assessment Workflow Data
- LS-DYNA with CSV Config File
- CSV Data
- ISO-MME Data

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.

3.9.4.5.2. LS-DYNA with CSV Config File

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.

3.9.4.5.3. CSV Data

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.

3.9.5. Euro NCAP Virtual Far Side VC1 (ISO Scores)

Euro NCAP Virtual Far Side VC1 (ISO Scores)

This topic focuses on the **automation** of the Validation Criterion 1 (ISO Scores) assessment using REPORTER. You can also use SimVT to perform the correlation interactively.

Introduction

The **Euro NCAP Virtual Far Side 2024 VC1 (ISO Scores)** REPORTER Template can be used to perform the Validation Criterion 1 (ISO Scores) check according to section 6.3.5 of the Euro NCAP VTC Simulation and Assessment Protocol v1.0:

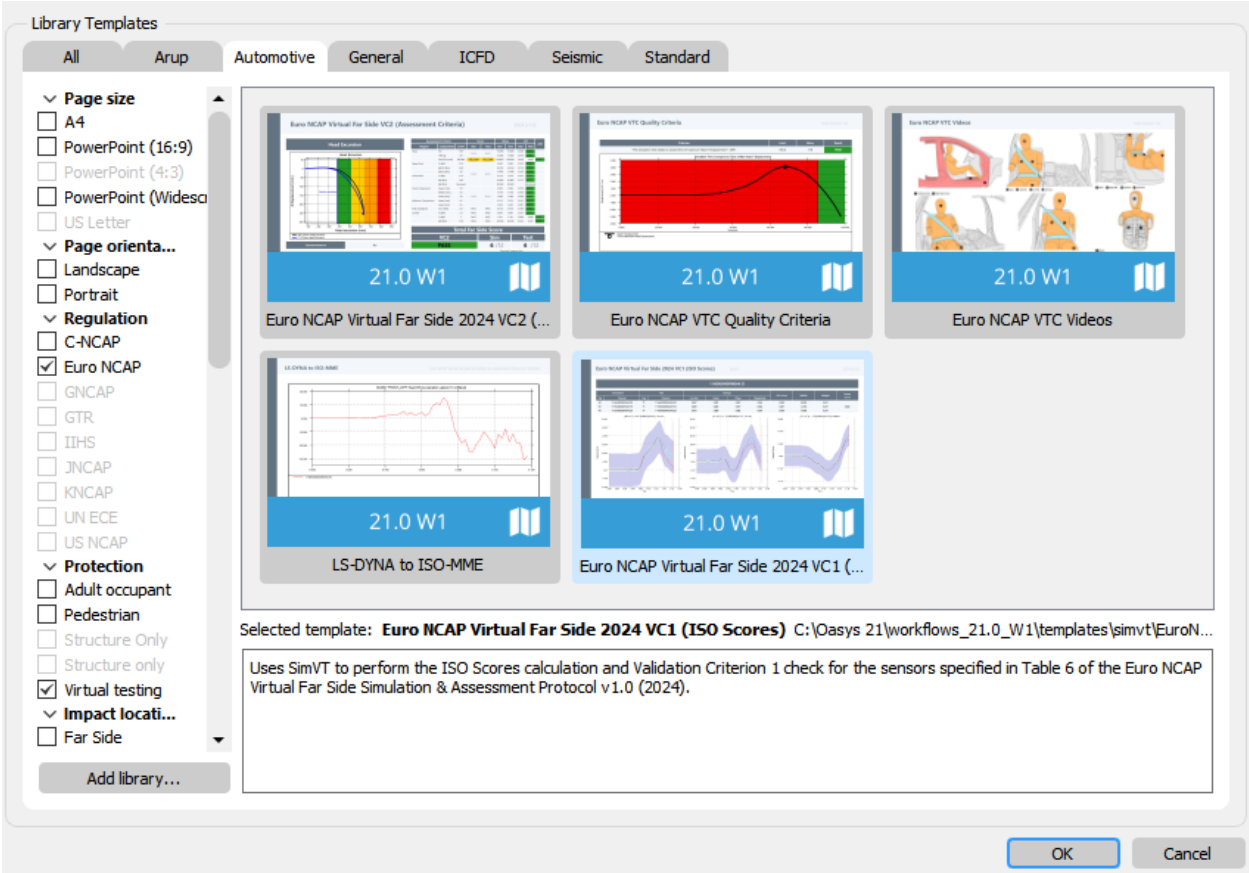
Validation Criterion 1 (ISO Scores): \(S_{Sensor} > 0.5\)

If the template is successfully generated it will show a report summarising the the sensor scores for all the mandatory and monitored channels (listed in Table 6 of the protocol) as well as the correlation graphs for each channel so that you can inspect any channels which perform poorly. Additionally, a SimVT settings file (*REPORTER_settings.simvt*) is created which can be loaded in to the SimVT workflow tool to interrogate the results interactively.

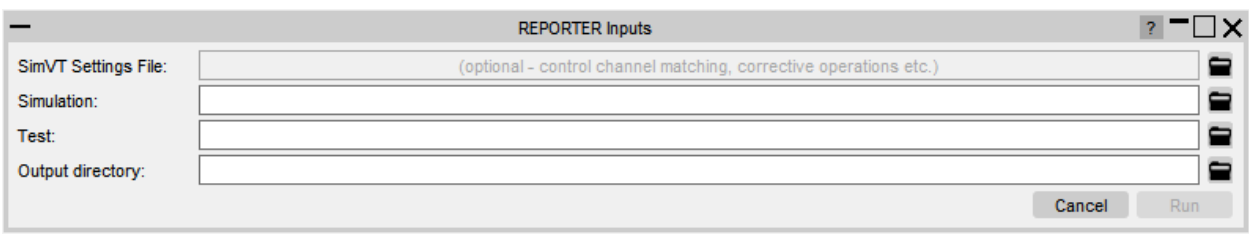
Before using the REPORTER template (either interactively or in batch), you should have defined the occupant and structures user data required for your model in PRIMER (see Automotive Assessments PRIMER for details).

Running the template interactively

In REPORTER, click the **Automotive** tab and then filter by Virtual Testing (under protection) and double-click **Euro NCAP Virtual Far Side 2024 VC1 (ISO Scores)** to open the template.



T/HIS will launch straight away and an input window will appear:



You can **optionally** select a previously-saved SimVT settings file (*.simvt) using the file explorer icon on the first row. This can be useful to define any channel matching rules and corrective operations you require. If the SimVT settings file is valid it will automatically populate the "Simulation" and "Test" file path rows.

