CAEfatigue Software (Cf)

USER GUIDE

This document provides the User with theoretical background information and practical examples via a Test Problem Library. The example problems are grouped into similar functions and provide the User with simple to complex templates on how to use the software in the Control File view.

This document also provides examples on how to use the TIME2PSD toolset to accurately convert time signals to power spectral densities (PSDs) as well as examples on how to use the Load Scheduler to make customer Time Domain loading events.

For information on the entries required to run a Cf analysis as well as the TIME2PSD tool set and Load Scheduler toolset, the User should refer to the Cf **Quick Reference Guide**. For detailed example from the Test Problem Library, the User should refer to the Cf **User Guide Examples** document.

Disclaimer

CAEfatigue Limited reserves the right to make changes in specifications and other information contained in this document without prior notice.

The concepts, methods, and examples presented in this text are for illustrative and educational purposes only and are not intended to be exhaustive or to apply to any particular engineering problem or design. CAEfatigue assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

User Documentation: Copyright © 2022 CAEfatigue Limited.

This notice shall be marked on any reproduction of this documentation, in whole or in part. Any reproduction or distribution of this document, in whole or in part, without the prior written consent of CAEfatigue Limited is prohibited.

Please consult the **Release Notes** file that comes with the installation package for latest software updates.

CAEfatigue is a trademark or registered trademark of CAEfatigue Limited. The Hexagon logo is a trademark or registered trademark of Hexagon AB. All other trademarks belong to their respective owners.

CAEF:V2022.1:Z:Z:Z:DC-USR-PDF

Table of Contents

1 | REQUIREMENTS AND SETTINGS INFORMATION 6
Getting the Correct Output from the Solver (Nastran, Abaqus, Ansys and LS-Dyna) 7
Setting the ABAQUS Library Directory Path 11
Solver Supported Elements 14
How are Units Handled in CF? 14
System Requirements 16
Licensing Requirements and Special Licensing Parameters 16
Noteworthy Limitations of the Software 17
Current Status of Interoperability with other MSC Software Products 18
Setting Paths for Material Center, Digimat and Nastran use with CAEfatigue 18
Connecting to MSC Material Center 19
Connecting to Digimat 21
Connecting to Nastran 21
3rd Party Software 21
About This Document 21
Contacts 22
2 | SOFTWARE INSTALLATION AND MSC ONE TOKEN USAGE 23
Installing the Software and Pointing to the License File 24
Testing the Installation by Running TPL1 24
CAEfatigue Packaging and MSC ONE Token Usage 27
Features / Capabilities Within Each Package and by Solver 27
MSC ONE Token Usage with CAEfatigue 29
CF IDLE Timeout to Release License 29
3 | INTRODUCTION TO GRAPHICAL USER INTERFACE LAYOUT 30
General Layout of the Software GUI 31
File Menu – Create, Open, Save Cf Jobs and Sessions Files 32
Tools Menu – Access to the Utilities, Fatigue Tools and Settings Menus 32
Help Menu – Access to the Training Material and Software Documentation 36
4 | INTRODUCTION TO THE CAEFATIGUE SOFTWARE (CF) COMMAND ENTRIES AND FIELDS 37
Entries (Commands) and Fields used in Frequency Domain Analysis 38
Brief Description of Entries and Fields used in Frequency Domain Analysis 41
Entries (Commands) and Fields used in Time Domain Analysis 46
Brief Description of Additional Entries and Fields used ONLY for Time Domain Analysis 48
Entries (Commands) and Fields used in a TIME2PSD Analysis with Descriptions 49
Rules for Entries and Fields in a Control File or TIME2PSD 54
Special Rules for the INCLUDE Entry in a Fatigue Control File 54

5 | CONTROL OF OUTPUT RESULTS AND LOADS SCALING 55
Output Control Options – OFILTYPE & LOGLVL (in vIBFAT), ELSET (in vFTGDEF) and EVNTOUT (in vFTGSEQ) 56
CSV Output File Formats – Main, PSD, DVA, D_REL, Spot, Seam, Force, etc. 59
Global Scaling Options - FE_L_MAG (in FE_UNIT), TSCALE (in vFTGSEQ) and SCALE (in vFTGLOAD) 61
Local Scaling Option – SCALE (in FNOTCH) 65
Post Processing Options 67
6 | INTRODUCTION TO THE FATIGUE CONTROL FILE VIEW 68
Overview and Introduction to the Fatigue Control File Wizard 69
Wizard Toolbox 70
Job Execution – Run, Edit View, Log View and Console View 76
Viewing Results – Frequency Domain Analysis Outputs 77
Viewing Results – Time Domain Analysis Outputs 91
Viewing Results – Fringe Plotter Properties Box: Viewing Options 91
Viewing Results – Fringe Plotter Selection Menu 97
7 | INTRODUCTION TO THE FATIGUE PROCESS FLOW VIEW 98
Current Limitations of the Process Flow in this Release 99
Overview and Introduction to the Process Flow Menu Items 99
A Frequency Domain Process Flow Example – TPL1.cfmodel 100
A Time Domain LQSTATIC, LMTRANS Analysis – TPL358 Example 112
Viewing Results – Same as Control File view 120
8 | INTRODUCTION TO THE TIME2PSD CONTROL FILE VIEW 121
Overview and Introduction to the TIME2PSD Wizard 122
More Information about SUBEV, AutoD and AutoT. 126
Viewing Results via the FFT Plotter and PSDM Plotter 127
RMS, E[0], E[P] and Irregularity Factor Statics Available in the FFT Plotter 131
Dealing with Signal Offsets: RMS versus Standard Deviation 132
9 | INTRODUCTION TO THE TIME2PSD PROCESS FLOW VIEW 134
Overview and Introduction to the TIME2PSD Template File 135
More Information about KEEP FILTER, AutoD and AutoT. 142
Viewing Results via the FFT Plotter and PSDM Plotter 143
10 | INTRODUCTION TO THE TIME DOMAIN LOADS SCHEDULER 147
Overview and Introduction to the Load Scheduler Menu Items 148
File Import of Existing Time Signals 149
File Export of Channels and Events 149
Generation of Unique Time Signals 150
Potential Issues with Screen Resolution for Display 152
Potential Issues with Signal Digitizing for Output 153

Example of Adding Additional Signals to Events *155*
11 | INTRODUCTION TO THE TIME DOMAIN LOAD SCENARIO CREATER **157**
Overview and Introduction to the Load Scenario Creator for Time Domain Linear or Nonlinear Static Analysis *158*
Proportional Loading Scenario *159*
Pre-stress Incorporated with Proportional Loading Scenario *160*
Cycle Between Stress States (ON/OFF) Scenario *161*
12 | INTRODUCTION TO THE RELIABILITY MODE FOR RUNNING MULTIPLE JOBS WITH VARYING INPUTS **163**
Overview and Introduction to the Reliability Mode *164*
Review of Reliability Mode Menus from within the Process Flow *165*
Options for Adjusting Variables and Selecting Outputs *166*
Review of Reliability Mode Progress Box and Output Plots *166*
Files Required and Generated by the Reliability Mode *168*
Contents of the JSON Configuration File *168*
Viewing and Plotting Results from the HDF5 file. *172*
Running Reliability Mode in BATCH within Windows or Linux *173*
13 | INTRODUCTION TO ROBUST DESIGN FOR RUNNING MULTIPLE MSC NASTRAN JOBS WITH VARYING INPUTS **174**
Overview and Introduction to the Robust Design *175*
Review of Robust Design Menus from within the Process Flow *176*
Running Robust Design in the Process Flow *180*
Understanding the Correlation Values *183*
Understand Quality and Robustness from the Ant Hill Plot *185*
So why is the Robust Design data important? *189*
What's Next? *189*
Running Robust Design in BATCH within Windows or Linux *190*
14 | INTRODUCTION TO THE TIME DOMAIN MULTIAXIAL ASSESSMENT **192**
Overview and Introduction to the Time Domain Multiaxial Assessment Feature *193*
Simple Biaxiality Assessment *195*
Standard Multiaxial Assessment *196*
Automatic Multiaxial Assessment – not yet supported *197*
15 | INTRODUCTION TO THE NEF DECK CREATION INTERFACE FOR TIME DOMAIN ANALYSIS **201**
Overview and Introduction to the NEF Deck Creator *202*
Review of NEF Deck Creation Menus *202*
Exporting or Running a NEF Analysis *203*
16 | APPENDIX 1: CSV FILE FORMATS **206**
Main CSV File Format *207*
PSD.CSV File Format *209*
_DVA.CSV File Format *210*

_D_REL.CSV File Format *211*
_SPOT.CSV File Format *212*
_SEAM.CSV File Format *213*
_FORCE.CSV File Format *214*
_CRITOUT.CSV File Format *215*
TIMEOUT_<filename>.h5 File Format *216*
17 | APPENDIX 2: EXAMPLE MSC NASTRAN SOL112 INPUT FILES WHEN USING FORCE INPUTS 218
18 | APPENDIX 3: EXAMPLE MSC NASTRAN SOL111 INPUT FILES WHEN USING FORCE INPUTS 220
19 | APPENDIX 4: MMPDS MATERIAL DATA TABLE 222
20 | APPENDIX 5: TYPICAL STRESS-LIFE (SN) MATERIAL DATA TABLE 224
21 | APPENDIX 6: TYPICAL STRAIN-LIFE (EN) MATERIAL DATA TABLE 227
22 | APPENDIX 7: CAEFATIGUE – OPEN SOURCE SOFTWARE 229

Theoretical Background Topics

WHAT VALUE SHOULD I USE FOR SAMPLING RATE IN THE LOAD SCHEDULER? 153

1 | REQUIREMENTS AND SETTINGS INFORMATION

The avoidance of fatigue cracks (fatigue damage) and excessive response due to resonance is a design requirement for nearly all mechanical engineering systems. In fact, for many everyday products like cars, aircraft, trains, etc., fatigue life (or durability) is the limiting design requirement. Testing plays an important role in determining the required level of durability, but analysis is also vital at all stages of product development.

Typical durability analysis processes involve the integration of Finite Element Analysis (FEA) methods to evaluate the structural stress conditions that are caused by real world loading conditions. When the loading source is random in nature, the stress response is then also random in nature and the fatigue damage can be called “vibration fatigue”, “acoustic fatigue”, “sonic fatigue”, “very high cycle fatigue”, etc. In order to evaluate this kind of phenomena analytically, FEA is usually used to calculate the stresses or strains caused by the random loading conditions.

There are several major codes in existence that are used to calculate these random stress or strain conditions. These include, but are not limited to, MSC Nastran (Hexagon), NX Nastran (Siemens), Ansys and LS-Dyna (Ansys Corp), Optistruct (Altair) and Abaqus (Dassault Systems). These codes incorporate analysis procedures to deal with the dynamic response (resonances) in structural systems and they also have vertical applications to perform fatigue life calculations.

These random response and fatigue analysis procedures are generally complete and fully featured when the dynamic behavior is defined in the time domain (using time histories for the loading conditions). However, most dynamic analysts prefer to work in the so-called frequency domain where loads are defined as a function of frequency. CAEfatigue Software (CF) has been created to satisfy the needs of Users where these existing codes are functionally limited. The concept of Cf is to combine stress solver environments with fatigue life calculations where dynamic response is important both in the frequency domain and time domain. The fundamental objective of Cf is that it shall be fast, easy to use and understand, be capable of handling very large models, be able to cater for multiple correlated loads, be able to us mixed random and deterministic loads, have advanced fatigue capabilities (including Strain-Life) and be able to utilize appropriate stress output formats. It is intended to work (as a vertical application) with all major stress solvers and in some cases, it has embedded into the stress solvers.

Getting the Correct Output from the Solver (Nastran, Abaqus, Ansys and LS-Dyna)

MSC Nastran, NX Nastran, NEi Nastran

General Information

- Nastran support is provided for **OP2** files or **HDF5**. NOTE: currently the HDF5 format can only be used for a single Input PSD analysis (using a Nastran SOL111) or for a linear static analysis (LQSTATIC) or for a non-linear static analysis (NQSTATIC) using a Nastran SOL400 analysis.
- Nastran HDF5 format is also support for SOL111 analysis of composite material in a CAEfatigue RANDOM analysis (only).
- The OP2 file must be specified using **PARAM,POST,-1** or **PARAM, POST, 1**.
- Stresses must be written using the default method **(SORT1).**
- Stresses must be requested in the **REAL / IMAGINARY format** i.e., STRESS(SORT1, REAL, PLOT) or **MAG / PHASE format** i.e., STRESS(SORT1, PHASE, PLOT).
- Stresses can be requested using the **CORNER** option (either BILIN, CUBIC or SGAGE) or the **CENTER** option, the following applies:
    - If corner results are selected in Nastran, you will also get center results. Cf will output results for both element center and element nodes (nodes).
    - If center results are used in Nastran, Cf will output results for the element center ONLY.
- Nastran output must include the **DISP or DISPLACEMENT output** request if displacement, acceleration or velocity response output is required from Cf at the nodes (i.e., D, V or A output for nodes). Reading the

node displacement outputs from the Nastran OP2 file is done within Cf by using the ABSRESP entry in the vFTGDEF entry. NOTE: Cf will calculate displacements, accelerations and/or velocities from the solver nodal displacements.

- If Force (vFTGDEF: ABSRESP: ABSVAR=F) is requested, then **MPCFORCES(SORT1)** for an element set must be specified in the Nastran file.
- If Monitor Point Force (vFTGDEF: ABSRESP: ABSVAR=M) is requested, then **PARAM,POST,1** and **PARAM,POSTEXT,YES** must be specified in the Nastran file.
- If Relative Response (vFTGDEF: RELRESP) is requested, then **PARAM,POSTEXT,YES** must be specified in the Nastran file.
- If Spot Welds (**vFTGDEF: SPOTW**) is requested then **PARAM,POSTEXT,YES** must be specified in the Nastran file.
- If Spot Welds (**vFTGDEF: SPOTW**) is requested then **MPCFORCES(SORT1)=SETID_NODES** must be specified, where SETID_NODES is the set number for the list of grids (nodes) that form the corners of the spot weld CHEXA's. Note: technically, we are getting the forces in the RBE3 elements at the CHEXA node locations.
    - Make sure all grids are in one set, e.g. MPCFORCES (SORT1)=22. NOTE: Nastran will not accept multiple set number i.e., 22, 23, 24.
    - Another element set containing all the CHEXA's will be needed for the Cf analysis. This should be an FSET3 format. This should be an element set, which directly corresponds with the grid set mentioned above.
    - If conventional stress based fatigue calculations and/or random analysis calculations are to be done on other parts of the model, then the appropriate STRESS request and DISP requests must also be made in parallel with the MPCFORCES request.
- If User Welds (**vFTGDEF: USERWHS**) is requested then **FORCE(PLOT)=SETID_ELEMS** must be specified, where SETID_ELEMS is the set number for the list of CBUSH elements that are used to represent the user weld.
- If Seam Welds (**vFTGDEF: SEAMW**) is requested then **STRESS(SORT1, PLOT, CUBIC)=SETID_ELEMS** must be specified, where SETID_ELEMS is the set number for the list of elements running along either side of the seam weld. This set should also be created in FSET3 format for use in CF.
    - If conventional stress based fatigue calculations and/or random analysis calculations are to be done on other parts of the model, then the appropriate STRESS request and DISP requests must also be made.

- It is recommended that only one data block containing stresses using STRESS or ELSTRESS (Physical Output) is requested. This can be achieved by using only one FREQUENCY (Dynamic Solutions Conditions) entry above the 1st subcase or by having the same FREQUENCY ID repeated within all subcases.
- Where a mean stress calculation is specified, the model configuration (elements included in SETS, etc.) must be identical for both the SOL111 (or SOL108) and SOL101 (or SOL106) jobs.
- Subcases in the Nastran file are numbered, and these numbers must be referenced in the mapping entry OP2MAP.
- Cf supports both multiple subcases being used consecutively, and multiple subcases being used simultaneously (with correlation).

Information specific to Time Domain Modal Transient Analysis (LMTRANS)

- CF expects the modal stresses (OP2) and modal loading (PUNCH) to be calculated and outputted from a single Nastran run. The modal stresses are outputted to the OP2 file using a subcase (typically subcase 1) that contains the entries ANALYSIS = MODES and STRESS (SORT1). The modal loadings are outputted to the PUNCH file additional subcases i.e., subcase 2, 3, 4, etc.
- The User can also achieve the same output by running the SOL103 analysis first (creating the modal stress OP2 file), followed by a SOL112 with a restart entry to create the modal loading PUNCH files.
- Conducting a SOL112 analysis without the subcase 1 entries, will results in an OP2 files that contains direct stresses (not modal stresses). This will result in an error message within Cf because the OP2 file and PUNCH files will have a mismatch in the data.

OPTISTRUCT

- Optistruct support is provided for OP2 files in the same way as Nastran. See MSC Nastran section above.
- Optistruct behaves in a very similar way to MSC Nastran with a couple of small exceptions.
- Firstly, when writing stresses for separate subcases, Optistruct will always write separate data blocks. This needs to be flagged in the Cf Control File by specifying the source entry as “OS” in the vIBFAT entry.
- Secondly, when comparing analysis runs with other MSC Nastran codes be careful to check which PARAM instructions are applicable. Not all PARAM codes are used in Optistruct.
- ABSRESP / RELRESP is only supported with the latest 2018 version of Optistruct.
- If center results are used in Optistruct, Cf will output results for the element center ONLY.
- If corner results are used in Optistruct, Cf will add additional results at the centers based on a linear interpolation from the corners results. This is done for purposes like hot spot detection but generally the use of these center results should be avoided (the corner results are more accurate).

LS-DYNA

- Ansys LS-DYNA support is provided for 3dplot files with center stresses for linear transient dynamic analyses only (LTRANS).
- CF does not currently support the addition of static offset
- Ansys LS-DYNA version from April 2020 is tested and supported. Most existing version are also expected to work with CAEfatigue but have not been tested.

ANSYS

- Ansys support is provided for RST files with single or multiple steps (subcases) within each RST.
- Multiple RST’s can be specified in order to apply multiple events (consecutively) but only single random loads are supported for this release (i.e., multiple correlated random inputs are not supported).
- It is assumed that Ansys steps (subcases) are numbered consecutively according to their order in which they are specified. Substeps in the Ansys RST file are not supported.

- Ansys will only output a result at corners (OUTRES,STRS,ALL). Cf will add additional results at the centers based on a linear interpolation from the corners results. This is done for purposes like hot spot detection but generally the use of these center results should be avoided (the corner results are more accurate).
- CF does not currently support the ABSRESP entry (in vFTGDEF) for Ansys so you cannot ask for PSDs of displacement, velocity or acceleration in Cf using the Ansys RST file.
- If a static load is applied this must be provided in a separate RST file. Note: the first subcase in the file will be taken as the Static subcase for the analysis.
- Ansys versions 14.5 and later are supported.

ABAQUS

- Abaqus support is provided for ODB files with single or multiple subcases.
- You can use the default Gauss point output results or set the Abaqus results output to either corners or centers (***Element Output, position = NODES** or ***Element Output, position = CENTROIDAL**) but not to both. Note that in the CSV Control File, the appropriate entry for GIDOUT (Center or Corner) should be specified on the vIBFAT entry.
- If corner results are used in Abaqus, Cf will add additional results at the centers based on a linear interpolation from the corners results. This is done for purposes like hot spot detection but generally the use of these center results should be avoided (the corner results are more accurate).
- Abaqus output must include the *Node Output request for TU displacements if random response output is required at the nodes (i.e., D, V or A) from the Cf run. Reading the node displacement outputs from the Abaqus ODB file is done within Cf by using the ABSRESP entry in the vFTGDEF entry.
- For Spot and Seam Weld analysis, the User must define element sets and the job parameters to be considered for the fatigue analysis and output desired.
- Make sure that the ODB file containing the required harmonic subcases contains only those subcases.
- An ODBMAP entry must be used in Cf to assign number to the STEP names. **This is REQUIRED**.
- Abaqus 6.14, 2016 thru 2022 are supported. Note: Linux Abaqus v2016 is only supported with the FP1825 Fix Pack or newer.

NOTE: If using Abaqus as the solver, you must start your input PSD definitions at f1=0, g1=0 to eliminate the influence of the stress output by Abaqus at 0 Hz. This is a meaningless output that is used in the transfer function and will cause errors in your response results if the input PSD is not set to zero as the initial point. You can also set the second point to amplitude 0 as described in the example.

Example: If your first actual input PSD point is at, say 8 Hz, 15 G^2/Hz, make the first two points f_1=0, g_1=0 and f_2=7.999, g_2=0. Make the third point f_3=8.0, g_3=15.0 and continue with your remaining PSD data points. This will make sure that the 0 Hz stress output point provided by Abaqus (and used in the transfer function) does not influence the response output.

On the following page is a matrix of supported capabilities for each supported solver.

Whichever stress solver is used great care should be taken to ensure there is an adequate distribution of solution frequencies in the transfer function. Solution frequencies need to be adequately spaced to pick up resonances (the FREQ1 and FREQ4 entries in MSC Nastran are a good tool for this) as well as a finely spaced set of solution frequencies in any location where the input load changes.

In all cases, be careful about specifying frequencies at or near zero hertz. Strange results can occur (depending on the unit of load applied) as the frequencies tend to zero. For example, with a multi input full car system it might be appropriate to start the frequencies at 0.5 Hz or 1.0 Hz.

Also, be careful what range of frequencies have been requested for the eigenvalues, especially if frequencies have been requested around the eigen frequencies, because a very large number of modes can occur as the frequency range is increased and the higher modes may have no influence. It is useless to specify solution frequencies in the solver run above the maximum loading frequency in the input PSD(s).

The HOTSPOT flag (in vFTGDEF) can now be used to determine a filtered set of elements, and this could be a logical way to reduce very large models before final processing with the Cf code.

*Force is only supported for MSC Nastran element CBUSH and Abaqus element CONN3D2.

	Single PSD inputs - stress output	Single PSDs plus static inputs - stress output	Single PSDs plus deterministic inputs - stress output	Single PSDs plus deterministic plus static inputs – stress output	Multiple correlated inputs - stress output	Multiple correlated plus static inputs - stress output	Multiple correlated plus deterministic inputs	Single PSD Inputs - D, V, A or F * output	Single PSD plus static Inputs - D, V, A or F output	Multiple correlated inputs - D, V, A or F * output	Relative random response output (RELRESP)	FLUCT type plus static loadings - stress output
Nastran	yes	yes	yes	yes	yes	yes	no	yes	yes	yes	yes	yes
Abaqus	yes	yes	yes	yes	yes	yes	no	yes	yes	yes	yes	no
Optistruct	yes	yes	yes	yes	yes	yes	no	yes	yes	yes	yes	yes
Ansys	yes	yes	yes	yes	no	no	no	no	no	no	no	no

Setting the ABAQUS Library Directory Path

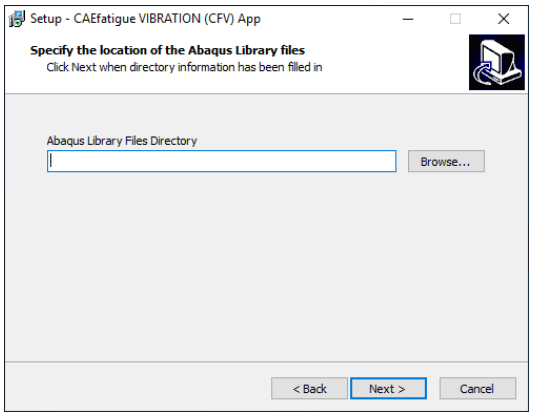
If you choose to use ABAQUS as a Solver, Cf requires access to the Abaqus Library files, which are needed to read in the frequency response function (FRF) ODB file as well as to view the Abaqus model.

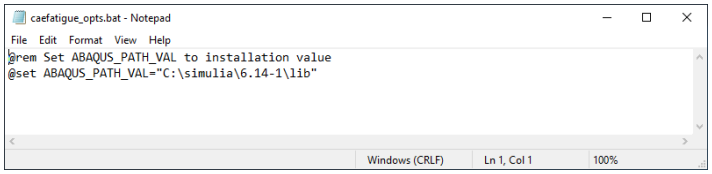
During the installation process, the User can choose to provide the location of the Abaqus Library directory by pointing to the individual directory for a certain version of Abaqus or by pointing to a text file that has multiple Abaqus Library directories already defined.

Below are 3 options to define the Abaqus Library directory location in the order used by CF.

Option 1 – Provide Directory Location during Cf Installation via **CAEFATIGUE_OPTS.TXT** file

The Abaqus Library Directory is an optional entry available during the Cf installation process. If you select the appropriate check box during the installation, you will be asked to enter the location of the Abaqus Library. This information will be entered into a new file called "***caefatigue_opts.txt***" and placed in the installation directory; by default, C:\Program Files\MSC.Software\CAEfatigue\<ver>\caefatigue_opts.txt.

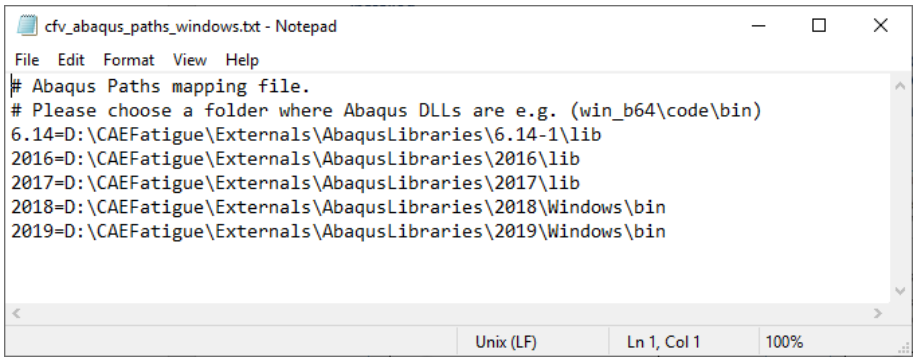




You can also set the path to point to a text file that defines multiple Abaqus Library directory locations for different versions of Abaqus. By doing this, the User can run Cf jobs with different Abaqus models and not worry about the version of those models because the text file will tell Cf where to find the needed libraries.

