USER MANUAL
REPORT AUTOMATION
V 1.01

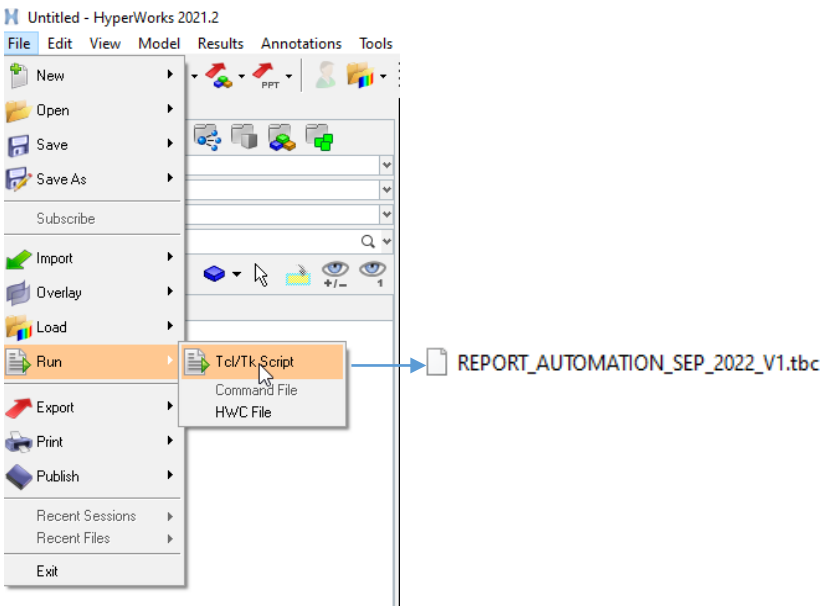
September - 2022

1.0.0 Launching the Automation.

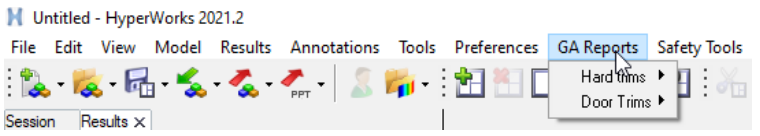
1.0.1 Launching the TBC file.

User can click the menu bar on file ->Run->Tcl/Tk script . At this point an explorer window shall pop up. Please locate the tbc file at The location you have placed it. The tbc file may be kept at any location. But please avoid network locations.
There are two options to launch the script. Option 1 and 2. With Option 2 , User can launch the automation directly from menu bar On Option GA Reports (Currently Option 2 is not activated).

Option 1



Option 2



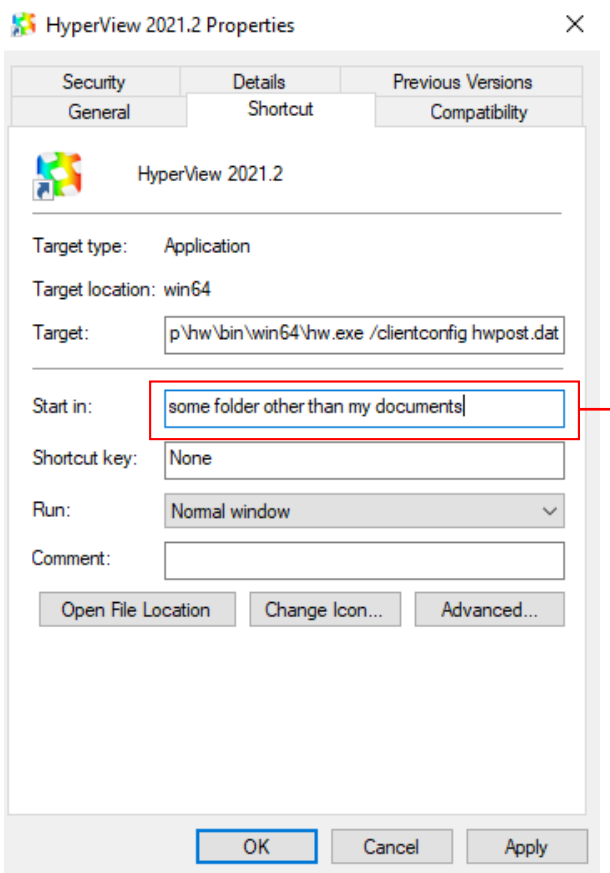
1.0.2 Status of Hyper view Session.

A New Hyper view window is mandatory to run the Automation. Clearing the Data on an existing window with File ->New->Session to launch the automation is not advised . Please do open a new hyper view window to launch the automation.

1.0.3 GA Hyper works installation issue.

Please create a shortcut for your Hyper works applications on desktop. For Hyper view, Hyper mesh etc. . If your start in dir ectory is set to my documents or one drive . This will cause the applications to slow down ,due to upload process happening every time any changes Are applied to applications. It is a general guideline ,This is not related to automations but if not done it would eventuall y slow down automations and general hyper works performance.

Right click on the shortcut icon of Hypermesh / Hyperview.



Please change this path to some location i

1.1.0 User Interface of Automation.

Project,Product, solver and Unit selector area

Files, Data Extract,Format and Non Plot image selector area

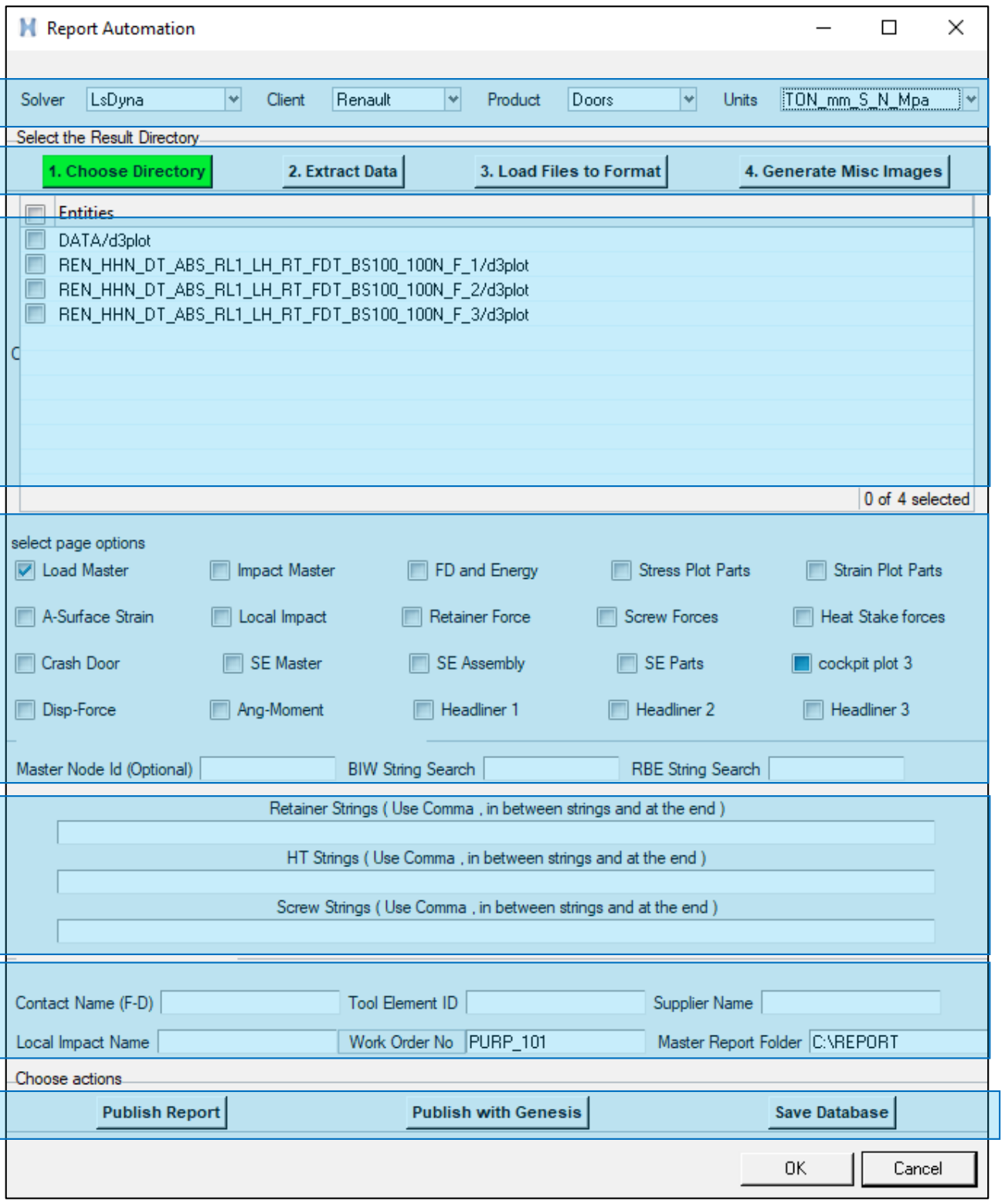
Files selector area.

Format selection Area

Deviation entry ,Additional connector data input area

Work Order, Report location and deviation area

Publish area



1.1.1 User Interface and Automation is written in TCL/TK 8.3.

The user interface and automation is written in TCL/TK 8.3.2 . It is written first with TCL version 8.5.9 and is later back ported to 8.3 to meet the encryption Requirements. The TCL files are converted to bytecode format. As direct encryption is not possible , A p ractical approach that's possible is deployed. Converting back Bytecodes to tcl files is extremely difficult if not impossible.
TCLPro is available for free download at https://www.tcl.tk/software/tclpro/eval/

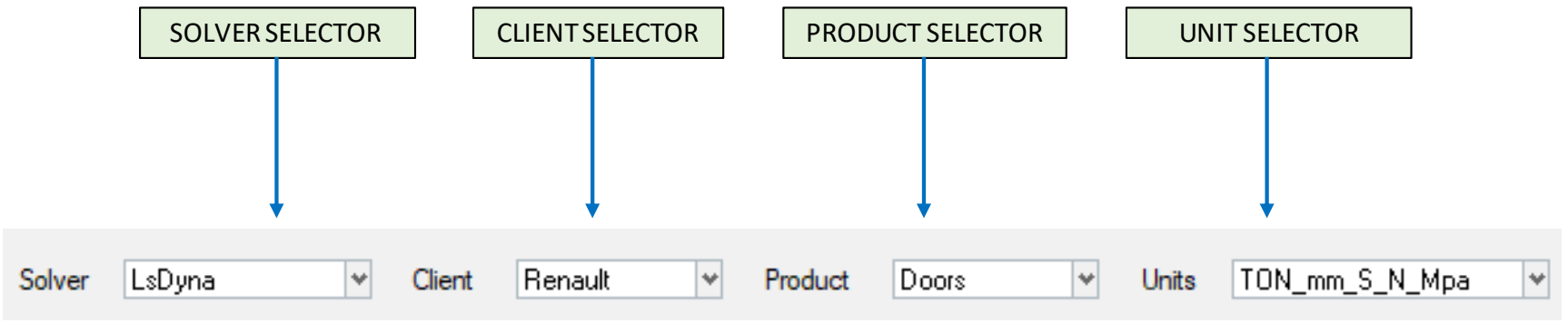
1.1.2 User Interface compatibility.

The user interface and automation is compatible with Altair 2021 and Altair 2022 versions. Please make sure you are using the correct version of Hyperworks to launch the automation. It is also compatible with the new hyperworks interface , however at present extensive testing on the new Hyperworks interface is not Carried out. Should you find any issues – Please contact the methodology department.

1.1.3 System Requirements.

The Automation currently supports windows 10 and 11. Linux support is unavailable for release1.
Recommended hardware – 32 GB RAM- 3200+ MHZ , intel 8th gen core i7, NVidia Quadro M series GPU, and SSD storage.
(The automations should perform reasonably well with similar AMD hardware)
The automations should work in ancient systems but the performance will be severely limited and could crash the system when m ultiple Result files are loaded.

1.2.0 Project ,Product , solver and Unit selector area



1.2.1 Solver Selector.
Select the appropriate solver from the drop down list. Currently supported solvers are Abaqus implicit , Abaqus Explicit , Pamcrash and LsDyna. There are many dependencies associated with this selection. So please make correct selections.

1.2.2 Client Selector.
Select the appropriate solver from the drop down list. If certain client seems to be unavailable in the list, Please select A ny.

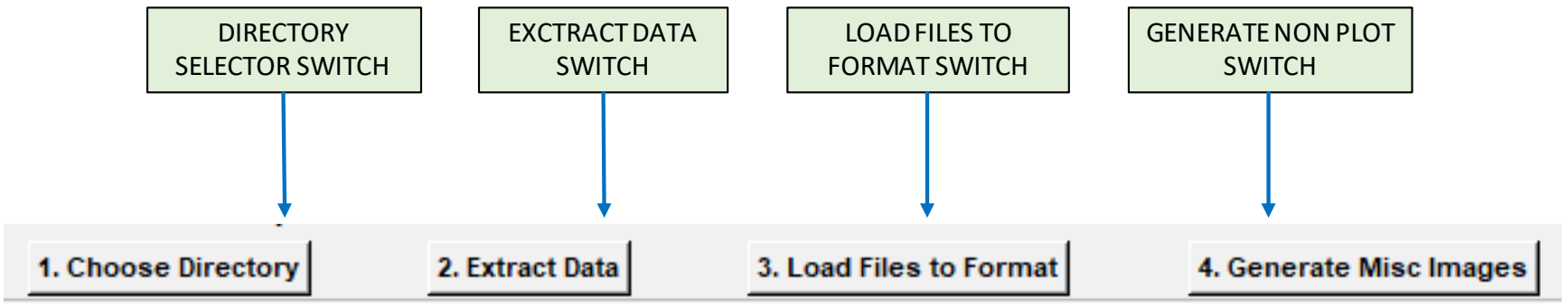
1.2.3 Product Selector.
Select the appropriate Product choice from the drop down list. Currently supported Products are Doors and Hard trims. However if Generic page formats are chosen , There shouldn't be any issues running the automation for any product.
An Example is – If user wish to generate stress plots of independent parts for a cockpit , Please select those page options in t he user interface of the automation.

1.2.4 Unit System Selector.
Select the appropriate Unit System from the drop down list. Currently supported Unit systems are Ton-mm-S-N-Mpa and Kg-mm-Ms-KN-GPa. If you need another unit system please inform the methodology department in advance , at the start of the project . If supp ort for unit system is not added, It is possible to use the replace all tool in PowerPoint to Swap a particular unit with the one of your choice.

Unit system reference table

MASS	LENGTH	TIME	FORCE	STRESS	ENERGY	DENSITY	YOUNG's	GRAVITY
kg	m	s	N	Pa	J	7.83E+03	2.07E+11	9.806
kg	cm	s	1.0e-02 N			7.83E-03	2.07E+09	9.81E+02
kg	cm	ms	1.0e+04 N			7.83E-03	2.07E+03	9.81E-04
kg	cm	us	1.0e+10 N			7.83E-03	2.07E-03	9.81E-10
kg	mm	ms	kN	GPa	kN-mm	7.83E-06	2.07E+02	9.81E-03
g	cm	s	dyne	dyne/cm²	erg	7.83E+00	2.07E+12	9.81E+02
g	cm	us	1.0e+07 N	Mbar	1.0e+07 Ncm	7.83E+00	2.07E+00	9.81E-10
g	mm	s	1.0e-06 N	Pa		7.83E-03	2.07E+11	9.81E+03
g	mm	ms	N	MPa	N-mm	7.83E-03	2.07E+05	9.81E-03
ton	mm	s	N	MPa	N-mm	7.83E-09	2.07E+05	9.81E+03
kgf-s²/mm	mm	s	kgf	kgf/mm²	kgf-mm	7.98E-10	2.11E+04	9.81E+03
kg	mm	s	mN	1.0e+03 Pa		7.83E-06	2.07E+08	9.81E+03
g	cm	ms	1.0e+1 N			7.83E+00	2.07E+06	9.81E-04

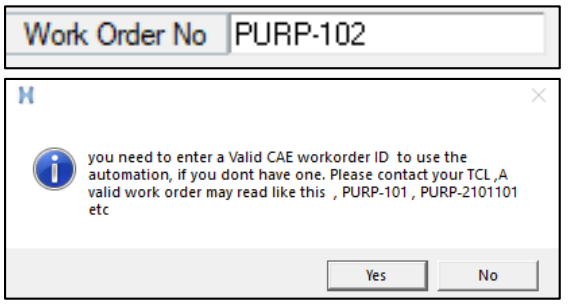
1.3.0 Files selector, Data selector , load files and Generate Non Plot switch.