If you select a SimVT settings file then you can **optionally change the "Simulation" and "Test" file paths to point to data sources that differ from those in the SimVT settings file**. For example, you may have a SimVT settings file which defines the channel matching rules and corrective operations to correlate *LS_DYNA_SIM_**1**.d3thdt* vs. *ISO_MME_TEST.chn*, but you want to correlate *LS_DYNA_SIM_**2**.d3thdt* vs. *ISO_MME_TEST.chn* so you could load the SimVT settings file in (which would populate the "Simulation" row

with *LS_DYNA_SIM_1.d3thdt* and the "Test" row with *ISO_MME_TEST.chn*) then change the "Simulation" path to *LS_DYNA_SIM_2.d3thdt* so that it will be used instead.

If you do not select a SimVT settings file then you must **input the file paths to the "Simulation" and "Test"** **data sources** (e.g. files ending with d3thdt, .thf, binout0000, .csv, .chn, .mmi, .iso).

Next, you need to input the "Output directory" which is where the results will be saved.

Once you are happy with your inputs, click **Run**.

REPORTER Results

Page 1 contains a summary of the results for all the assessed and monitored channels. It shows the PASS/FAIL status of Validation Criterion 1 as well as the run duration PASS/FAIL status that requires the simulation time to be greater than 1.2 times the time of maximum head excursion.

Euro NCAP Virtual Far Side 2024 VC1 (ISO Scores) 2024 (v1.0)

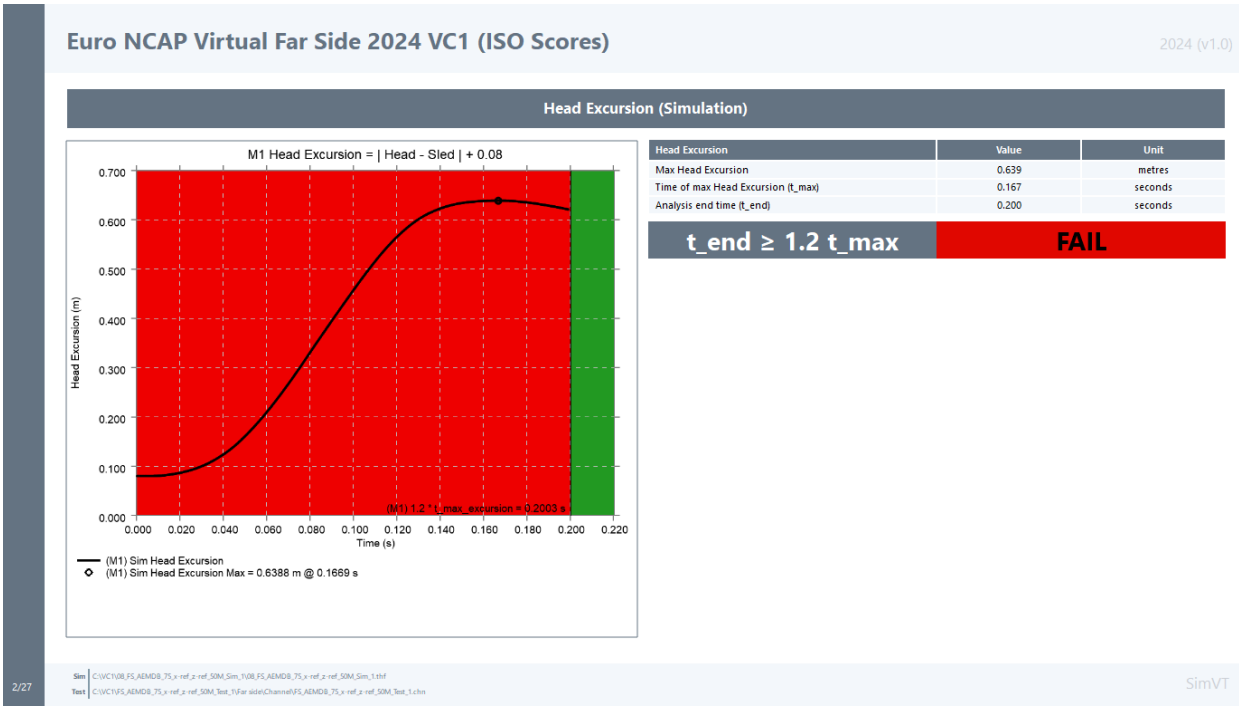
Results Summary

Sensor		1D or X Axis		Y Axis		Z Axis		Sensor Score	Mandatory in monitoring phase
Description	ISO Code	ISO Score	|Max|	ISO Score	|Max|	ISO Score	|Max|		
Head CoG Angular velocities	1_HEAD0000WSAV_D	0.866	38.252	0.887	14.355	0.956	16.842	0.892	YES
Head CoG Accelerations	1_HEAD0000WSAC_A	0.668	33.264	0.797	126.611	0.648	314.365	0.689	
Head CoG Accelerations (derived from velocities)	1_HEAD00VEWSAC_A	0.672	29.782	0.804	129.048	0.645	315.975	0.690	
Upper neck Forces	1_NECKUP00WSF0_A	0.722	164.005	0.809	582.462	0.642	1240.491	0.697	
Upper neck Moments	1_NECKUP00WSM0_B	0.779	19.259	0.707	22.694	0.813	6.978	0.751	
Lower neck Forces	1_NECKL000WSF0_A	0.704	467.050	0.693	1022.484	0.641	1285.656	0.671	
Lower neck Moments	1_NECKL000WSM0_B	0.833	129.110	0.766	61.047	0.787	6.388	0.811	
Spine – T4 Accelerations	1_THSP0400WSAC_C	0.705	69.759	0.714	165.482	0.631	124.467	0.684	YES
Spine – T12 Accelerations	1_THSP1200WSAC_C	0.767	124.022	0.747	185.241	0.615	79.222	0.726	YES
Pelvis accelerations	1_PELV0000WSAC_B	0.805	216.938	0.748	301.571	0.695	106.102	0.759	YES
Lumbar spine loadcell Forces	1_LUSP0000WSF0_B	0.653	343.551	0.618	970.252	0.713	2183.339	0.681	
Lumbar spine loadcell Moments	1_LUSP0000WSM0_B	0.713	63.431	0.699	16.936	0.765	21.252	0.721	
Shoulder joint Forces	1_SHLD__00WSF0_B	0.745	431.442	0.774	985.000	0.668	886.773	0.728	
Shoulder – rib Displacement (corrected)	1_SHRI__00WSDS0C	0.799	0.006					0.799	
Thorax - Upper rib Displacement (corrected)	1_TRRI__01WSDS0C	0.710	0.002					0.710	
Thorax - Mid rib Displacement (corrected)	1_TRRI__02WSDS0C	0.744	0.002					0.744	
Thorax - Lower rib Displacement (corrected)	1_TRRI__03WSDS0C	0.805	0.004					0.805	
Abdomen – Upper rib Displacement (corrected)	1_ABRI__01WSDS0C	0.774	0.003					0.774	
Abdomen – Lower rib Displacement (corrected)	1_ABRI__02WSDS0C	0.560	0.005					0.560	
Pubic Symphysis Loadcell Forces	1_PUBC0000WSF0YB	0.694	584.021					0.694	
B-Pillar (non-struck side) Accelerations	1_BPILL00000AC_O	0.638	73.623	0.637	210.811	0.466	88.679	0.597	YES
Lap Belt (B6) Force	1_SEBE0003B3F000	0.631	2107.680					0.631	
Shoulder Belt (B3) Force	1_SEBE0003B6F000	0.599	1826.294					0.599	YES