Below is an example of the text file format. This can be done during the installation (if you have the text file already created) or afterwards by simply editing the OPTS file to replace the contents with the contents of the text file.

Note: please point to the “LIB” directory for Abaqus versions 6.14, 2016 and 2017. Point to the “BIN” directory for Abaqus versions 2018 thru 2021.



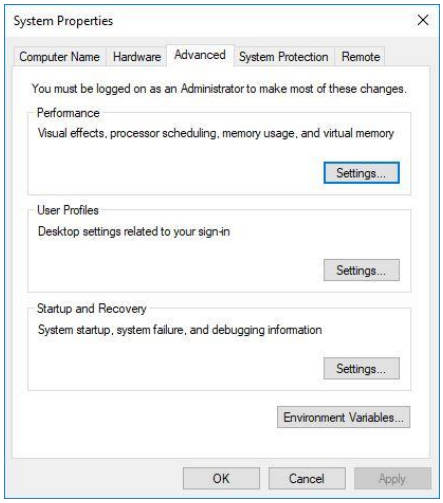
Option 2 – Provide Directory Location via the ABAQUS_PATH Environment Variable

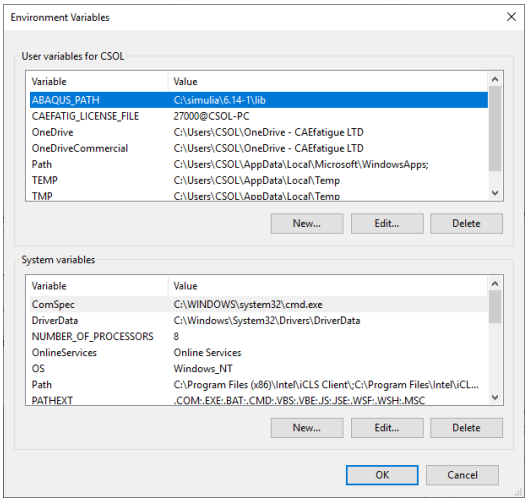
If you choose not to set the directory location via Option 1, the User can choose to create an environment variable called ABAQUS_PATH through the System Properties window. As above, the User can set the variable to point to the location of a Abaqus library directory for a specific version of Abaqus OR the User can point to a text file that contains the Abaqus library directory locations for multiple versions.

Below is an example of the ABAQUS_PATH for a single directory location and an example of the ABAQUS_PATH for a text file that defining the directory paths to multiple Abaqus versions.

ABAQUS_PATH=c:\simulia\6.14-1\lib

ABAQUS_PATH=c:\Abaqus\Abaqus_Directory_Definitions.txt



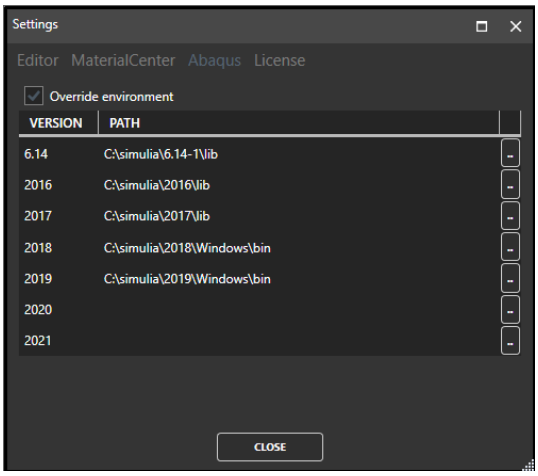


Option 3 - Provide Directory Location via the TOOLS> SETTING menu

If you do not set the Abaqus Library Files directory via Option 1 or Option 2, you can set the directory paths using the SETTINGS menu in the CAEfatigue GUI (CFG).

To do this, select the SETTING icon when the software is up and running. This will open a SETTINGS menu box where you can set individual paths to individual Abaqus versions. Please ensure the “OVERRIDE ENVIRONMENT” box is CHECKED. This will tell Cf to override both Option 1 and Option 2 (if they are set) and use the locations defined in the Abaqus Setting window. If the User wishes to use Option 1 or Option 2 at a later date, the “OVERRIDE ENVIRONMENT” checkbox must be UNCHECKED.

By default, the directory locations defined in the Abaqus Settings window are stored in a hidden text file called **…\AppData\Roaming\CAEfatigue\CF_abaqus_paths.txt.** If there is an error with a directory path specified in this window, Cf will report the error in the LOG file and reference this hidden file. If you see this error, the issue is within the Abaqus Setting window. Example of Abaqus settings box is shown below.



Solver Supported Elements

Nastran

Name	Type	Number of GID's	
CQUAD4/8	Shell	4 corners + center on 2 layers	= 10 GID's
CTRIA6	Shell	3 corners + center on 2 layers	= 8 GID's
CTRIA3	Shell	3 corners + center on 2 layers	= 8 GID's
CTETRA	Solid	4 corners + center	= 5 GID's
CHEXA	Solid	8 corners + center	= 9 GID's
CPENTA	Solid	6 corners + center	= 7 GID's
CBUSH	Generalized Spring and Damper (for forces)	Used to connect 2 grids of other elements	= none
CPYRAM	Solid (pyramid)	5 corners + 8 midside, if desired	= 5 to 13 GID's

LS-DYNA

The elements supported are NEL8, NEL4, NELT, NEL2 and NEL10.

Ansys

The elements supported are 181, 281 (quads) and 185, 186, 285, 187 (solids)

Abaqus

Triangular Shell (3 main nodes) - CPE3**, CPE6**, CPS3**, CPS6**, CPEG3**, CPEG6**, STRI3**, STRI6**, SC3**, SC6**, CAX3**, CAX6**).
Rectangular Shell (4 main nodes) - CPE4**, CPE8**, CPS4**, CPS8**, CPEG4**, CPEG8**, SC4**, SC8**, S4**, S8**, CAX4**, CAX8**).
Solids (4 main nodes) – C3D4**, C3D10**.
Solids (6 main nodes) – C3D6**, C3D15**.
Solids (8 main nodes) – C3D8**, C3D20**, C3D27**.
Membrane elements M3D3, M3D6 M3D4 M3D8.
CONN3D2 element is supported for force output.

How are Units Handled in CF?

CF assumes the following default units come from the Solver:

Stress => MPa
Force => N
Displacement => mm

If this is not the case, the User must include an **FE_UNITS** entry to define the actual stress units coming from the Solver i.e., Pa, PSI, KSI, or User Defined units. Cf will write out all results in the FE_UNITS or units provided by the FE model when dealing with displacement, velocity, acceleration or force.

Any material data is also expected to be in the correct units to work with stress in MPa. If they are not, the User must also define the actual material units in the **vFTGMAT** entry.

NOTE: Cf expects a consistent system for units and will output results in the following units:

Examples of Consistant Systems of Units for Structural Analysis											
System of Units	*Input*								*Output*		
	Length	Force	Elastic Modulus	Mass	Mass Density	WTMASS Param	Acceleration (1 G)		Disp	Force	Stress
1 Metric meter-kg	m	N	Pa	kg	Kg/m^3	1.0	9.807	m/sec^2	m	N	Pa
2 Metric mm-ton	mm	N	MPa	t or Mg	t/mm^3 or Mg/mm^3	1.0	9807	mm/sec^2	mm	N	MPa
3 English ft-lb	ft	lb_f	psf	slug	$slug/ft^3$	1.0	32.17	ft/sec^2	ft	lb_f	psf
4 English in-lb (mass)	in	lb_f	psi	lb_f - sec^2/in	lb_f - sec^2/in^4	1.0	386.1	in/sec^2	in	lb_f	psi
5 English in-lb (force)	in	lb_f	psi	lb_f	lb_f $/in^3$	0.00259	386.1	in/sec^2	in	lb_f	psi

Result Quantity	Units
Moment – M0	FE_UNITS or default MPa2
Moment – M1	FE_UNITS or default MPa2 * Hz
Moment – M2	FE_UNITS or default MPa2 * Hz2
Moment – M4	FE_UNITS or default MPa2 * Hz4
E[P]	Hz or (cycles / sec)
E[0]	Hz or (cycles / sec)
Irregularity Factor	No units
Displacement	Units from FE model
Velocity	Units from FE model
Acceleration	Units from FE model
Force	Units from FE model

Result Quantity	Units
RMS stress = √ (M0)	FE_UNITS or default MPa
RMS strain	No units
Mean stress	FE_UNITS or default MPa
Damage	No units
Life	Life per DUR_UNITS
Margin of Safety (MOS)	Percent
Plasticity Index	No units
Freq at Peak Response	Hz
Clearance (collision detection)	Units from FE model
Collision Probability	Percent

System Requirements

The Cf application runs on 64-bit Intel and Intel-compatible processors running a 64-bit version of either Windows or Linux.

- For Windows, Windows 8 or later is supported. The Graphical User Interface support program also requires the Microsoft .NET 4.5 (or later) framework and several Microsoft Visual C++ Runtime Libraries depending on the version of Windows being used. Note that these files will be installed automatically if they are not already installed.

- For Linux, SUSE 12 SP4 or later, RHEL 7.7 or later and compatible systems are supported.

The system should have a minimum of 16 GB RAM. Approximately 4 GB of free disk space is required for the installation and additional disk space will be required to run the Cf application. It is **highly** recommended that additional RAM be available for large models. Cf can handle virtually any size results file provided the hardware has sufficient memory to do the solver analysis.

Licensing Requirements and Special Licensing Parameters

Cf requires the installation of MSC Licensing Helium which is available under the MSC Licensing product page within the MSC Software Download Center. Note: if after January 2021, any new installation will require MSC **Licensing Lithium**.

Job Execution / Loss of Connection Timeout Parameter - Command Prompt / Linux Build

If you wish to run consecutive jobs, perhaps in batch mode through your own script, Cf has a special parameter in the script file (caefatigue.bat) that allows Cf to pause execution of the next analysis in the queue while waiting for a license / tokens to free up.

To invoke the CAEfatigue executable with the TIMEOUT parameter, whether from the Command Prompt or in Linux, use the line below.

CAEFatigue -license-timeout <value> *filename.txt*

Where <value> means

- If -1, the software will wait indefinitely for the next available license / tokens to run the next job in the queue.
- If 0, the software will not wait and will terminate if a license or tokens are not available to output the results of an ongoing analysis or for a new analysis.
- If any other value, the software will wait that value in minutes for an available license / tokens to write out the results of an ongoing analysis or to run the next job in the queue.

By using the above, the User can add several CAEfatigue.bat entries into a User defined script and if a license is not available, the software will wait for a license to free up instead of terminating Cf because no license was available.

This parameter is also invoked if the connection to the license server is lost. CAEfatigue will check for a license just before writing out results from an ongoing analysis or before starting a new analysis. If a license cannot be found, the job will suspend for the TIMEOUT period and if no license is available during that time, the software will present a message to Resume or Terminate.

NOTE: this adjustable Timeout parameter is only available for scripting i.e., from the Command Prompt or from Linux.

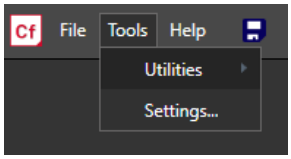
Loss of Connection Timeout Parameter - Windows

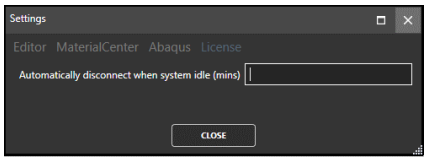
This parameter is invoked if the connection to the license server is lost. If the connection is lost, CAEfatigue will display a warning in the software banner that the license is unavailable, but the software will continue working for 60 additional minutes. The banner warning will stay present for the entire 60 minutes, even if the connection is reestablished.

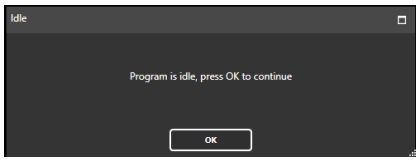
At the end of the 60 minute period, CAEfatigue will provide the User with an option to attempt a reconnection to the license server or to close the software.

Software Idle Timeout Parameter - Windows

CAEfatigue offers an IDLE timeout parameter that can be set in the TOOLS> Settings box. The IDLE timeout will recognize that you have stepped away from the computer if there is no activity (i.e., keystrokes or mouse clicks) for a period of time and will automatically disconnect you from the license so it can by freed up for others to use. Note: A value of zero (0) or a blank field, means the parameter is disabled.







ADMINISTRATIVE USE: The Idle Timeout can be set by the Administrator as an Environment Variable. Simply create the variable shown below and set the VALUE to the number of minutes desired before idling the software and releasing the license.

CFV_LICENSING_TIMEOUT = <value in minutes>

Noteworthy Limitations of the Software

The following limitations exist in this release:

- The Ansys solver is not supported for ABSRESP and RELRESP in the vFTGDEF entry. It is also not support for multi-input PSD analysis.
- The use of Deterministic loads (DETLOADS, SINGSINE and SINESW) and the use of principal stresses (ABSMAXPR or MAXPREST) are not supported for multi-input PSDs analysis. If principal stress output is requested by mistake, CAEfatigue will change the output to sign von Mises and put a not in the LOG file that this was done.
- The response time history created by TIMEOUT is in FEA stress and not Neuber stress.
- Biaxiality Assessment (vFTGPARM>MULTIAX) cannot be run along with Factor of Safety (vFTGPARM>FOS) at the same time.
- Factor of Safety (FOS) does not yet work with LTRANS, or analyses involving critical plane, spotwelds or user welds.

Current Status of Interoperability with other MSC Software Products

CAEfatigue is continually working to improve our connections to existing MSC software products. Below is our status as of this release.

MSC Nastran: Cf is embedded into MSC Nastran for both Time Domain and Frequency Domain. The time domain interoperability has been improved with the upcoming release of MSC Nastran to the use of multiple SN curves. The frequency domain interoperability is currently very old (NEVF) but will be updated in the future. Cf also provides a mechanism to generate / run NEF time domain run from within CF.

ADAMS: Cf will accept RSP files from Adams with time domain acceleration loading for a time domain or frequency domain analysis (after loads have been converted to PSDM files with TIME2PSD). MNF (modal stresses from Nastran) and RSP (modal loads) from Adams can also be read into Cf to allow a modal transient time domain analysis.

Romax: Cf will accept a linear Enduro stress OP2 file for a modal transient analysis (MTRANS) within CF.

Marc: Cf will accept a nonlinear HDF5 stress file for a nonlinear static analysis (NQSTATIC), but only for model with solid elements. When using Marc with the embedded Cf product, we also allow a nonlinear transient analysis (NLTRANS) analysis. NOTE: The OP2 file format from Marc is not supported.

Apex: There is currently a Cf button in APEX that will open the Cf interface.

Digimat: Cf will connect to Digimat to obtain SN curves for a time domain static Cf analysis (LQSTATIC) using 2 methods:

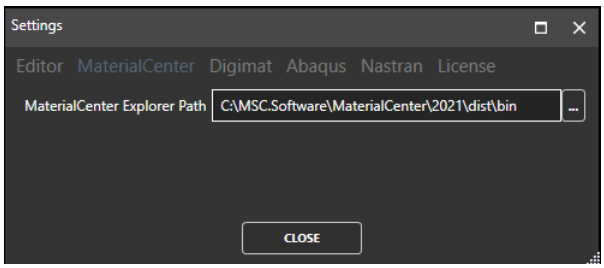
- Time Domain Critical Plane: Requires a MAT and DOF file from Digimat to be specified by the User and Cf will request the SN curve for every location and direction of the critical plane analysis.
- Fix R-Ratio: Requires a MAT and DOF file from Digimat to be specified by the User plus an R-Ratio value from a single on/off time domain loading.

Material Center: Cf has a connection to Material Center, but further work is required by the Material Center team to make the interoperability more functional.

Future planning includes improvements to the above connection as well as connections with Simufact, Actran, Cradle and Volume Graphics.

Setting Paths for Material Center, Digimat and Nastran use with CAEfatigue

In order to make use of the interoperability with other MSC product, Cf requires the User provide various paths to the product. Cf provide a mechanism for this through the Tools>Settings menu.



EDITOR: Allows options for the automatic saving of Cf files when the Run button is pushed. Options are to (1) Always Ask to save the file when Run is selected, (2) to Always Save the file when Run is selected, and (3) to Never Save the file when Run s selected.

Material Center: Requires the User to set a path to the file that starts Material Center Explorer. See further information below.

Digimat: Requires the User to set a path to the Digimat executable file.

Abaqus: Requires the User to set a path to the Abaqus libraries needed to read in an OBD file.

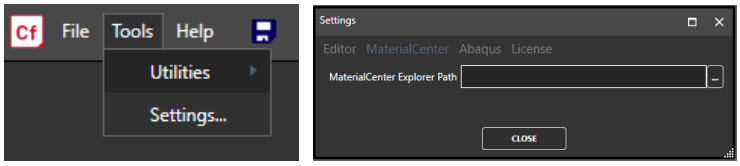
Nastran: Requires the User to set a path to the Nastran executable file.

License: Allow the User to set an IDLE time in minutes. If the PC remain idle for that period of time, the Cf software will disconnect from the license server.

Connecting to MSC Material Center

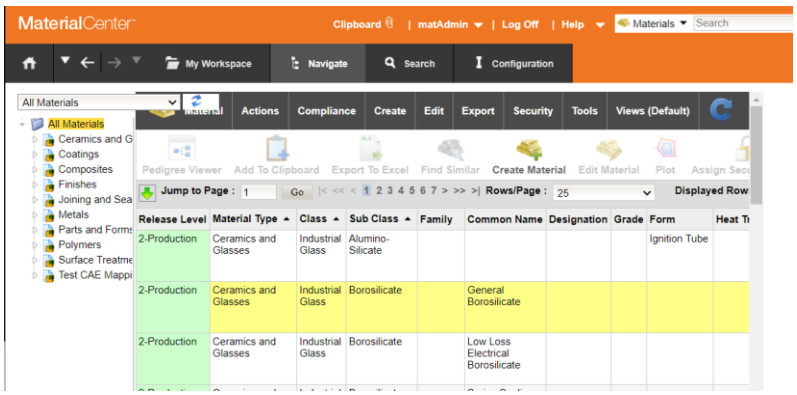
CAEfatigue offers the User a connection to **MSC Material Center** via a plugin called **Material Center Explorer** (available for download from withing Material Center).

Material Center is a database of materials properties that resides on a Server (not on a local PC). Material Center Explorer is a Material Center plugin but must be installed on the local PC. Once installed, the User must provide CAEfatigue with a path to the executable from within the TOOLS> SETTINGS.

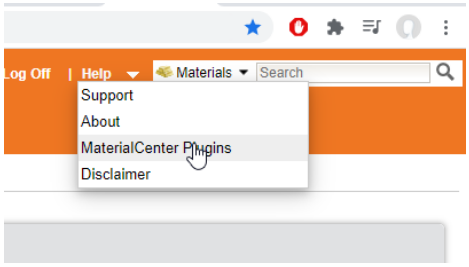


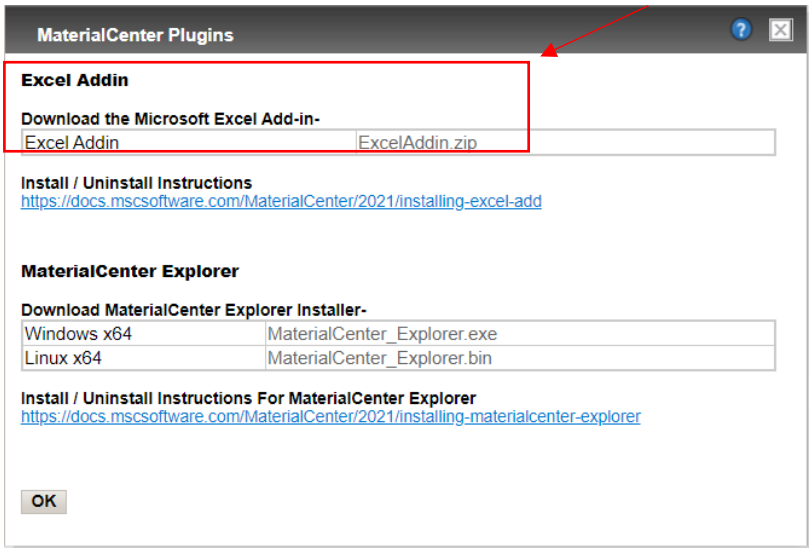
The steps required to activate Material Center are as follows:

Step 1) Download Material Center from the MSC Software Download Center and install the software on the server.



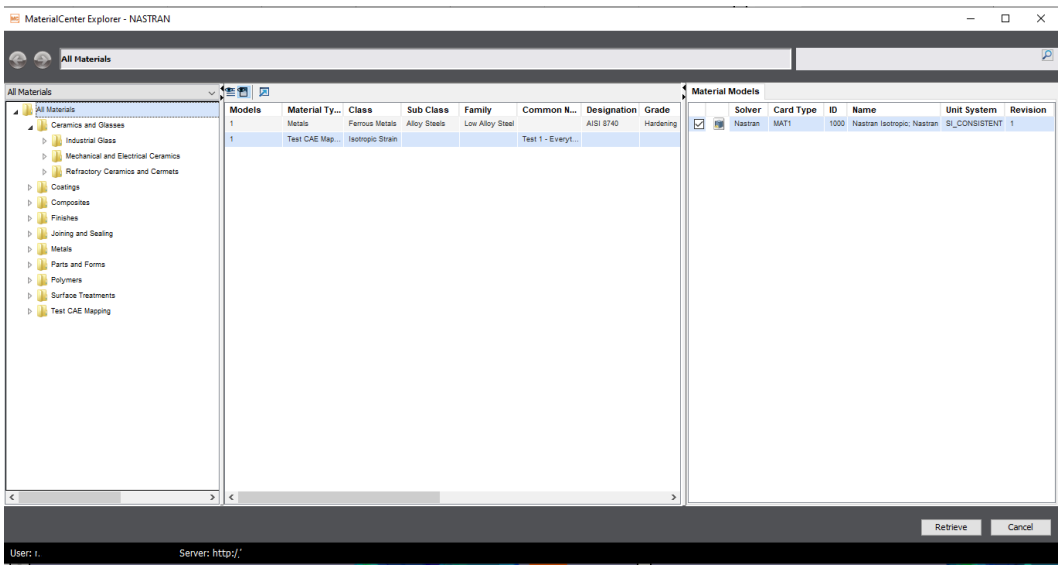
Step 2) Download the Material Center Explorer plugin from the Material Center Help menu.





Step 3) Make note of the installation location on the local PC and enter that location into CAEfatigue via the TOOLS> SETTINGS window as show earlier. Typically, the location **c:\Program Files\MSC.Software** is sufficient enough for CAEfatigue to find MC Explorer.

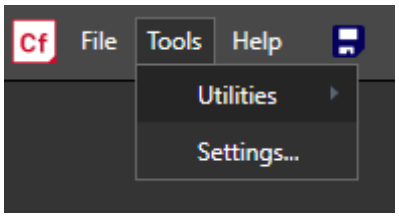
Step 4) Select Material Center from the Materials box in the Process Flow interface. This will open the Material Center Explorer interface (after entering your username and password). Select a material and it will be imported into CAEfatigue.



Connecting to Digimat

CAEfatigue offers the ability to obtain SN curves from Digimat. To do this, Cf must know where the **digi2caefatigue** interface file is located and the User must have a Digimat license. The path to the Digimat interface file is set in the CAEfatigue **Tools>Settings** window as discussed earlier. Typically, this path is similar to:

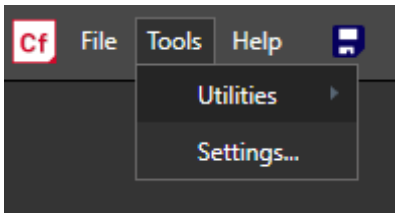
```
C:\MSC.Software\Digimat\2021.<ver>\DigimatCAE\exec\digi2CAEfatigue
```

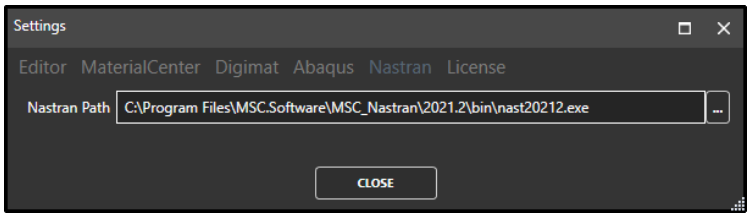


Connecting to Nastran

CAEfatigue offers the ability to run a NEF job from within the Cf GUI. To do this, Cf must know the executable file for MSC Nastran and the User must have a MSC Nastran license. The path to the MSC Nastran executable is set in the **Tools>Settings** window as discussed earlier. Typically, this path is similar to the example shown below for MSC Nastran v2021.2.

```
C:\Program Files\MSC.Software\MSC_Nastran\2021.2\bin\nast20212.exe
```





3rd Party Software

This software makes use of various 3rd party libraries as referenced in the Appendix section.

About This Document

This document forms part of the documentation for the Cf software product. The reader is also referred to the Cf ***Quick Reference Guide*** and the ***User Guide Examples*** document. This User Guide is intended to be used by engineers or designers with a basic knowledge of fatigue, FEA and Dynamics.

The Cf software program also includes a Windows™ graphical User interface for visualizing output results. The User can also use other GUI's programs such as Patran, or HyperView to see output results or the results can be visualized using standard tools like EXCEL™ with our CSV output option.