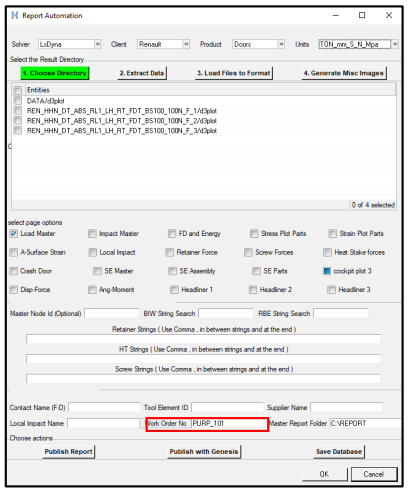


1.3.1 Choose Directory.

Select the folder where you have kept the result files and input files with this switch. Always keep the files in SSD. Please do not choose the folders on network or hard disk . This will result in a very slow loading process. This has nothing to do with the automation. It's a general guideline.

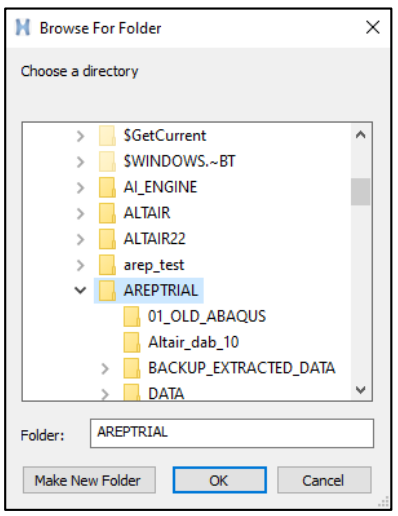
In order to load the files , Entry of a valid Work order number is mandatory. If you Don't have a work order number, Please contact the TCL. You can enter a work order ID that you will create later as well. But please do make sure that the work order entered is valid. This gets written to the log files. And during tracking process it shouldn't happen that the work order id is invalid.

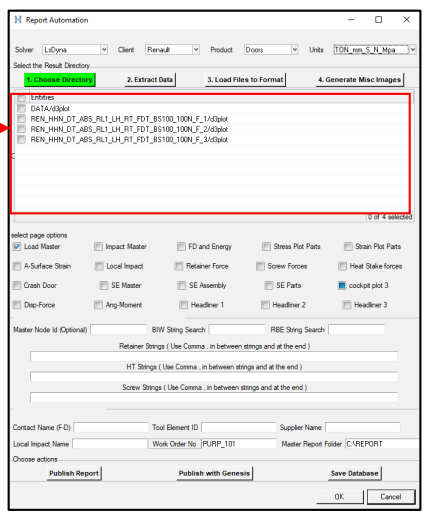




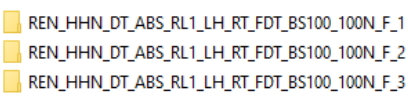
1.3.2 What to select.

Once you have entered a valid work order number the switch will be active . Click on the switch and Navigate to the folder where you have the results and click On it . Now the result files should appear at files selector area.
User can choose the report files that they intend to make report with . For LS-Dyna the automation shall publish the folder names . User need to select the folders that they need to Load for Ls-Dyna.

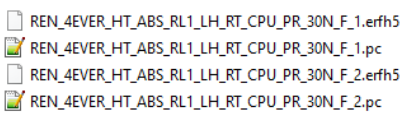




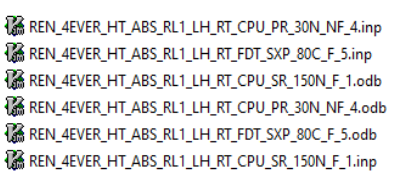
Example for Dyna Results.
Each folder has Dyna input and Result files.
So please select the folder once step above this.



Example for PamCrash Results.
The master folder has PC files and erfh5 files.
Please note:-DSY and THP files are not supported.



Example for Abaqus Results.
The master folder has inp and ODB files.



1.3.3 Mandatory Requirements.

It is mandatory to have the corresponding input files master present at the result folder. You may have your own include structure formats. But please make sure that the master input files can read all the include files if opened in hyper mesh through a normal procedure.

An example of Pamcrash include format, to read the include files one folder up the structure is given below.

INCLU / ../INCLUDE1/

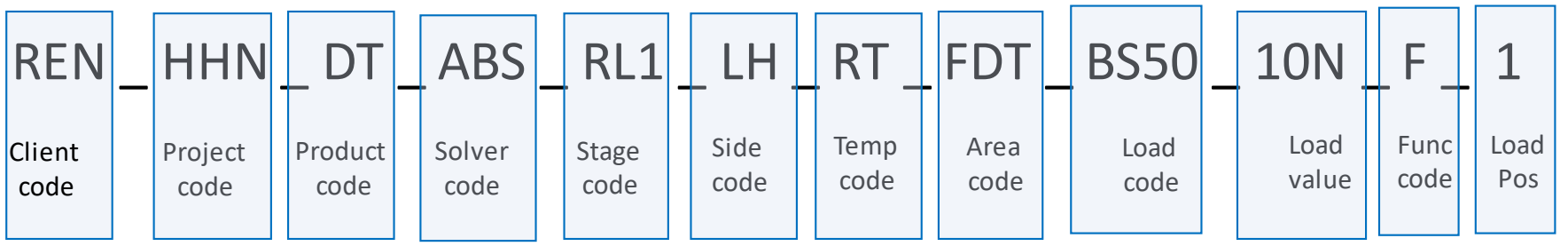
```
INCLU / ../INCLUDE1/material_compos_2019_composite_____________________00274_6850214f.inc
INCLU / ../INCLUDE1/material_db_____2019_connection_____________________00274_548514dd.inc
```

1.4.0 File and Folder Naming.

1.4.1 Naming Terminology.

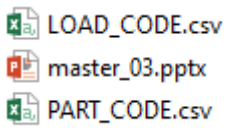
The naming terminology is explained below. It is very similar to the existing methodology . It is standardized little deeper.
In case the supplier or engineer did not follow the naming protocols, The output files and input master files can be renamed to reflect Methodology before automation use.

REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS50_10N_F_1



1.4.2 Report Folder and master PowerPoint file.

You will find at every installation location ,The below given files. The master file is a Power point file of Grupo Antolin report template. You can use any PowerPoint file instead. An example is you can rename a file to master_03 and keep at C:\REPORT folder. The automation will append the slides to this master file. But it is recommended that the given file is used to avoid technical issues. Methodology will not provide support for PowerPoint master files you have chosen.



1.4.3 Load Code and Part code CSV files.

It will be responsibility of the CAE department to update and maintain these files. The CAE department can choose the Client , product and area code of their choice. Make sure there are no spaces in between words . Use an underscore wherever you would like to separate words .

The area code file and Part code files are to be in CSV format. Please do check the CSV file in a text editor also once you h ave done updating with Microsoft Excel or a similar tool. Please make sure there are no spaces in the file. This can happen due to dif ferences in encoding.

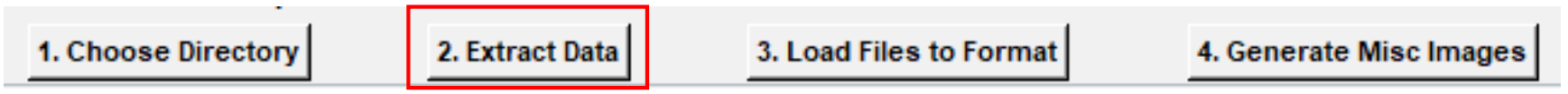
Part Code File

CLIENT	PRODUCT	CODE	AREA
REN	Hard_Trim	BDT	Back_Door_Trim
REN	Hard_Trim	BDU	Back_Door_Upper_Trim
REN	Hard_Trim	FKP	Front_Kicking_Plate
REN	Hard_Trim	ALU	A_Pillar_Upper
REN	Hard_Trim	APL	A_Pillar_Lower
REN	Hard_Trim	BPU	B_Pillar_Upper
REN	Hard_Trim	BPL	B_Pillar_Lower
REN	Hard_Trim	CPU	C_Pillar_Upper
REN	Hard_Trim	CPL	C_Pillar_Lower
REN	Hard_Trim	FKP	Front_Kicking_Plate
REN	Hard_Trim	RKP	Rear_Kicking_Plate
REN	Hard_Trim	TKP	Trunk_Kicking_Plate
REN	Hard_Trim	BPR	B_Pillar_Ramp
REN	Hard_Trim	CPR	C_Pillar_Ramp
REN	Door_Trim	FDT	Front_Door_Trim
REN	Door_Trim	RDT	Rear_Door_Trim
REN	Door_Trim	MPT	Map_Pocket
REN	Door_Trim	BLT	Belt_Line
REN	Door_Trim	PCP	PullCup
REN	Door_Trim	HT	Hard_Trim
REN	Door_Trim	DT	Door_trim

Area Code File

CLIENT	CODE	AREA	LIMIT	FORCE	TOOL_DESCRIPTION
REN	PR30	Percieved_Rigidity	1	30	30mm_Disk
REN	PR50	Percieved_Rigidity	1	50	30mm_Disk
REN	PR	Percieved_Rigidity	1	50	30mm_Disk
REN	BS100	Bending_Strength	1	180	12mm_Disk
REN	BS180	Bending_Strength	1	180	12mm_Disk
REN	BS500	Bending_Strength	1	500	50mm_Disk
REN	BS800	Bending_Strength	1	800	40mmX80mm_Plate
REN	BS1000	Bending_Strength	1	1000	40mmX80mm_Plate
REN	BS	Buckling_Strength	1	1000	40mmX80mm_Plate
REN	SR	Slipping_Resistance	1	150	30mm_Disk
REN	ARS	ArmRest_Stiffness	1	500	40mmX80mm_Plate
REN	GHT	GrabHandle_Torsion	1	10	Cylinder
REN	THM	TibHandle_MisUse	1	250	Cylinder
REN	THF	Thorax_Impact_Frontal	1	5	Dummy Form
REN	THR	Thorax_Impact_Rearward	1	5	Dummy Form
REN	ABF	Abdomen_Impact_Frontal	1	5	Dummy Form
REN	ABR	Abdomen_Impact_Rearward	1	5	Dummy Form
REN	PLF	Pelvis_Impact_Frontal	1	5	Dummy Form
REN	PLR	Pelvis_Impact_Rearward	1	5	Dummy Form
REN	SHF	Shoulder_Impact_Frontal	1	5	Dummy Form
REN	SHR	Shoulder_Impact_Rearward	1	5	Dummy Form
REN	SXP	Sun_Exposure	3	80C	Thermal_Load

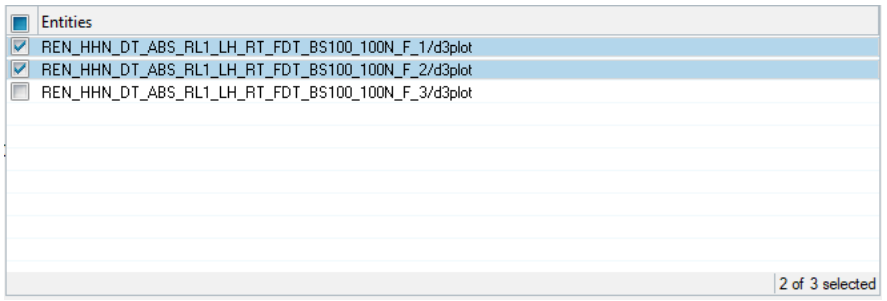
1.5.0 Extract Data.



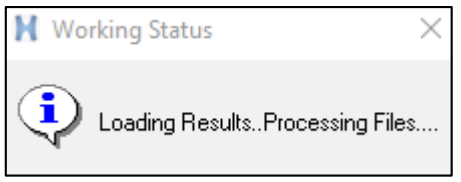
1.5.1 Extract Data.

Extract Data button collects information useful to the engineer and for the subsequent operation of the automation.
Once user has selected the files of interest – Please click this button to generate data. This operation creates a folder named DATA
In the folder you have kept the result and input files. Please make sure you don't already have a file named DATA in the folder.
If you are running a second round of report generation in a folder, Please delete the DATA folder or rename it to something else.

File Selector Area



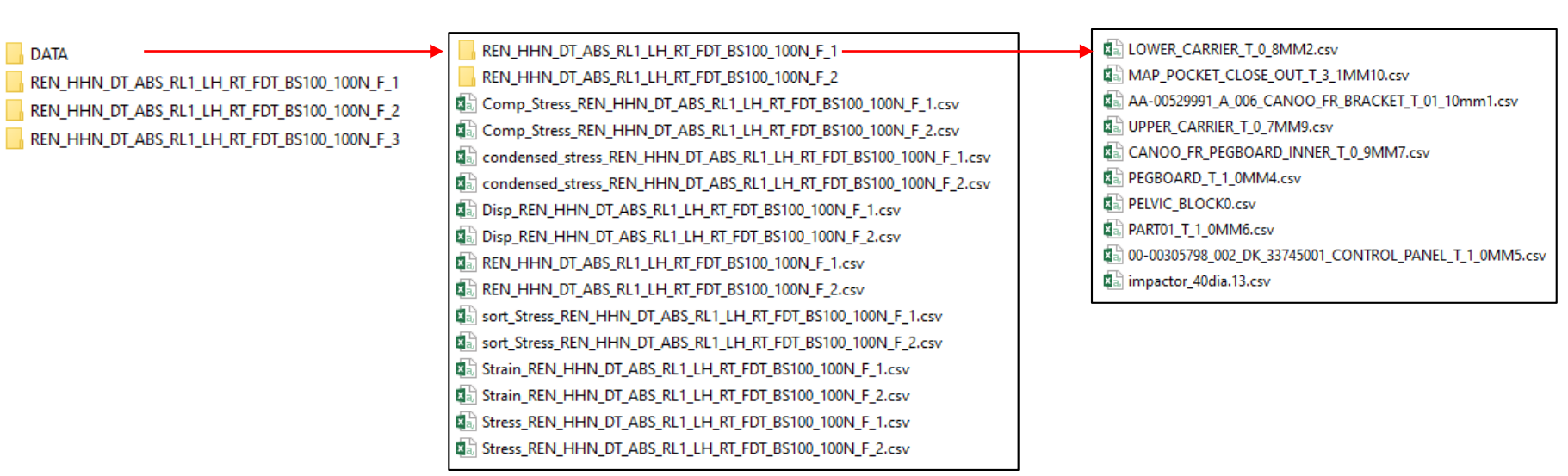
Process status Popup



Proceed to next step, Only after popup window closes

1.5.2 Generated Files.

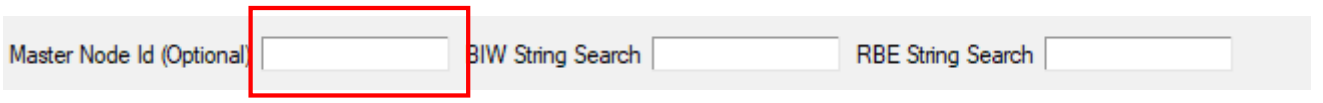
Once the Pop Up window closes, You will find that several files are generated at this location. These files contain most information related to stress, strains and displacement of the simulation. It will be a good idea to save it for later reference. The total file sizes are Very less compared to a result database.
Inside the DATA folder, There will be folders which has names of chosen result files. These folders have csv files which contain element Ids of each independent part (not to be confused with component) , Power users may use this for various purposes.