Validation criterion 1	PASS	t_end ≥ 1.2 t_max	FAIL

1/27 Sim C:\VC1\08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1\08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1.thf
Test C:\VC1\FS_AEMDB_75_x-ref_z-ref_50M_Test_1\Far side\Channel\FS_AEMDB_75_x-ref_z-ref_50M_Test_1.chn SimVT

Page 2 shows the trace of simulation head excursion versus time. The trace must extend in to the green zone in order for the check to pass.



Pages 3-25 show the corridor plots for each sensor. 3D sensors will have X, Y and Z plots:

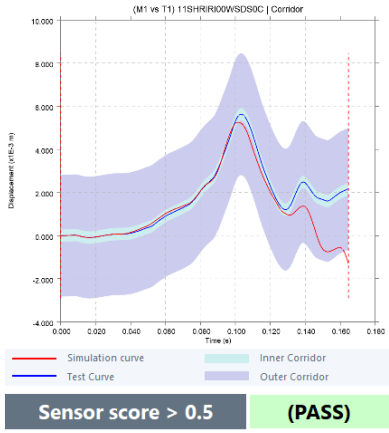
1D Sensors will have a single plot:

Euro NCAP Virtual Far Side 2024 VC1 (ISO Scores)

2024 (v1.0)

11SHRIRI00WSDS_C

Simulation		Test		Ratings				ISO Score	|MAX|	Weight	Sensor Score
Tag	Channel	Tag	Channel	Corridor	Slope	Phase	Magnitude				
M1	11SHRIRI00WSDS0C	T1	11SHRIRI00WSDS0C	0.828	0.767	0.921	0.651	0.799	0.006		0.799



15/27

Sim C:\VC1\08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1\08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1.thf
Test C:\VC1\FS_AEMDB_75_x-ref_z-ref_50M_Test_1\Far side\Channel\FS_AEMDB_75_x-ref_z-ref_50M_Test_1.chn

SimVT

Monitored sensors are shown with the PASS/FAIL status in parenthesis (e.g. "(PASS)" in the image above means that the seatbelt B3 force sensor passed the check, but the Euro NCAP protocol only requires it for monitoring purposes at the moment).

The final pages of the report contain a table showing the detailed results for each correlation so that they can all be viewed in one place.

Euro NCAP Virtual Far Side 2024 VC1 (ISO Scores)

SimVT

2024 (v1.0)

Detailed Results

Simulation		Test		Ratings				ISO Score	|MAX|	Weight	Sensor Score
Tag	Channel	Tag	Channel	Corridor	Slope	Phase	Magnitude				
M1	11HEAD0000WSAVXD	T1	11HEAD0000WSAVXD	0.857	0.822	0.857	0.934	0.866	38.252	0.551	0.892
M1	11HEAD0000WSAVYD	T1	11HEAD0000WSAVYD	0.859	0.885	0.967	0.864	0.887	14.355	0.207	0.892
M1	11HEAD0000WSAVZD	T1	11HEAD0000WSAVZD	0.973	0.896	0.982	0.954	0.956	16.842	0.243	0.892
M1	11HEAD0000WSACXA	T1	11HEAD0000WSACXA	0.714	0.353	0.860	0.698	0.668	33.264	0.070	0.689
M1	11HEAD0000WSACYA	T1	11HEAD0000WSACYA	0.891	0.406	0.921	0.875	0.797	126.611	0.267	0.689
M1	11HEAD0000WSACZA	T1	11HEAD0000WSACZA	0.699	0.437	0.812	0.596	0.648	314.365	0.663	0.689
M1	11HEAD00VEWSACXA	T1	11HEAD00VEWSACXA	0.710	0.362	0.863	0.715	0.672	29.782	0.063	0.690
M1	11HEAD00VEWSACYA	T1	11HEAD00VEWSACYA	0.896	0.416	0.918	0.891	0.804	129.048	0.272	0.690
M1	11HEAD00VEWSACZA	T1	11HEAD00VEWSACZA	0.700	0.422	0.812	0.590	0.645	315.975	0.665	0.690
M1	11NECKUP00WSFOXA	T1	11NECKUP00WSFOXA	0.760	0.385	0.924	0.781	0.722	164.005	0.083	0.697
M1	11NECKUP00WSFOYA	T1	11NECKUP00WSFOYA	0.907	0.448	0.885	0.898	0.809	582.462	0.293	0.697
M1	11NECKUP00WSFOZA	T1	11NECKUP00WSFOZA	0.697	0.444	0.812	0.558	0.642	1240.491	0.624	0.697
M1	11NECKUP00WSMOXB	T1	11NECKUP00WSMOXB	0.805	0.528	0.918	0.838	0.779	19.259	0.394	0.751
M1	11NECKUP00WSMOYB	T1	11NECKUP00WSMOYB	0.715	0.435	1.000	0.673	0.707	22.694	0.464	0.751
M1	11NECKUP00WSMOZB	T1	11NECKUP00WSMOZB	0.847	0.445	0.979	0.946	0.813	6.978	0.143	0.751
M1	11NECKL000WSFOXA	T1	11NECKL000WSFOXA	0.745	0.438	0.985	0.606	0.704	467.050	0.168	0.671
M1	11NECKL000WSFOYA	T1	11NECKL000WSFOYA	0.812	0.421	0.812	0.610	0.693	1022.484	0.368	0.671
M1	11NECKL000WSFOZA	T1	11NECKL000WSFOZA	0.698	0.405	0.794	0.609	0.641	1285.656	0.463	0.671
M1	11NECKL000WSMOXB	T1	11NECKL000WSMOXB	0.926	0.628	0.803	0.884	0.833	129.110	0.657	0.811
M1	11NECKL000WSMOYB	T1	11NECKL000WSMOYB	0.792	0.567	0.967	0.711	0.766	61.047	0.311	0.811
M1	11NECKL000WSMOZB	T1	11NECKL000WSMOZB	0.822	0.450	0.927	0.913	0.787	6.388	0.032	0.811
M1	11THSP0400WSACXC	T1	11THSP0400WSACXC	0.702	0.554	0.961	0.609	0.705	69.759	0.194	0.684
M1	11THSP0400WSACYC	T1	11THSP0400WSACYC	0.701	0.476	0.982	0.713	0.714	165.482	0.460	0.684
M1	11THSP0400WSACZC	T1	11THSP0400WSACZC	0.640	0.456	0.824	0.594	0.631	124.467	0.346	0.684
M1	11THSP1200WSACXC	T1	11THSP1200WSACXC	0.697	0.639	0.961	0.840	0.767	124.022	0.319	0.726
M1	11THSP1200WSACYC	T1	11THSP1200WSACYC	0.780	0.485	0.927	0.764	0.747	185.241	0.477	0.726