Theoretical Background Sections

A limited amount of theoretical background detail is also provided, where it makes the User examples easier to understand or validate. These sections are marked using a red background. Again, these sections are not needed to run each example but do provide useful background context for the examples being run.

Contacts

Please submit customer support requests through the email ***caefatigue.support@mscsoftware.com***.

2 | SOFTWARE INSTALLATION AND MSC ONE TOKEN USAGE

Installing the Software and Pointing to the License File

Full instructions for installing the software are included in the Installation and Operations Guide (IOG).

Testing the Installation by Running TPL1

Once installation has completed, the Application can be located in the directory ***C:\Program Files\MSC.Software\CAEfatigue\<ver>*** (if the default installation was used). The installation will also (by default) create the TPL directory ***C:\Documents\MSC.Software\CAEfatigue\{ver}\TPL.*** Please locate the **Test Problem Library (TPL)** and run one of the test problems to confirm the application and licensing are working correctly.

MAKE SURE YOU INSTALL THE TPL IN A FOLDER WHERE YOU HAVE READ AND WRITE PERMISSION.

For example, to run TPL1 simply start the application by selecting the Cf desktop icon and select FILE>OPEN to select TPL1. Select the “run” button so execute. The User can also open a CMD prompt in the TPL library and then type ***CAEfatigue*** followed by a space, followed by the TPL file name. For example, type:

caefatigue TPL1.txt

and hit enter (return) key to start the job.

Depending on the options chosen in the Control File there will be a mixture of TXT, CSV, FEF, H3D and/or LOG files written. For the TPL1 example, the following text files will be written. Note: LOGLVL is set to 0 for this example.

Results1.log - log file containing job execution runtime information and details on the top 10 damaging elements

Results1.csv – CSV file containing results for all elements in the model

Results1PSD.csv – CSV file containing PSD information for the element with the most damaging grid (node), **ONLY**.

Results1_vmatftg_60.txt – detailed copy of the SN material data

*Additional binary files are created such as **Results1.Cfvdata** and **Results1.Cfvdata2** that are used for the RESTART entry and by CFG to display results information.*

The Results1.csv file is a comma separated (CSV) file that contains a large amount of data. When reviewing the contents of this file, it is best to use Microsoft Excel (or similar), which allows formatting and sorting into a more readable form such as below. Excel has been used extensively throughout this User Guide to review and analyze output data from the example problems.

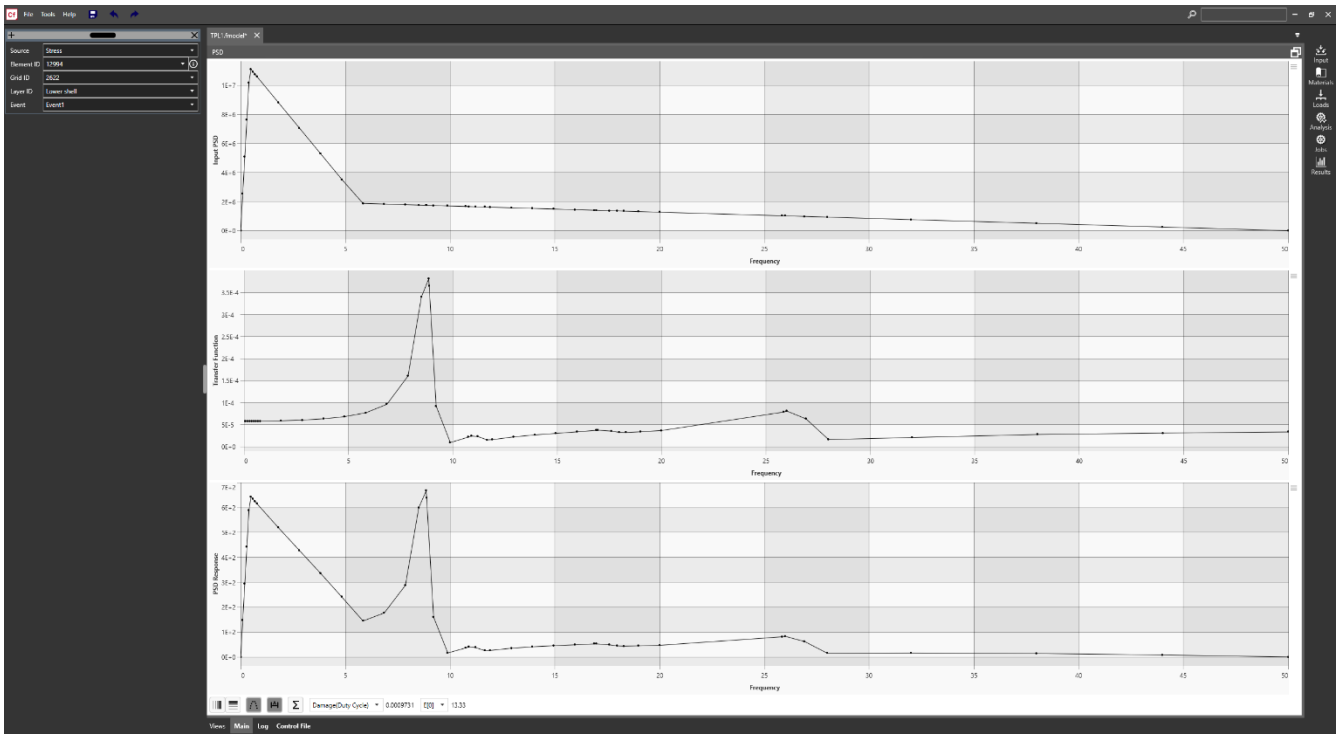
Element	Grid	Layer	Elset Id	Material ID	Event	m0	m1	m2	m4	rms	rms_strain	E[0]	E[p]	Irr_Factor	Mean Stress	Mean+4*rms	Mean-4*rms	Mean+4*rms_strain	Mean-4*rms_strain	Damage (Duty Cycle)	Log Damage (Duty Cycle)	Life(seconds)	Log of Life(seconds)	MOS	Plasticity Index	Freq at peak response
12226	0	lower	0	60	100	1.97E+00	2.61E+01	6.23E+02	6.18E+05	1.40E+00	0.00E+00	1.78E+01	3.15E+01	5.65E-01	0.00E+00	4.21E+00	-4.21E+00	0.00E+00	0.00E+00	3.15E-17	-1.65E+01	3.18E+16	1.65E+01	1.00E+02	0.00E+00	4.88E-01
12226	0	upper	0	60	100	1.97E+00	2.61E+01	6.23E+02	6.18E+05	1.40E+00	0.00E+00	1.78E+01	3.15E+01	5.65E-01	0.00E+00	4.21E+00	-4.21E+00	0.00E+00	0.00E+00	3.15E-17	-1.65E+01	3.18E+16	1.65E+01	1.00E+02	0.00E+00	4.88E-01
12226	225	lower	0	60	100	4.59E+00	5.86E+01	1.33E+03	1.27E+06	2.14E+00	0.00E+00	1.70E+01	3.09E+01	5.49E-01	0.00E+00	6.43E+00	-6.43E+00	0.00E+00	0.00E+00	3.09E-17	-1.65E+01	3.23E+16	1.65E+01	7.66E+01	0.00E+00	8.87E+00
12226	225	upper	0	60	100	4.59E+00	5.86E+01	1.33E+03	1.27E+06	2.14E+00	0.00E+00	1.70E+01	3.09E+01	5.49E-01	0.00E+00	6.43E+00	-6.43E+00	0.00E+00	0.00E+00	3.09E-17	-1.65E+01	3.23E+16	1.65E+01	7.66E+01	0.00E+00	8.87E+00
12226	828	lower	0	60	100	1.14E+00	1.54E+01	3.69E+02	3.67E+05	1.07E+00	0.00E+00	1.80E+01	3.15E+01	5.70E-01	0.00E+00	3.21E+00	-3.21E+00	0.00E+00	0.00E+00	3.15E-17	-1.65E+01	3.17E+16	1.65E+01	1.00E+02	0.00E+00	4.88E-01
12226	828	upper	0	60	100	1.14E+00	1.54E+01	3.69E+02	3.67E+05	1.07E+00	0.00E+00	1.80E+01	3.15E+01	5.70E-01	0.00E+00	3.21E+00	-3.21E+00	0.00E+00	0.00E+00	3.15E-17	-1.65E+01	3.17E+16	1.65E+01	1.00E+02	0.00E+00	4.88E-01
12226	2301	lower	0	60	100	1.44E+00	1.90E+01	4.52E+02	4.47E+05	1.20E+00	0.00E+00	1.77E+01	3.14E+01	5.64E-01	0.00E+00	3.60E+00	-3.60E+00	0.00E+00	0.00E+00	3.14E-17	-1.65E+01	3.18E+16	1.65E+01	1.00E+02	0.00E+00	4.88E-01
12226	2301	upper	0	60	100	1.44E+00	1.90E+01	4.52E+02	4.47E+05	1.20E+00	0.00E+00	1.77E+01	3.14E+01	5.64E-01	0.00E+00	3.60E+00	-3.60E+00	0.00E+00	0.00E+00	3.14E-17	-1.65E+01	3.18E+16	1.65E+01	1.00E+02	0.00E+00	4.88E-01

A Control File with LOGLVL=2 will also write additional CSV files with detailed results for:

- input PSD data,
- transfer functions,
- response PSD's,
- rainflow cycle counts, and
- damage results.

Note that if LOGLVL is set to either 1 or 2, then the ELSET entry should be used with the FSET3 entry to reduce the number of output points as defined by the element numbers specified. Otherwise, the additional files will contain information for ALL output points, which will significantly degrade the performance of the analysis.

If required, the input PSD, transfer function and response PSD can be plotted using the ***PSD Plotter*** in the GUI as shown below.



The LOG file will contain information about how the job ran as well as a summary of the 10 worst damage results for each element set type, as shown below.

SECTION OF OUTPUT FROM RESULTS1.LOG

```
Beginning Frequency Domain Damage Computations

Number of points in model after restricting to FSET3 elements (if used):  32480

Damage computed
Present   time: <day / time / date>
Beginning output to csv/fef/h3d

Element centers - Worst Damage sum (duty cycle) over all element centers
-----------------------------------------------
Rank     Elem    Grid   Layer      DamageSum
```

```
   1    13897  center   lower      2.881470e-007
   2    13897  center   upper      2.881470e-007
   3    16225  center   lower      6.283916e-008
   4    16225  center   upper      6.283916e-008
   5    12895  center   upper      1.969500e-008
   6    12895  center   lower      1.969500e-008
   7    13887  center   lower      1.192074e-008
   8    13887  center   upper      1.192074e-008
   9    12897  center   upper      9.769580e-009
  10    12897  center   lower      9.769580e-009

Element nodes - Worst Damage sum (duty cycle) over all element nodes
-------------------------------------------------
Rank     Elem    Grid   Layer      DamageSum
   1    12994    2622   lower      1.008048e-003
   2    12994    2622   upper      1.008048e-003
   3    13511    2616   lower      4.598827e-004
   4    13511    2616   upper      4.598827e-004
   5    13897    2756   lower      4.586307e-004
   6    13897    2756   upper      4.586307e-004
   7    14906    2751   upper      3.276262e-004
   8    14906    2751   lower      3.276262e-004
   9    13324    2757   upper      7.980962e-005
  10    13324    2757   lower      7.980962e-005

3d Element centers - Worst Damage sum (duty cycle) over 3d element centers
-------------------------------------------------
Rank     Elem    Grid   Layer      DamageSum

3d Element nodes - Worst Damage sum (duty cycle) over 3d element nodes
-------------------------------------------------
Rank     Elem    Grid   Layer      DamageSum

2d Element centers - Worst Damage sum (duty cycle) over 2d element centers
-------------------------------------------------
Rank     Elem    Grid   Layer      DamageSum
   1    13897  center   lower      2.881470e-007
   2    13897  center   upper      2.881470e-007
   3    16225  center   lower      6.283916e-008
   4    16225  center   upper      6.283916e-008
   5    12895  center   upper      1.969500e-008
   6    12895  center   lower      1.969500e-008
   7    13887  center   lower      1.192074e-008
   8    13887  center   upper      1.192074e-008
   9    12897  center   upper      9.769580e-009
  10    12897  center   lower      9.769580e-009

2d Element nodes - Worst Damage sum (duty cycle) over 2d element nodes
-------------------------------------------------
Rank     Elem    Grid   Layer      DamageSum
   1    12994    2622   lower      1.008048e-003
   2    12994    2622   upper      1.008048e-003
   3    13511    2616   lower      4.598827e-004
   4    13511    2616   upper      4.598827e-004
   5    13897    2756   lower      4.586307e-004
   6    13897    2756   upper      4.586307e-004
   7    14906    2751   upper      3.276262e-004
   8    14906    2751   lower      3.276262e-004
   9    13324    2757   upper      7.980962e-005
  10    13324    2757   lower      7.980962e-005


All MAXPEAK*RMS stresses are less than UTS

Finished with output

End   time: <day / time / date>
SUCCESS
```

CAEfatigue Packaging and MSC ONE Token Usage

The CAEfatigue software is packaged in 4 different configurations

CAEfatigue – TIME	15 tokens
CAEfatigue – RANDOM	15 tokens
CAEfatigue – FREQUENCY	25 tokens
CAEfatigue – PREMIUM	48 tokens

ALL packages include a **Graphical User Interface (GUI) for Microsoft Windows™**.

CAEfatigue **TIME**: This package is used to calculate time domain durability and fatigue damage using results from Nastran, Ansys and/or Abaqus. This package also includes the TIME2PSD toolset to allow the User to condition and reduce (if permitted) raw time signals to generate new time signals of shorter duration for testing purposes that produce the same damage as the original time signals.

CAEfatigue **RANDOM**: This package is used to calculate frequency domain random response from Base Shake (single input) or Multi input loading using system properties (transfer functions) from a variety of different solvers (e.g. Nastran, Abaqus, Ansys). This package also includes the TIME2PSD toolset for converting time domain data into frequency domain PSD matrix file.

CAEfatigue **FREQUENCY**: This package includes the features in TIME and RANDOM. It is used to calculate frequency domain random response and fatigue damage from Base Shake (single input) or Multi input loading using system properties (transfer functions) from a variety of different solvers (e.g. Nastran, Abaqus, Ansys). This package also includes the TIME2PSD toolset for converting time domain data into frequency domain PSD matrix file.

CAEfatigue **PREMIUM**: This package includes the features in FREQUENCY and is used to calculate time domain fatigue damage or frequency domain random response and fatigue damage. It also includes advanced random outputs like displacements, velocities, accelerations and forces, allows for the use of standalone deterministic loading or mixed deterministic loading with random loading, the calculation of Pseudo-Damage, the calculation of Surrogate loads, the calculation of Spot/Seam welds in the time domain and the calculation of Spot/Seam/User welds in the frequency domain.

Features / Capabilities Within Each Package and by Solver

CAEfatigue has a large number of features and capabilities that are available depending on the package chosen by the User, if working from a Standalone license, and the Solver providing the stress fields. If working with MSC ONE tokens, the token draw will correspond to the feature or capability defined in the analysis.

The following page contains a summarized listing of features and capabilities available in each package and Solver. This is not an exhaustive list so see the Quick Reference Guide or contact Customer Support for further information.

Summary of CAEfatigue Compatibility by Package and Solver

Data based on v2022.1 Release

				PACKAGE					NASTRAN (all forms)		ABAQUS		ANSYS		LS-DYNA	
			Control File Entry	TIME	RANDOM	FREQUENCY	PREMIUM	Control File	Process Flow	Control File	Process Flow	Control File	Process Flow	Control File	Process Flow
GUI				✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
Time Domain Fatigue	Loading	Linear Static Superposition Analysis	LQSTATIC	✓		✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Modal Participation Factor Analysis	LMTRANS	✓		✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Direct Stress Recovery Analysis	LTRANS	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Nonlinear Static Superposition Analysis	NQSTATIC	✓		✓	✓	✓	✓	X	X	X	X	X	X
		Support for MARC HS/OP2 solid element support	LTRANS	✓		✓	✓	✓	✓	N/A	N/A	N/A	N/A	N/A	N/A
		Mean Loading for TD	MEAN	✓		✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Time Domain Load Scheduler - Ability to create Time History events using RSP loading, deterministic loading, block loading, XY pairs of loading, etc.		✓		✓	✓	N/A	✓	N/A	✓	N/A	✓	N/A	X
	Features / Capabilities	Spot / Seam Weld Fatigue for TD	SPOTW, SEAMW	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Ability to create and run a Nastran Embedded Fatigue (NEF) from within CAEfatigue. Needs a Nastran license.	multiple	✓		✓	✓	✓	✓	N/A	N/A	N/A	N/A	N/A	N/A
		Ability to specify separate element/node groups and apply material properties to those groups.	ELSET	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Built-in Material Database	vUDNAME	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Build-in autogenerate material properties from static UTS material data	AUTOSN / AUTOEN	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Ability to use SN and/or EN material data in the same analysis	vMATFTG> EN/SN	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Ability to use multiple SN curves including Haigh Life, Mean Stress and R-Ratio.	vMATFTG> SN_Type	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Ability to connect with MSC DIGIMAT for SN material properties. Needs a DigiMAT license.	vMATFTG> digimat / digidof	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Control over Processors for TD	PARALLEL	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Control over RAM for TD only	MEMORY	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Automatic Load Scenario time history creation	n/a	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	X
	Output	Equivalent Stress outputs including Principal stress, von Mises and Critical Plane)	vFTGPARM> STRESS> COMB	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		3D Critical Plane sphere plot output	CRITOUT	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Mean Stress Correction including versions of Goodman, Gerber, SWT, Morrow, Walker and MMPDS	vFTGPARM> STRESS> CORR	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Multiaxial Assessment	vFTGPARM> MULTIAX	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Factor of Safety (FOS) for TD - except LTRANS	vFTGPARM> FOS	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Damage / Life output		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Time Signal Response output	TIMEOUT	✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Reliability Mode multiple simultaneous CF job simulation in TD		✓		✓	✓	✓	✓	✓	✓	✓	✓	✓	✓
		Robust Design multiple simultaneous Nastran job simulation in TD		✓		✓	✓	✓	✓	N/A	N/A	N/A	N/A	N/A	N/A
		Nastran Embedded Fatigue (NEF) Deck Creator for TD		✓		✓	✓	✓	✓	N/A	N/A	N/A	N/A	N/A	N/A
Frequency Domain Fatigue	Loading	Time History to PSD Matric conversion (TIME2PSD) with features including Enveloping and Bounding Diagonals	TIME2PSD	✓	✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Single PSD Loading			✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Multi PSD Loading			✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Random Loading w/wo Mean Loading			✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
	Features / Capabilities	Control over Processors for FD	PARALLEL		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Built-in Material Database	vUDNAME		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Build-in autogenerate material properties from static UTS material data	AUTOSN / AUTOEN		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to use SN and/or EN material data in the same analysis	vMATFTG> EN/SN		N/A	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to use multiple SN curves including Haigh Life, Mean Stress and R-Ratio.	vMATFTG> SN_Type		N/A	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to connect with MSC DIGIMAT for SN material properties. Needs a DigiMAT license.	vMATFTG> digimat / digidof		N/A	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to use surface finish / surface treatment corrections to alter SN curve* *defined using SN option	KADJ		N/A	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to add a Standard Error to adjust material curve	SE		N/A	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to specify separate element/node groups and apply material properties to those groups.	ELSET		N/A	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to use stresses from Nastran Composite analysis			✓	✓	✓	✓	✓	N/A	N/A	N/A	N/A	N/A	N/A
		Built-in Random PSD Database			✓	✓	✓	N/A	✓	N/A	✓	N/A	✓	X	X
		Loads cascading from base inputs to subcomponent locations	PSDMOUT		✓	✓	✓	✓	✓	✓	✓	X	X	X	X
		HOTSPOT calculation to generate the highest stress locations	HOTSPOT		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Apply a local filtering on output RMS values to reduce output file sizes	FILTER		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Apply Notch loading to Transfer Function	RESTART		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to do a RESTART that will avoid reading the Solver files on successive CF runs	FNOTCH		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Ability to limit calculations to only the elements / nodes specified using the ELSET entry	vIBFAT> OPTIONS> STRLIM		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
	Output	Equivalent Stress outputs including Principal stress*, von Mises and Component stress * not for multiple input	vFTGPARM> STRESS> COMB		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Calculation of Neuber Stress (True Stress)	CYCPROC		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Mean Stress Correction including versions of Goodman, Gerber, SWT, Morrow, Walker and MMPDS	vFTGPARM> STRESS> CORR		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Factor of Safety (FOS) for FD	vFTGPARM> FOS		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		PSD response output at element / node locations	PSDOUT		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Loading Channel diagnostics to determine contribution to response output	DIAGNOS		✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Advanced Random DVA / Force output	ABSRESP		✓	✓	✓	✓	✓	✓	✓	X	X	X	X
		Collision (rattle) Detection	RELRESP		✓	✓	✓	✓	✓	✓	✓	X	X	X	X
		Relative Response between 2 nodes	RELRESP2		✓	✓	✓	✓	✓	✓	✓	X	X	X	X
		Robust Design for running multiple simultaneous Nastran job simulations in FD			✓	✓	✓	✓	✓	N/A	N/A	N/A	N/A	X	X
		Reliability Mode for running multiple simultaneous CF job simulations in FD			✓	✓	✓	✓	✓	✓	✓	✓	✓	X	X
		Damage / Life output				✓	✓	✓	✓	✓	✓	✓	✓	X	X
	Advanced Loading	Deterministic and/or Single Input* Random PSD w/wo Mean Loading *Deterministic with multi PSD loading is not supported	SINGSINE, DETLOAD, NBLOAD, SINESW, vFTGLOAD				✓	✓	✓	✓	✓	✓	✓	X	X
	Advanced Features / Capabilities	Pseudo Damage Calculation using Acceleration Time Signals	Pseudo				✓	✓	✓	✓	✓	✓	✓	X	X
		Spot / Seam Fatigue for FD	SPOTW, SEAMW				✓	✓	✓	✓	✓	✓	✓	X	X
		User Weld Fatigue for FD	USERWHS				✓	✓	✓	✓	✓	X	X	X	X
		Surrogate Loads calculation to simply complicated loading to single PSD / sine waves	SGATE, VECTOR, PERTURB				✓	✓	✓	✓	✓	✓	✓	X	X
		Vector feature to allow loading in a direction other than X, Y and Z	SGATE, VECTOR, PERTURB				✓	✓	X	✓	X	✓	X	X	X
		Vector with Perturb to calculate the best angle for applying test loading	SGATE, VECTOR, PERTURB				✓	✓	X	✓	X	✓	X	X	X
		Surrogate Loads w/wo Vector and Perturb	SGATE, VECTOR, PERTURB				✓	✓	X	✓	X	✓	X	X	X

n/a - not applicable (will not be implemented) X - applicable but no implemented as yet

MSC ONE Token Usage with CAEfatigue

CAEfatigue will pull 15 tokens when the software is opened. This token draw will allow the user to view results from any number of jobs that have already been run by using the job tabs across the top of the GUI window. The User is also permitted to run a single **CAEfatigue TIME** or **RANDOM** analysis with this token count.

If the User wishes to run (for example) a FREQUENCY or PREMIUM analysis, the software will drop the 15 tokens and pull the required tokens needed to run the Frequency or Premium job i.e., 25 or 48 tokens respectively. The software will continue to hold these tokens after the job is complete i.e., the software holds last highest token count anticipating another analysis (excluding tokens drawn for simultaneous jobs – see “Simultaneous Jobs” below).

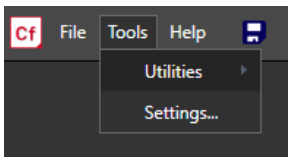
If the computer remains inactive for a period of time (see IDLE Timeout parameter below and in the IOG document), the software will automatically release all tokens or standalone license and wait for User input to (A) resume and get a license again, or (B) shutdown / close the software.

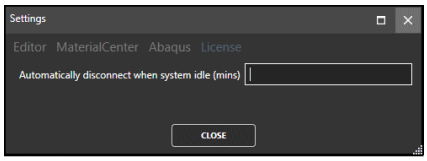
Running 2 or more jobs at the same time (Simultaneous Jobs):

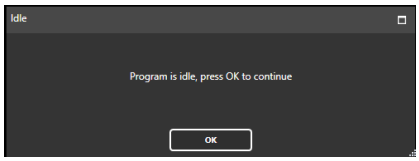
The User is permitted to run as many simultaneous jobs as they have computer resources to do so. For each additional job that is run, the software treats those jobs as separate analyses and will pull / release the required tokens as needed or licenses, if working with Standalone licenses.

CF IDLE Timeout to Release License

CAEfatigue offers an IDLE timeout parameter that can be set in the TOOLS> Settings box. The IDLE timeout will recognize that you have stepped away from the computer if there is no activity (i.e., keystrokes or mouse clicks) for a period of time and will automatically disconnect you from the license so it can by freed up for others to use. Note: A value of zero (0) or a blank field, means the parameter is disabled.







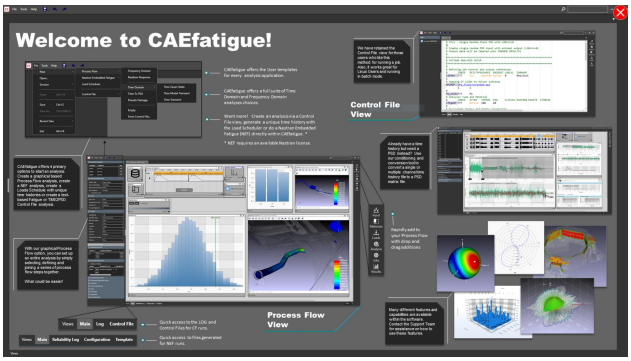
3 | INTRODUCTION TO GRAPHICAL USER INTERFACE LAYOUT

NOTE: The images shown in this section reflect CAEfatigue v2021 and may change slightly with future releases.