An Example of a Condensed Stress file.

LOAD_CASE	INCREMENT	COMPONENT_ID	COMPONENT_NAME	MAXIMUM_ID	MAXIMUM_VALUE	COMPONENT_NAME	LOAD_STEP_ITERATION
Loadcase1	0.13	1	00-00305798_002_DK_33745001_CONTROL_PANEL_T_1_0MM	189948	2.1561	00305798_002_DK_33745001_CONTROL_PANEL	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.13	7	AA-00529991_A_006_CANOO_FR_BRACKET_T_02_50mm	216736	9.84716	00529991_A_006_CANOO_FR_BRACKE	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.07	11	AA-00531969_A_004_CANOO_FR_DECO_STRIP_T_1_0MM	26822	9.82994	00531969_A_004_CANOO_FR_DECO_STRIP	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.18	25	AA-00531972_A_005_CANOO_FR_PEGBOARD_ASSEMBLY_T_1_2MM	4780	9.94687	00531972_A_005_CANOO_FR_PEGBOARD_ASSEMBLY	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.13	47	CANOO_FR_PEGBOARD_INNER_T_2_5MM	223455	9.54022	CANOO_FR_PEGBOARD_INNER	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.12	52	LOWER_CARRIER_T_1_0MM	67464	9.64256	LOWER_CARRIER	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.12	81	MAP_POCKET_CLOSE_OUT_T_3_1MM	249743	0.62143	MAP_POCKET_CLOSE_OU	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.1	85	PART01_T_1_8MM	218916	3.13556	PART01	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.3	91	PEGBOARD_T_1_2MM	104736	4.97783	PEGBOARD	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.1	253000	PELVIC_BLOCK	254632	0.01987	PELVIC_BLOCK	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0	125	PegBoard_Carrier_Screws_XXSCR	N/A	N/A	PegBoard_Carrier_Screws_XXSCR	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0	124	Screw_6m_XXSCR	N/A	N/A	Screw_6m_XXSCR	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1
Loadcase1	0.12	119	UPPER_CARRIER_T_2_5MM	193114	9.97503	UPPER_CARRIER	1_REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_1

1.6.0 Register Inputs.



1.6.1 Master Node ID.

Entry of Master node ID is optional . Master node ID of the constrain equation element connected to Load application entity is 1001 as per methodology . If the input file and result database follow the defined rules, There is no need to enter the Master Node ID. Displacement ,Force or any output that need drawing of a curve in Load master page is to be defined in output block with this ID.

If the master node has a different number in the input and output files , It can be typed to the Box given. But it is highly recommended that the methodology is followed by engineers.

1.6.2 Accepted output Quantities.

Entry of Master node ID is optional . Master node ID of the constrain equation element connected to Load application entity is 1001 as per methodology . If the input file and result database follow the defined rules, There is no need to enter the Master Node ID. Displacement ,Force or any output that need drawing of a curve in Load master page is to be defined in output block with this ID.

If the master node has a different number in the input and output files , It can be typed to the Box given. But it is highly recommended that the methodology is followed by engineers.
The Force and Displacement request for PAMCRASH must be defined as IMPACTOR_TRIM_CONTACT. Other Names are not supported.

Output Entry accepted for Load curve

OUTPUT	ABAQUS	PAMCRASH	LS-DYNA
DISPLACEMENT	Displacement	Node (Time History)	Displacement
FORCE	CF-Point loads	Contact Variables (Time History)	discrete/nodes
FORCE COMPONENT	MAG	Contact_Force-Magnitude	resultant_force
DISPLACEMENT COMPONENT	MAG	Translational_Displacement-Magnitude	Mag
FORCE REQUEST	N1001	IMPACTOR_TRIM_CONTACT	N1001
DISPLACEMENT REQUEST	N1001	IMPACTOR_TRIM_CONTACT	N1001

1.6.3 Accepted Output Formats.

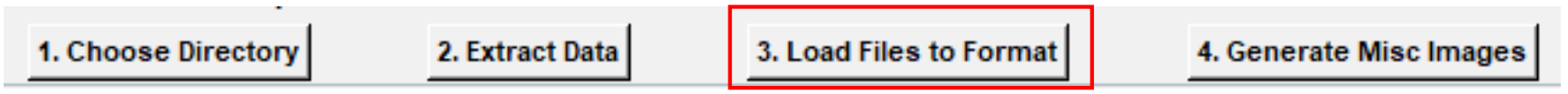
List of accepted output formats are given below.
Dyna input files are to be in DYN format, If not please rename to DYN format before launching the automation.

OUTPUT	ABAQUS	PAMCRASH	LS-DYNA
INPUT FORMAT	INP	PC	DYN
FIELD OUTPUT	ODB	ERFH5	D3PLOT
TIME HISTORY	ODB	ERFH5	BINOUT0000

1.6.4 Unsupported Input and Output Formats.

DSY and THP format outputs for Pamcrash are not supported. Please make sure Pamcrash outputs are in ERFH5 format.
Dyna master input files cannot have a key extension.

1.7.0 Load Files to Format.

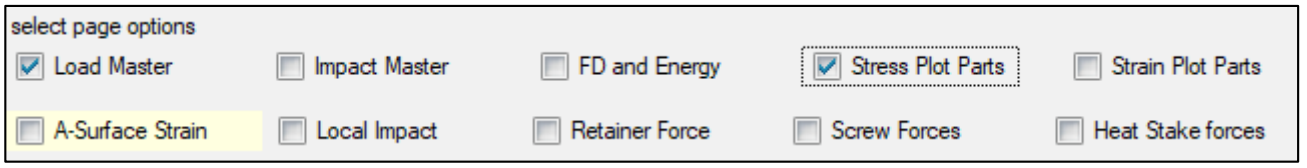


1.7.1 Loading files to specific formats.

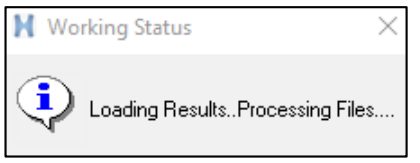
Once the Process status popup closes, The format selection is ready to launch . Please choose the appropriate formats applicable for the report to be made. Check the Boxes adjacent to description of the formats. Please take care not to select conflicting formats or formats not compatible with the result output.

It is important to make sure that Hyperview can successfully load the input files also, Not just the output files. The include structure and corresponding folders are to be placed correctly .

Format selection Area

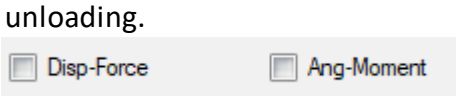


Process status Popup



1.7.2 Load Master Page.

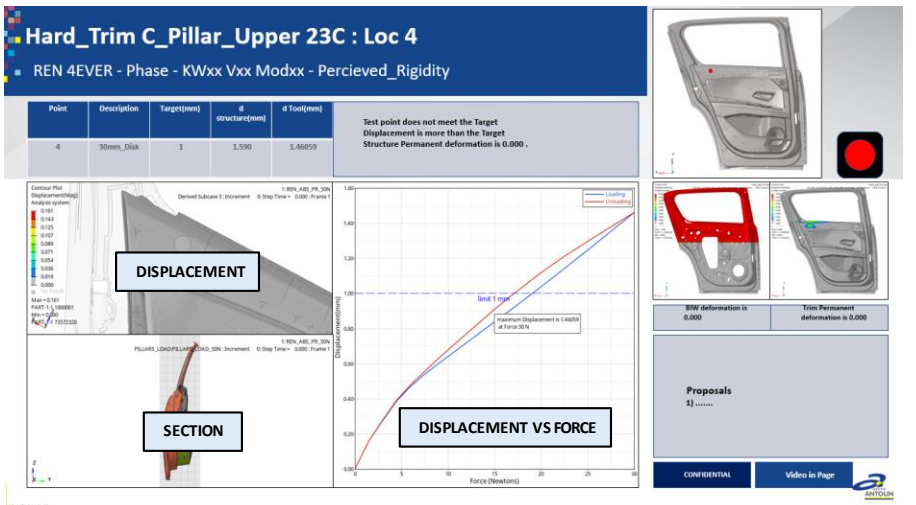
The Load master Page provides a format for reporting stiffness related load cases. It provides displacement ,section animations and a force displacement curve for loading and unloading.



User has the option to have F-D or D-F curve.

By default the application plots Displacement vs force. By checking The Force- Disp check button, This can be reversed. The same applies for Angle moment as well.

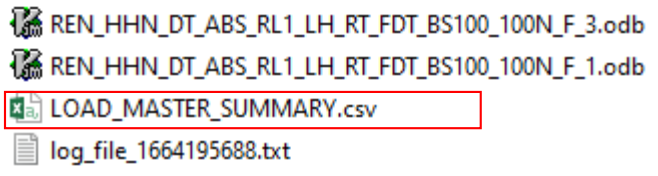
A sample output presentation file is provided at the automation folder in PUNCAE.



1.7.3 Load Master Summary

The Load master summary writes out important information from the results . This information can be copied to report summary page. Information about impactor coordinates, Maximum displacement, permanent deformation etc. are recorded. A sample summary file is provided at \\punnscaddata01\PUN_CAE\2022\20.AUTOMATION_AI

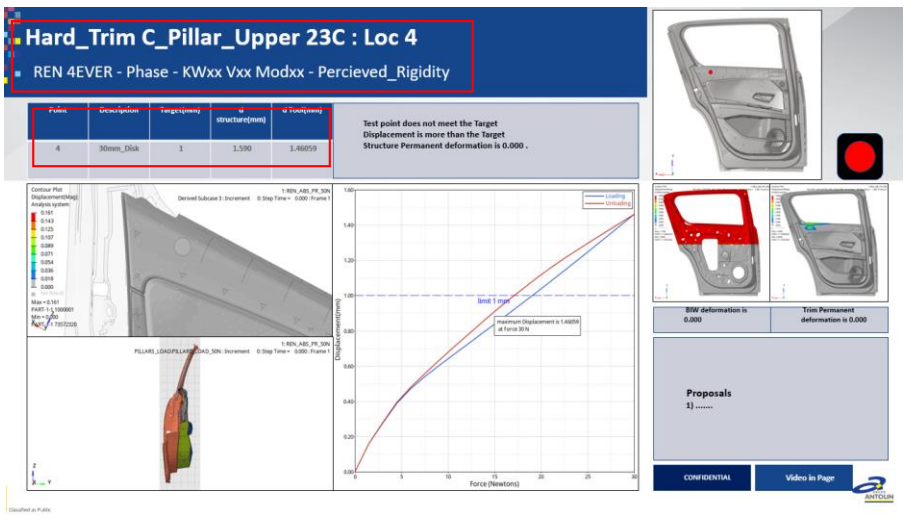
LOC_NUM	FILE_NAME	TEMP	AREA	LOAD	TOOL	FORCE	LIMIT	X	Y	Z	TRIM_MAX	PERM_TRIM	BIW_MAX
4	REN_4EVER_HT	23C	C_Pillar_Upper	Percieved_Rigidity	30mm_Disk	50	1	2137.666	-656.748	786.643	1.59	0	0
1	REN_4EVER_HT	23C	C_Pillar_Upper	Slipping_Resistance	30mm_Disk	150	1	1887.713	-697.371	612.593	5.188	0.045	0



1.7.4 Load master page - Dependencies

Many values of Headings and Tables are derived from the load code and Part code files mentioned in **1.4.3.**

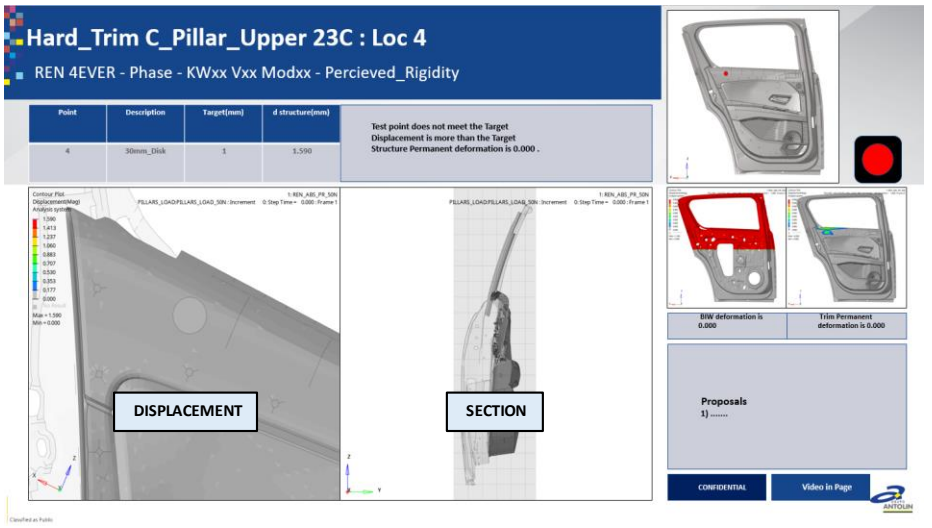
If information printed in PowerPoint does not match with the contents of your input file or result file name , Please update the contents of the Load code and Part code files as mentioned in section **1.4.3** of the document.



1.7.0 Load Files to Format.

1.7.5 Impact Master Page.

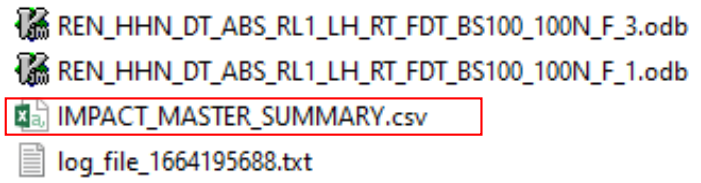
The Impact master Page provides a format for reporting Animations and displacement data for Impact related load cases. The BIW deformations and Trim Permanent deformation is reported on left side of the Page. This page is active for Abaqus, Pamcrash and LS-Dyna .



1.7.6 Impact Master Summary

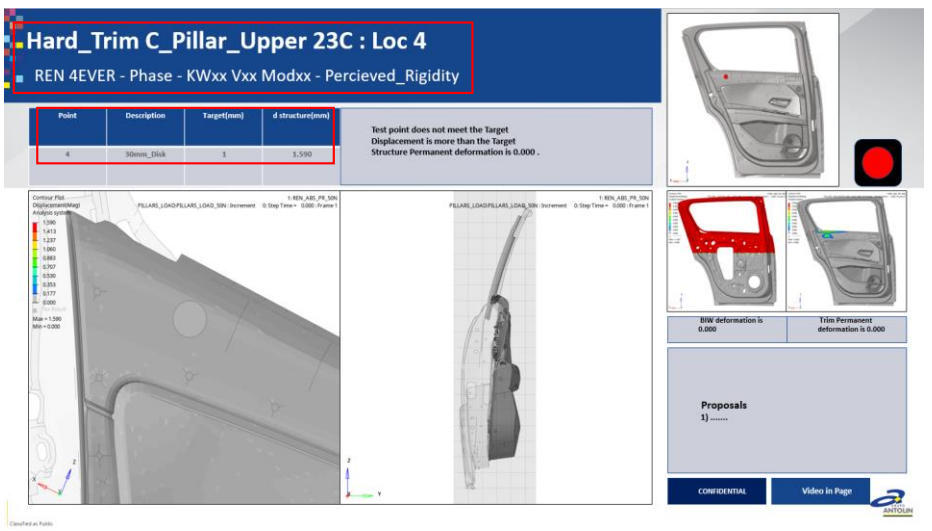
The Impact master Page provides a format for reporting Animations and displacement data for Impact related load cases. The BIW deformations and Trim Permanent deformation is reported on left side of the Page. This page is active for Abaqus, Pamcrash and LS-Dyna .