26/27

Sim C:\Users\Rory.Bradshaw\Desktop\VC1\08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1\08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1.thf
Test C:\Users\Rory.Bradshaw\Desktop\VC1\FS_AEMDB_75_x-ref_z-ref_50M_Test_1\Far side\Channel\FS_AEMDB_75_x-ref_z-ref_50M_Test_1.chn

Missing Data

If all the channel data required to calculate a given Sensor Score is found, then that Sensor Score will be displayed on the summary table (page 1) and the cell will be coloured to indicate the PASS/FAIL status. Monitored sensors are shown in lighter colours as their status does not impact the overall Validation Criterion 1 PASS/FAIL status. If channel data required to perform a correlation is missing then the corresponding results will be shown as "Missing". The different cell colours and their corresponding status are shown in the table below.

Colour	Status
0.389	Mandatory sensor failed
0.315	Monitored sensor failed
0.528	Mandatory sensor passed
0.648	Monitored sensor passed
Missing	Missing data to calculate sensor score

If any of the mandatory sensors fail to pass Validation Criterion 1 then the overall result will be FAIL. Otherwise the overall result will be a PASS (regardless of the status on monitored sensors) unless any of the mandatory sensors have missing data, in which case the Validation Criterion 1 result will show "Missing". Below is an example results table showing all the possible statuses.

Results Summary									
Sensor		1D or X Axis		Y Axis		Z Axis		Sensor Score	Mandatory in monitoring phase
Description	ISO Code	ISO Score	|Max|	ISO Score	|Max|	ISO Score	|Max|		
Head CoG Angular velocities	1_HEAD0000WSAV_D	0.215	34.108	0.508	6.374	0.650	19.777	0.389	YES
Head CoG Accelerations	1_HEAD0000WSAC_A	0.161	28.130	0.164	220.294	0.419	363.533	0.315	
Head CoG Accelerations (derived from velocities)	1_HEAD00VEWSAC_A	Missing	Missing	Missing	Missing	Missing	Missing	Missing	
Upper neck Forces	1_NECKUP00WSFO_A	0.139	196.694	0.169	642.701	0.323	646.029	0.232	
Upper neck Moments	1_NECKUP00WSMO_B	0.249	28.756	0.392	16.998	0.428	25.878	0.347	
Lower neck Forces	1_NECKL000WSFO_A	0.194	214.467	0.158	888.656	0.259	591.782	0.198	
Lower neck Moments	1_NECKL000WSMO_B	0.251	157.027	0.272	41.231	0.348	19.440	0.264	
Spine – T4 Accelerations	1_THSP0400WSAC_C	0.419	78.787	0.204	315.700	0.443	69.240	0.276	YES
Spine – T12 Accelerations	1_THSP1200WSAC_C	0.235	79.171	0.298	584.429	0.257	23.860	0.289	YES
Pelvis accelerations	1_PELV0000WSAC_B	0.345	85.166	0.213	361.673	0.339	225.130	0.272	YES
Lumbar spine loadcell Forces	1_LUSP0000WSFO_B	0.198	127.993	0.405	1693.410	0.249	2201.297	0.313	
Lumbar spine loadcell Moments	1_LUSP0000WSMO_B	0.128	58.860	0.300	30.872	0.218	10.483	0.190	
Shoulder joint Forces	1_SHLD__00WSFO_B	Missing	Missing	Missing	Missing	Missing	Missing	Missing	
Shoulder – rib Displacement (corrected)	1_SHRI__00WSDS0C	Missing	Missing					Missing	
Thorax - Upper rib Displacement (corrected)	1_TRRI__01WSDS0C	Missing	Missing					Missing	
Thorax - Mid rib Displacement (corrected)	1_TRRI__02WSDS0C	Missing	Missing					Missing	
Thorax - Lower rib Displacement (corrected)	1_TRRI__03WSDS0C	Missing	Missing					Missing	
Abdomen – Upper rib Displacement (corrected)	1_ABRI__01WSDS0C	Missing	Missing					Missing	
Abdomen – Lower rib Displacement (corrected)	1_ABRI__02WSDS0C	Missing	Missing					Missing	
Pubic Symphysis Loadcell Forces	1_PUBC0000WSFOYB	0.648	1348.730					0.648	
B-Pillar (non-struck side) Accelerations	1_BPILL00000AC_0	0.390	210.393	0.379	671.766	0.441	469.883	0.402	YES
Lap Belt (B6) Force	1_SEBE0003B3F000	0.024	404.135					0.024	
Shoulder Belt (B3) Force	1_SEBE0003B6F000	0.528	2213.210					0.528	YES
Validation criterion 1	FAIL	t_end ≥ 1.2 t_max	PASS						

Running the template in Batch

The template can also be run in batch mode, specifying the required information through command line arguments.