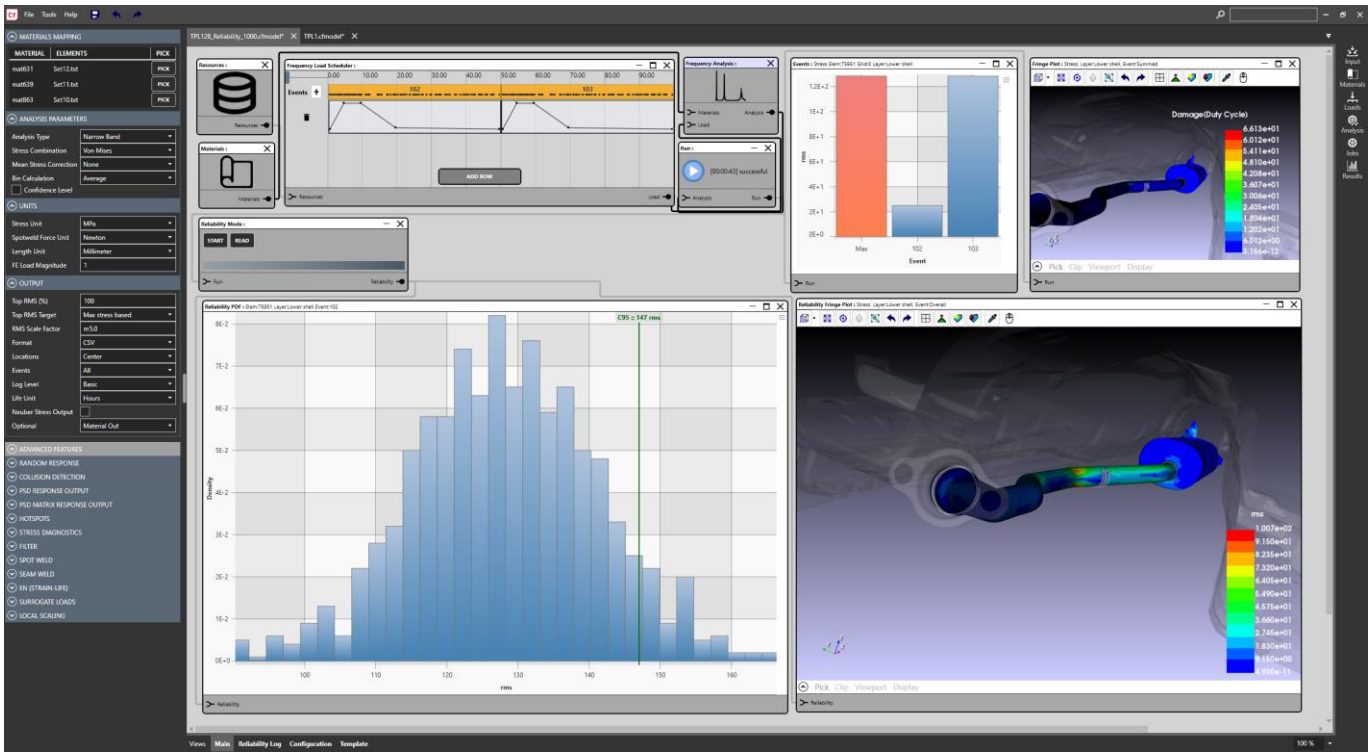
General Layout of the Software GUI

When the CAEfatigue software is first opened, the “work surface” is blank because no previous jobs have been opened, however, the next time the software is opened it will remember any previous jobs that were opened when the software was closed, and it will return the User to a condition where those jobs are shown as tabs over the work surface.

Note: the first time the software is opened the User will see the HELP> Introduction Card shown as the right image below. This image shows some of the features / capabilities of CAEfatigue. You can see this card in the Help menu at any time.

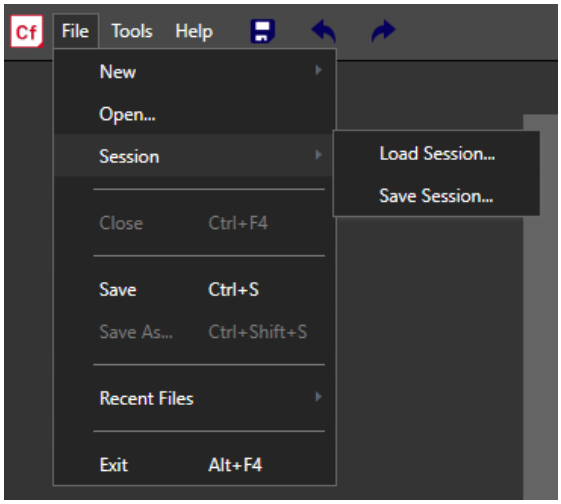


An example of how the software will open after you have previously opened one or more jobs. In the case below, 2 jobs were in use when the software was previous closed so those 2 jobs will be visible when the software next opens.



File Menu – Create, Open, Save Cf Jobs and Sessions Files

Besides the normal activities of saving, closing and exiting, the **FILE** menu allows the User to open a new file template or blank job (**NEW**), an existing job (**OPEN**), an existing session (**SESSION**) or recently opened jobs (RECENT FILES).



Explanation of Session files:

A SESSION file (*.cfsession) is a way to group a number of CAEfatigue jobs so that the group of jobs can be reopened at any time. It is a great way to group a number of related analyses so that they can be opened together in a single “Load Session” operation instead of opening each individual *.cfmodel file.

To create a session file, simple open all the *.cfmodel files you would like to group together. Once opened (i.e. once visible as tabs over the work surface), go to “Save Session” and give the file a name. It would be good practice to label the session file with a familiar / appropriate name to understand what files are contained in the group. Also, it would be good to also store your session files in a directory so they can be easily found as needed in the future.

Tools Menu – Access to the Utilities, Fatigue Tools and Settings Menus

Selecting the **TOOLS** menu will provide access to the CAEfatigue Utility tools menu, the legacy MSC Fatigue Tools menu and the CAEfatigue software Settings menu.

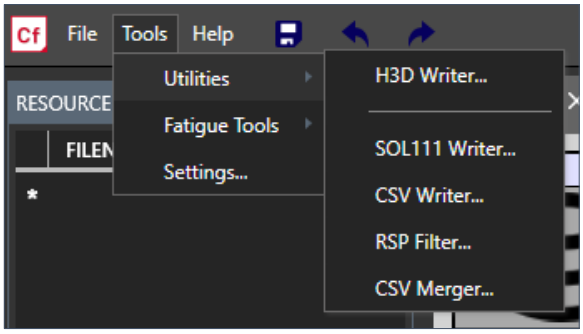
Utilities Menu

Within the **Utilities menu**, the User is provided with a selection of additional functionalities. Note: These functionalities can also be called from the Command Prompt. See the separate document - Cf ***Utilities User Guide*** - for further information.

The available Utilities are:

- **Combine H3D File Writer** – A utility to a single H3D file from the data within the CSV files.
- **BDF Translator to SOL111 File** – A utility to translate a MSC Nastran or OptiStruct SOL112 BDF control deck into a SOL111 control deck that can be run in MSC Nastran / OptiStruct to product the FRF stress data needed to execute a Cf analysis.

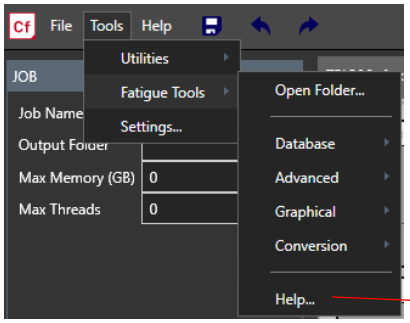
- **BDF Time History Writer to CSV File** – A utility to extract the time history loading entries from a SOL112 control deck into a CSV file. This CSV file of time histories can then be read into TIME2PSD and converted into PSDM files for subsequent use in a multi input Cf analysis.
- **RSP Time History Filter** – A utility to filter an RSP file with respect to time signal channels and/or a specific time duration or range. This allows a User to reduce the size of an RSP file to the information needed for the TIME2PSD conversion.
- **CSV File Merger** – A utility to combine data from several different CSV files into single merged files of different result types. This can be a useful utility when two separate, but related analyses are completed, and the User wants to combine the output data into a single CSV file.

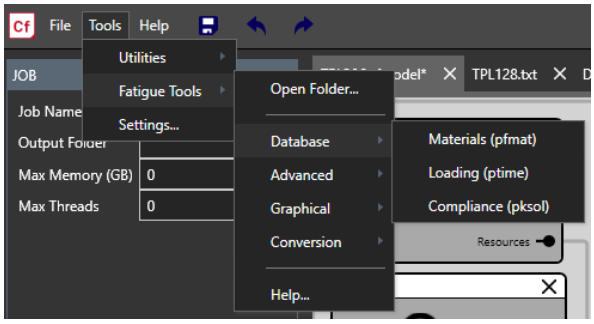


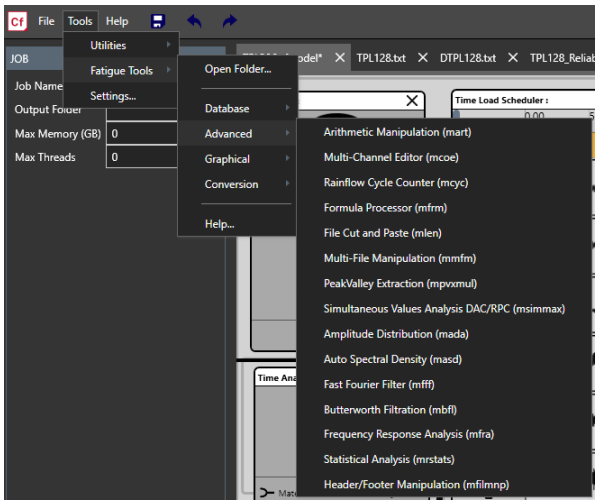
Fatigue Tools Menu

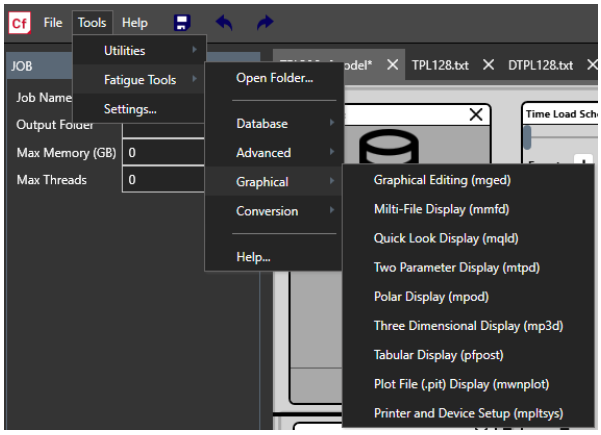
Within the **Fatigue Tools** menu, the User is provided with a selection of additional functionalities that have been migrated from **MSC Fatigue**. These functionalities are grouped in a similar way to how they are grouped within MSC Fatigue, i.e., DATABASE, ADVANCED, GRAPHICAL and CONVERSION. See submenus for each below.

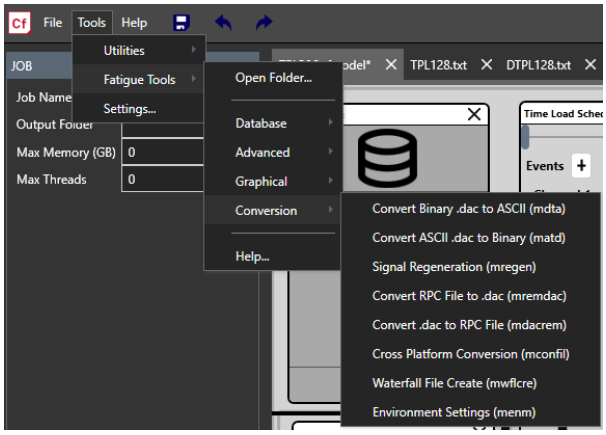
For further information about these tools, please see the **Tools> Help> CAEfatigue Fatigue Tools** document.









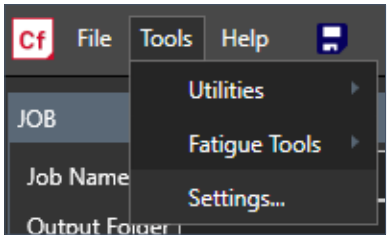


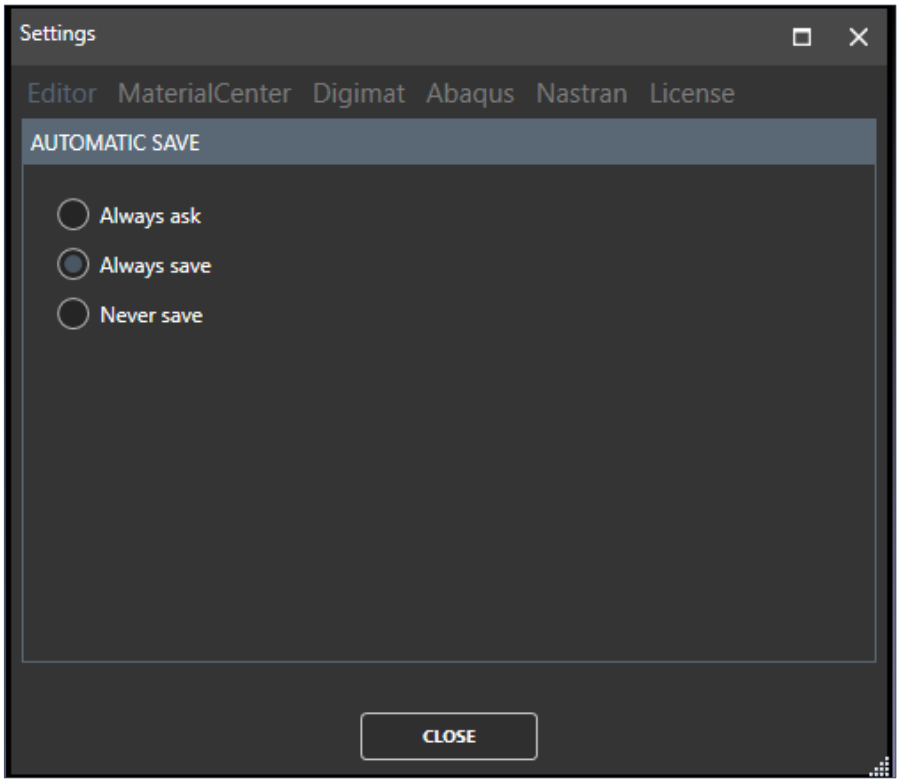
Note: These functionalities can also be called as separate utility tools. By default, the location of these tools is

C:\Program Files\MSC.Software\CAEfatigue\{ver}\Windows\tools\bin.

Settings Menu

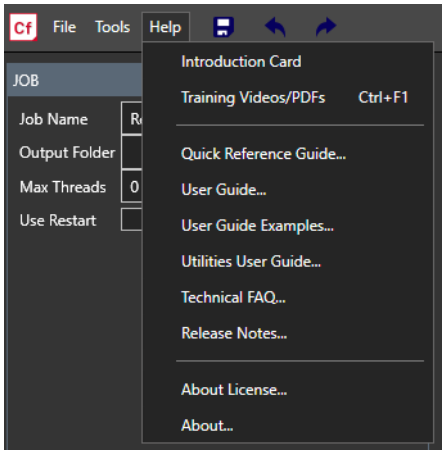
Within the **Settings Tools** menu, the User is provided with a selection of settings that can be set within CAEfatigue. These settings include that ability to automatically save jobs when the execute button is selected, the ability to set various parameters need to link with and/or use files from Material Center, Digimat, Abaqus and MSC Nastran and finally the ability to disconnect the license file when the software remains idle for a period of time.


File
Tools
Help
Utilities
Fatigue Tools
Settings...
JOB
Job Name

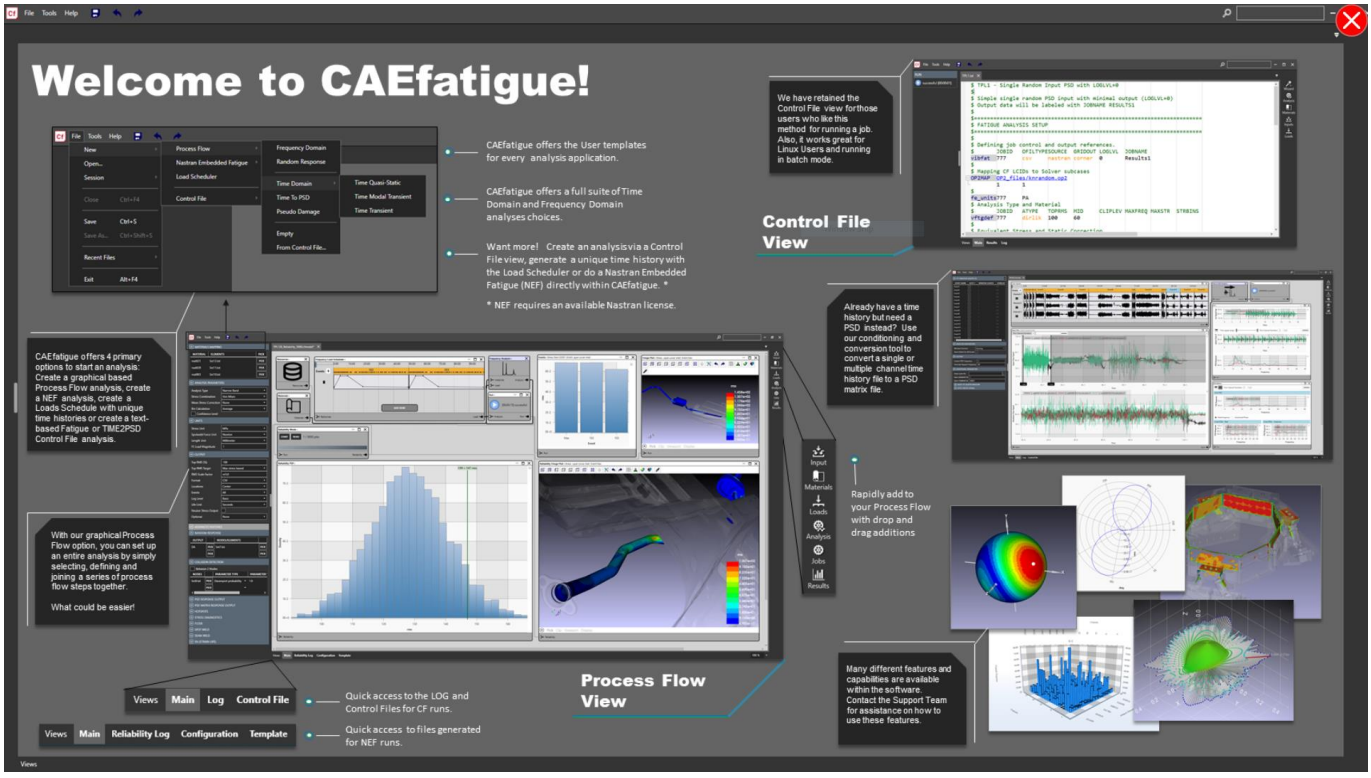

Settings
Editor
MaterialCenter
Digimat
Abaqus
Nastran
License
AUTOMATIC SAVE
Always ask
Always save
Never save
CLOSE

Help Menu – Access to the Training Material and Software Documentation

Selecting the **HELP** menu will provide access to the CAEfatigue training videos and PDF file for the User to conduct self-training. The menu also provides access to the available software documentation, information about the license file currently in use and the version of the software currently in use.



The **INTRODUCTION CARD** will be shown the first time the software is opened and then can be accessed at any other time the User wishes to see the card.



4 | INTRODUCTION TO THE CAEFATIGUE SOFTWARE (CF) COMMAND ENTRIES AND FIELDS

CF operates like Nastran, meaning, the operation of the software is based on the instructions found in a Control File or Process Flow view. This text file generated from either view uses commands or ENTRIES (rows) with numerous FIELDS that provide the software with all the parameters needed to execute a random response or fatigue calculation. This section will introduce the large array of instructional entries and fields that are available for both Cf and the time history conversion toolset - TIME2PSD – that is also included in the Cf software.

An introduction to the Control File User Interface and Process Flow User Interface are discussed in later chapters.

Entries (Commands) and Fields used in Frequency Domain Analysis

Below is the complete set of available entries that can be used to do a Cf frequency domain analysis (either in the Control File view or Process Flow view). In a typical analysis, only small selection of these items are used. Please note: Entries in *GREY* are not currently supported.

DATA COLUMN 1	DATA COLUMN 2	DATA COLUMN 3	DATA COLUMN 4	DATA COLUMN 5	DATA COLUMN 6	DATA COLUMN 7	DATA COLUMN 8	DATA COLUMN 9
vIBFAT	JOBID	OFILTYPE *(CSV)*	SOURCE *(NASTRAN)*	GIDOUT *(CENTER)*	LOGLVL *(0)*	JOB_NAME *(control file name)*		
	"OPTIONS"		STRLIM *(0)*			OUTDIR		
PARALLEL	TH_CAP							
OP2MAP(S)	FILE_NAME (for dynamic and static (S) stresses from NASTRAN)							
	$CFLCID_i$	$OP2LCID_i$						
ODBMAP(S)	FILE_NAME (for dynamic and static (S) stresses from ABAQUS)							
	$CFLCID_i$	$ODBLCID_i$						
vFTGDEF	JOBID	ATYPE *(DIRLIK)*	TOPRMS *(100)*	MID	CLIPLEV *(same as MAXSTR)*	MAXFREQ *(99.9)*	MAXSTR *(3 for SINES 10 Otherwise)*	STRBINS *(1280 for Sines 32 Otherwise)*
	"ELSET"	$ELSID_1$	MID_1	$ELSID_2$	MID_2	CONTINUE		
	"HOTSPOT"	NHS *(1)*	HSGATE *(1)*	HSFLAG	STOP *(NO)*			
	"FILTER"	TOPRMS	TOPRMSD	TOPRMSV	TOPRMSA			
	"DIAGNOS"	*NHSD*	HSSET	*CHSET*	*VAR (STRESS)*			
	"PSDOUT"	$PSDVAR_1$	$PSDID_1$	CONTINUE				
	"CRITOUT"	CRITVAR1	CRITID1	CONTINUE				
	"PSDMOUT"	OUTVAR	G_1	DOF_1	CONTINUE			
	"SGATE"	METHOD *(4)*	*EVOPT*	*MODE*	*TYPE*	*T_ACC*	*PL_LIM*	
	TARGET_FILE (results that SGATE will try to achieve)							
	"USERWHS"	$HLSID_1$	MID_1	$TENSID_1$	$HLSID_2$	MID_2	$TENSID_2$	
		$HLSID_3$	MID_3	$TENSID_3$	CONTINUE			
	"SPOTW"	SPSID1	MID1	DIAM	T1	T2		
		SPSID2	MID2	DIAM	T1	T2	CONTINUE	
	"SEAMW"	SWSIDi	MID1	NDSID1	*BTHRESH (0.5)*	*RTHICK (1.0)*	nTHICK *(0.16667)*	WTYPE (Toe)

	"ABSRESP"	$ABSVAR_1$ *(DVAF)*	$SETID_1$	$ABSVAR_2$ *(DVAF)*	SETID2	CONTINUE		
	"RELRESP"	RELVAR	SETID	PARAM1 *(same as MAXPEAK)*	PARAM2			
	"RELRESP2"	RELVAR	$NODEID_1$	$NODEID_2$				
FE_UNIT	JOBID	CNVRTS *(MPa)*	FE_L_MAG *(1.0)*	CONJCON *(B)*	CNVRTF *(N)*	CNVRTL *(mm)*		
vMATFTG	MID	CNVRTM *(MPa)*	AMP_RNG *(RANGE)*					
	"STATIC"		UTS	E	Code		SE	
	"CYCLIC"	K'	n'					
	"SN"	SRI1	b1	Nc1	b2 *(0.0)*	Nfc *(1.0E30)*	SE	
	"KADJ"	KU	KT	KRNUM	KRDESC			
	"SN0" "SN1" "SN2"	SRI1	b1	Nc1	b2	Nfc	MSS	SE
		SF-FXY	DE-FXY	TE-FXY	SF-MXY	DE-MXY	TE-MXY	
		SF-FZ	DE-FZ	TE-FZ	SF-MZ	DE-MZ	TE-MZ	
	"SNx_FKM"	M1	M2	M3	M4			
	"SNBR1" "SNBR0"	SRI1	b1	Nc1	b2	Nfc	SE	
	"SEAM_FKM"	M1	M2	M3	M4	MSS		
	"EN"	σ'_ϕ	b	ε'_ϕ	c	K'	n'	Y *(0.5)*
		SEe	SEp	Sec				
	"AUTOSN" or "AUTOEN"	Can be used instead of SN or EN						
	"MMPDS"	A1	A2	A3	A4			
	"TABLE"	TID_1	T_1	CONTINUE				
	"FILE"	UDID	Type	XML_File_Reference				
	"DIGIMAT"	Type	File_Location					
vMATSTAT	MID	CNVRTM *(Mpa)*						
	"STATIC"		UTS	E				
	"CYCLIC"	K'	n'					
vFTGPARM	JOBID							
	"STRESS"	COMB *(SGVON)*	CORR *(NONE)*	NCALC *(NAVER)*	DEBUG *(NONE)*	TOLER *(1.0E-5)*	MAXPEAK *(CLIPLEV / 2)*	RCCT *(60.0)*
	"CYCPROC"	TRSTIN *(NO)*	*EVALL (NO)*	HYST *(CORR)*	TRSTOUT *(NO)*			

	"SPOTW"	COMB *(CRITICAL PLANE)*	CORR *(None)*	NANGLE *(18)*	SWLOC *(0)*	*ZCOMP (1)*		
	"SEAMW"	COMB *(ABS MAXPR)*	CORR *(None)*	THICK *(0)*	WELDTOP *(Upper)*			
	"CERTNTY"	CERTSV *(50.0)*						
	"USERWHS"	COMB *(SGVON)*	CORR *(NONE)*	NANGLE *(18)*	SWLOC *(0)*			
	"FOS"	LIFE	BACKACC *(5.0)*	MAXFAC *(5.0)*	MINFAC *(0.2)*			
vFTGSEQ	JOBID	EVNTOUT *(0)*	DUR_UNITS *(SECONDS)*	MOS_DUR *(SUM of EVENTS)*	TSCALE *(1.0)*	NREP *(YES)*		
	$EVID_1$	EVT_1 *(1.0)*	$EVID_2$	EVT_2 *(1.0)*	CONTINUE			
	"UNITS"	EQUIV	EQNAME					
vFTGEVNT	EVID	$FLOAD_1$	$FLOAD_2$	CONTINUE				
	"NAME"	EVENT_NAME						
vFTGLOAD	FLOAD	TYPE *(PSD)*	SID	SCALE *(1.0)*	OFFSET			
LCID	FILE_NAME							
vRANDPS	SID	J	K	X *(1.0)*	Y *(0.0)*	TID		
vTABRND	TID	XAXIS *(LINEAR)*	YAXIS *(LINEAR)*	UDID				
	f_1	g_1	f_2	g_2	endt			
vUDNAME	UDID							
	FILE_NAME							
vRANDT	SID	T	Units *(SECONDS)*					
NBLOAD	SID	LCID	f_1	b_1	h_1	CONTINUE		
DETLOAD	SID	LCID	f_1	g_1	CONTINUE			
SINGSINE	SID	LCID	f_1	g_1				
SINESW	SID	LCID	f_1	g_2	f_n	g_n	SWTYPE *(LINEAR)*	SWNUM *(50)*
	"PARAMS"	SWRATE	N_SWEEPS	FRF_UNIT	LD_UNIT			
KTDATA	JOBID	ELSETi	SCALEi	OFFSETi				
FSET3	ELSID	DES	ID_1	ID_2	ID_3	ID_4	ID_5	CONTINUE
INCLDIR	JOBID	DIR_NAME *(LOCAL DIRECTORY)*						
INCLUDE	FILE_NAME							

RESTART	FILE_NAME							
VECTOR	JOBID	VXCOMP	VYCOMP	VXCOMP	VINCR			
PERTURB	PTYPE	VALUE						
FNOTCH	FNID							
	$LCID_i$	f_{ia}	f_{ib}	$SCALE_i$ *(0.0)*				

A brief description of each entry is given in the table below. For a full explanation of the individual entries please refer to the **Quick Reference Guide** document.