LOC_NUM	FILE_NAME	TEMP	AREA	LOAD	TOOL	FORCE	LIMIT	X	Y	Z	TRIM_MAX	PERM_TRIM	BIW_MAX
4	REN_4EVER_HT	23C	C_Pillar_Upper	Percieved_Rigidity	30mm_Disk	50	1	2137.666	-656.748	786.643	1.59	0	0
1	REN_4EVER_HT	23C	C_Pillar_Upper	Slipping_Resistance	30mm_Disk	150	1	1887.713	-697.371	612.593	5.188	0.045	0



1.7.7 Impact master page - Dependencies

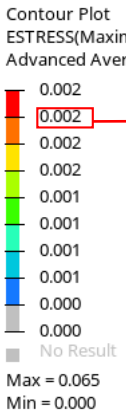
Many values of Headings and Tables are derived from the load code and Part code files mentioned in **1.4.3.**
If information printed in PowerPoint does not match with the contents of your input file or result file name , Please update the contents of the Load code and Part code files as mentioned in section **1.4.3** of the document.



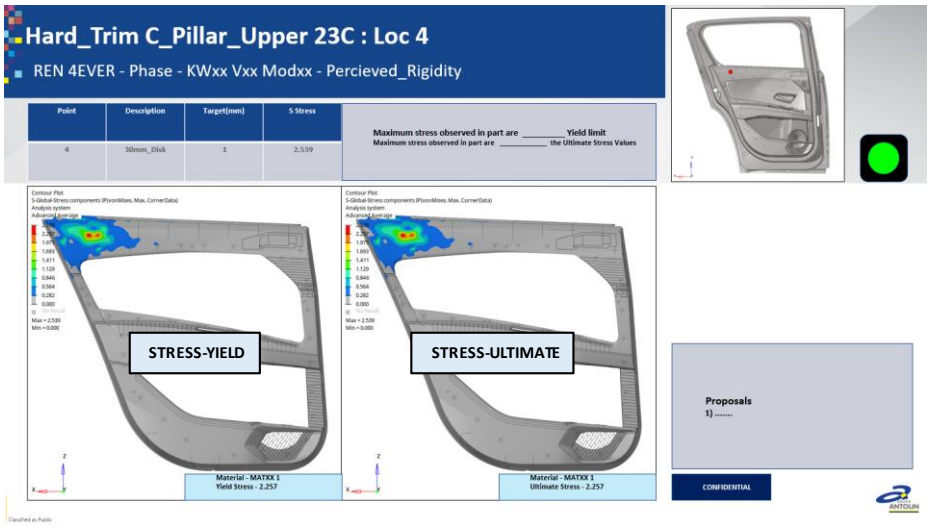
1.7.0 Load Files to Format.

1.7.8 Stress Plot Parts.

The Stress Plot Page Records the Vonmises stress for each part in the assembly. The actions required for selection of parts are Explained in next paragraph.



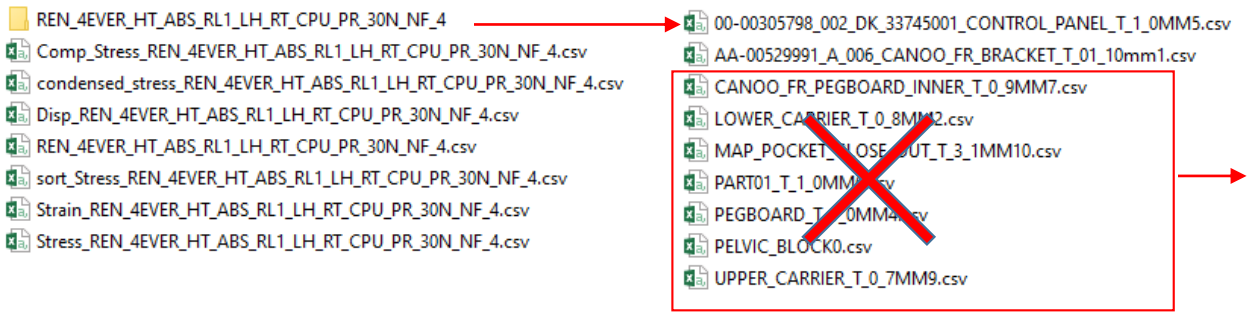
Window 1 - Update the second value in legend as Yield stress
Window 2 - Update the second value in legend as Ultimate stress
The values updated here will show up in the summary table file.



1.7.9 Selection of required parts for Stress Plot .

The Automation can identify each independent part in the assembly. Details of such parts are written in Data folder in directories named after output files. Users can open those folders and delete the csv files corresponding to parts which are not to be used forplot generation.

An example is given below. Here only first two files are retained, Rest of the files are deleted. This means Parts stress, Parts strain and A-surface strain plots will be generated for these two parts only. Please ignore the thickness in the name, The components correspond to Remaining part of naming.



Delete the files that correspond To Part names that are not to be Included in the Partwise stress Plots.

In the example , Plots will be made For parts corresponding to first two Files.

Do this for other results files by selective Deletion of files from respective folders

1.7.10 Stress Data Types and Data components .

The Table given below explains the types of Data components that will be used for Stress plots. If users need to use a different quantity Please contact the methodology department in advance before project commence.

It is recommended that this data is converted to a dynamic input for subsequent revisions of Automation.

Table of reported Stress Entry

SOLVER	CONTOUR	DATA TYPE	DATA COMPONENT
ABAQUS	STRESS	vonMises	S-Global-Stress components IP
LS-DYNA	STRESS	Stress	vonMises
PAMCRASH	STRESS(2D)	ESTRESS	Maximum equivalent stress in shell element
	STRESS(3D)	SXYZ/3D/Stress	vonMises

1.7.11 Stress Plot Parts - Dependencies

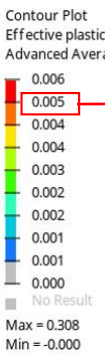
User Many values of Headings and Tables are derived from the load code and Part code files mentioned in **1.4.3.**

If information printed in PowerPoint does not match with the contents of your input file or result file name , Please update the contents of the Load code and Part code files as mentioned in section **1.4.3** of the document.

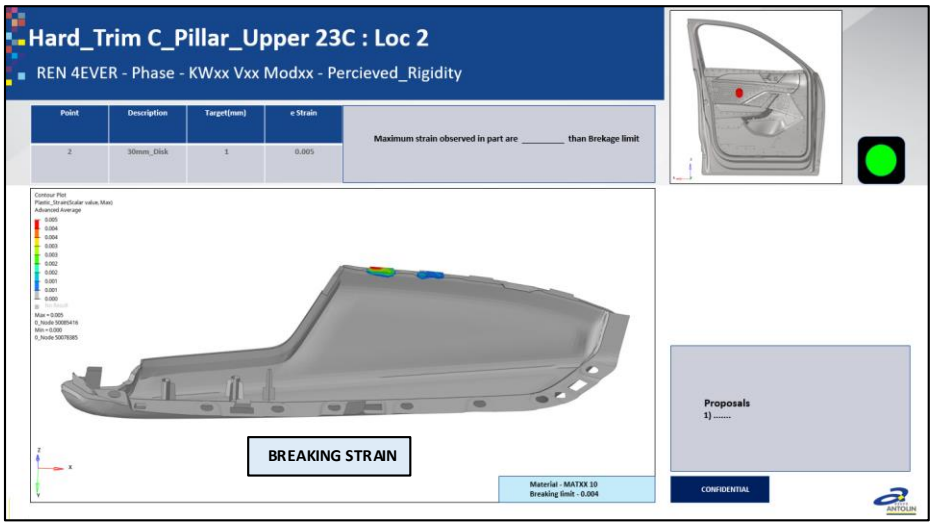
1.7.0 Load Files to Format.

1.7.12 Strain Plot Parts.

The Strain Plot Page Records the Plastic strains for each part in the assembly. The actions required for selection of parts are Explained in next paragraph.



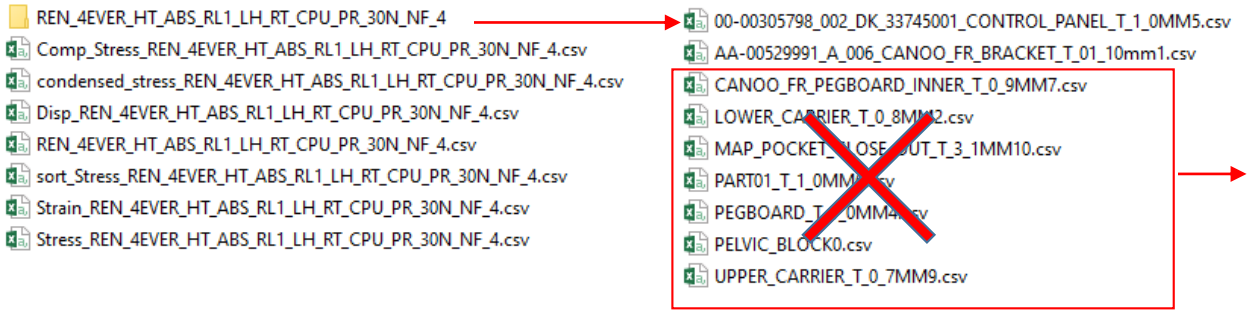
Window 1 - Update the breaking strain value in legend
The values updated here will show up in the summary table file.



1.7.13 Selection of required parts for Strain Plot .

The Automation can identify each independent part in the assembly. Details of such parts are written in Data folder in directories named after output files. Users can open those folders and delete the csv files corresponding to parts which are not to be used forplot generation.

An example is given below. Here only first two files are retained, Rest of the files are deleted. This means Parts stress, Parts strain and A-surface strain plots will be generated for these two parts only. Please ignore the thickness in the name, The components correspond to Remaining part of naming.



Delete the files that correspond To Part names that are not to be Included in the Partwise stress Plots.

In the example , Plots will be made For parts corresponding to first two Files.

Do this for other results files by selective Deletion of files from respective folders

1.7.14 Strain Data Types and Data components .

The Table given below explains the types of Data components that will be used for Stress plots. If users need to use a different quantity Please contact the methodology department in advance before project commence.

It is recommended that this data is converted to a dynamic input for subsequent revisions of Automation.

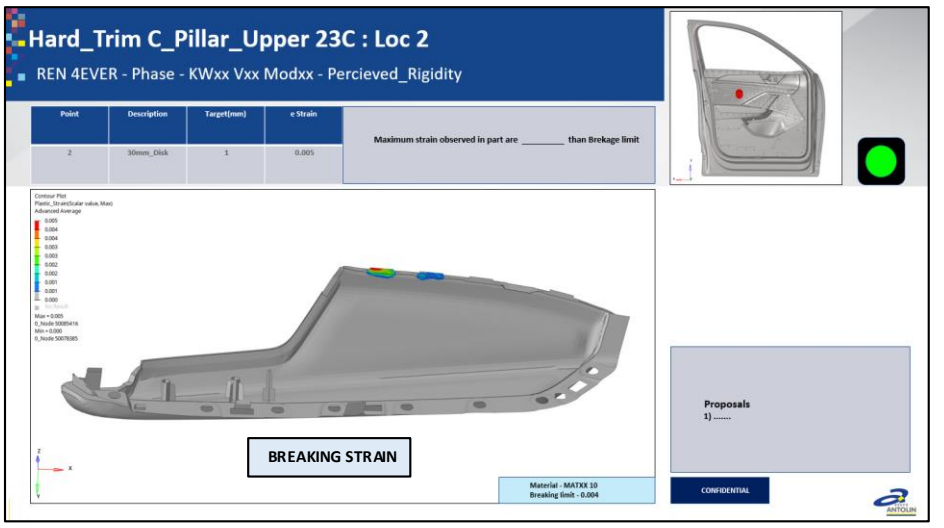
Table of reported Strain Entry

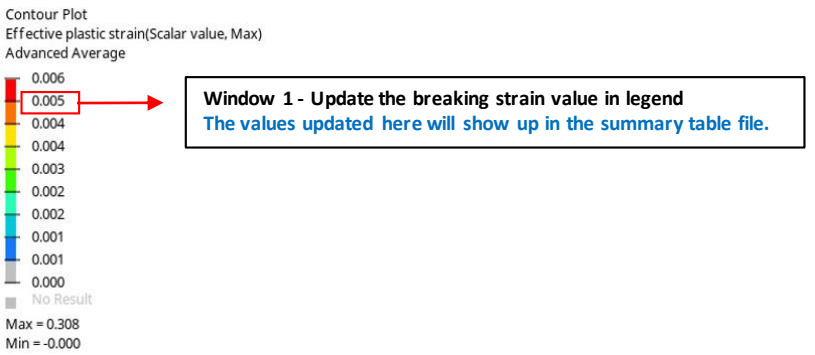
SOLVER	CONTOUR	DATA TYPE	DATA COMPONENT
ABAQUS	STRAIN	vonMises	PE-Global-Plastic strain components IP
LS-DYNA	STRAIN	Stress	vonMises
PAMCRASH	STRAIN(2D)	EPLE/2D	Scalar value
	STRAIN(3D)	EPLE/3D/Equivalent Plastic Strain	Scalar value

1.7.0 Load Files to Format.

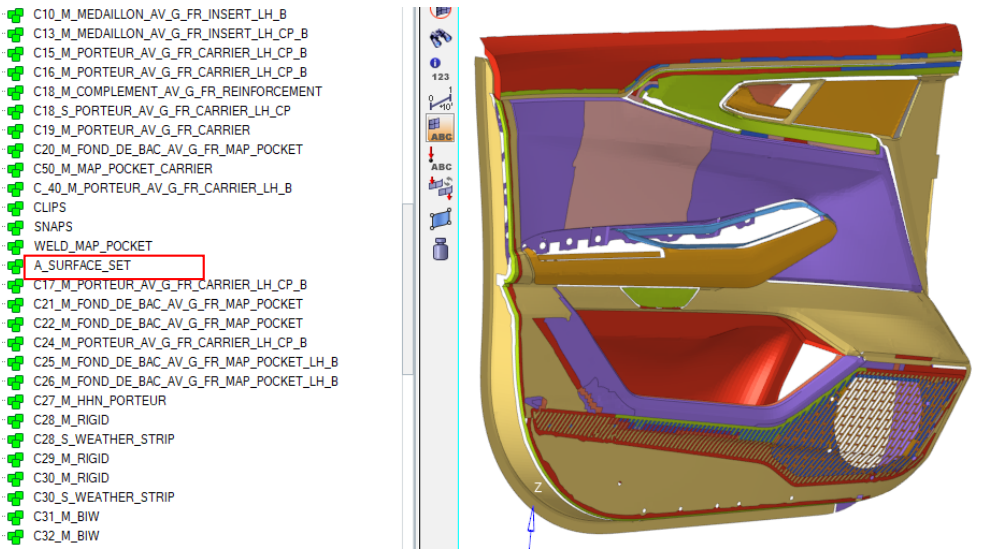
1.7.15 A-Surface Strain Plot Parts.

The A-Surface Strain Plot Page Records the Plastic strains for A-Surface of each part in the assembly. The actions required for selection of parts are Explained in next paragraph.

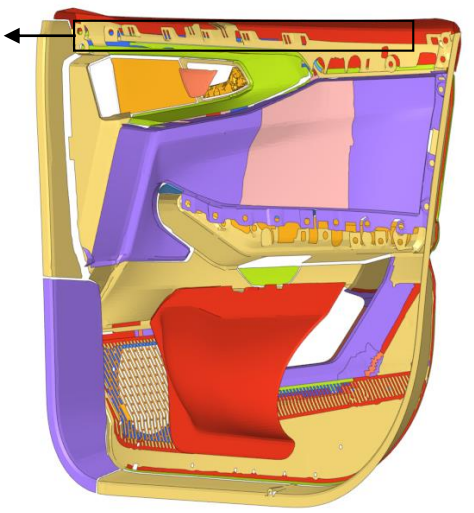




Example of an A-Surface strain set is given below.