If you want to use the simulation and test data specified by the SimVT settings file then you only need to specify the output directory and SimVT settings file file on the command line:

<*reporter_exe*> -batch -file=<*template_name*> -varOUTPUT_DIR=<*output_directory*> -varSIMVT_SETTINGS_FILE=<simvt_settings_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 to the EuroNCAP_Virtual_Far_Side_2024_VC1_ISO_Scores.ort template: ***$OA_WORKFLOW/templates/simvt/EuroNCAP_Virtual_Far_Side_2024_VC1_ISO_Scores.ort*** Note that **$OA_WORKFLOW** should be substituted with the full path to your workflows 21.0 W1 directory. Note **$OA_WORKFLOW** is set using the *workflow_definitions_directory* preference

reporter_exe	The full path and filename to the REPORTER executable
output_directory	The directory where the correlation graphs images, results.csv and REPORTER_settings.simvt are written to (it must exist and you must have write permissions).
settings_file	The full path and filename of the SimVT settings file. The SimVT settings file argument is optional, but if it is provided it will be used to configure the channel matching rules, corrective operations and correlation method* used. *Note that the Euro NCAP protocol mandates that the default ISO/TS 18571:2024 protocol is used for correlations.

If you want to override the simulation and test data specified by the SimVT settings file then you only need to specify the extra arguments, varSIMULATION_DATA_PATH and varTEST_DATA_PATH respectively:

<*reporter_exe*> -batch -file=<*template_name*> -varOUTPUT_DIR=<*output_directory*> -varSIMVT_SETTINGS_FILE=<simvt_settings_file> **-varSIMULATION_DATA_PATH=<*simulation_file*> -varTEST_DATA_PATH=<*test_file*>** -exit

Where:

simulation_file	The full path and filename of the simulation data source. This is optional if -varSIMVT_SETTINGS_FILE is defined, but it can be used to override the simulation data source specified in the SimVT settings file.
test_file	The full path and filename of the test data source. This is optional if -varSIMVT_SETTINGS_FILE is defined, but it can be used to override the simulation data source specified in the SimVT settings file.

If you do not want to use a SimVT settings file you can remove -varSIMVT_SETTINGS_FILE provided that both varSIMULATION_DATA_PATH and varTEST_DATA_PATH are defined:

<*reporter_exe*> -batch -file=<*template_name*> -varOUTPUT_DIR=<*output_directory*> **-varSIMULATION_DATA_PATH=<*simulation_file*> -varTEST_DATA_PATH=<*test_file*>** -exit

3.9.6. Euro NCAP Virtual Far Side VC2 (Assessment Criteria)

Euro NCAP Virtual Far Side VC2 (Assessment Criteria)

This topic focuses on the **automation** of the Validation Criterion 2 (Assessment Criteria) assessment using REPORTER. You can also use Automotive Assessments to perform the correlation interactively.

Introduction

The **Euro NCAP Virtual Far Side 2024 VC2 (Assessment Criteria)** REPORTER Template can be used to perform the Validation Criterion 2 (Assessment Criteria) check according to section 6.3.10 of the Euro NCAP VTC Simulation and Assessment Protocol v1.0:

Validation Criterion 2 (Assessment Criteria): $d_{AC} < 30\%$

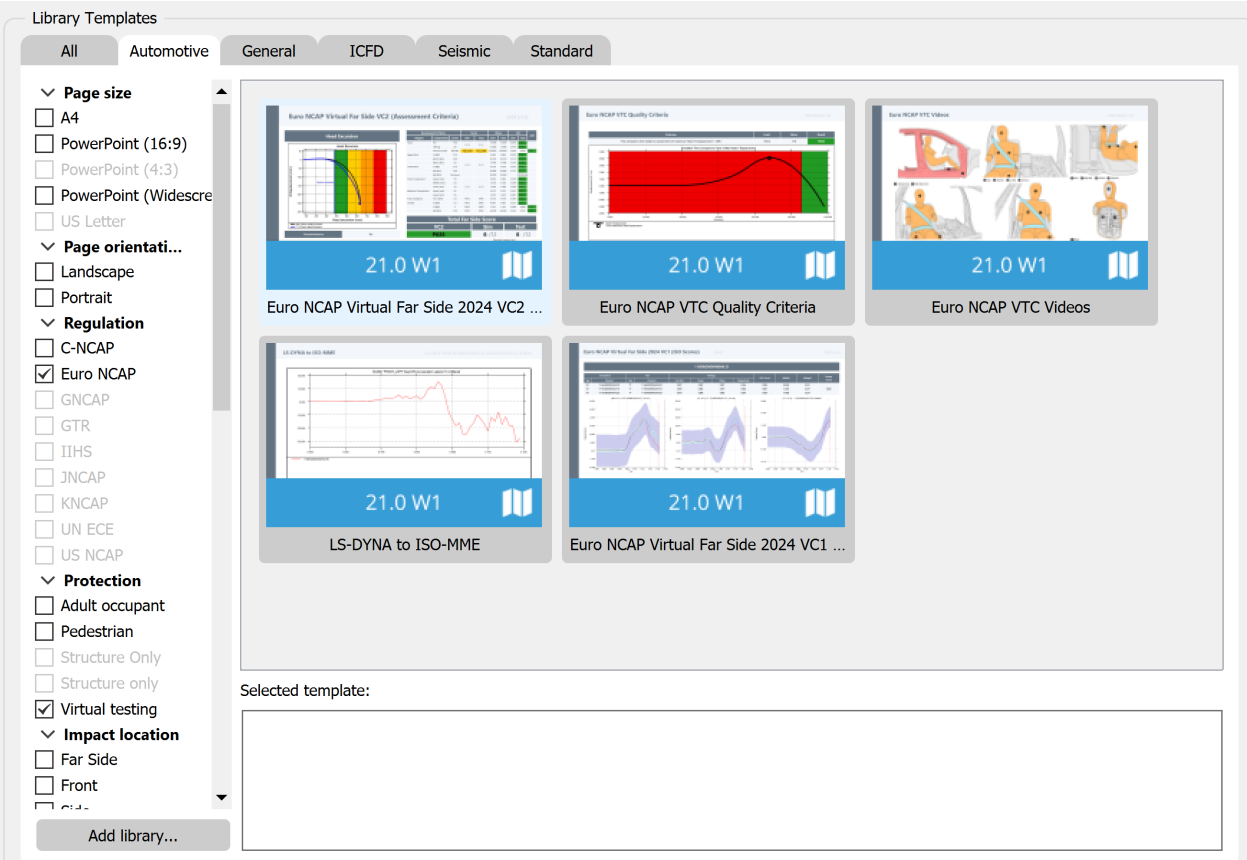
If the template is successfully generated it will show a report summarising the assessment criterion limits, values, $r_{AC_{sim}}$, $r_{AC_{test}}$ and d_{AC} as well as Far Side scores. The graphs for all the assessment data and structures are plotted to allow you to understand the results in more detail.