Brief Description of Entries and Fields used in Frequency Domain Analysis

ENTRY NAME	DESCRIPTION
vIBFAT	**Vibration Fatigue Output Request**
JOBID	CF job ID.
OFILTYPE	Choice of CSV, FEF, CSVFEF, H3D, FEFH3D, CSVH3D, H3Dxx, CSVH3Dxx, FEFH3Dxx (where xx = nb/nt/nw), FEFyy, CSVFEFyy (where yy =eb/et/ew/nb/nt/nw). FEF is a flat ASCI Patran Neutral file. H3D is an Altair binary format file.
SOURCE	Must be Nastran, Ansys, Abaqus, LSDyna or Optistruct.
GIDOUT	CORNER, NODAV, CENTER, CENTUPP or CENTLOW.
LOGLVL	Level of messaging; 0 = standard, 1 = +PSD, 2 = +Fatigue.
JOB_NAME	Used to tag all output files. No character limit for entry.
"OPTIONS"	FLAG. Indicates various options are to follow.
STRLIM	Set to "1" to only use elements from ELSETS. Set to "0" to use all elements in Solver file.
OUTDIR	Directory location where all output files should be written. No character limit on entry.
OP2MAP(S)	**Nastran Subcase Mapping Entries**
FILE_NAME	File name of Nastran OP2 file.
$CFLCID_i$	CF Load case ID number. Must start at 1.
$OP2LCID_i$	Subcase ID in OP2 file.
ODBMAP(S)	**Abaqus STEP Mapping Entries**
FILE_NAME	File name of Abaqus ODB file.
$CFLCID_i$	CF load case ID number. Must start at 1.
$ODBLCID_i$	Name of relevant STEP in Abaqus ODB file.

vFTGDEF	Fatigue Element Definitions
JOBID	CF job ID.
ATYPE	Dirlik, NB (narrow band), Stein (Steinberg), Sines (sines only), or Simsine.
TOPRMS	Top RMS stress percentage used to filter analysis set.
MID	ID of a vMATFTG entry (leave blank for **Random Only Response**).
CLIPLEV	Value of stress clipping as a function of RMS.
MAXFREQ	Frequency value for moment integration.
MAXSTR	MAX stress as function of RMS, to be used in RCC.
STRBINS	Number of bins to use in RCC histogram.
"ELSET"	FLAG. Element set ($ELSID_i$) and property (MID_i) pairs to follow.
$ELSID_i$	Unique FSET3 ID.
MID_i	Unique vMATFTG ID (leave blank for **Random Only Response**).
"HOTSPOT"	FLAG. Conduct Hot Spot detection with following parameters.
NHS	Number of hot spots to be detected.
HSGATE	Number of elements to show around hot spot.
HSFLAG	Set Name.
STOP	Terminate job after hot spot 1st pass.
"FILTER"	FLAG. Filter element / grids based on RMS.
TOPRMS	Pass through element based on RMS stress. Alternate location to input TOPRMS.
TOPRMSD	Pass through grids based on % RMS displacement.

TOPRMSV	Pass through grids based on % RMS velocity.
TOPRMSA	Pass through grids based on % RMS acceleration.
"DIAGNOS"	FLAG. Write output diagnostics results to diagnose.csv file.
NHSD	*Number of monitoring location (hotspots) to use.*
HSSET	FSET3 ID that list elements to be used.
CHSET	*FSET3 ID that list channels to be used*
VAR	*Output options including Stress, Stain, Damage.*
"PSDOUT"	FLAG. Indicates PSD response output.
$PSDVAR_i$	Choice of S, D, V, A, F and/or M output requested in this order.
$PSDID_i$	FSET3 set of nodes or elements to be used in PSD response assessment. MONPNT3 Set Name for OUTVAR = "M".
"CRITOUT"	FLAG. Indicates 2D/3D critical plane output parameters to follow.
$CRITVAR_i$	Choice of DS (damage and RMS stress). Only RMS in Frequency Domain.
$CRITID_i$	Set of Element ID for output
"PSDMOUT"	FLAG. Indicates PSD matrix response output.
$OUTVAR_i$	Choice of D, V, A or M.
G_i	Grid IDs for output matrix or MONPNT3 Set Name for OUTVAR = "M".
DOF_i	Degrees of Freedom for output.
"SGATE"	FLAG. Conduct Surrogate Load analysis with the following parameters.
METHOD	Method used for the optimization.
EVOPT	*Choice of optimization type; Stress, Damage or EV_NUM*
MODE	*Choice of optimization type; Stress, Damage or EV_NUM*
TYPE	*Type of Surrogate Load to be created.*
T_ACC	*Test acceleration percentage.*
PL_LIM	*Limit of test acceleration.*
TARGET_FILE	Name of target results file that feature will try to achieve with Surrogate Load.
"USERWHS"	FLAG. Element set ($HSSID_i$), property (MID_i) and Tensor data (TENSIDi) to follow.
$HSSID_i$	FSET3 ID of the CBUSH or CONN3D2 entries to be used in the fatigue analysis.
MID_i	Unique vMATFTG ID for Heat Stakes.
$TENSID_i$	ID of vUDNAME filename entry specifying tensor data.
"SPOTW"	FLAG. Conduct Spot Weld analysis with the following parameters.
SPSIDi	ID of an FSET3 entry listing the elements that represent spot welds.
MIDi	ID of vMATFTG definition for spot weld elements.
DIAM, T1, T2	User specified values for nugget diameter, top shell thickness and bottom shell thickness.
"SEAMW"	FLAG. Conduct Seam Weld analysis with the following parameters.
SWSIDi	ID of an FSET3 entry listing the elements that represent seam welds.
MIDi	ID of vMATFTG definition for seam weld elements.
NDSIDi	ID of an FSET3 entry listing grids of the elements to be retrained for the analysis.
BTHRESH	*Threshold value of the bending ratio used in interpolation between stiff and flexible SN curves.*
RTHICK	*Reference thickness / threshold for sheet thickness correction used for seam welds.*
nTHICK	Sheet thickness correction exponent used seam welds.
WTYPE	Location on weld for results output.
"ABSRESP"	FLAG. Calculate absolute random response.
$ABSVAR_i$	Character combination of D, V, A or M.
$SETID_i$	ID of a FSET3 entry with list of grid IDs for DVA data output or MONPNT3 Set Name for "M".
"RELRESP"	FLAG. Calculate relative random response.
RELVAR	Choice of D only.
SETID	ID of FSET3 entry with list of grid IDs to be used in relative response assessment.
PARAM1	Parameter for 3rd stage filtering to create pairs of nodes for relative displacement assessment.
PARAM2	Parameter for 2nd stage filtering to create pairs of nodes for relative displacement assessment.
"RELRESP2"	FLAG. Calculate distance only.
RELVAR	Choice of D only.
$NODEID_1$	ID of first node
$NODEID_2$	ID of second node
FE_UNIT	**FE Stress Units**
JOBID	CF job ID.

CNVRTS	FE stress unit's conversion.
FE_L_MAG	FRF load magnitude used in solver.
CONJCON	Choice for how multi-input PSD's are calculated.
CNVRTF	FE force unit's conversion.
vMATFTG	**Fatigue Materials Properties**
MID	Material ID.
CNVRTM	S-N units' conversion.
AMP_RNG	S-N data format in AMP or RANGE.
"STATIC"	FLAG. Static material properties UTS, E Code and SE to follow.
"CYCLIC"	FLAG. Cyclic stress-strain properties K' and n' to follow.
"KADJ"	FLAG. Surface finish / treatment corrections to follow.
"SN"	FLAG. S-N curve parameters SRI1, b1, Nc1, b2, Nfc, SE and M1-M4 to follow in future release.
"SN0", "SN1", "SN2"	FLAG. S-N spot weld curve parameters SRI1, b1, Nc1, b2 Nfc, MSS, SE, SF, DE, TC, M1-M4 to follow in future release.
"SNx_FKM"	FLAG. Mean stress slope parameters M1 through M4 representing sensitivity to mean stress.
"SNBR1", "SNBR0"	FLAG. S-N seam weld curve parameters SRI1, b1, Nc1, b2, Nfc and SE.
"SEAM_FKM"	FLAG. Mean stress slope parameters M1 through M4 representing sensitivity to mean stress along with MSS.
"EN"	FLAG. E-N curve parameters σ'_f, b, c, ε'_f, K', n', Y to follow.
AUTOSN or AUTOEN	FLAG. Allow for automatically generated SN or EN curves to be used instead of defined SN or EN curves.
"MMPDS"	FLAG. MMPDS data coefficients to follow.
"TABLE"	FLAG. Tabular S-N data provided in TID_i table and SE to follow.
"FILE"	FLAG. Material data will be referenced in external file.
UDID	ID of vUDNAME that references XML file
Type	Type of data
XML_File Reference	Reference to unique material entry in XML file.
"DIGIMAT"	FLAG. Material data to come from DIGIMAT
Type	Type of Digimat file imported
File_Location	Path to Digimat file

vMATSTAT	**Random Response Materials Properties**
MID	ID.
CNVRTM	S-N units' conversion.
'STATIC"	FLAG. Static material properties UTS and E to follow.
"CYCLIC"	FLAG. Cyclic stress-strain properties K' and n' to follow.
vFTGPARM	**Fatigue Parameters**
JOBID	CF Job ID.
"STRESS"	FLAG. Indicates stress is used in the fatigue calculation.
COMB	Stress combination MAXPREST, ABSMAXPR, SGVON, COMPX, COMPY, COMPZ, COMPXY, COMPXZ, COMPYZ, CRIT30, CRIT90, CRIT180, CRIT360.
CORR	Mean stress correction method for SN or EN. Can be None, Goodman, Modgood, Gerber, Gdmant, Grbert, MMPDS, Interp-, Interp, SOD, SWTIT, SWTITC, Walker, Morrow.
NCALC	Average (NAVER), Middle (NMID) or Upper (NUPPER).
DEBUG	FRF set to 1.0 (UNITTF) or detailed S-N output (MATOUT).
TOLER	Tolerance used when performing Neuber Rule correction.
MAXPEAK	Calculates MAX stress/strain or MIN stress/strain for output to CSV file.
RCCT	Duration for Time History with vFTGDEF ATYPE = SIMSINE.
"CYCPROC"	FLAG. Indicates rainflow cycle counting parameters are to follow.
TRSTIN	Indicates nonlinear or elastic-plastic stress is coming from Solver. Also called Neuber Stress.
EVALL	Options for calculating Mean Stress and largest rainflow cycle.
HYST	Options for dealing with hysteresis loops.
TRSTOUT	Indicates nonlinear or elastic-plastic stress is written to the output files. Also called Neuber Stress.
"SPOTW"	FLAG. Indicates spot weld analysis parameters are to follow.
COMB	Equivalent stress to use in the spot weld fatigue calculation.
CORR	Mean stress correction. Can be None, Simple, FKM (TD only).

NANGLE	The number of calculation angles in 360 degrees around the spot weld.
SWLOC	Location to report fatigue life.
ZCOMP	Whether to include the compressive part of the axial force steady state response.
"SEAMW"	FLAG. Indicates seam weld analysis parameters to follow.
COMB	Stress / strain combination to use in the seam weld fatigue calculation.
CORR	Mean stress correction. Can be None, Simple, FKM (TD only).
THICK	Thickness correction to be applied.
WELDTOP	Indicates which side of weld is top.
"CERTNTY"	FLAG. Indicates that a confidence level parameter will follow.
CERTSV	Confidence Level
"USERWHS"	FLAG. Indicates parameters for User Welds to follow.
COMB	Equivalent stress
CORR	Mean stress correction
NANGLE	The number of calculation angles in 360 degrees around the User weld.
SWLOC	Location to report fatigue life.
"FOS"	FLAG. Indicates parameters for factor of safety calculation.
LIFE	Target life to set for analysis
BACKACC	Back calculation accuracy for iterations
MAXFAC	Max factor of safety to calculate
MINFAC	Min factor of safety to calculate
vFTGSEQ	**Fatigue Load Sequence**
JOBID	CF job ID.
EVNTOUT	Output for each individual event. 0 = none or 1 = yes.
DUR_UNITS	Duration units in seconds, minutes, hours or days.
MOS_DUR	Required duration for complete duty cycle of testing.
TSCALE	Global scale factor to apply to all loads.
NREP	If YES, values in EVT are Event Duration in REPEATS.
$EVID_i$	ID of a FTGEVNT.
EVT_i	Event duration.
"UNITS"	FLAG. Indicating a fatigue equivalent unit is to be used.
EQUIV	Number of equivalent units.
EQNAME	Equivalent Name.
vFTGEVNT	**Fatigue Load Events**
EVID	Event ID.
$FLOAD_i$	ID of an FTGLOAD entry or SID of a DETLOAD or NBLOAD entry.
"NAME"	FLAG. Indicating an event name is to be used.
EVENT_NAME	Event name.
vFTGLOAD	**Fatigue Load Variation**
FLOAD	ID.
TYPE	PSD, STATIC, ZEROPSD, FLUCT, MEANLDS, TIME112, TIME103, TIME101
SID	CF set ID of a VRANDPS or NBLOAD.
SCALE	Scale factor.
OFFSET	Offset to apply to static stress; read from static stress OP2 file.
LCID	CF load case ID mapped to solver subcase ID. Use MULTI for multiple inputs.
File_Name	Location and name of the stress file.
vRANDPS	**Power Spectral Density Specification**
SID	Set ID.
J	CF load case ID that will be applied to the transfer function.
K	CF load case ID that will be applied to the transfer function. For single input PSD, J equals K.
X, Y	Real and imaginary components of the complex number. For single input PSD, X=1.0, Y=0.0. Note: X cannot equal Y.
TID	Unique ID for associated vTABRND
vTABRND	**PSD and Material Definition Table**
TID	ID.
XAXIS	"LINEAR" or "LOG" X axis.
YAXIS	"LINEAE" or "LOG" Y axis.
UDID	ID of a vUDNAME entry.
f_i	Frequency value in cycles per unit time.
g_i	Y value of table entry.
endt	Signifies end of input for vTABRND.
vUDNAME	**User Defined File Name**
UDID	Unique ID (Integer>0)
File_Name	Location and name of user defined file.
vRANDT	**PSD Time Specification**
SID	Random analysis set ID
T	Time of event

Units	Units of T
NBLOAD	**Narrow Band PSD Load Specification**
SID	CF set ID referenced in vFTGEVNT.
LCID	Load Case ID mapped to solver subcase ID
f_i	Frequency value in cycles per unit time.
b_i	Width of narrow band frequency block.
h_i	Y value of table entry (PSD units).
DETLOAD	**Deterministic Load Specification**
SID	CF set ID referenced in vFTGEVNT.
LCID	Load Case ID mapped to solver subcase ID
f_i	Frequency value in cycles per unit time.
g_i	Y value of table entry (sine amp units).
SINGSINE	**Single Sine Wave Deterministic Load Specification**
SID	CF set ID referenced in vFTGEVNT.
LCID	Load Case ID mapped to solver subcase ID
f1	Frequency value in cycles per unit time.
g1	Y value of table entry (sine amp units).
SINESW	**Swept Sine Wave Deterministic Load Specification**
SID	CF set ID referenced in vFTGEVNT.
LCID	Load Case ID mapped to solver subcase ID
f_1	Frequency value in cycles of first sine wave.
g_1	Peak value (amplitude) of first sine wave.
f_n	Frequency value in cycles of last sine wave.
g_n	Peak value (amplitude) of last sine wave.
SWTYPE	The sweep type in Decibel (LOG/DB), Octave (OCT) or Linear (HZ).
SWNUM	Number of sine bands to use.
"PARAMS"	FLAG. Indicating additional sweep parameters to follow.
SWRATE	Sweep rate per second in Hz, DB, or OCT.
N_SWEEPS	Number of times the sine is swept through the frequency range.
FRF_UNIT	Units of loading used to create FRF.
LD_UNIT	Units of loading used to define sweep.
FSET3	**KT Material Parameters**
JOBID	CF Job ID.

$ELSID_i$	ID of FSET3 file containing elements
$SCALE_i$	Scale factor to apply to elements.
$OFFSET_i$	Offset factor to apply to elements.
FSET3	**Element or Grid Set Definition**
ELSID	ID.
DES	Must be "ELEM" or "NODE".
Id_i	Element or Node numbers.
INCLDIR	**Directory Location for "INCLUDE" files**
JOBID	CF Job ID.
DIR_NAME	Directory location for the INCLUDE files. No character limit for entry.
INCLUDE	**External Text File to be Used for "includes"**
FILE_NAME	Name of include file.
RESTART	**Option to request restart from intermediate data file.**
FILE_NAME	Name of restart file.
VECTOR	**Vector Application of Solver Results**
JOBID	CF Job ID.
VXCOMP, VYCOMP, VZCOMP	X, Y and Z components of a vector that will be created from the X, Y and Z Solver FRF values. Values for components must add up to 1.0.
VINCR	Number of angle increments. Check multiple vectors through an increment of angles.
FNOTCH	**Method to apply filters (notches) to transfer function inputs.**
FNID	ID number for FNOTCH
$LCID_i$	Solver subcase ID number for Transfer Function
f_{ia}	Start frequency for notch.
f_{ib}	End frequency for notch.
$SCALE_i$	Scale factor to apply to notch. Default is none.
PERTURB	Option to create damage increments for incoming load changes.
PTYPE	Relative or absolute.
VALUE	Value of perturbation.

Entries (Commands) and Fields used in Time Domain Analysis

Below is the complete set of available entries that can be used to do a Cf time domain analysis (either in the Control File view or Process Flow view). In a typical analysis, only small selection of these items are used. Please note: Entries in *GREY* are not currently supported. See descriptions in previous section for more information or refer to the **Quick Reference Guide** document.

DATA COLUMN 1	DATA COLUMN 2	DATA COLUMN 3	DATA COLUMN 4	DATA COLUMN 5	DATA COLUMN 6	DATA COLUMN 7	DATA COLUMN 8	DATA COLUMN 9
vIBFAT	JOBID	OFILTYPE *(CSV)*	SOURCE *(NASTRAN)*	GIDOUT *(CENTER)*		JOB_NAME *(control file name)*		
	"OPTIONS"		STRLIM *(0)*			OUTDIR		
MEMORY	MEM_CAP							
PARALLEL	TH_CAP							
PARAM	NAME	SETTING						
OP2MAP(S)	FILE_NAME (for dynamic and static (S) stresses from NASTRAN)							
	$CFLCID_i$	$OP2LCID_i$						
ODBMAP(S)	FILE_NAME (for dynamic and static (S) stresses from ABAQUS)							
	$CFLCID_i$	$ODBLCID_i$						
vFTGDEF	JOBID			MID				STRBINS *(32)*
	"ELSET"	$ELSID_1$	MID_1	$ELSID_2$	MID_2	CONTINUE		
	"TIMEOUT"	TIMEVAR1	TIMEID1 or RUNTIME					
	"CRITOUT"	CRITVAR1	CRITID1					
	"HOTSPOT"	NHS *(1)*	HSGATE *(1)*	HSFLAG	STOP *(NO)*			
	"FILTER"					TOPDMG		
	"DIAGNOS"	*NHSD*	HSSET	*CHSET*	*VAR (STRESS)*			
	"RCCOUT"	*$RCCVAR_1$*	*$RCCID_1$*	*CONTINUE*				
	"USERWHS"	*$HLSID_1$*	*MID_1*	*$TENSID_1$*	*$HLSID_2$*	*MID_2*	*$TENSID_2$*	
		$HLSID_3$	*MID_3*	*$TENSID_3$*	*CONTINUE*			
	"SPOTW"	SPSID1	MID1	DIAM	T1	T2		
		SPSID2	MID2	DIAM	T1	T2	CONTINUE	
	"SEAMW"	SWSIDi	MID1	NDSID1	*BTHRESH (0.5)*	*RTHICK (1.0)*	nTHICK *(0.16667)*	WTYPE (Toe)
FE_UNIT	JOBID	CNVRTS *(MPa)*	FE_L_MAG *(1.0)*		CNVRTF *(N)*	CNVRTL *(mm)*		

vMATFTG	MID	CNVRTM *(MPa)*	AMP_RNG *(RANGE)*					
	"STATIC"		UTS	E	Code		SE	
	"CYCLIC"	K'	n'					
	"SN"	SRI1	b1	Nc1	b2 *(0.0)*	Nfc *(1.0E30)*	SE	
		M1	*M2*	*M3*	*M4*			
	"KADJ"	KU	KT	KRNUM	KRDESC			
	"SN0" "SN1" "SN2"	SRI1	b1	Nc1	b2	Nfc	MSS	SE
		SF-FXY	DE-FXY	TE-FXY	SF-MXY	DE-MXY	TE-MXY	
		SF-FZ	DE-FZ	TE-FZ	SF-MZ	DE-MZ	TE-MZ	
		M1	*M2*	*M3*	*M4*			
	"SNBR1" "SNBR0"	SRI1	b1	Nc1	b2	Nfc	SE	
	"SEAM_FKM"	M1	M2	M3	M4	MSS		
	"EN"	σ'_ϕ	b	ε'_ϕ	c	K'	n'	Y *(0.5)*
		SEe	SEp	SEc				
	"AUTOSN" or "AUTOEN"	Can be used instead of SN or EN						
	"MMPDS"	A1	A2	A3	A4			
	"TABLE"	TID_1	T_1	CONTINUE				
	"FILE"	UDID	Type	XML_File_Reference				
vFTGPARM	JOBID							
	"STRESS"	COMB *(SGVON)*	CORR *(NONE)*		DEBUG *(NONE)*	TOLER *(1.0E-5)*		
	"**SPOTW**"	COMB *(CRITICAL PLANE)*	CORR *(None)*	NANGLE *(18)*	SWLOC *(0)*	ZCOMP *(1)*		
	"SEAMW"	COMB (ABS MAXPR)	CORR *(None)*	THICK *(0)*	WELDTOP *(Upper)*			
	"FOS"	LIFE	BACKACC *(5.0)*	MAXFAC *(5.0)*	MINFAC *(0.2)*			
	"MULTIAX"	METHOD *(None)*	GATE *(20)*					
vFTGSEQ	JOBID	EVNTOUT *(0)*	DUR_UNITS *(SECONDS)*	MOS_DUR *(SUM of EVENTS)*	TSCALE *(1.0)*	NREP *(YES)*		
	$EVID_1$	EVT_1 *(1.0)*	$EVID_2$	EVT_2 *(1.0)*	CONTINUE			
	"UNITS"	EQUIV	EQNAME					

vFTGEVNT	EVID	$FLOAD_1$	$FLOAD_2$	CONTINUE				
	"NAME"	EVENT_NAME						
vFTGLOAD	FLOAD	TYPE		SCALE *(1.0)*	OFFSET			
LCID	FILE_NAME							
vUDNAME	UDID							
	FILE_NAME							
KTDATA	JOBID	ELSETi	SCALEi	OFFSETi				
FSET3	ELSID	DES	ID_1	ID_2	ID_3	ID_4	ID_5	CONTINUE
INCLDIR	JOBID	DIR_NAME *(LOCAL DIRECTORY)*						
INCLUDE	FILE_NAME							

A brief description of additional entries and fields available for a time domain analysis is given in the table below. For a full explanation of all entries please refer to the **Quick Reference Guide** document.