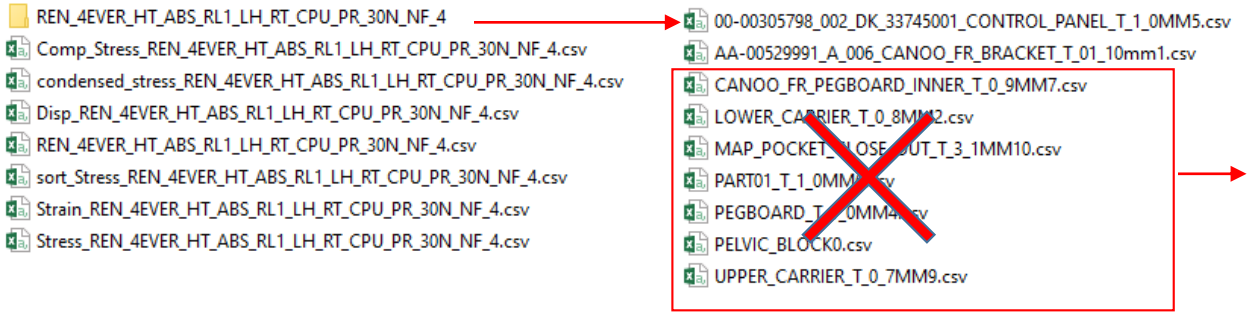
Avoid selecting Areas like this.
This surface is not Part of A-Surface.



1.7.16 Selection of required parts for Strain Plot .

The Automation can identify each independent part in the assembly. Details of such parts are written in Data folder in directories named after output files. Users can open those folders and delete the csv files corresponding to parts which are not to be used forplot generation.

An example is given below. Here only first two files are retained, Rest of the files are deleted. This means Parts stress, Parts strain and A-surface strain plots will be generated for these two parts only. Please ignore the thickness in the name, The components correspond to Remaining part of naming.



Delete the files that correspond To Part names that are not to be Included in the Partwise stress Plots.

In the example , Plots willbe made For parts corresponding to first two Files.

Do this for other results files by selective Deletion of files from respective folders

1.7.17 Strain Data Types and Data components .

The Table given below explains the types of Data components that will be used for Stress plots. If users need to use a differ ent quantity Please contact the methodology department in advance before project commence.

It is recommended that this data is converted to a dynamic input for subsequent revisions of Automation.

Table of reported Strain Entry

SOLVER	CONTOUR	DATA TYPE	DATA COMPONENT
ABAQUS	STRAIN	vonMises	PE-Global-Plastic strain components IP
LS-DYNA	STRAIN	Stress	vonMises
PAMCRASH	STRAIN(2D)	EPLE/2D	Scalar value
	STRAIN(3D)	EPLE/3D/Equivalent Plastic Strain	Scalar value

1.7.0 Load Files to Format.

1.7.18 Retainer Force Plot .

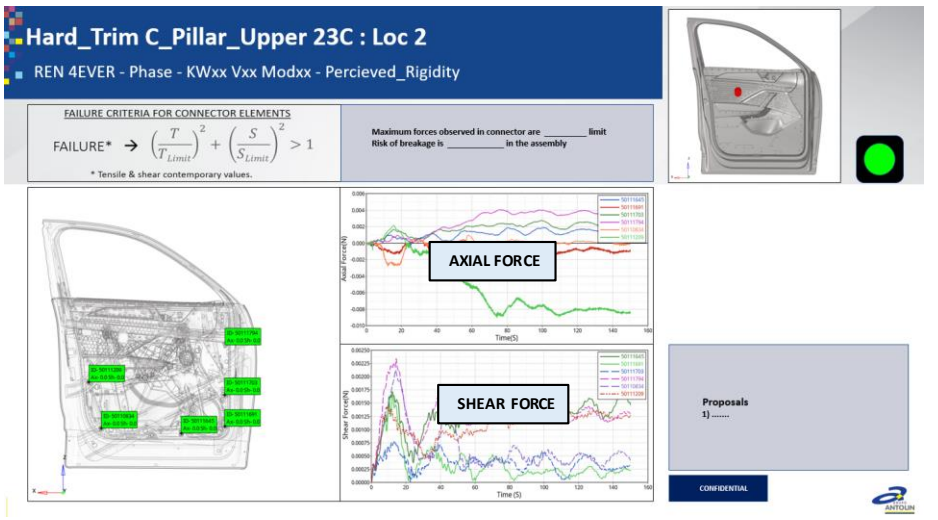
The Retainer force Plot Page Records the Axial force and Shear force values from the result data for components of relative motion present in the model. There are few naming rules defined for connector parts to achieve this.

RETAINER	_XXRET_
SCREW	_XXSCR_
HEAT STAKE	_XXHT_

FAILURE CRITERIA FOR CONNECTOR ELEMENTS

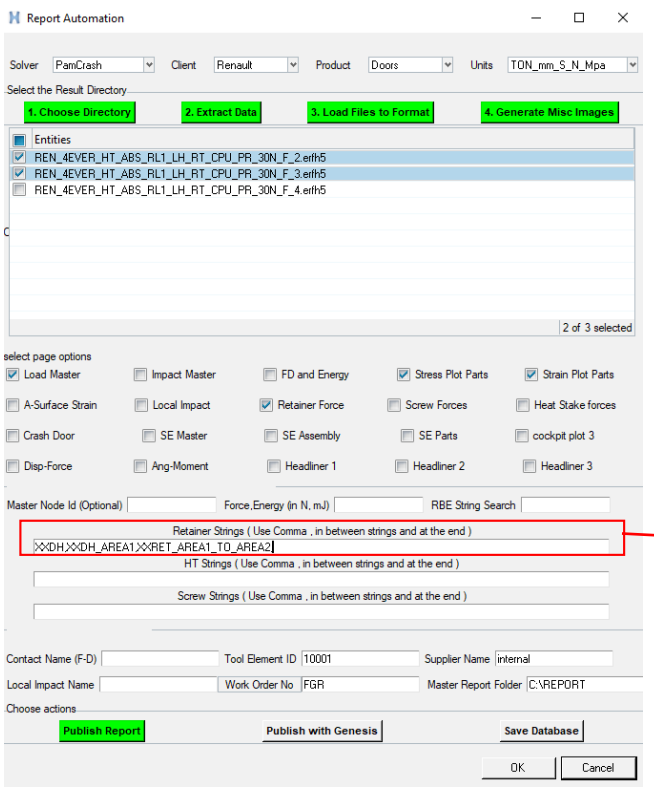
$$\text{FAILURE*} \rightarrow \left(\frac{T}{T_{Limit}}\right)^2 + \left(\frac{S}{S_{Limit}}\right)^2 > 1$$

* Tensile & shear contemporary values.



1.7.19 Selection of required connectors for Retainer force Plot .

Different types of connectors are to have specific strings in their component names. The user has the freedom to use strings of their choice to differentiate different connector groups , Such sets are useful to group connectors specific to two parts or an area.
User need not enter full names of such a group, Only a unique portion is required. For example XXHT_AREA4



EXAMPLE FOR CONNECTOR NAMING

- SWDM_TVKL_Pos1_RAHMEN_an_ALF-8mmPP_XXHT_AREA1
- SWDM_TVKL_Pos1_RAHMEN_an_WARNWESTENFACH-8mmPP_XXHT_AREA2
- SWDM_TVKL_Pos1_RAHMEN_an_TIEFTOENER-8mmPP_XXHT_AREA3
- SWDM_TVKL_Pos1_RAHMEN_an_LAUTSPRECHERGITTER_ZZG-8mmPP_XXHT_AREA4
- SWDM_TVKL_Pos1_RAHMEN_an_ZZG-8mmPP_XXHT_AREA5
- SWDM_TVKL_Pos1_RAHMEN_an_ARMAUFLAGE-NULLBAR-8mmPP_XXHT_AREA6
- SWDM_TVKL_Pos1_ZZG_an_LAUTSPRECHERGITTER-8mmPP_XXHT_AREA7
- SWDM_TVKL_Pos1_AAL_an_ZZG-8mmPP_XXHT_AREA8
- SWDM_TVKL_Pos1_LAUTSPRECHERGITTER_zu_AUFNAHME_MT_FS-8mmPP_XXHT_AREA9

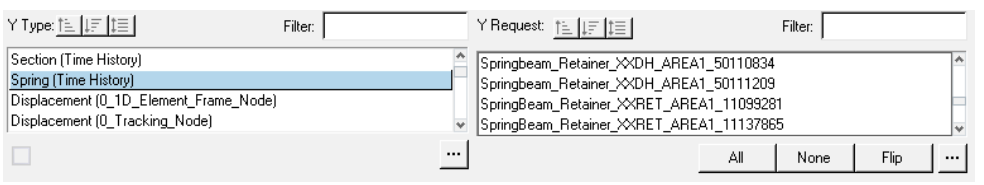
SPECIFYING FAILURE

If a connector fails at 20N in axial and 15N in shear ,please add *_AXFVAL_20_SHFVAL_15*
To the component name of the connector.
The units are always in N, Automation will convert it to your system of choice.
Example -CONNECTOR_WELD_CARRIER_INSERT_TOP_ROLL_XXHT_AREA4_AXFVAL_20_SHFVAL_15

Text area to enter additional specific connector groups
XXDH,XXDH_AREA1,XXRET_AREA1_TO_AREA2,

Failed connector elements will be highlighted in Red.
Others in Green Color

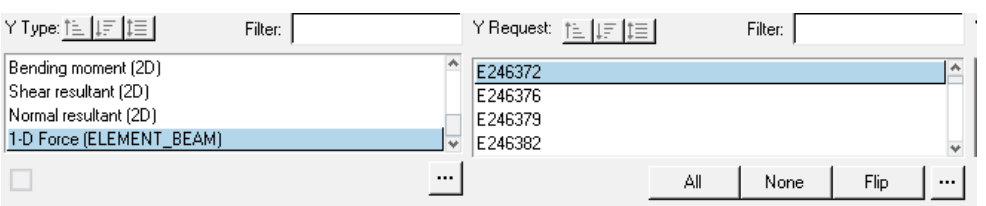
Output Example for Hyperview - PAMCRASH



Here element ID is auto added by Pamcrash –Not to enter by user

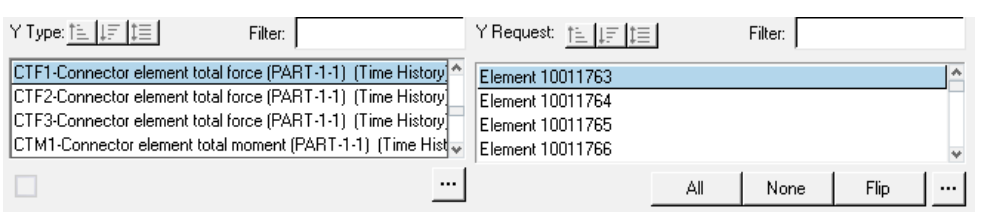
```
THELE /      0
NAME SWRP_TVKL_Pos1_RAHMEN_an_INSERT-15mmPP_XXHT_AREA21
ELE    50080929 50080964 50080995
 END
```

Output Example for Hyperview - LSDYNA




```
*DATABASE_HISTORY_BEAM
$HMNAME  OUTPUTBLOCKS                1Screws
$     ID      ID      ID      ID      ID      ID      ID      ID
246740  246372  246376  246379  246382  246385  246429  246462
```

Output Example for Hyperview - ABAQUS




```
*OUTPUT,  HISTORY, TIME INTERVAL = 0.0
*ELEMENT  OUTPUT, ELSET = CONNECTOR_OP,  VARIABLE = XX
```

1.7.0 Load Files to Format.

1.7.20 Screw Force Plot .

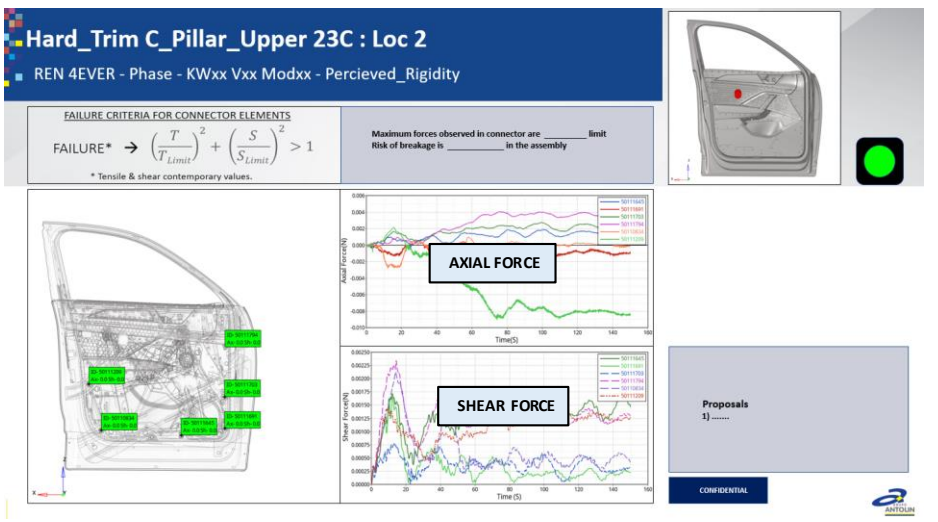
The Retainer force Plot Page Records the Axial force and Shear force values from the result data for components of relative motion present in the model. There are few naming rules defined for connector parts to achieve this.

RETAINER	_XXRET_
SCREW	_XXSCR_
HEAT STAKE	_XXHT_

FAILURE CRITERIA FOR CONNECTOR ELEMENTS

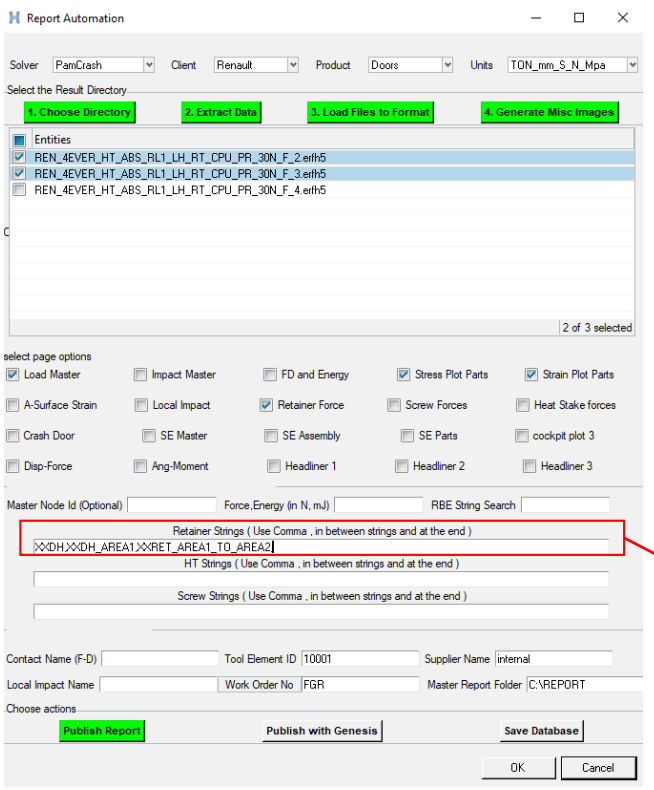
$$\text{FAILURE*} \rightarrow \left(\frac{T}{T_{Limit}}\right)^2 + \left(\frac{S}{S_{Limit}}\right)^2 > 1$$

* Tensile & shear contemporary values.