Firstly, in PRIMER, you should have set up the occupant and structures required for your Simulation model. This can be found in Workflows in the tools menu, then Automotive Assessments and under the Crash Test dropdown find 'Far Side + VTC' and fill out the Driver and Structures (Airbag is optional). Then save the created Workflows data using the Save to File or Save to Model buttons. For more information see the 'Automotive Assessments PRIMER' manual.

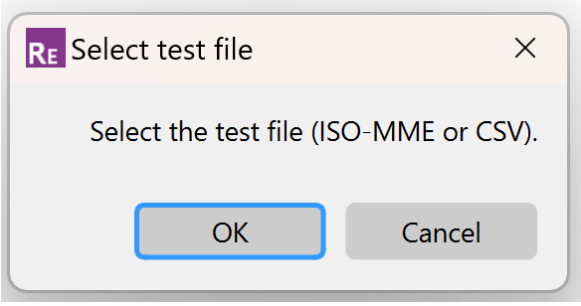
Using the REPORTER template

Before using the REPORTER template, you should have defined the occupant and structures user data required for your model in PRIMER (see Automotive Assessments PRIMER for details).

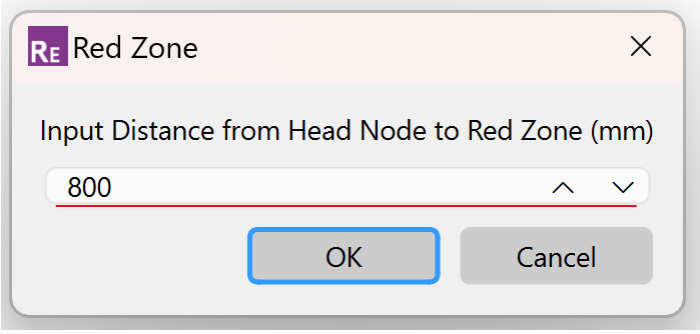
In REPORTER, click the **Automotive** tab and then filter by Virtual Testing (under protection) & Euro NCAP (under regulation) and double-click **Euro NCAP Virtual Far Side 2024 VC2 (Assessment Criteria)** to open the template.



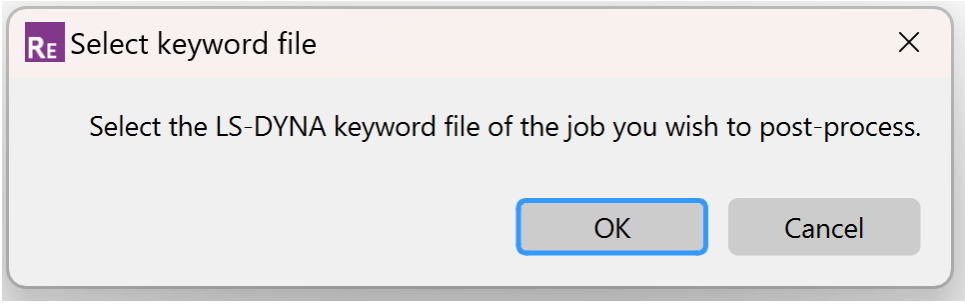
You will first be prompted to select your Test file, this can be .iso, .mme, .chn or .csv format:



Next you will be prompted for the distance between the Head CoG and the Red Line which marks the inboard intrusion from your previous Side MDB or Side Pole simulation.



Finally you will be prompted for the Simulation (.key) file. Once selected the REPORTER Template will run and produce the results.



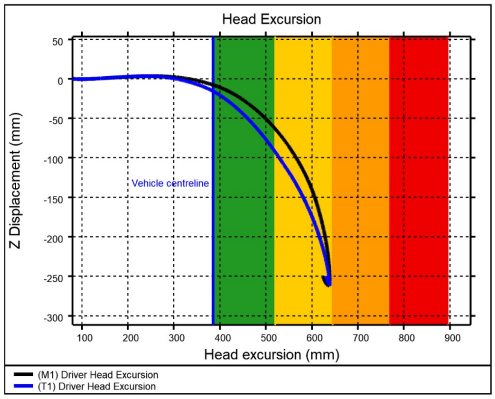
REPORTER Results

Page 1 contains the DRIVER_HEAD_EXCURSION, and all the required assessment data (Values, Far Side Scores, Assessment Criteria Limits, \(r_{AC_{sim}}\), \(r_{AC_{test}}\) and \(d_{AC}\) values). The total Validation Criterion 2 pass or fail is then displayed next to the Far Side Assessment Criteria scores out of 12.

Euro NCAP Virtual Far Side VC2 (Assessment Criteria)

2024 (v1.0)

Head Excursion



Countermeasure	No

Assessment Criteria			Score		Value		rAC		dAC
Region	Component	Limit	Sim	Test	Sim	Test	Sim	Test	
Head	HIC	700	2 / 2	2 / 2	14.296	14.296	0.020	0.020	
	TMS [g]	80			15.686	15.686	0.196	0.196	
	Excursion [mm]	690.340	YELLOW	YELLOW	638.887	638.892	0.810	0.810	0.000
Upper Neck	Fx [kN]	3.74	2 / 2	2 / 2	0.682	0.682	0.182	0.182	
	MxOC [Nm]	248			29.193	29.193	0.118	0.118	
	MyOC [Nm]	50			17.947	17.946	0.359	0.359	
Lower Neck	Fz [kN]	3.74			0.730	0.730	0.195	0.195	
	Mx [Nm]	248			92.886	92.886	0.375	0.375	
	My [Nm]	Monitored			40.560	40.560	Monitored		
Chest Compression	Upper [mm]	50	2 / 2	2 / 2	0.962	0.962	0.019	0.019	
	Middle [mm]	50			1.878	1.878	0.038	0.038	
	Lower [mm]	50			1.896	1.896	0.038	0.038	
Abdomen Compression	Upper [mm]	65			0.755	0.755	0.012	0.012	
	Lower [mm]	65			5.292	5.292	0.081	0.081	
Pubic Symphysis	Force [kN]	2.8	PASS	PASS	0.576	0.576	0.206	0.206	
Lumbar	Fy [kN]	3.5	PASS	PASS	0.901	0.901	0.257	0.257	
	Fz [kN]	2	PASS	PASS	1.361	1.361	0.680	0.680	0.000
	Mx [Nm]	120	PASS	PASS	64.236	64.236	0.535	0.535	0.000

Total Far Side Score

VC2	Sim	Test
PASS	6 /12	6 /12

*Reached Capping Limit

1/24

Sim C:/Users/harry.graham/Documents/Work/08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1/08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1.key
Test C:/Users/harry.graham/Documents/Work/08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1/isomme/Far_side/Channel/FS_Pole_75_x-ref_z-ref_50M_Sim_1.chn

Page 2 contains more in depth results for the Simulation Model and Page 3 contains the same set of results for the Test Model.