Brief Description of Additional Entries and Fields used ONLY for Time Domain Analysis

ENTRY NAME	DESCRIPTION
MEMORY	**MAX Memory for Time Domain Analysis**
MEM_CAP	Maximum memory that can be used in Gb
PARAM	**Setting Parameters**
NAME	Name of Parameter
SETTING	New value for Parameter
PARALLEL	**Parallel Processing for Time Domain Analysis**
TH_CAP	Maximum threads that can be used.
vFTGDEF	**Fatigue Element Definitions**
TOPSTR	Show top % of model based on largest stress range. Options available.
"TIMEOUT"	FLAG. Indicates time history output parameters to follow.
$TIMEVAR_i$	Choice of S.
$TIMEID_i$ OR RUNTIME	Set of Element ID for output OR various options to automatically output stress time histories at different elements.
"FILTER"	FLAG. Indicates ability to filter damage results.
TOPDMG	Show top % of model based on damage.
"RCCOUT"	*FLAG. Indicates RCC and damage response output.*

RCCVAR$_i$	*Option is BOTH (RCC and Damage).*
RCCID$_i$	*FSET3 set of nodes or elements to be used in RCC and damage output.*
vFTGPARM	**Fatigue Parameters**
"MULTIAX"	FLAG: Indicated that a Multiaxial assessment is required.
METHOD	The type of assessment.
GATE	A filter to ignore low stress elements.

Entries (Commands) and Fields used in a TIME2PSD Analysis with Descriptions

Below is the complete set of available entries that can be used to do a Cf TIME2PSD analysis. Please note: Entries in *GREY* are not currently supported. See descriptions information that follows or refer to the **Quick Reference Guide** document for further information.

DATA COLUMN 1	DATA COLUMN 2	DATA COLUMN 3	DATA COLUMN 4	DATA COLUMN 5	DATA COLUMN 6	DATA COLUMN 7	DATA COLUMN 8	DATA COLUMN 9
vIBFAT	JOBID				LOGLVL (0)	JOBNAME		
TIME2PSD	SRATE (*1 for RPC or RSP format*)	EVIDST	TABIDST	EVENT_N (*1*)	WINDOW (*Hanning*)	FORMAT (*CSV*)	MINF	MAX (*the Nyquist Frequency*)
	"FILEDIR"	TS_file_directory						
	"dcyfile"	*DCY_filename*						
	"MAPPING"	skip	CHAN_N	*T_UNITS (seconds)*	chan1	chan2	chan3	chan4
	chan5	chan6	chan7	chan8	chan9	chan10	chan11	cont
	"autoT"	T_Init (*0.5*)	K_PTS (*200*)	K_PCENT (*2*)	Level (*50*)	Method (*5*)		
	"autoD"	T_DEL (*0.5*)	RMS_FL (*50.0*)	HPFILT (1.0)				
	"Post"	Operation (Diagonal)	Type (if Scale)	Event	Channel			Scale (1.0)
	"Post"	Operation (Diagonal)	Type (if Envelope or Average)	Action (Scale)	Envelope Channel	*StartF*	*EndF*	Scale (1.0)
	"Post"	Operation (Target)	Type (if stress)	Event	Channel	Element	Node	Layer
	"Post"	Operation (Target)	Type (if acce)	Event	Channel	Direction		
	"EV_OPTS"	EV_NUM	*NSI (1)*	*RMSI (1)*	*TSMOOT (1)*	SF (*1.0*)	T	δ (*0.0*)
	Load_name							

	"SUBEV"	S1	S2	S3	S4	S5	S6	
	T1	T2	T3	T4	T5	T6	T7	T8

GREEN = Control Entry or Flag **BLACK = Enter User Data** **GREY = Not Currently Supported** **RED = Special User data fields can be any number of characters** **ORANGE – Default entries**

NOTE:

See vIBFAT entry for details regarding the first row. Note the following difference:

a. LOGLVL=0: Produces PSDM files with the following format "***load_name" _ "event number" _PSDM.txt***.

b. LOGLVL=1 or 2: Produces PSDM files mentioned above PLUS the files below. These files are needed if you wish to use the FFT PLOTTER to view results.

i. ***_autoPSD_debug.txt*** files that provide XY data to plot Channel PSD data versus Frequency.

ii. ***_time_data.txt*** files that provide XY data to plot Channel Time History data versus Time (for the original time histories).

iii. ***_time_data_del.txt*** files that provide XY data to plot Channel Time History data versus Time (after DELETES has modified time histories).

iv. ***_subev_time_data.txt*** files that provide XY data to plot Channel Time History data verse Time (after SUBEV has modified the time histories).

- "auto" is a legacy entry that will only be supported for a short time. "auto", "autoT" and "autoD" applies only to events where the values for buffer window length, overlap and T are not specified. If these values are specified, TIME2PSD will do calculations based on these inputs.
- TIME2PSD can only be used as a pre-processing step. It must be run before the main fatigue (CF) job is executed.

ENTRY NAME	DESCRIPTION
SRATE	If using an RPC (RSP) file, SRATE will automatically be obtained from the file and this entry will be ignored. If using an CSV file, the User must input the sampling rate (SRATE) of the time history. -- Required, Real>0, Default=1.
EVIDST	Start (vRANDPS) ID for events. -- Required, Integer>0, no default.
TABIDST	Start ID for vTABRND tables. -- Required, Integer>0, no default.
EVENT_N	Number of time history event files. -- Required, Integer>0, Default=1.
WINDOW	Window function to use. Options are Hanning Window or no window. This is applied to the "block" of data extracted from the total time signal. -- Optional, Character, Default=Hanning.
FORMAT	Format of time signal files. Options are RPC or RSP for binary format road load data. CSV for text format. **NOTE: the CSV file must only contain columns of Y data i.e., no X value starting column.** Header rows of text are allowed. -- Required, Character, Default=CSV.

MINF	Frequency below which the PSD data will be set to zero. Must be less than MAXF. – Optional, Real≥0.0, Default=0.0.
MAXF	Max frequency in output (used to override the Nyquist frequency when outputting PSD data). This is useful when PSDM files are much larger than necessary due to frequency content in the time history that is well above the last FRF frequency of interest. This entry has the effect of suppressing the writing of all PSD and cross PSD frequencies above MAXF. Must be greater than MINF. – Optional, Real>0.0, Default = the Nyquist Frequency.
"FILEDIR"	FLAG. Used to specify the directory where all input time history files are located and where output data files will be written when vIBFAT > LOGLVL=1.
TS_filedirectory	This should correspond to the name of the directory. – Required, Character, no default.
"dcyfile"	*FLAG. Name of the DCY file used to specify repeats of each event, the event names and order of events.*
DCY_filename	*This should correspond to the DCY filename. If no file exists, then the vFTGSEQ entry in the Cf analysis must be used to specify the time durations. – Optional, Character, no default.*
"**MAPPING**"	FLAG. Used to specify the order of channel data. Required when not all channels are used from the RPC/RSP/CSV file. Otherwise, this entry it optional – Optional, Integer>0, no default.
skip	Number of header lines to skip in an asci file. – Optional, Integer>0, no default.
CHAN_N	Number of channels in event file to use. Required if not all channels from RPC/RSP/CSV file are used in conversion. Otherwise, format is -- Optional, Integer>0, no default.
T_UNITS	*Time units. Options are seconds and hours. – Optional, Character, Default=seconds.*
$chan_i$	Location in asci input file for channel "i" of data. – Optional, Integer>0, no default.
"EV_OPTS"	FLAG. Optional Event parameters are to follow (one set for each Event).
EV_NUM	Number of Event. First EV_NUM must equal EVIDST number and each consecutive EV_NUM must increase by 1. Example: EVIDST=55 so therefore, EV_NUM_1=55, EV_NUM_2=56, EV_NUM_3=57, etc. The number of EV_OPTS entries must match the number specified in EVENT_N. – Required, Integer>0, no default.
NSI	*Number of non-stationary intervals for this Event. – Optional, Integer>0, Default=1.*
RMSI	*Number of rms scaling intervals for this Event. – Optional, Integer>0, Default=1.*
TSMOOTH	*Number of adjacent time points to be used for temporal smoothing of response PSD for this Event. – Optional, Integer>0, Default=1.*
SF	Scale factor to apply to time signals in this Event before FFT. – Optional, Real>0.0, Default=1.0.

T	Length of FFT buffer window function in time for this Event. NOTE: if using AutoT, this entry must be left blank otherwise this entry will override AutoT. - Required, Real>0.0, no default.
δ	Overlap or gap in time between windows for this Event. Positive value mean overlap. Negative value means gap. Value must be less than T. NOTE: if using AutoT, this entry must be left blank otherwise this entry will override AutoT. – Required, Real≥0.0, Default=0.0.
Load_name	Name of the input time history loading file used for this event (e.g. “load.rsp”). – Required, Character, no default.
“SUBEV”	FLAG indicating that a subevent will be created using the specified times S_1, S_2, etc. This entry must be placed below the Load_Name entry of the Event of interest along with the T values (if used).
S_i, S_{i+1}	Used to separate time segments within an Event (defined by pairs of time values S_1-S_2, S_3-S_4, S_5-S_6, etc.) that become additional events in the analysis. – Required, Real>0.0, no defaults.
MANUAL DELETES	
T_i, T_{i+1}	These entries are used to manually delete sections in an Event (or subevent) and are defined by pairs of time values t_1-t_2, t_3-t_4, t_5-t_6, t_7-t_8, etc. These portions of the Event are removed before the FFT process is applied. The T1 value can be set to START if the User wishes to use the first point in the Event. The last value can be set to END if the User wishes the deleted section to span to the end of the Event. These T values must be listed directly below the Load_Name to which it applies. IF SUBEV is being used, then SUBEV would be below the Load_Name entry **first**, followed by the Delete T values. NOTE: if using AutoD, this field must be left blank otherwise this field will override AutoD. – Optional, Real>0.0 and Character=Start or END, no default.

Additional Automatic Functions - AutoT, AutoD.

“autoT”	FLAG indicating that an automatic FFT buffer window length “T” will be calculated.
T_Init	Initial value of the FFT window length in seconds. – Required, Real>0.0, Default=0.5.
K_PTS	Target value / range to use for calculation of K_PTS based on Methods 1 to 6 described below. If a single value is specified, then the range to be used is determined using K_PCENT above and below the single value. Example: 3,4 for range input or 3 for target value (best values to use for Method 1 through 4). – Required, Integer>0, Default=200 (for Method 5).
K_PCENT	Percentage used to define a range of K_PTS when only a single value is specified for

	K_PTS. – Required, Real>0.0, Default=2 (2%).
Level	Percentage of maximum used as the reference line when evaluating K_PTS above and below the reference line. – Required, Integer>0, Default=50 (50%).
Method	The method to use when evaluating K_PTS. – Required, Integer=1 to 6, Default=5. 1 – Number of K_PTS above a line defined as a percentage (level) of the maximum. 2 – As Method 1 but normalized by total number of points in PSD plot. 3 – As Method 1 but where the Level used is the PSD average defined as the sum of the PSD values divided by the total number of points in the PSD plot. 4 – As Method 3 but where the result calculated for K_PTS is normalized by the total number of points in the PSD plot. 5 – K_PTS is calculated as the number of peaks or valleys in the PSD plot. 6 – As Method 5 but where the result calculated for K_PTS is normalized by the total number of points in the PSD plot.
“autoD”	FLAG indicating that automatic deletes based on signal intensity will be calculated. Note: TIME2PSD will look for Ti values FIRST to use for the manual deletion of time history segments from within an event. If Ti values are not present, the software will use AutoD to make automatic deletes within each event using the parameters defined for AutoD.
T_DEL	Value of window length in seconds used for deletes assessment. – Required, Real>0.0, Default=0.5.
RMS_FL	RMS level of window (of length T_DEL) as a percentage of PSD file average below which deletes will apply. – Required, Real>1.0, Default=50.0 (50%).
HPFILT	High pass filter value used to pre-filter the time data before deletes are assessed. Any data removed by the filter is restored before the FFT processing. – Required, Real>0.0, Default=1.0.

Entries required for Bounding Diagonals

“**Post**”	Optional FLAG indicating that a Bounding Diagonals request has been made.
Operation	Type of operation to be carried out. Current options are Diagonal and Target. – Required, Character, no default.
Type	The type of action to be carried out. Options are Scale and Envelope for Operation=Diagonal and Stress and Acce for Operation=Target. – Required, Character, no default.
Event	Event ID. – Required, Integer, no default.
Action	Action to be carried out. Only option is SCALE when using Operation=Diagonal and Type=Envelope or Average. – Required, Character, no default.
Channel or Envelope Channel	Channel or Envelope Channel ID. – Required, Integer, no default.

Element	Element ID. – Required, Integer, no default.
Node	Node ID. – Required, Integer, no default.
Layer	Layer ID. Options are None, Lower and Upper. Used only for Target>Stress selection – Required, Character, no default.
Direction	Acceleration direction required when Operation=Target and Type=acce. Options are X, Y, Z, Mx, My and Mz. – Required, Character, no default.
Scale	Scale factor to apply. Used only for Operation=Diagonal – Required, Real>0.0, Default=1.0.

Rules for Entries and Fields in a Control File or TIME2PSD

If the User is going to use the Control File view to run a fatigue analysis or TIME2PSD analysis, with few exceptions, the Control File is 10 columns wide with 8 characters per column (called a field). An exception is made for identifying the filenames or locations where you are allowed more than 8 characters. The case of a field does not matter: DIRLIK is the same as dirlik.

The 10th field is left blank even in the case of continuations. Continuations are detected automatically because more information is expected by the program.

Comment lines are to begin with $, % or #. In-line comments (comments on the same line as an entry line) also supported. Tab characters are not supported so in the case of blank entries, the User must use blank spaces to create the blank field (not tabs).

Special Rules for the INCLUDE Entry in a Fatigue Control File

INCLUDE entries permit the addition of a text string (saved as a text file) into the fatigue control file. No nesting of includes is allowed and the "Include" must be a whole line containing only the include instruction,

e.g., ***INCLUDE LOADFILE_A*** or ***INCLUDE MAT_A***. Any included filename name must not have spaces.

If needed, an optional **INCLDIR** entry can be used to define the directory location of the “Include” file as follows:

INCLDIR JOBID "address of directory for include files"

Note: ALL Include entries that follow will use this directory location. This may be changed before further includes are specified by adding another INCLDIR entry. In case of errors during input or during the solution phase, the log file (extension .log) is to be consulted for the cause of the error and to see where the included files were being pulled from.

5 | CONTROL OF OUTPUT RESULTS AND LOADS SCALING

Output Control Options – OFILTYPE & LOGLVL (in vIBFAT), ELSET (in vFTGDEF) and EVNTOUT (in vFTGSEQ)

Great care should be taken with the control of outputs because using various settings can mean that important output is not obtained, or enormous, unnecessary volumes of output can be written to the hard drive.

There are 4 variables that control output.

1. The OFILTYPE (vIBFAT entry) controls whether a CSV file, FEF file (Patran Neutral), H3D file or combinations of these files are written. The CSV file is the most complete version of results. When written to FEF and H3D files the results are split into subcategories for nodes and elements, and then into top, bottom and worst. It is also possible to limit which of these FEF files are written, for example element top only. In the current release, HyperView (H3D) files can only be written for element results. The Cf interface also includes a CSV to H3D translator that converts the results into a single H3D file for reading into HyperView. This is a much more elegant option than specifying H3D as part of OFILTYPE. The H3D translator is accessed from the “utilities” icon in the top right menu bar.

2. **FREQUENCY DOMAIN ONLY**: The LOGLVL (vIBFAT entry) controls whether additional data is written to the output file (0=Basic Output, 1=Basic plus PSD Output and 2=Basic plus PSD and Damage Output). Care should be taken when specifying LOGLVL=1 or 2 to restrict the element locations where results are requested (via ELSET) otherwise there will be a large amount of output created for every element / node in the model. Even for LOGLVL=0, PSD response and transfer function output is still written for the element with the highest RMS stress in the model. It is this element that is plotted by default in the PSD Plotter. Please note, however, that if the highest stress element is not included in the Users ELSET sets then the PSD.csv file may be blank.

3. ELSET (vFTGDEF entry) combined with FSET3, can be used to limit results output to the element groups listed on an FSET3 entry. It is **highly recommended** that an ELSET / FSET3 combination be used if LOGLVL is set to 1 or 2. Otherwise, Cf will write out PSD and RCC data for **ALL** elements in the model file. This can significantly delay the Cf analysis while the software writes out this data.

4. EVNTOUT (vFTGSEQ entry) controls whether individual event results are written (1) or just results for the sum of events (0). Requested summed results (EVNTOUT=0), will reduce the output data written to the main results CSV file. NOTE: If FEF/H3D files are requested for EVNTOUT=0, there will be 6 FEF/H3D files written - top (T), bottom (B) and worst (W) for both nodes and element centers. If FEF/H3D files are requested for EVNTOUT=1, there will be 6 files for each event and the sum of the events, which will be a significant increase in output. **A better option for writing H3D files can be found in the UTILITIES menu of the GUI**.

Standard Cf output goes to the CSV / FEF / H3D files, as requested, and will include the items listed in the table below for each Element ID, Grid ID, Layer ID and Event. All results are written to one CSV file. Separate FEF/H3D files are written for each event and the sum of events. Results are also written separately for top layer, bottom layer or worst. Where solids elements are included in the model, these results are written to all 3 (top, bottom and worst). Results are also written separately for element center results and element nodal results. For example, with a QUAD 4-noded shell element there will be a center result written to both top and bottom layer FEF/H3D files. There will also be 4 nodal results written to both the top and bottom FEF/H3D files. The following options are possible.

Note that *** can be FEF or H3D.

Table 5-1 Options for OFILTYPE (note that the CSV file can be translated into an H3D file in the Utilities section of the GUI).

	*.log	*.csv	PSD.csv (Frequency Domain Only)	RCC.csv (Frequency Domain Only)	Element FEF/H3D results			Nodal FEF/H3D results		
OFILTYPE					Top	Bottom	Worst	Top	Bottom	Worst
CSV	✓	✓	✓	If LOGLVL=1 or 2						
FEF or H3D	✓		✓	If LOGLVL=1 or 2	✓	✓	✓	✓	✓	✓
CSV***	✓	✓	✓	If LOGLVL=1 or 2	✓	✓	✓	✓	✓	✓
CSV***ET	✓	✓	✓	If LOGLVL=1 or 2	✓					
CSV***EB	✓	✓	✓	If LOGLVL=1 or 2		✓				
CSV***EW	✓	✓	✓	If LOGLVL=1 or 2			✓			
CSV***NT	✓	✓	✓	If LOGLVL=1 or 2				✓		
CSV***NB	✓	✓	✓	If LOGLVL=1 or 2					✓	
CSV***NW	✓	✓	✓	If LOGLVL=1 or 2						✓
***$$	✓		✓	If LOGLVL=1 or 2	same options as CSV***$$ where $$ can be ET, EB, EW, NT, NB, NW					

If vIBFAT: GIDOUT=NODAV output is requested, it is assumed the FE model results file contains element corner results. In this case each set of results at a node (keeping layers separate) will be combined separately at each frequency. For a shell, this will result in one single corner result for top layer and another for the bottom layer (at each frequency). For a CQUAD4 mesh there will generally be 4 or more results for each node (at both top and bottom). The example below shows the results obtained with GIDOUT=CORNER and GIDOUT=NODAV for example TPL1.txt (output files will start with the text Result1). Depending on LOGLVL, additional CSV files are created and contain the following for each frequency, element, Grid ID, and layer.

NOTE: In a **Time Domain** analysis LOGLVL is ignored and only the LOG and main CSV files are produced.

LOGLVL = 0: writes the following files:

- **Results1.log** – log file containing job execution runtime information and details on the top 10 damaging elements
- **Results1.csv** – CSV file containing the information listed below for all elements and grids (nodes) in the model.
- **Results1PSD.csv** – CSV file containing input (load) PSD, equivalent stress (transfer function), response PSD and moment information for the element with highest RMS stress output **ONLY**.
- **Results1.cfvdata** & **Results1.cfvdata2** – these are binary files required by the program do display GUI information. This is also the files that are read for the RESTART entry.

LOGLVL = 1: writes the same output as LOGLVL=0 with the following changes/additions:

- **Results1PSD.csv** – CSV file now contains the PSD and moment information for all elements in the model.
- **Results1RCC.csv** – new CSV file that contains a Palmgren-Miner cumulative damage sum (Damage) at each element / grid (node) location.

LOGLVL = 2: writes the same output as LOGLVL=1 with the following changes:

- **Results1RCC.csv** – CSV file now contains (1) Rainflow Cycle Count (actual number of cycles to failure “n”) for each specified grid ID (node) as a function of stress (BIN). (2) The stress value (S) at the center of each BIN. (3) Allowable number of cycles to failure “N” for each specified GID as a function of stress (BIN). (4) Palmgren-Miner cumulative damage sum for each stress (BIN).

Table 5-2 Control of output information using vIBFAT entry LOGLVL.

	Log file	Main CSV file	PSD.csv file LOGLVL=0	PSD.csv file LOGLVL=1	RCC.csv file LOGLVL=1	RCC.csv file LOGLVL=2
Run time information	✓					
All 4 spectral moments		✓				
Cumulative moments for specified GID's			✓	✓		
Input PSD, Transfer Function, Response PSD for worst element node			✓	✓		
Input PSD for specified GID's (as function of frequency)				✓		
Transfer function for specified GID's (as function of frequency)				✓		
Response PSD for specified GID's (as function of frequency)				✓		
RMS stress		✓				
RMS strain		✓				
E[0]		✓				
E[P]		✓				
Irregularity factor		✓				
Mean Stress		✓				
Mean + 3 Sigma stress		✓				
Mean – 3 Sigma stress		✓				
Mean + 3 Sigma strain		✓				
Mean – 3 Sigma strain		✓				
Damage/event		✓				
Damage/duty cycle		✓				
Log Damage/event		✓				
Log Damage/duty cycle		✓				
Life (User units)/event		✓				
Life (User units)/duty cycle		✓				
Life (cycles)/event		✓				
Life (cycles)/duty cycle		✓				
MOS/event		✓				
MOS/duty cycle		✓				
Plasticity Index		✓				
Frequency at Max Response		✓				
Actual "n" - Rainflow cycle count for specified GID's (as function of stress BIN)						✓
Allowable "N" for specified GID's (as function of stress BIN)						✓
Palmgren Miner damage sum for specified GID's (as function of stress BIN)					✓	✓

CSV Output File Formats – Main, PSD, DVA, D_REL, Spot, Seam, Force, etc.

The Cf program will write out **many** types of output files depending on the output requests specified in the control deck. The most common (and by default) is the MAIN CSV file, which is detailed below. See **Appendix Section** for details on all the type of output files including (but not limited to):

- filename**.csv** – contain default output data including spectral moment, basis random response, stress, strain, damage, life, etc.
- filename**PSD.csv** – contains Input PSD, Transfer Function and Response PSD output data generated for the PSD Plotter.
- Filename**RCC.csv** – contains stress bin values, actual cycles (small n) and cycles to failure (large N) and Palmgren Miner damage sums.
- filename_**dva.csv** – contains displacement, velocity and/or acceleration output data generated when request is for advanced random response output (vFTGDEF: ABSRESP)
- filename_**d_rel.csv** – contains the output data generated when request is for collision detection (vFTGDEF: RELRESP).
- filename_**spot.csv** – contains the output data generated when request is for spot weld (vFTGDEF: SPOTW).
- filename_**seam.csv** – contains the output data generated when request is for seam weld (vFTGDEF: SEAMW).
- filename_**force.csv** – contains the output data generated when requesting force data (vFTGDEF: ABSRELP: ABSVAR=F and vFTGDEF:USERWHS).
- Filename_**CRITOUT.csv** – contains the output data generated when requesting 3D critical plane data (CRITOUT) for a time domain analysis.
- **TIMEOUT**_Filename.**h5** – contains the output data in an HDF5 format that is generated when requesting stress response time history data (TIMEOUT) for a time domain analysis.

Main CSV File Format

The results within the **filename.csv** output file are listed in the following order:

A	B	C	D	E	F	G	H	I	J
Element	Grid	Layer	Elset ID	MID	Event	M0	M1	M2	M4
K	L	M	N	O	P	Q	R	S	T
rms	rms_strain	E[0]	E[p]	Irr_ Factor	Mean Stress	Mean + MAXPEAK * rms	Mean - MAXPEAK * rms	Mean + MAXPEAK * rms strain	Mean – MAXPEAK * rms strain
U	V	W	X	Y	Z	AA	AB		
Damage (for Duty Cycle)	Log Damage (for Duty Cycle)	Life (in Duration Units)	Log of Life (in Duration Units)	MOS / FOS *	Plasticity Index	Frequency at Max Response	Critical Plane Angle	**	

* Margin of Safety (MOS) is replaced by Factory of Safety (FOS) when FOS is selected.

** EXTRA COLUMNS of data will be in the CSV file for a Time Domain analysis if vFTGPARM>MULTIAX is set to any value other than NONE. See User Guide “Introduction to Multiaxial Assessment” for further information on the additional columns added to the CSV file depending on the multiaxial assessment method chosen.