1.7.21 Selection of required connectors for Screw force Plot .

Different types of connectors are to have specific strings in their component names. The user has the freedom to use strings of their choice to differentiate different connector groups , Such sets are useful to group connectors specific to two parts or an area.
User need not enter full names of such a group, Only a unique portion is required. For example XXHT_AREA4



EXAMPLE FOR CONNECTOR NAMING

- SWDM_TVKL_Pos1_RAHMEN_an_ALF-8mmPP_XXHT_AREA1
- SWDM_TVKL_Pos1_RAHMEN_an_WARNWESTENFACH-8mmPP_XXHT_AREA2
- SWDM_TVKL_Pos1_RAHMEN_an_TIEFTOENER-8mmPP_XXHT_AREA3
- SWDM_TVKL_Pos1_RAHMEN_an_LAUTSPRECHERGITTER_ZZG-8mmPP_XXHT_AREA4
- SWDM_TVKL_Pos1_RAHMEN_an_ZZG-8mmPP_XXHT_AREA5
- SWDM_TVKL_Pos1_RAHMEN_an_ARMAUFLAGE-NULLBAR-8mmPP_XXHT_AREA6
- SWDM_TVKL_Pos1_ZZG_an_LAUTSPRECHERGITTER-8mmPP_XXHT_AREA7
- SWDM_TVKL_Pos1_AAL_an_ZZG-8mmPP_XXHT_AREA8
- SWDM_TVKL_Pos1_LAUTSPRECHERGITTER_zu_AUFNAHME_MT_FS-8mmPP_XXHT_AREA9

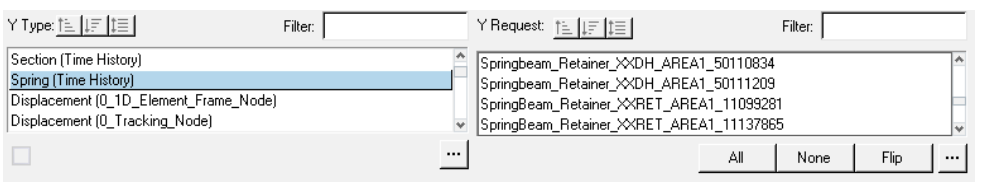
SPECIFYING FAILURE

If a connector fails at 20N in axial and 15N in shear ,please add *_AXFVAL_20_SHFVAL_15*
To the component name of the connector.
The units are always in N, Automation will convert it to your system of choice.

Here is an example.
CONNECTOR_WELD_CARRIER_INSERT_TOP_ROLL_XXHT_AREA4 _AXFVAL_20_SHFVAL_15

Text area to enter additional specific connector groups
XXDH,XXDH_AREA1,XXRET_AREA1_TO_AREA2,

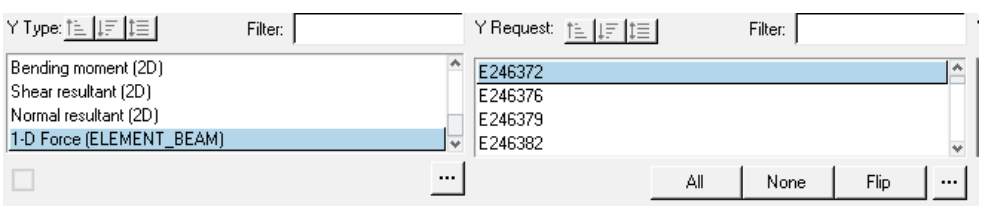
Output Example for Hyperview - PAMCRASH



Here element ID is auto added by Pamcrash –Not to enter by user

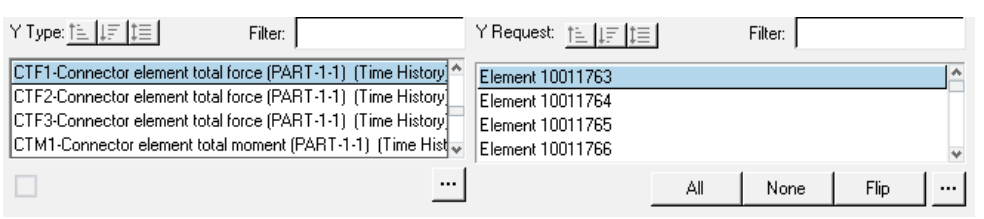
```
THELE /      0
NAME SWRP_TVKL_Pos1_RAHMEN_an_INSERT-15mmPP_XXHT_AREA21
ELE    50080929 50080964 50080995
 END
```

Output Example for Hyperview - LSDYNA




```
*DATABASE_HISTORY_BEAM
$HMNAME  OUTPUTBLOCKS                1Screws
$     ID      ID      ID      ID      ID      ID      ID      ID
246740  246372  246376  246379  246382  246385  246429  246462
```

Output Example for Hyperview - ABAQUS




```
*OUTPUT,  HISTORY, TIME INTERVAL = 0.0
*ELEMENT  OUTPUT, ELSET = CONNECTOR_OP,  VARIABLE = XX
```

1.7.0 Load Files to Format.

1.7.22 Heat Stake Force Plot .

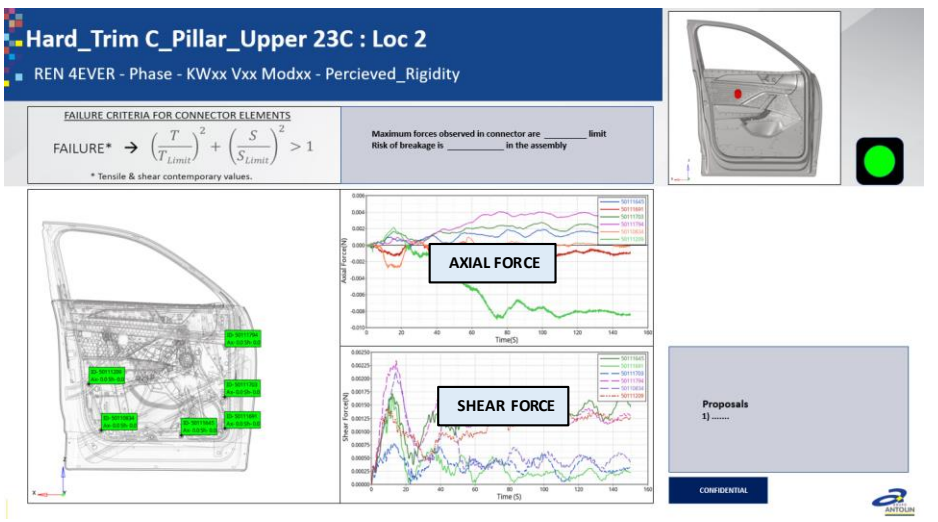
The Retainer force Plot Page Records the Axial force and Shear force values from the result data for components of relative motion present in the model. There are few naming rules defined for connector parts to achieve this.

RETAINER	_XXRET_
SCREW	_XXSCR_
HEAT STAKE	_XXHT_

FAILURE CRITERIA FOR CONNECTOR ELEMENTS

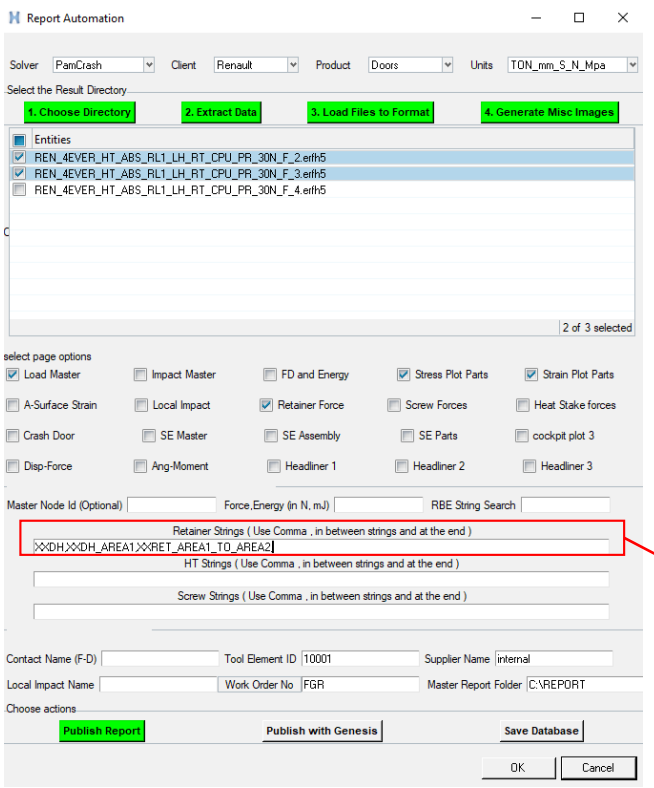
$$\text{FAILURE*} \rightarrow \left(\frac{T}{T_{Limit}}\right)^2 + \left(\frac{S}{S_{Limit}}\right)^2 > 1$$

* Tensile & shear contemporary values.



1.7.23 Selection of required connectors for Heat Stake force Plot .

Different types of connectors are to have specific strings in their component names. The user has the freedom to use strings of their choice to differentiate different connector groups , Such sets are useful to group connectors specific to two parts or an area.
User need not enter full names of such a group, Only a unique portion is required. For example XXHT_AREA4



EXAMPLE FOR CONNECTOR NAMING

- SWDM_TVKL_Pos1_RAHMEN_an_ALF-8mmPP_XXHT_AREA1
- SWDM_TVKL_Pos1_RAHMEN_an_WARNWESTENFACH-8mmPP_XXHT_AREA2
- SWDM_TVKL_Pos1_RAHMEN_an_TIEFTOENER-8mmPP_XXHT_AREA3
- SWDM_TVKL_Pos1_RAHMEN_an_LAUTSPRECHERGITTER_ZZG-8mmPP_XXHT_AREA4
- SWDM_TVKL_Pos1_RAHMEN_an_ZZG-8mmPP_XXHT_AREA5
- SWDM_TVKL_Pos1_RAHMEN_an_ARMAUFLAGE-NULLBAR-8mmPP_XXHT_AREA6
- SWDM_TVKL_Pos1_ZZG_an_LAUTSPRECHERGITTER-8mmPP_XXHT_AREA7
- SWDM_TVKL_Pos1_AAL_an_ZZG-8mmPP_XXHT_AREA8
- SWDM_TVKL_Pos1_LAUTSPRECHERGITTER_zu_AUFNAHME_MT_FS-8mmPP_XXHT_AREA9

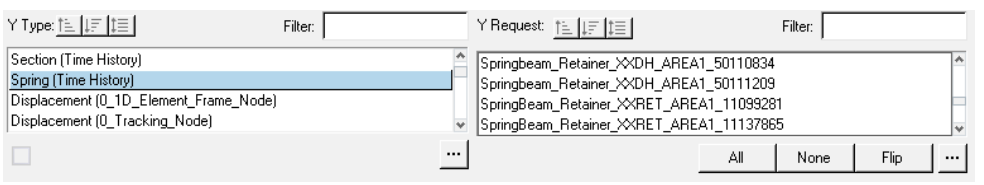
SPECIFYING FAILURE

If a connector fails at 20N in axial and 15N in shear ,please add *_AXFVAL_20_SHFVAL_15*
To the component name of the connector.
The units are always in N, Automation will convert it to your system of choice.

Here is an example.
CONNECTOR_WELD_CARRIER_INSERT_TOP_ROLL_XXHT_AREA4_AXFVAL_20_SHFVAL_15

Text area to enter additional specific connector groups
XXDH,XXDH_AREA1,XXRET_AREA1_TO_AREA2,

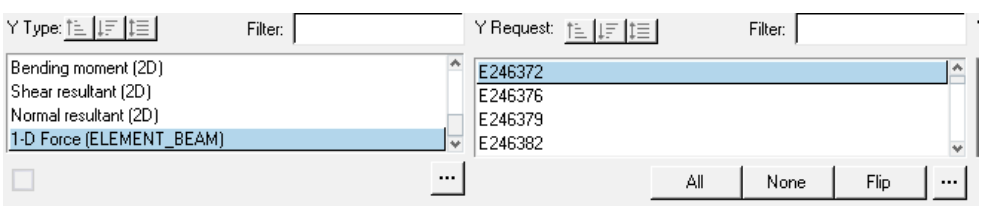
Output Example for Hyperview - PAMCRASH



Here element ID is auto added by Pamcrash –Not to enter by user

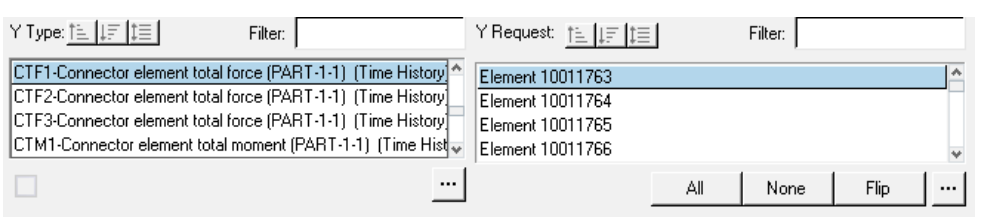
```
THELE /      0
NAME SWRP_TVKL_Pos1_RAHMEN_an_INSERT-15mmPP_XXHT_AREA21
ELE    50080929 50080964 50080995
 END
```

Output Example for Hyperview - LSDYNA




```
*DATABASE_HISTORY_BEAM
$HMNAME  OUTPUTBLOCKS                1Screws
$     ID      ID      ID      ID      ID      ID      ID      ID
246740  246372  246376  246379  246382  246385  246429  246462
```

Output Example for Hyperview - ABAQUS

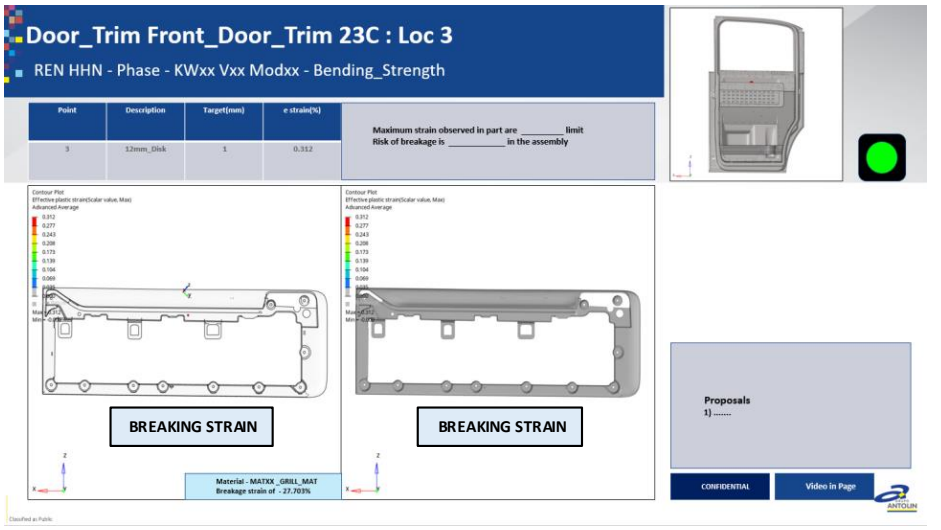



```
*OUTPUT,  HISTORY, TIME INTERVAL = 0.0
*ELEMENT  OUTPUT, ELSET = CONNECTOR_OP,  VARIABLE = XX
```

1.7.0 Load Files to Format.

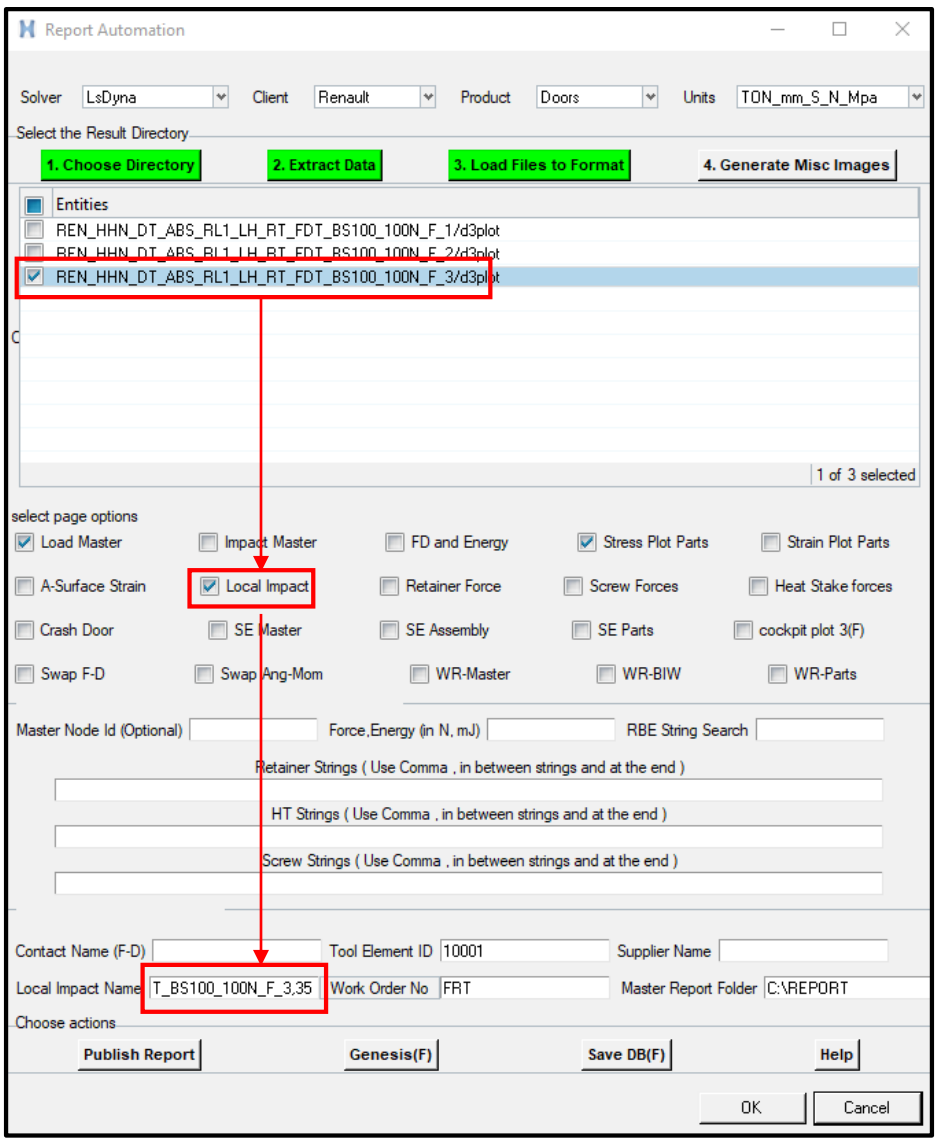
1.7.24 Local Impact Master Page.