Euro NCAP Virtual Far Side VC2 (Assessment Criteria)

2024 (v1.0)

Simulation Results

Head	Value	Higher Limit	Lower Limit*	Score
Excursion [mm]	638.887			
HIC15	14.296	500	**700**	2.000
Resultant 3ms acceleration [g]	15.686	72	**80**	2.000
Head Score				2.000

*Lower performance limit in bold if it is also a capping limit

Neck	Value	Higher Limit	Lower Limit	Score
Upper tension [kN]	0.682		3.74	2.000
Upper lateral flexion [Nm]	29.193	162	248	2.000
Upper extension negative [Nm]	17.947		50	2.000
Lower tension [kN]	0.730		3.74	2.000
Lower lateral flexion [Nm]	92.886	162	248	2.000
Lower extension negative [Nm]	40.560		100	2.000
Neck score				2

Chest and Abdomen	Value	Higher Limit	Lower Limit*	Score
Chest lateral compression [mm]	1.896	28	**50**	2.000
Abdomen lateral compression [mm]	5.292	47	**65**	2.000
Chest and abdomen score				2.000

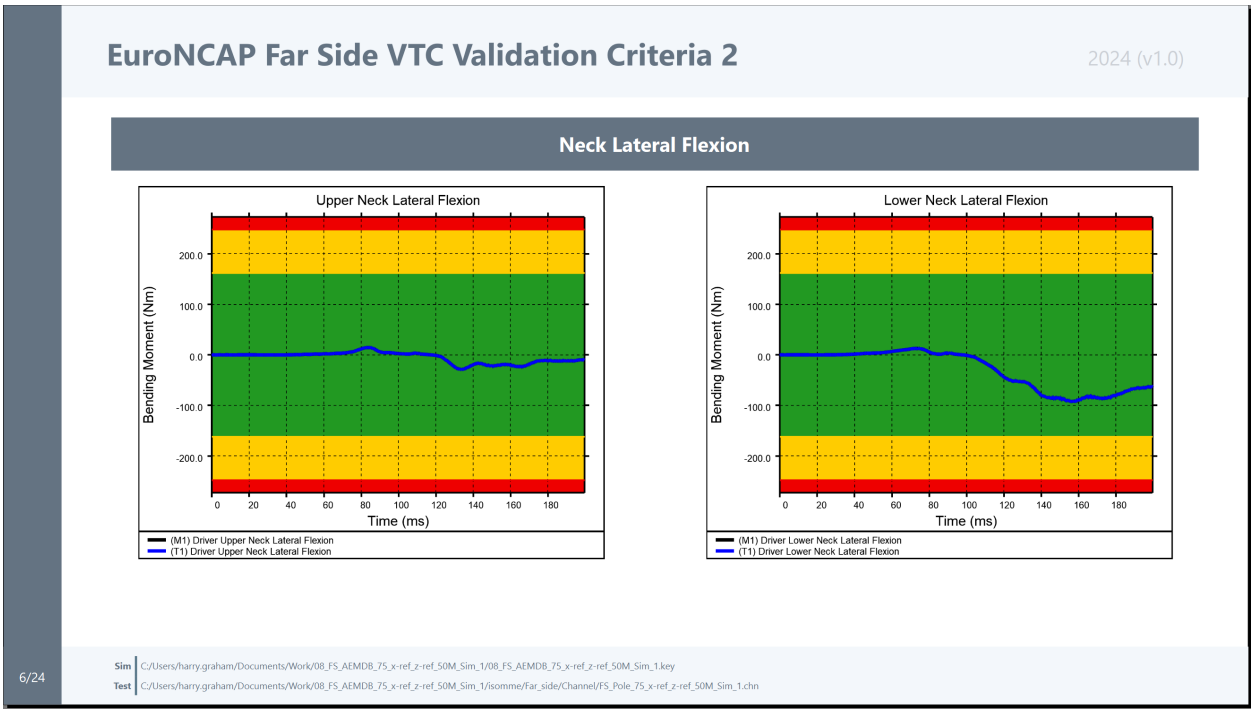
*Lower performance limit in bold if it is also a capping limit

Pelvis and Lumbar	Value	Limit	Performance
Pubic symphysis [kN]	0.576	2.8	PASS
Lumbar Fy [kN]	0.901	2.0	PASS
Lumbar Fz [kN]	1.361	3.5	PASS
Lumbar Mx [Nm]	64.236	120	PASS
Modifier		0	

2/24

Sim C:/Users/harry.graham/Documents/Work/08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1/08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1.key
Test C:/Users/harry.graham/Documents/Work/08_FS_AEMDB_75_x-ref_z-ref_50M_Sim_1/isomme/Far_side/Channel/FS_Pole_75_x-ref_z-ref_50M_Sim_1.chn

Pages 4 to 10 display the graphs for the required assessment data and pages 11 to 24 display the structure data from Table 6 in the Euro NCAP VTC protocol.