Some comments:

- The order and listing of output will change with various selected parameters (i.e. selection of SIMSINE, inclusion of Event name, etc.). However, the above listing is typical for stress output.
- **RMS Strain** (column L) is the elastic plastic strain calculated using the elastic FEA stress results and the Neuber Rule for conversion. This column will only be populated if the cyclic strength coefficient (K') and strain hardening exponent (n') are provided in the vFTGMAT or VFTGSTAT entries.
- For a **Time Domain** analysis, columns Q, R, S, T are replaced by MAX Stress, MIN Stress, MAX Strain and MIN Strain.
- The **Margin of Safety - MOS** (column Y) is a percentage value that can be positive or negative. If positive, the User can increase the loading until the positive percentage becomes zero. At this point, the model will fail within the required duration specified by the User (vFTGSEQ: MOS_DUR). If the percentage is negative, the model will fail sooner than the required duration.
- The **Plasticity Index** (column Z) is defined as the real (elastic plastic) max strain (***Mean + MAXPEAK*rms_strain***) divided by the FEA (elastic) strain (***Mean + MAXPEAK * rms_stress*** / ***E***). There is no Plasticity Index for a Time Domain analysis.
- The **Critical Plane Angle** (column AB) is populated if vFTGPARM>COMB = critical plan angle when doing a Time Domain analysis. Also, the MEAN + / - columns are replaced with MAX / MIN data when doing a Time Domain analysis.
- **Damage** values are presented as Damage for the Duty Cycle i.e., damage / sec * event duration (converted to seconds from duration units), whereas **Life** is presented as life per duration units. LIFE is the number of duration units required to FAIL due to the input loading applied in the Cf file.
- A summary of the fatigue damage results (for top 10 elements) is presented in the LOG file.

Example Output:

Table 5-3 Frequency Domain analysis output results for elements 13897 and 5971 from TPL2 in User Guide.

Element	Grid	Layer	Elset Id	Material ID	Event	m0	m1	m2	m4	rms	rms_strain	E[0]	E[p]	Irr_Factor	Mean Stress	Mean+ 3*rms	Mean- 3*rms	Mean+ 3*rms_strain	Mean- 3*rms_strain	Damage (Duty Cycle)	Log Damage (Duty Cycle)	Life (seconds)	Log of Life (seconds)	MOS	Plasticity Index	Freq at peak
5971	0	none	600	60	100	4.40E+02	4.08E+03	8.01E+04	6.74E+07	2.10E+01	0.00E+00	1.35E+01	2.90E+01	4.65E-01	0.00E+00	6.29E+01	-6.29E+01	0.00E+00	0.00E+00	1.40E-11	-1.09E+01	7.12E+10	1.09E+01	7.67E+01	0.00E+00	4.88E-01
5971	2658	none	600	60	100	8.06E+02	7.23E+03	1.37E+05	1.14E+08	2.84E+01	0.00E+00	1.30E+01	2.89E+01	4.51E-01	0.00E+00	8.52E+01	-8.52E+01	0.00E+00	0.00E+00	1.61E-09	-8.79E+00	6.19E+08	8.79E+00	7.28E+01	0.00E+00	8.85E+00
5971	2622	none	600	60	100	3.75E+02	4.12E+03	8.60E+04	7.65E+07	1.94E+01	0.00E+00	1.51E+01	2.98E+01	5.08E-01	0.00E+00	5.81E+01	-5.81E+01	0.00E+00	0.00E+00	4.65E-12	-1.13E+01	2.15E+11	1.13E+01	7.66E+01	0.00E+00	8.85E+00
5971	2671	none	600	60	100	3.63E+02	3.24E+03	6.06E+04	4.87E+07	1.91E+01	0.00E+00	1.29E+01	2.84E+01	4.55E-01	0.00E+00	5.72E+01	-5.72E+01	0.00E+00	0.00E+00	2.94E-12	-1.15E+01	3.40E+11	1.15E+01	7.67E+01	0.00E+00	8.85E+00
5971	2616	none	600	60	100	6.27E+03	5.57E+04	1.03E+06	8.15E+08	7.92E+01	0.00E+00	1.28E+01	2.81E+01	4.56E-01	0.00E+00	2.37E+02	-2.37E+02	0.00E+00	0.00E+00	8.40E-03	-2.08E+00	1.19E+02	2.08E+00	2.65E+01	0.00E+00	8.85E+00
13897	0	lower	600	60	100	1.57E+03	1.36E+04	2.55E+05	2.30E+08	3.97E+01	0.00E+00	1.27E+01	3.00E+01	4.24E-01	0.00E+00	1.19E+02	-1.19E+02	0.00E+00	0.00E+00	2.88E-07	-6.54E+00	3.47E+06	6.54E+00	6.20E+01	0.00E+00	8.85E+00
13897	0	upper	600	60	100	1.57E+03	1.36E+04	2.55E+05	2.30E+08	3.97E+01	0.00E+00	1.27E+01	3.00E+01	4.24E-01	0.00E+00	1.19E+02	-1.19E+02	0.00E+00	0.00E+00	2.88E-07	-6.54E+00	3.47E+06	6.54E+00	6.20E+01	0.00E+00	8.85E+00
13897	2858	lower	600	60	100	1.72E+03	1.48E+04	2.73E+05	2.43E+08	4.15E+01	0.00E+00	1.26E+01	2.98E+01	4.22E-01	0.00E+00	1.24E+02	-1.24E+02	0.00E+00	0.00E+00	5.76E-07	-6.24E+00	1.74E+06	6.24E+00	6.03E+01	0.00E+00	8.85E+00
13897	2858	upper	600	60	100	1.72E+03	1.48E+04	2.73E+05	2.43E+08	4.15E+01	0.00E+00	1.26E+01	2.98E+01	4.22E-01	0.00E+00	1.24E+02	-1.24E+02	0.00E+00	0.00E+00	5.76E-07	-6.24E+00	1.74E+06	6.24E+00	6.03E+01	0.00E+00	8.85E+00
13897	2859	lower	600	60	100	2.29E+02	1.87E+03	3.15E+04	2.62E+07	1.51E+01	0.00E+00	1.17E+01	2.88E+01	4.07E-01	0.00E+00	4.54E+01	-4.54E+01	0.00E+00	0.00E+00	4.38E-14	-1.34E+01	2.28E+13	1.34E+01	7.67E+01	0.00E+00	8.85E+00
13897	2859	upper	600	60	100	2.29E+02	1.87E+03	3.15E+04	2.62E+07	1.51E+01	0.00E+00	1.17E+01	2.88E+01	4.07E-01	0.00E+00	4.54E+01	-4.54E+01	0.00E+00	0.00E+00	4.38E-14	-1.34E+01	2.28E+13	1.34E+01	7.67E+01	0.00E+00	8.85E+00
13897	2756	lower	600	60	100	4.09E+03	3.59E+04	6.91E+05	6.36E+08	6.39E+01	0.00E+00	1.30E+01	3.03E+01	4.29E-01	0.00E+00	1.92E+02	-1.92E+02	0.00E+00	0.00E+00	4.59E-04	-3.34E+00	2.18E+03	3.34E+00	3.90E+01	0.00E+00	8.85E+00
13897	2756	upper	600	60	100	4.09E+03	3.59E+04	6.91E+05	6.36E+08	6.39E+01	0.00E+00	1.30E+01	3.03E+01	4.29E-01	0.00E+00	1.92E+02	-1.92E+02	0.00E+00	0.00E+00	4.59E-04	-3.34E+00	2.18E+03	3.34E+00	3.90E+01	0.00E+00	8.85E+00

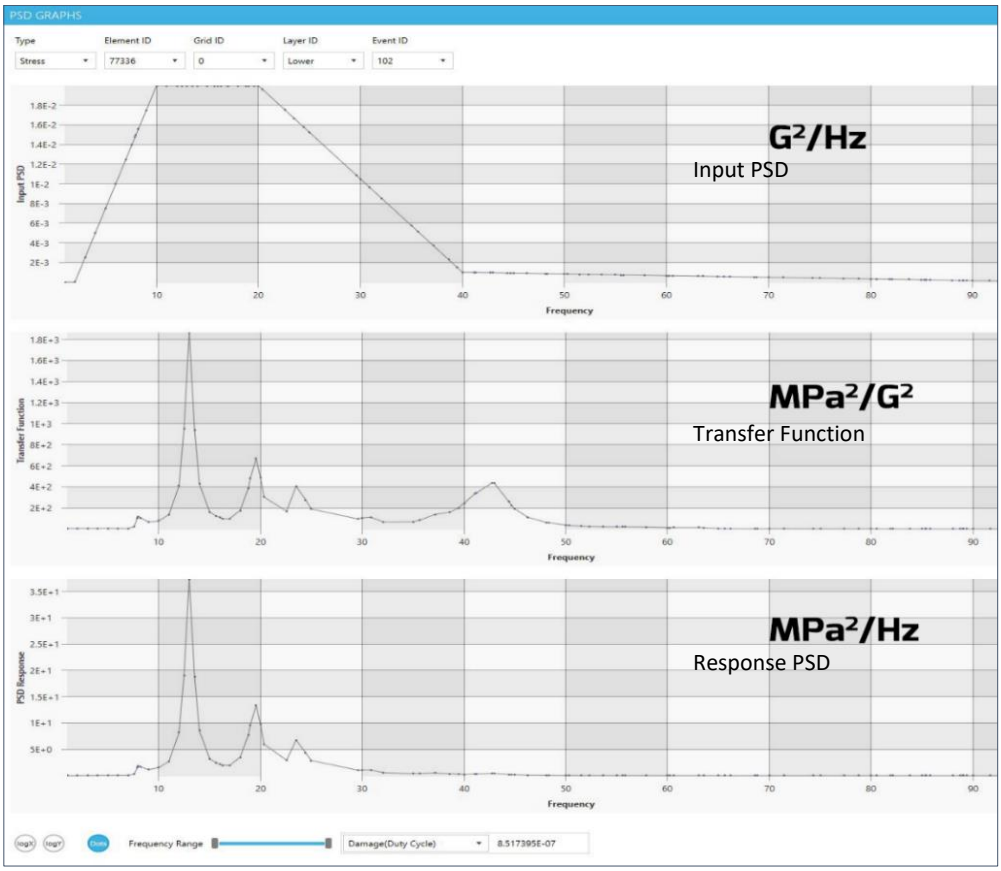
Global Scaling Options - FE_L_MAG (in FE_UNIT), TSCALE (in vFTGSEQ) and SCALE (in vFTGLOAD)

It is often necessary to scale the output provided by the solver or scale the input defined by the User in CF. Below are examples of where and how to scale these loads.

First, the general assumptions are:

- A one-unit gravity load (**G**) generates a FRF stress output (**MPa**) within the OP2, ODB or RST file.
- A **G^2/Hz** input PSD (defined within CF) MULTIPLIED by a **MPa^2/G^2** transfer function (generated by Cf from solver MPa output) EQUALS a **MPa^2/Hz** response PSD generated by CF. See following image for graphical representation.

Figure 5-1 Example of Input PSD, Transfer Function, Response PSD and the units for each.



SCALING OPTION 1: **FE_L_MAG entry in the FE_UNITS entry**:

FE_L_MAG is a scale factor applied to the equivalent stress calculated within CAEfatigue from the complex stress tensor coming from the Solver. It is used to adjust the stress if they were created using the wrong magnitude or unit within the solver and therefore, do not match the units in CAEfatigue. After this scale factor is applied, the equivalent stress is squared to create the transfer functions.

Wrong Magnitude Example: If something other than unity (1) was used to calculate the stress output in the solver for the FRF and time domain analyses, we can use FE_L_MAG to adjust the equivalent stresses to accommodate for this.

Setting FE_L_MAG=0.5 tell Cf that 0.5 G load was used and the original output stresses (MPa) from the solver need to be doubled to represent the stress that would have resulted if 1.0 G had been used instead of 0.5 G. This can be shown by the following

$$\frac{New\ Stress}{Original\ Stress} = \frac{Unit\ Load}{Original\ Load}$$

$$New\ Stress = \ Original\ \ Stress * \frac{Unit\ Load}{Original\ Load}$$

$$New\ Stress = \ Original\ Stress * \frac{1\ G}{0.5\ G}$$

Where **FE_L_MAG** equals the Original Load used of 0.5 G. In other words, the equivalent stress output (MPa) will be DIVIDED by the value entered for FE_L_MAG to calculate the new equivalent stress that would be generated from a Unit Load (G).

Wrong Unit Example : If the unit loading used in the solver was in different units than what is being used in CAEfatigue then we can use FE_L_MAG to adjust the equivalent stresses to accommodate for this.

Let's assume that 1 in/s^2 was used as the unit load in the solver, but the PSD has units of G in in/s^2. That means that 1G (or 386.1 in/s^2) should have been used in the solver instead of 1 in/s^2. We could rerun the solver, but it is easier to set **FE_L_MAG= (1/386.1) or 0.00259** to tell CAEfatigue that the stresses coming over from the Solver were actually made using 1 in/s^2 and not 1G. Knowing this, CAEfatigue will divide the stresses by this FE_L_MAG value to get stresses that are much larger; as if they had been generated by 1G loading instead of 1 in/s^2 loading This action will adjust the stress to match the G unit of the PSD.

FE_L_MAG is a global scale factor for all original FRF stress output points (MPa) from the solver and is applied BEFORE the transfer function is generated within CF.

Format:

Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Column 7	Column 8	Column 9
FE_UNIT	JOBID	CNVRTS	**FE_L_MAG**	CONJCON				

Format in Process Flow:

(Found in **Frequency Analysis** box in PSD analysis or **Time Analysis** box in LQSTATIC and LMTRANS analysis)

SCALING OPTION 2: TSCALE entry in the vFTGSEQ entry:

a. TSCALE also tells Cf that you want to scale the Cf equivalent stress values. For example, **TSCALE=2** will multiply the stress output by 2 prior to the calculation of the Transfer Function; the exact opposite of FE_L_MAG. This can be shown by

$$\frac{NewStress}{OriginalStress} = \frac{UnitLoad}{OriginalLoad}$$

$$NewStress = Original\frac{Stress * UnitLoad}{OriginalLoad}$$

$$NewStress = OriginalStress * \left(\frac{1G}{0.5G}\right) \vee OriginalStress * 2$$

Where **TSCALE** equals the Unit Load / Original Load, which equals 2. In other words, the original stress output (MPa) from the solver will be MULTIPLIED by the value entered for TSCALE to calculate the new equivalent stress that would be generated from a unit G load. TSCALE is the reciprocal of FE_L_MAG.

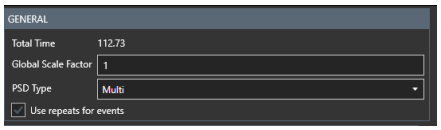
b. CF then takes this new scaled stress value and converts it to a point on the transfer function by squaring it (MPa^2) and dividing by G^2.

c. TSCALE is a global scale factor for all original stress output points (MPa) from the solver and is applied BEFORE the transfer function is generated within CF.

Format:

Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Column 7	Column 8	Column 9
vFTGSEQ	JOBID	EVNTOUT	DURATION UNIT	MOS_DUR	**TSCALE**			
	EVID1	EVT1	EVID2	EVT2	EVID3	EVT3	…	…
	"UNITS"	EQUIV	EQNAME					

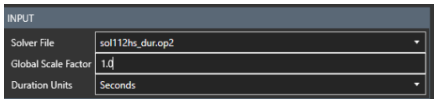
Format in Process Flow (Frequency Domain):



(Frequency Load Scheduler in PSD analysis)

Format in Process Flow (Time Domain):

(Time Load Scheduler box in LQSTATIC analysis)



(Modal Transient Loads box in LMTRANS analysis)

Revisit previous example of unit load used but in the wrong units.

In some cases, the FRF calculation is done within the solver using a 1 **m/s²** unit load, however, the input PSD being specified is in units of **G** (gravity). As stated in the previous example, the units between INPUT PSD and TRANSFER FUNCTION **must match up** in order to get the correct units for RESPONSE PSD. To correct this mismatch of units, we can use TSCALE to scale the equivalent stress to get results that would have been calculated if we had used a 1 G unit load in the original FRF calculation.

Since 9.81 m/s² is equivalent to 1 G, we set **TSCALE = 9.81**. This tells Cf that the original unit load equivalent stress values need to be multiplied by 9.81 in order to get the equivalent stresses to be equal to those what would have been generated using a 1 G unit load instead of a 1 m/s² unit load.

$$NewStress = OriginalStress * \left(\frac{1G}{0.102G}\right) \vee OriginalStress * 9.81$$

SCALING OPTION 3: **SCALE entry in the vFTGLOAD entry**:

a. Be careful as this is very different from FE_L_MAG and TSCALE.

b. This is a scale factor applied to the input PSD (G^2/Hz), not to the original FRF stresses as in the case for FE_L_MAG and TSCALE. For example, applying **SCALE=2** will multiply each point on the input PSD by 2 but will not change the original stress output from the solver (MPa) that was read from the OP2, ODB or RST file.

c. Multiplying the input PSD by 2 is actually multiplying each G^2/Hz point in the PSD by 2, which will create a different damage outcome than multiplying the original stress output (MPa) by 2, followed by squaring the MPa value to make the transfer function.

Format:

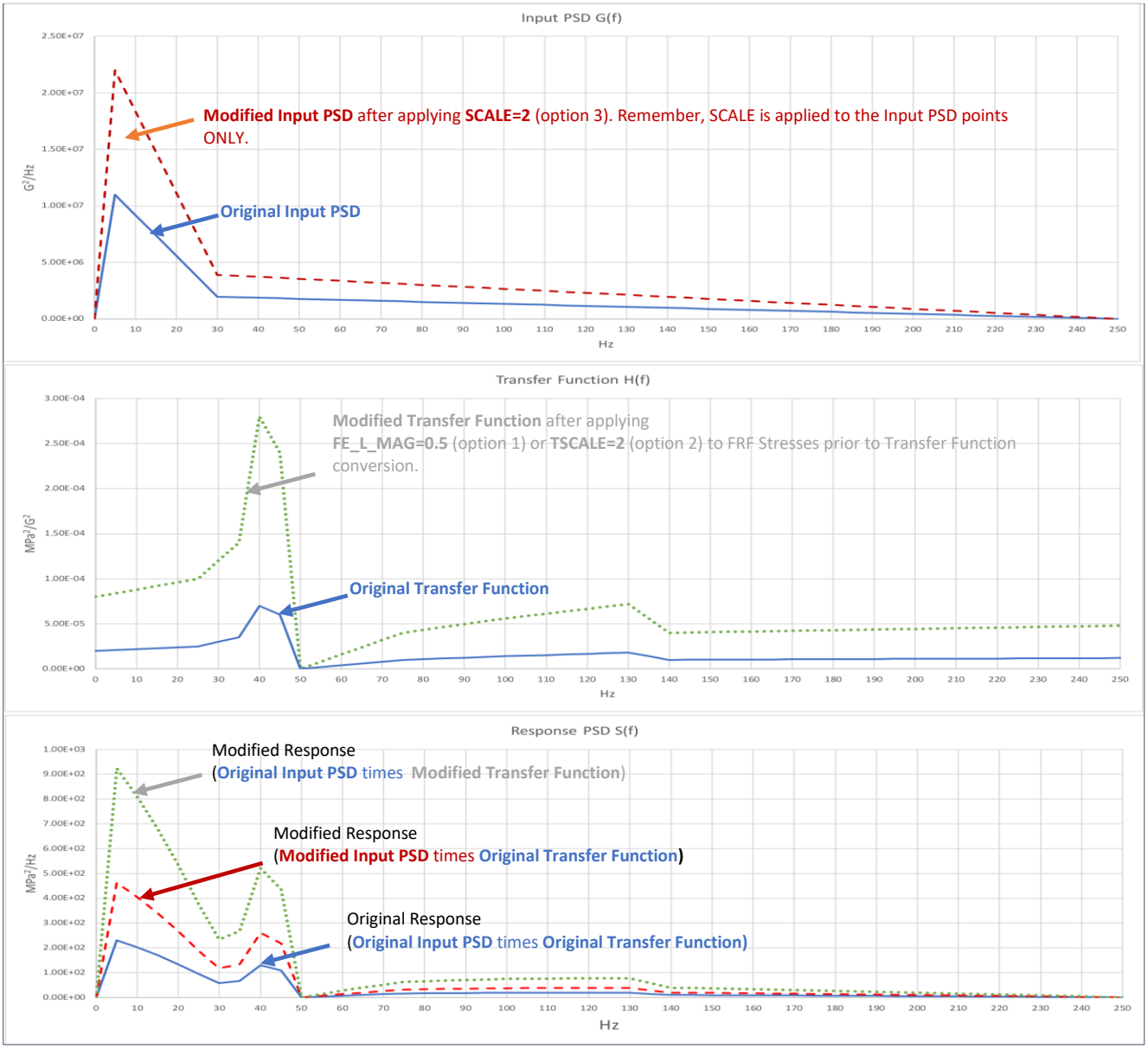
Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Column 7	Column 8	Column 9
vFTGLOAD	FLOAD	TYPE	SID	**SCALE**	OFFSET			
LCID	"Address of solver stress file" (only needed if not using mapping entry).							

Another Example: A unit Load was used to calculate original FRF stress, but the User is required to used different INPUT PSDs of scaled intensities.

In this example, the FRF calculation was done within the solver using a 1 G unit load, and the INPUT PSD being specified in Cf is also in units of G (gravity). In this case, the units between INPUT PSD and TRANSFER FUNCTION match up and will create the correct units for RESPONSE PSD. No FE_L_MAG or TSCALE is needed in this example. However, the User is given two INPUT PSDs to use in the analysis where one PSD is simply a scaled-up version of the other.

In this case, we could create two separate PSD definitions (using vTABRND entries) along with two vFTGLOAD entries or it may be easier to create one vTABRND entry for the first PSD definition and use SCALE=2 in the second vFTGLOAD entry to simply scale up the PSD described in the vTABRND that defines the first PSD. In this way, if the PSD shape changes for any reason, one change to the vTABRND entry will satisfy the changes for both Input PSD conditions in the example.

The image below is intended to show the differences between using **FE_L_MAG** (option 1), **TSCALE** (option 2), or **SCALE** (option 3). It is vital that the User clearly understands these scaling options and how it will affect their analysis. As can be seen, the OUTPUT RESPONSE varies significantly depending on the scaling used, which will significantly change the fatigue life calculation.



Local Scaling Option – SCALE (in FNOTCH)

SCALING OPTION 4: SCALE entry in the FNOTCH entry

a. SCALE in the FNOTCH entry is a **LOCAL** scaling method use to scale up or down, a section of the Transfer Function.

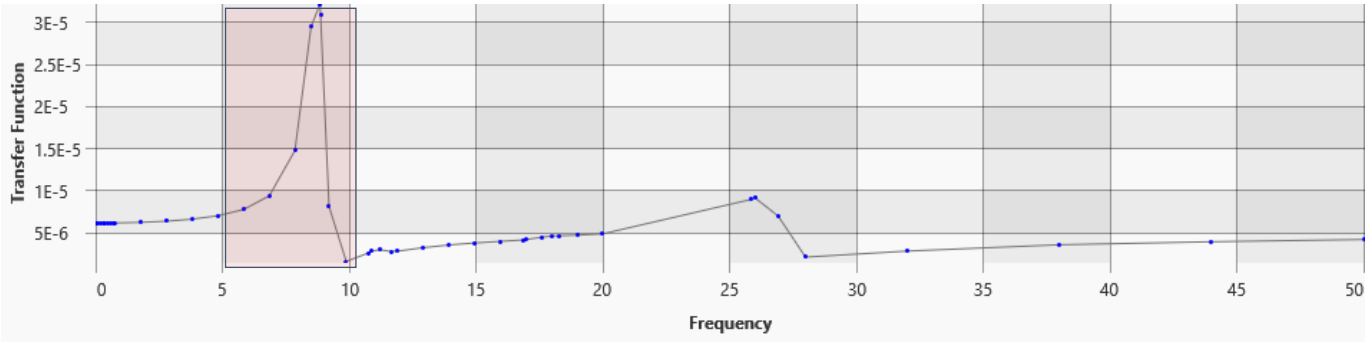
b. If the User provides a starting frequency values (fia) and an ending frequency value (fib), FNOTCH will apply a local scale value (SCALEi) to only that section of the Transfer Function.

Column 1	Column 2	Column 3	Column 4	Column 5	Column 6	Column 7	Column 8	Column 9
FNOTCH	EVID							
	$LCID_i$	f_ia	f_ib	**$SCALE_i$**				

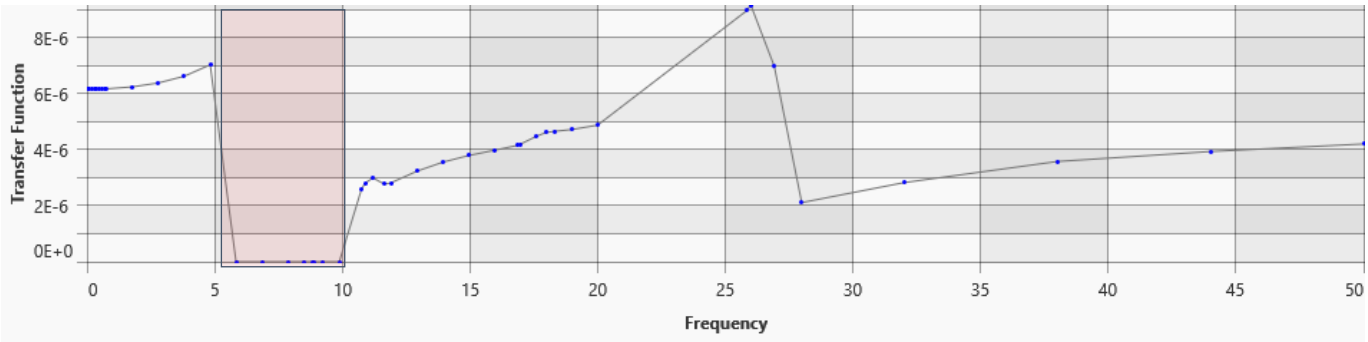
c. NOTE: unlike FE_L_MAG or TSCALE, this FNOTCH SCALE factor is applied directly to the Transfer Function values (MPa^2/G^2) and not just the solver stress values (MPa).

d. Applying a SCALE value of 0.0, can allow a User to remove the influence of a specific frequency range.