The Local impact page can report an impact area of interest. An example is example speaker grill. An ISO plastic strain plot and Plastic strain plot are generated for the component ID provided.



1.7.25 Local Impact page Input.

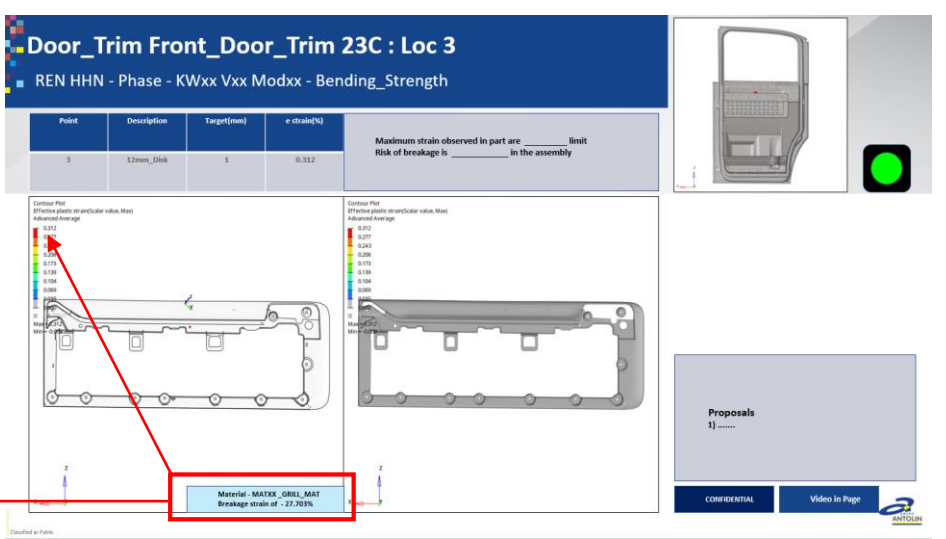
Choose the local Impact page format. It is mandatory to Fill the Local Impact Name . The input file which needs local impact among the list followed by comma and the component ID needs to be filled in .
Component name as per input files is valid.
Examples are given below for supported solvers.
Only the input file and component specified will be considered for the plot generation. Currently multiple files cannot be specified.



LSDYNA- REN_HHN_DT_ABS_RL1_LH_RT_FDT_BS100_100N_F_3,35
Abaqus -REN_4EVER_HT_ABS_RL1_LH_RT_CPU_SR_150N_F_1.odb,10000292
Pamcrash - REN_4EVER_HT_ABS_RL1_LH_RT_CPU_SR_150N_F_1.erfh5,101

1.7.26 Local Impact page - Dependencies

Local impact page is currently not tested for Pamcrash.
There is a requirement of proper input file loading for generation of local impact page. If the input file is not loaded properly to hyper view due to improper include structure or due to parameters present in the input files , Local impact page will not be generated.
Please let the methodology team in advance – When there is a requirement of local impact plot generation for PamCrash solver.

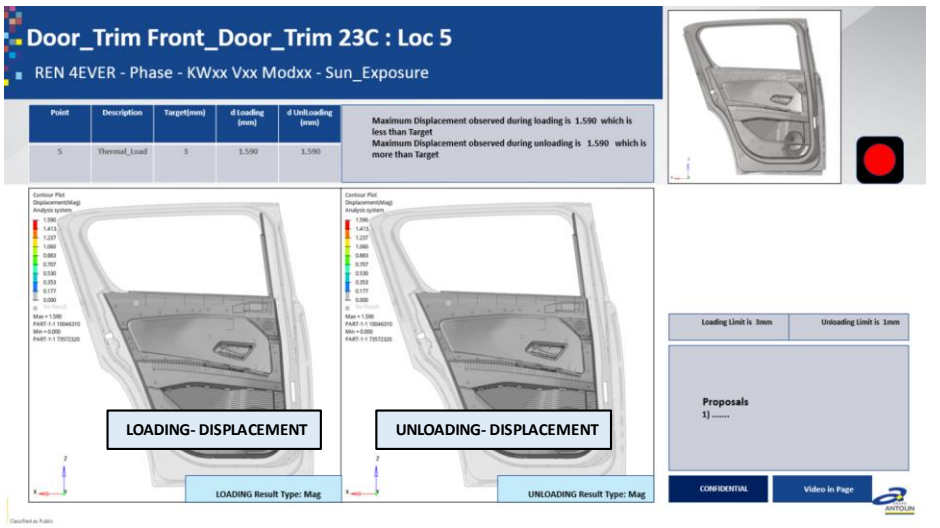


Value specified at #spot 2 will be printed in CAE report As Breaking strain

1.7.0 Load Files to Format.

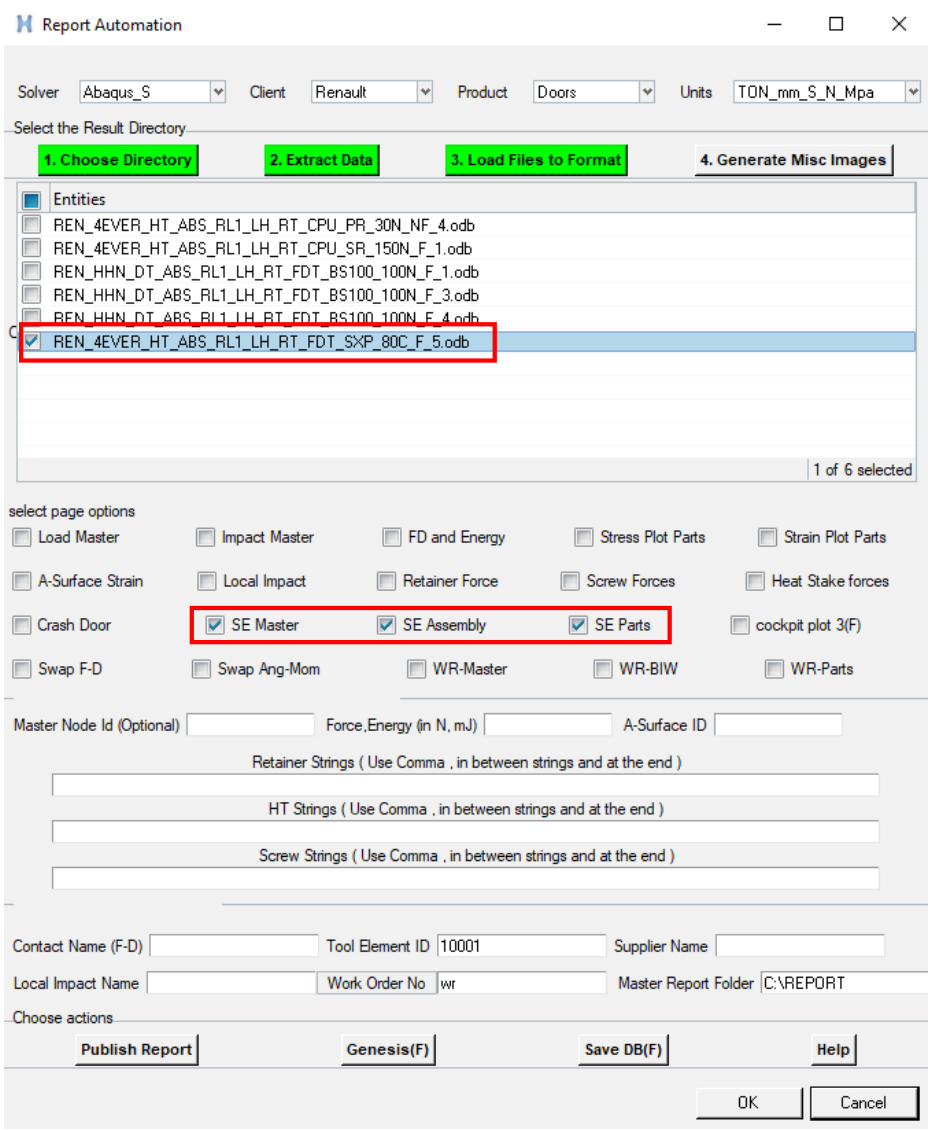
1.7.27 Sun Exposure Master Page.

The Sun exposure master page generates two windows ,Both of which are Displacement magnitude animations. First window records loading load step and second window records unloading step.



1.7.28 Sun Exposure Input.

Choose the Sun exposure master ,Sun exposure assembly and Sun exposure parts selection boxes .
The other inputs required are work order number and report folder input.



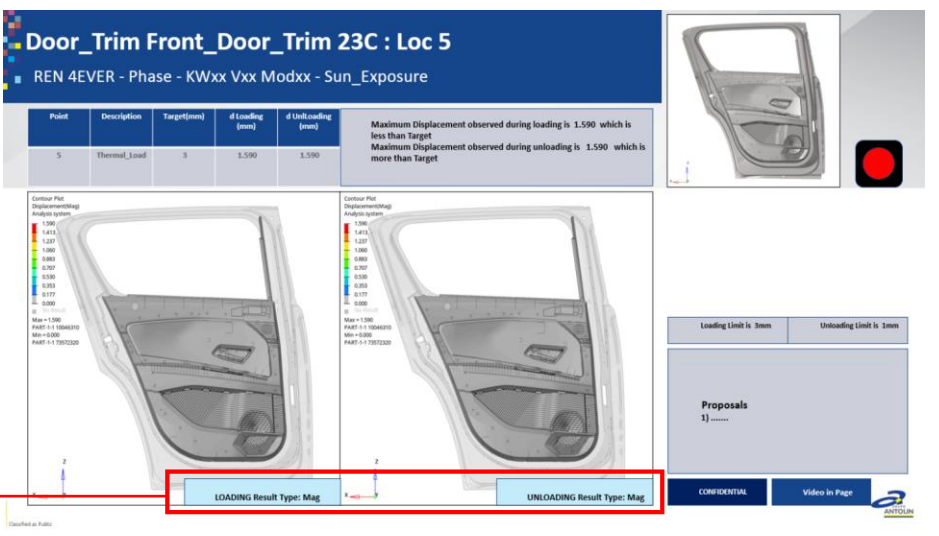
1.7.29 Sun Exposure Summary Table.

Summary of Sun Exposure is generated on the folder containing result files. Summary table records displacement outputs of Assembly , and Part files chosen.

FILENAME	AREA	LOAD_CASE_ DESCRIPTIO N	LOADING TARGET	UNLOADING TARGET	LOADING DISPLACEME NT	UNLOADING DISPLACEME NT	LOADING MAX_DIRECT ION	UNOADING MAX_DIRECT ION	DIRECTION
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	1.59	1.59	-	-	Mag
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	1.59	0	0	0	Mag
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.159	0	-0.124	0	X
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.191	0	-1.521	0	Y
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	1.59	0	-0.36	0	Z
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.153	0	0.013	0	Mag
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	-0.003	0	-0.121	0	X
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.004	0	-0.007	0	Y
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.064	0	-0.153	0	Z
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.182	0	0.089	0	Mag
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.033	0	0.01	0	X
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	-0.03	0	-0.135	0	Y
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	-0.064	0	-0.126	0	Z
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.225	0	0.052	0	Mag
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.065	0	0.011	0	X
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	-0.02	0	-0.169	0	Y
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	-0.002	0	-0.149	0	Z
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.202	0	0.06	0	Mag
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.048	0	0.019	0	X
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	-0.033	0	-0.153	0	Y
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	-0.032	0	-0.141	0	Z
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.082	0	0	0	Mag
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.009	0	-0.035	0	X
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.005	0	-0.029	0	Y
REN_4EVER_HT_ABS	Front_Door_Trim	Sun_Exposur	3	1	0.019	0	-0.075	0	Z

1.7.30 Sun Exposure - Dependencies

Sun Exposure is supported for Abaqus Solver only.
Master Node id and Tool element numbering is irrelevant for sun exposure page.



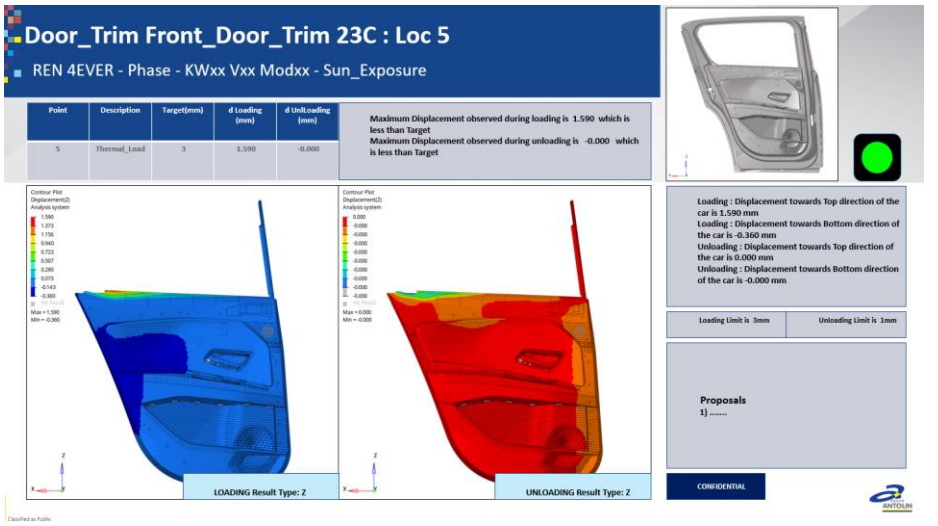
Type of displacement is printed here

1.7.0 Load Files to Format.

1.7.32 Sun Exposure Assembly Page.

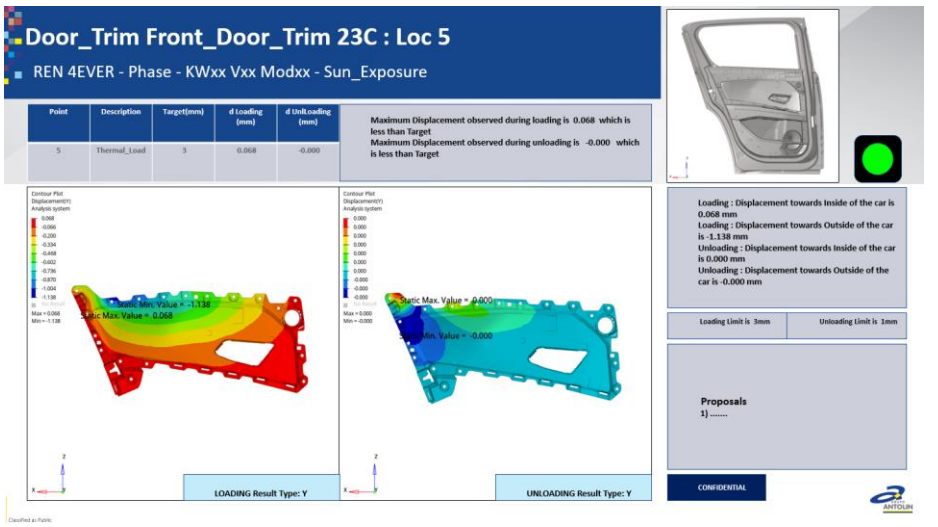
The Sun exposure Assembly page generates 4 Pages.
Displacement Magnitude followed by Displacements in Global X,Y and Z for the complete assembly .
The application will Hide components with the following strings present in component names (Caps or small caps).
BIW, FAKE,SEAL,NULL,SHEET,PATCH.