Running in Batch

The template 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 and the test file and red zone which are specific to this template:

`<reporter_exe> -batch -file=<template_name> -varKEYWORD_FILE=<keyword_file> -varTEST_FILE=<test_file> -varRED_ZONE=<red_zone> -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
test_file	The full path to the ISO-MME or CSV file to be used for the Test model

reporter_exe	The full path and filename to the REPORTER executable
red_zone	Distance between the Head CoG and the Red Line

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> -varTEST_FILE=<test_file> -varRED_ZONE=<red_zone> -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> -varTEST_FILE=<test_file> -varRED_ZONE=<red_zone> -varOUTPUT_DIR=<output_dir> -exit
```

Where:

output_dir	The full path to the output directory

4. Changelog

This section was imported from CHANGELOG.zip.

This section contains:

- Oasys 21.0 W1 (Workflows Update 1)

4.1. Oasys 21.0 W1 (Workflows Update 1)

Oasys 21.0 W1 (Workflows Update 1)

This section lists all the changes made to **Virtual Testing** Workflows since the Oasys 21.0 release.

Automotive Assessments

Enhancements

1. All the Structure channels which take PARTs as input in the Far Side + VTC crash test now have a single entity selection textbox. This means you only have to select entity IDs once for such channels and eliminates the need to perform the same repetitive selection multiple times.
2. Graphs for failed occupant assessments (e.g. missing channel data) are now displayed with a helpful title explaining that there was no output for that particular assessment.
3. Structure assessments can now be plotted for imported data (e.g. ISO-MME or CSV). The structures are derived from ISO-MME channel data (by matching channel codes to Structure names).
4. When clicking EDIT for a entity textbox that contains a list of parts, the Edit window will appear for each part successively after clicking Yes on a window that asks if you want to keep editing (previously only the first part was shown).
5. Removed restriction of suppliers to just the supported list: ATD-MODELS, DYNAmore, Humanetics, LSTC, PDB. Users can now define their own custom supplier name.
6. Added a new Euro NCAP Virtual Far Side VC2 (Assessment Criteria) REPORTER template for Validation Criterion 2.
7. Added Head Excursion assessment type for Far Side + VTC crash test.
8. Added support for the Validation Criterion 2 ratio calculation and included the results in the T/HIS results table.

Bug fixes

1. Made Clutch pedal an optional input for Automotive Assessment REPORTER templates. You are no longer prompted for missing Clutch Pedal input data.
2. Fixed an issue where selecting an entity with a comma in the title would be incorrectly considered invalid as commas were assumed to only be in part lists.

3. Fixed an issue where Occupant assessment graphs with no curves were deleted which meant that all subsequent assessments failed as they referenced a non-existent graph ID.
4. Fixed an issue where CSV data could not be imported.
5. In PRIMER, if an entity textbox contained a heading (or database history title) string that started with a number, the corresponding entity ID was incorrectly taken as the leading number characters which would lead to the incorrect entity being edited/sketched or a "does not exist" warning. Now fixed.
6. Fixed an issue where a capping limit asterisk was incorrectly being appended to the Euro NCAP MDB Shoulder Lateral Force assessment score when the limit had not been exceeded.

Euro NCAP VTC Quality Criteria

Enhancements

1. Results generation is now significantly faster.
2. Entity selection improved to match Automotive Assessment methods and allows DATABASE_HISTORY headings as well as IDs.
3. The PRIMER GUI is now automatically pre-populated with any data found from Automotive Assessments.
4. For 10% Max Internal vs Hourglass curves, the Internal Energy curve is removed from the graph (if test is a pass) so that the critical Hourglass Energy curve is shown in more detail.

Euro NCAP VTC Videos

Enhancements

1. Entity selection improved to match Automotive Assessment methods and allows DATABASE_HISTORY headings as well as IDs.
2. The PRIMER GUI is now automatically pre-populated with any data found from Automotive Assessments.
3. The three Shift Deform reference nodes have been changed to one Fixed reference node.
4. An Output Interval option has been added in PRIMER that sets the *DATABASE_BINARY_D3PLOT DT field to ensure output interval is 2 ms or less.
5. Added simulation start, interval and end time options to give more control over video export.
6. File size visibility added to the REPORTER Template so you don't have to search through your file system to check.

7. Euro NCAP Cut Section view now uses true thickness rather than fixed fixness.
8. New option added in PRIMER to allow you to blank parts such as the windscreen during video export.

Bug fixes

1. The Cut Section view is now normal to the x-axis.

LS-DYNA to ISO-MME

Enhancements

1. When writing out user data, the default Required output channel csv path is now written out using $OA_WORKFLOW to make the user data more portable across different machines.

SimVT

Enhancements

1. The new, recently published ISO/TS 18571:2024 method replaces ISO/TS 18571:Euro NCAP v1.0.
2. The Correlation Setup window layout has been changed to landscape with a larger, clearer and more informative Channels Table.
3. Evaluation intervals can be controlled individually for each Sim vs Test model pair.
4. A Weight column has been added to Correlation Table window.
5. Channel selection from search improved. e.g. search for HEAD, select all, search for LUSP, select all.
6. Reverse button added to channel selection.
7. Evaluation interval from Head Excursion calculation is automatically applied to relevant Sim vs Test model pair.
8. Added support for multiple channel matching rules for same subject.
9. Added a derived channels counter to model hover text.
10. Updated the Protocol explainer text.
11. Improved the Channel Table update time.
12. Added ability to load new LS-DYNA models in to T/HIS from SimVT
13. Added functionality that enables SimVT to check if a new LS-DYNA model has associated Automotive Assessments Workflow Data (AAWD) and to assign AAWD to the new model.
14. Added checkboxes to the Model Mapping window so that only the selected rows will be loaded from the SimVT settings file.

15. Added labels to the Model Mapping window to make it clearer which entries correspond to "Reference (test)" and "Simulation(s)".
16. The Protocol option now defaults to "Euro NCAP Virtual Far Side v1.0" when SimVT is loaded as most users are expected to use SimVT for the Euro NCAP Far Side Validation Criterion 1 check.

Bug fixes

1. The "Save SimVT settings" button is now disabled when no channels are selected, removing the possibility of saving invalid SimVT settings files.
2. Fixed an issue where Protocol change did not update model tag channels counter.
3. Fixed an issue where evaluation intervals saved to a SimVT settings file were not validated during reload.
4. Fixed an issue where ISO-MME or CSV data file was interpreted as an LS-DYNA model path if it was in the same directory as an existing Model in T/HIS.
5. The number of channels shown in brackets after the "LS-DYNA model" and "Imported Data" sources was not updated correctly when the Protocol option was selected. This has been fixed and more information is shown in data source hover text too.

T/HIS CORA

Enhancements

1. The new, recently published ISO/TS 18571:2024 method replaces ISO/TS 18571:Euro NCAP v1.0.
2. Updated the T/HIS CORA tutorial to reflect the latest changes.

Bug fixes

1. Fixed an issue where CSV export would omit some column titles.