Currently the Automatic displacement summary is printed for LH side only.



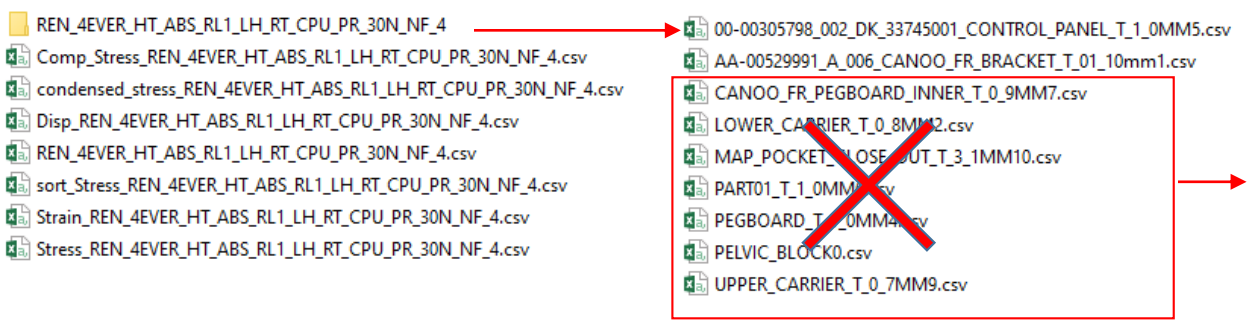
1.7.32 Sun Exposure Parts Page.

If Sunexposure Parts are chosen four pages each will be generated for every parts selected by the mechanism given below.
The Pages are displacement magnitude , X, Y and Z plots.
Left side window records the loading values and right side window records the unloading values.



1.7.33 Selection of required parts for Sun Exposure Page .

The Automation can identify each independent part in the assembly. Details of such parts are written in Data folder in directories named after output files. Users can open those folders and delete the csv files corresponding to parts which are not to be used forplot generation.
An example is given below. Here only first two files are retained, Rest of the files are deleted. This means Sun Exposure parts plots will be generated for these two parts only. Please ignore the thickness in the name, The components correspond to Remaining part of name.



Delete the files that correspond To Part names that are not to be Included in the Partwise stress Plots.

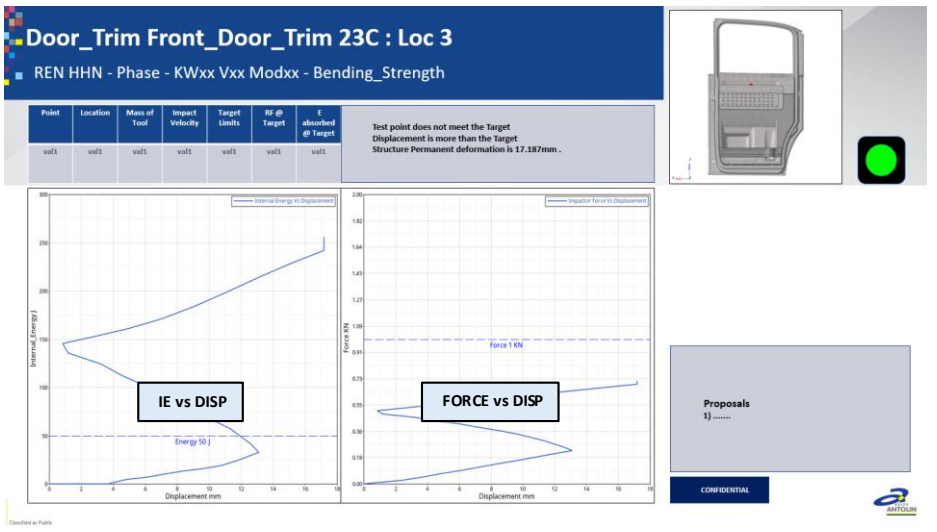
In the example , Plots willbe made For parts corresponding to first two Files.

Do this for other results files by selective Deletion of files from respective folders

1.7.0 Load Files to Format.

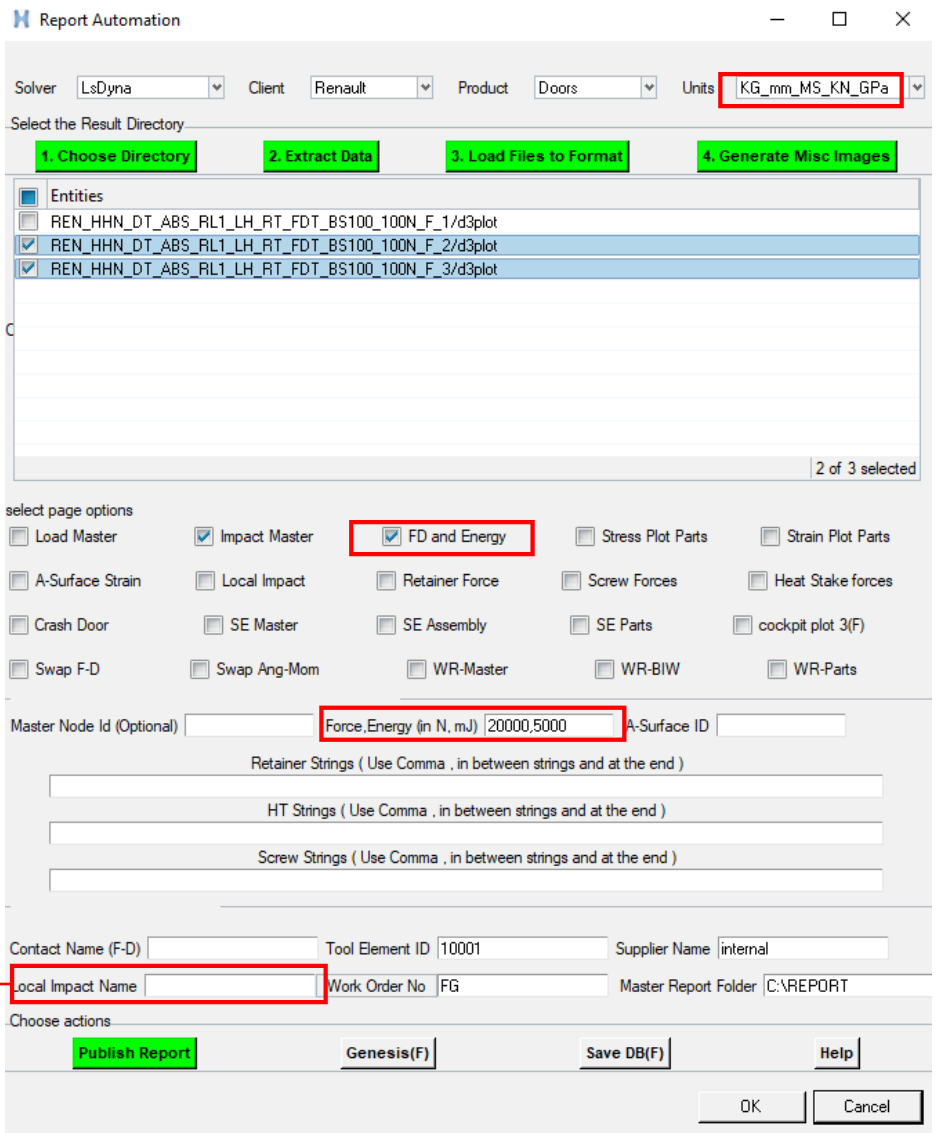
1.7.34 FD and Energy Page.

FD and Energy Page generates Internal energy Vs Displacement plot in Window 1 and Force vs Displacement plot on window 2.
At present this page supports LSDYNA and PAMCRASH.
If this page is required for Abaqus – Please contact methodology department.



1.7.35 FD and Energy Page input.

Select the FD and Energy page tick box . There is a mandatory requirement to enter the limits for force and internal energy.
The units for entry are N and mJ.
Even if the unit system of your model is a different one , Stick with the system mentioned here for FD and Energy entry.
Choosing the overall general unit system will automatically convert the limit values to model unit system.

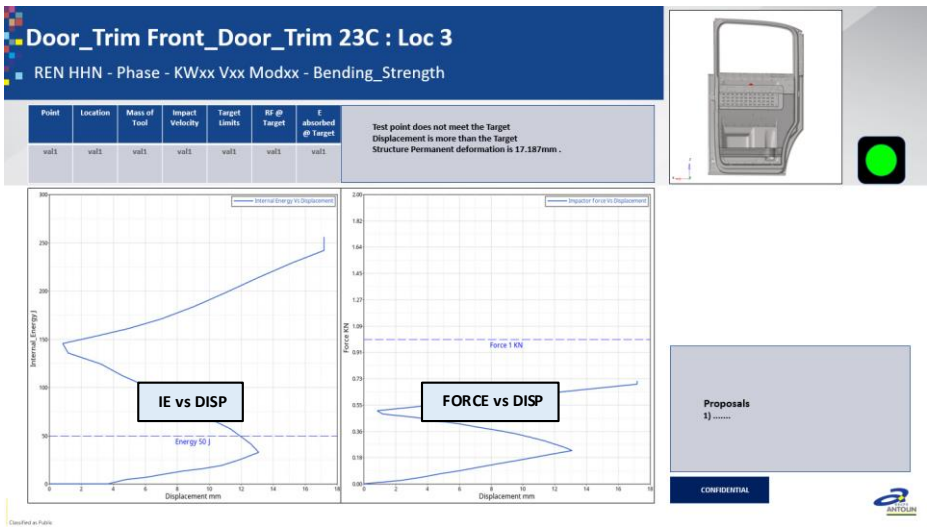


Type in contact name , if there is a deviation from IMPACTOR_TRIM_CONTACT

1.7.0 Generate Misc Images.

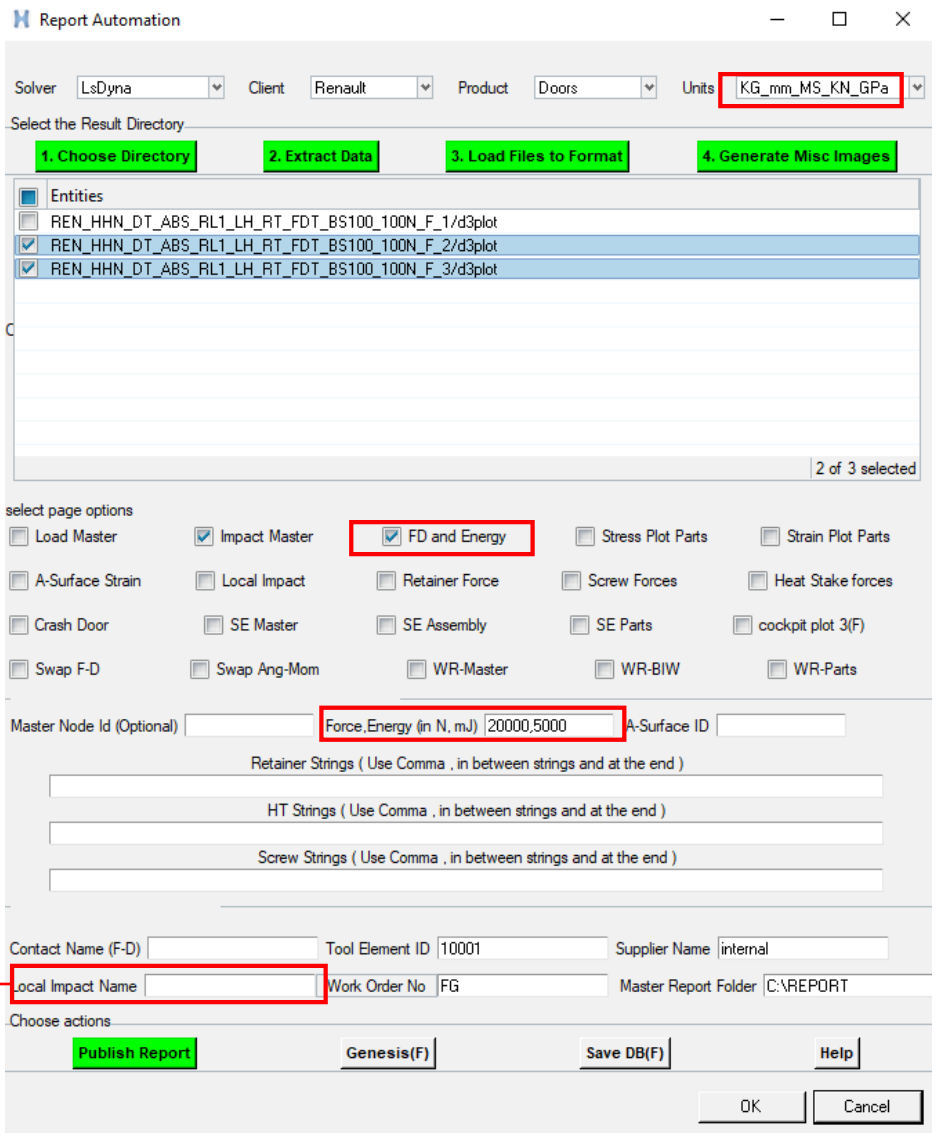
1.7.34 FD and Energy Page.

FD and Energy Page generates Internal energy Vs Displacement plot in Window 1 and Force vs Displacement plot on window 2.
At present this page supports LSDYNA and PAMCRASH.
If this page is required for Abaqus – Please contact methodology department.



1.7.35 FD and Energy Page input.

Select the FD and Energy page tick box . There is a mandatory requirement to enter the limits for force and internal energy.
The units for entry are N and mJ.
Even if the unit system of your model is a different one , Stick with the system mentioned here for FD and Energy entry.
Choosing the overall general unit system will automatically convert the limit values to model unit system.



Type in contact name , if there is a deviation from IMPACTOR_TRIM_CONTACT

1.7.0 Open Topics.

1) FD and Energy Page for Abaqus Solver.
CAE team to inform methodology team when an FD-Energy curve needs to be generated for Abaqus solver.
Pending due to unavailability of proper input and output files.

2) A-Surface Strain for LS-Dyna solver does not work on the workshop data set.
CAE team to inform methodology team when an A –Surface plot need to be generated for Dyna solver.
Pending due to unavailability of proper input and output files.

3) A-Surface strain for Pam Crash Solver
CAE team to inform methodology team when an A –Surface plot need to be generated for PamCrash solver.
Pending due to unavailability of proper input and output files compatible with Hyper view.

4) Local Impact for Pam Crash solver.
CAE team to inform methodology team when a Local Impact need to be generated for PamCrash solver.
Pending due to unavailability of proper input and output files compatible with Hyper view.

5) Torque load case.
CAE team to inform methodology team when an torque curve needs to be generated for any solver.
Pending due to unavailability of proper input and output files.