Below is the original Transfer Function for TPL109 (see User Guide example).

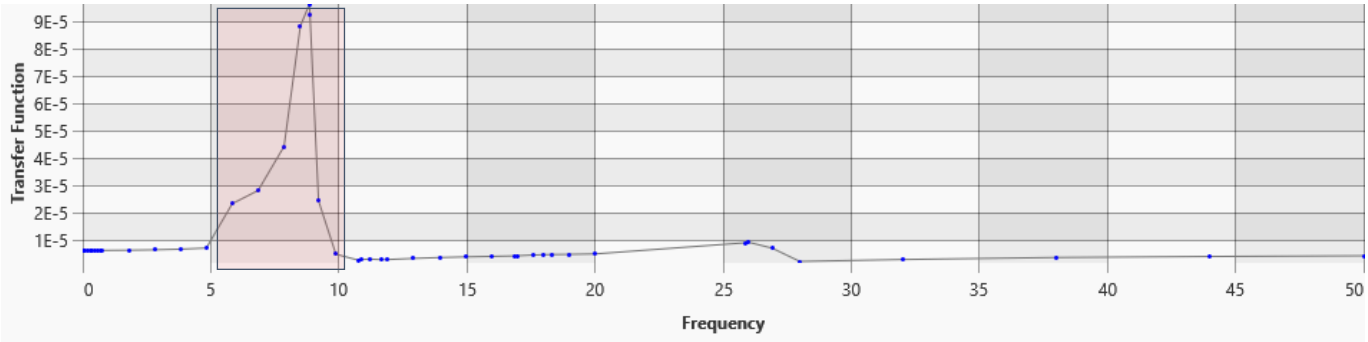


Below is the same Transfer Function with FNOTCH > SCALE=0.0 applied. This means the points in the Transfer Function between 5 Hz and 10 Hz are multiplied by 0.0, which effectively removed this section from the Transfer Function.



Note: FNOTCH scale factor is **only** applied to actual points in the Transfer Function. In the example above, the scale factor was applied to the first point above 5 Hz since no point existed directly at 5 Hz. This function does not interpolate between values to scale at exactly 5 Hz.

Below is the original Transfer Function with FNOTCH > SCALE=3.0 applied. This means the points in the Transfer Function between 5 Hz and 10 Hz are multiplied by 3.0, which effectively scales up this local section of the Transfer Function.



Why Use FNOTCH?

FNOTCH is a very quick and powerful way for the User to determine the influence on stress, damage, etc. for a very specific frequency range in the analysis. FNOTCH coupled with RESTART, can allow the User to perform sensitive studies to determine the best approach to resolve a frequency based issue with the model.

Post Processing Options

Throughout this document H3D results are post processed with HyperView and FEF results are post processed with Patran; however, any other compatible 3rd party post processor can be used. Results can also be post processed using the included graphical interface supplied within CAEfatigue.

6 | INTRODUCTION TO THE FATIGUE CONTROL FILE VIEW

NOTE: The images shown in this section reflect CAEfatigue v2020 and may change slightly with future releases.

CAEfatigue operates by executing a series of commands laid out in a “control file”. The actual control file can be defined within CAEfatigue via the Control File view. Alternatively, CAEfatigue allows the User to define the analysis via a graphical Process Flow view. This view will then create the control file when the User is ready to execute the job.

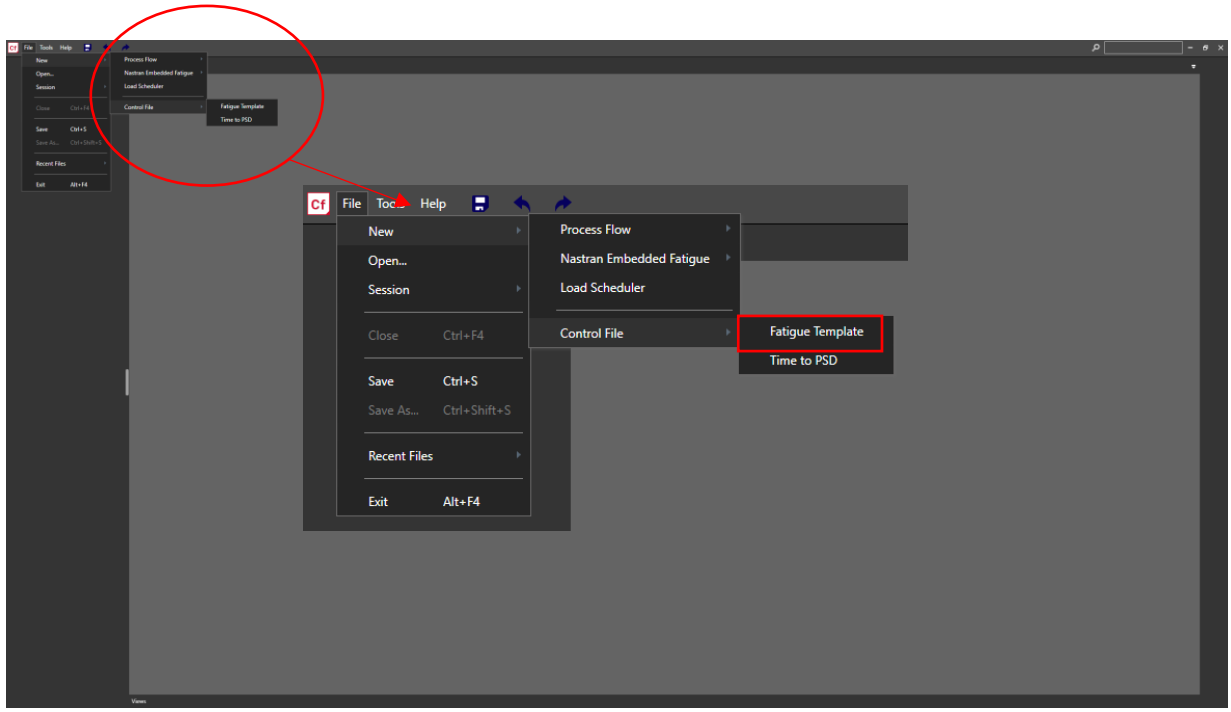
This chapter will explain the **Control File view** using an example for reference.

CF includes a graphical user interface application for executing the program. For the interface to run successfully, ".NET 4.5" (or later – see latest installation guide) must be installed on the system (this should normally be the case).

When the interface is opened for the first time the User should load an existing TPL from the test Problem Library (e.g. TPL1.txt) in order to initiate the functionality shown below.

Overview and Introduction to the Fatigue Control File Wizard

To start CF, click on the desktop Cf icon (see below) or find the ProcessApp.exe file for the program in the default installation directory "C:\Program Files\MSC.Software\CAEfatigue\<ver>\Windows\".



This will open Cf in the interface application window as shown above. To highlight the different features of the fatigue control file wizard, we will load and run a job. To do this we can either create a NEW Control File as shown above to generate the initial base structure of our Control File; or we can select ""FILE>OPEN" to select an existing Control File that was already created, which can then be edited to match the analysis we are trying to accomplish.

In the image below, we selected FILE> OPEN > TPL128.txt. This will open the test problem in the editor window.

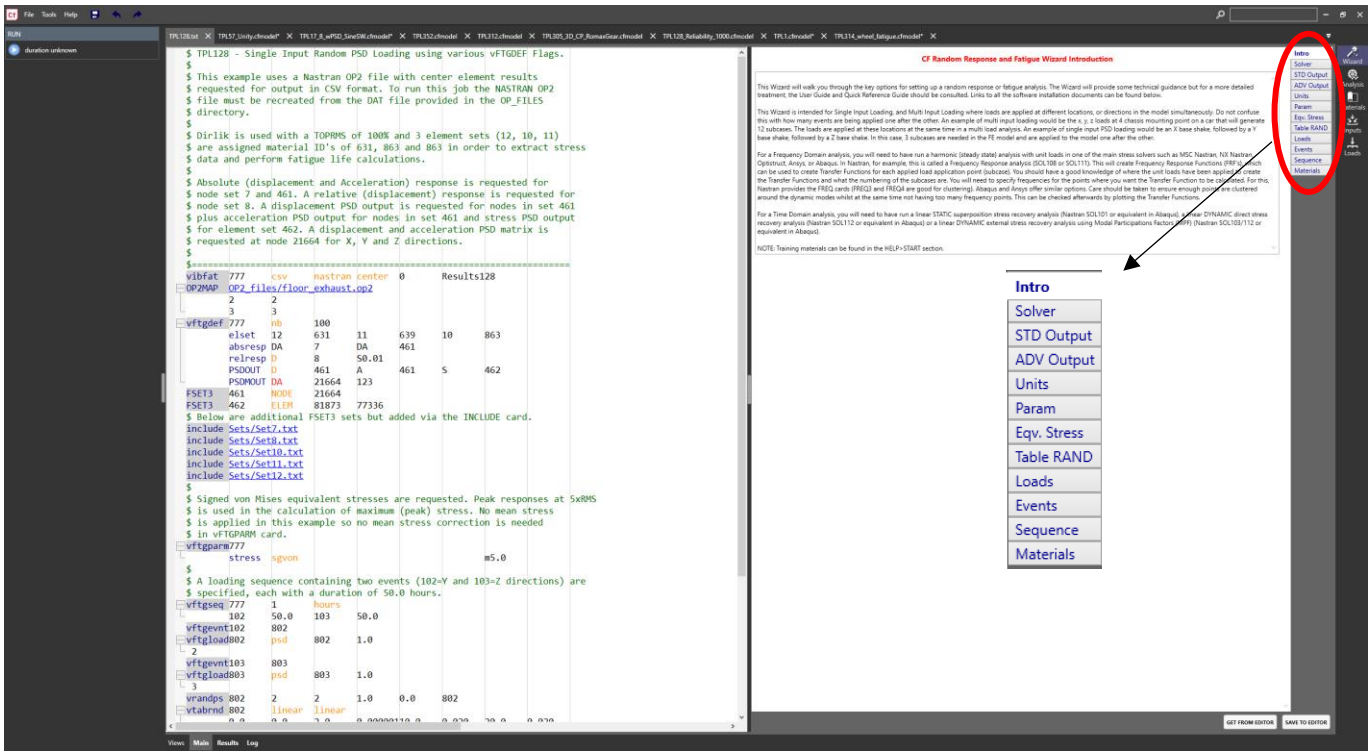
Please Note: The floor_exhaust.op2 file referenced in TPL128 Control File is not included in the TPL library due to the size of the OP2 file. The MSC Nastran file (floor_exhaust.dat) must be run to recreate the floor_exhaust.op2 file if you wish to duplicate this example on your own.

Wizard Toolbox

Select the **Wizard** icon to open the Wizard toolbox as shown on the right-hand side of the window below. We will use the Wizard toolbox to examine the parameters chosen for TPL128 as well as explain many of these parameters and their interaction.

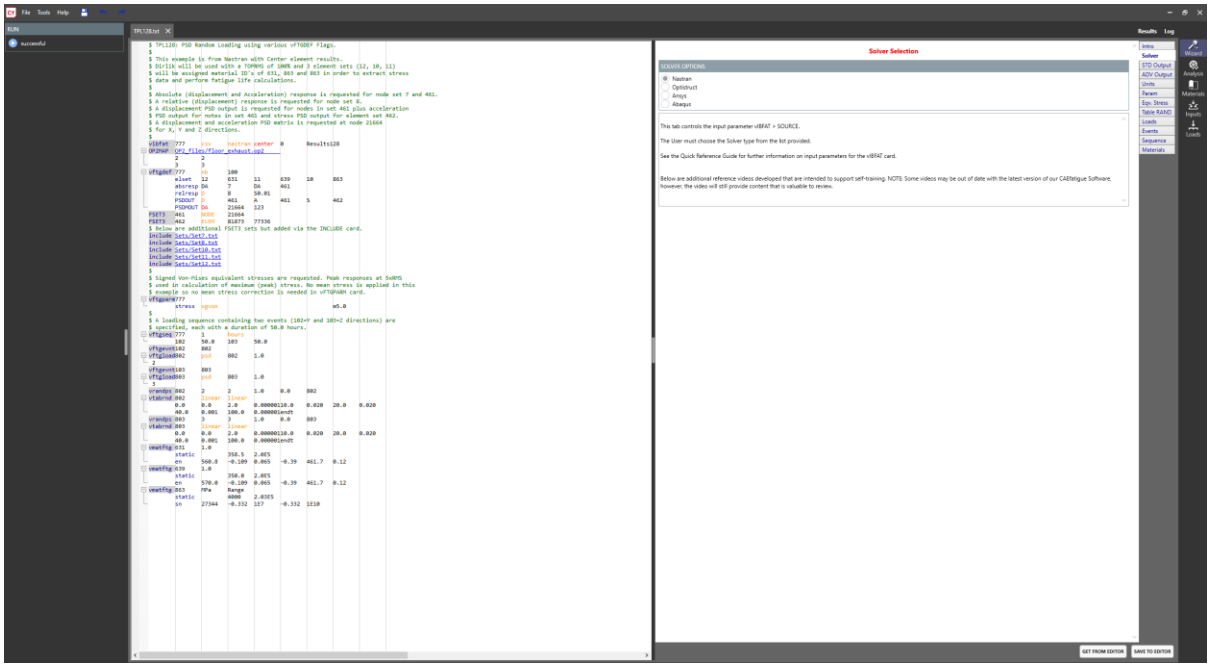
NOTE: If the User wishes to bypass the Wizard, they can select the menu items below the Wizard to get access to ENTRY / FIELD combinations for direct drag and drop insertion into the Control File editor window (left). For an experienced user, this can be faster than typing in the entries and fields manually. Also, when available, the User should consider starting with an existing Test Problem as a template that closely matches the type of analysis the User wishes to do. Often existing jobs created by the User become the templates for future work.

The Wizard contains tabs that are used to specify the key input parameters for executing a new job or changing an existing job. The Wizard also provides some information on the key parameters to help with the understanding.
Note: The toolbox makes use of horizontal and vertical scroll bars for some tabs.

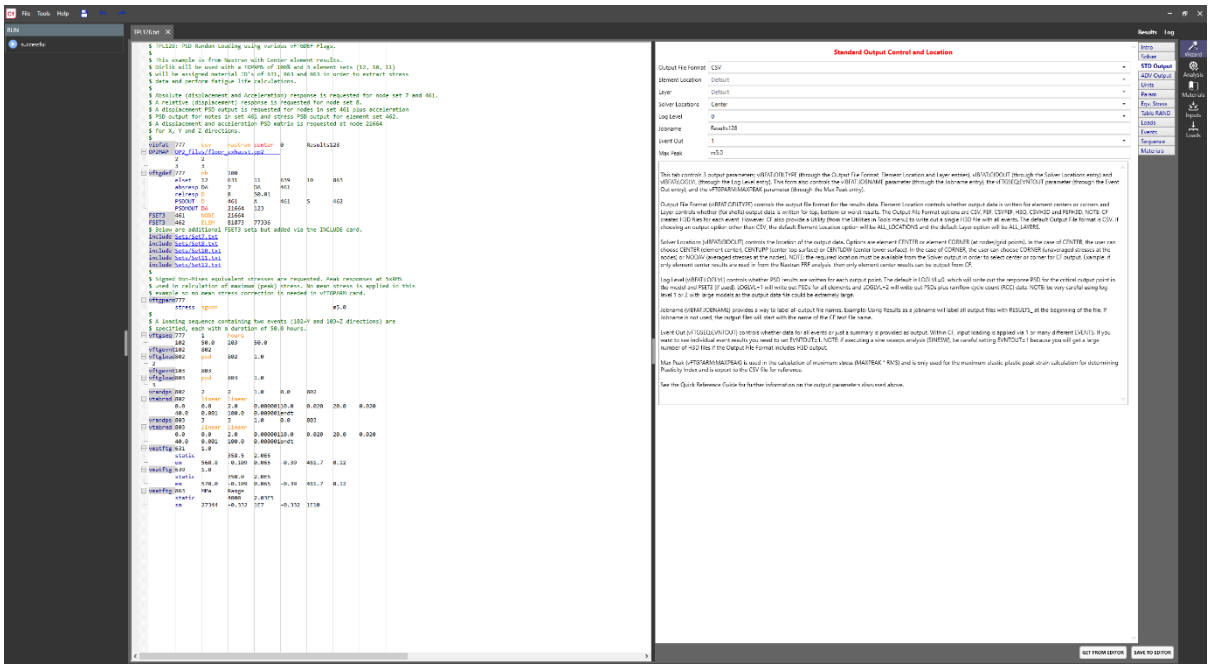


The **INTRO 0** tab (as shown above) has no parameters to specify but there are links to documentation including the User Guide, Quick Reference Guide and the Latest Release Notes.

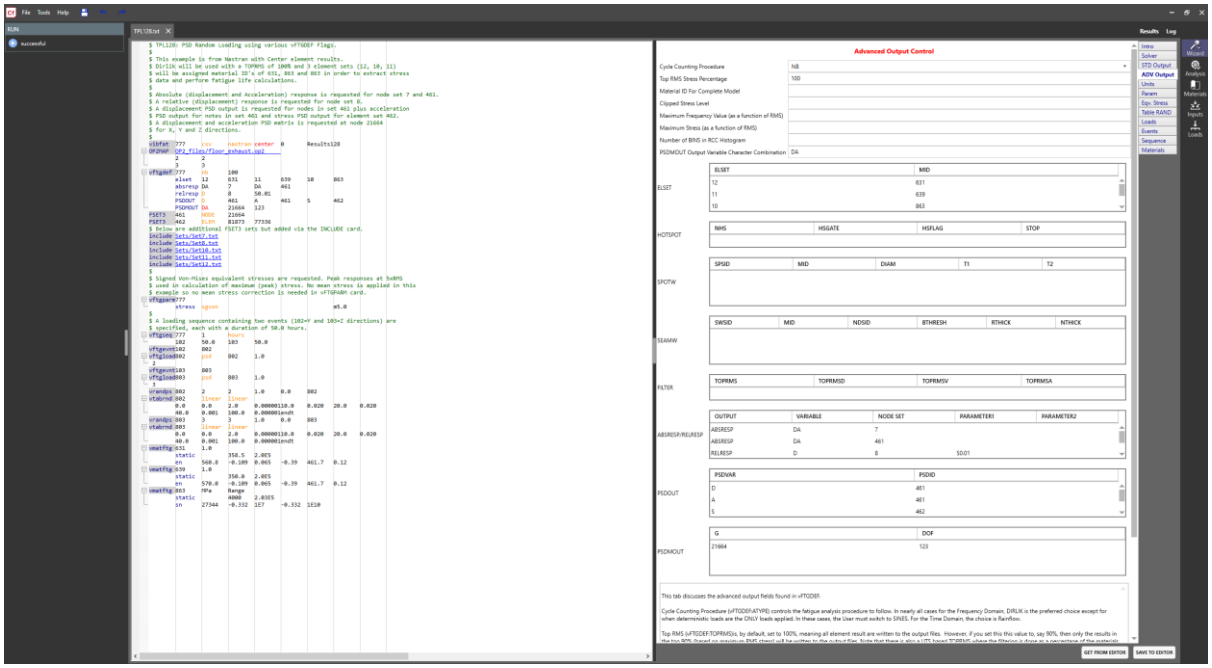
The **SOLVER** tab specifies the solver used in the analysis and the stress output location (GIDOUT). The most common options include CORNER (un-averaged corner node results), NODAV (averaged corner nodal results from surrounding elements) and CENTER (element center results). This tab also provides links to a collection of online training which covers the disciplines of fatigue, dynamics, FEA and random vibration analysis.



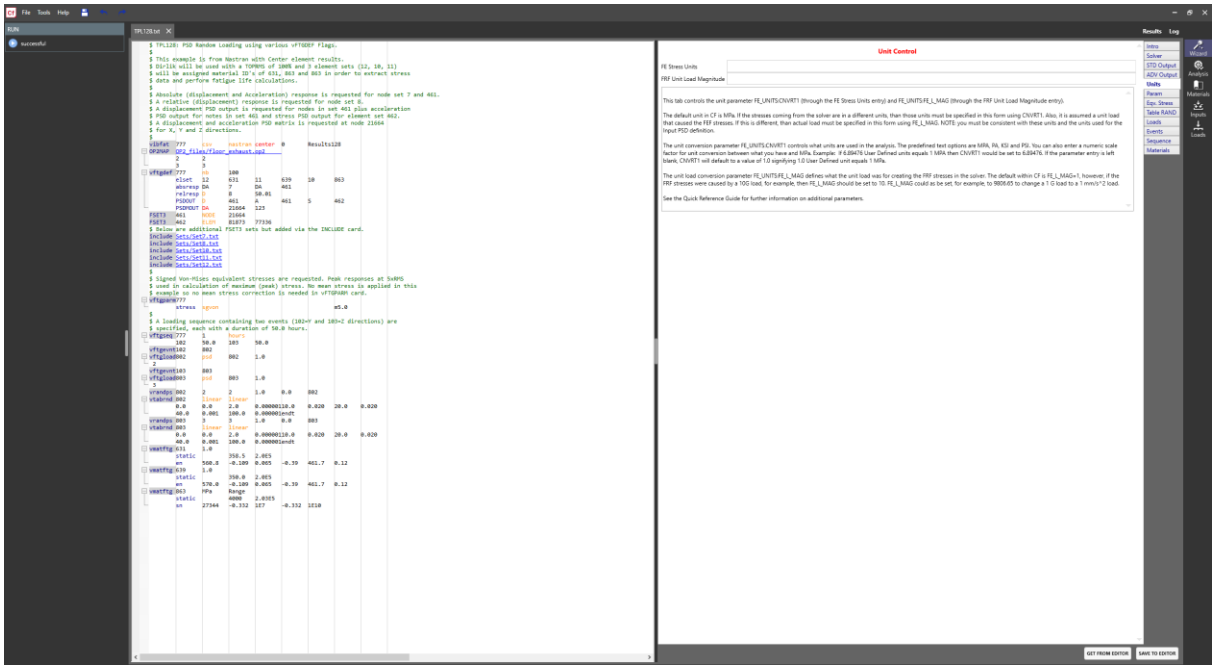
The **STD OUTPUT** tab controls much of the program output. This includes what kind of output format is needed (e.g. CSV, FEF and/or H3D) and what locations are to be used for the writing of output. This includes the position on the element, the layer to select (for shells) and whether results are required for individual elements (EVNTOUT=1) or just the summed results (EVNTOUT=0). The output control also provides for a MAXPEAK values that will be used for elastic plastic stress / strain output.



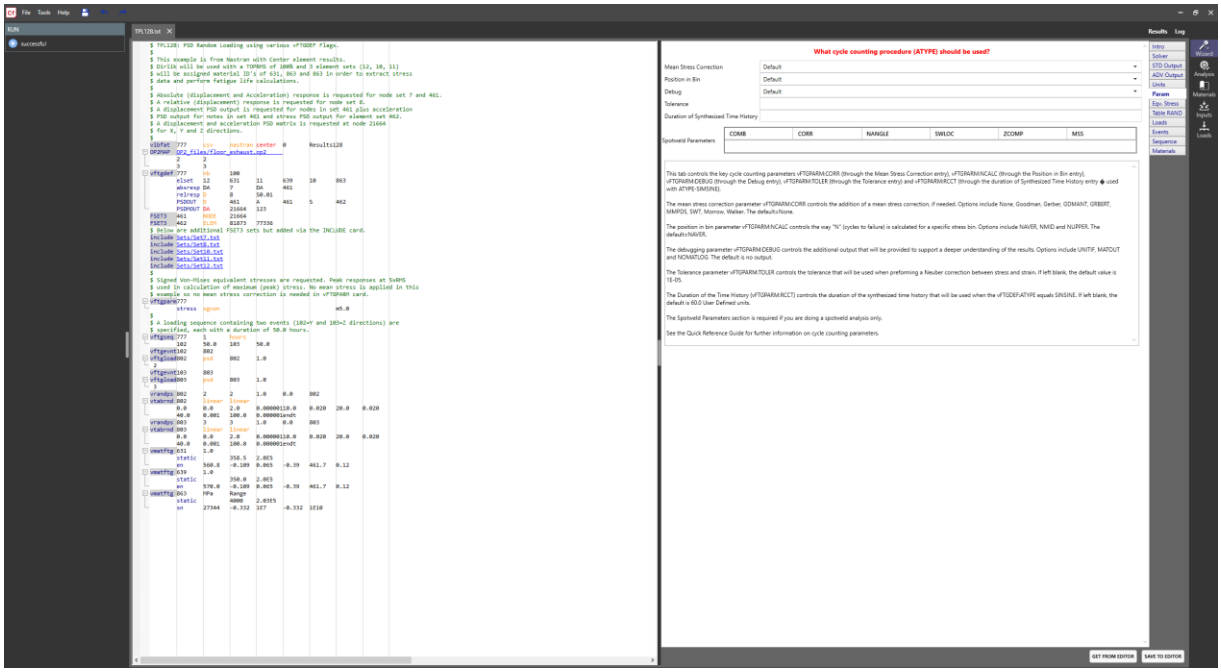
The **ADV OUTPUT** tab provides for additional output that is mostly found in the vFTGDEF entry. This screen provides entries for several cycle counting parameters along with entries for ELSETS, HOTSPOTS, SPOTW, SEAMW, various filters, collision detection parameters ABSRESP / RELRESP, PSDOUT and PSDMOUT.



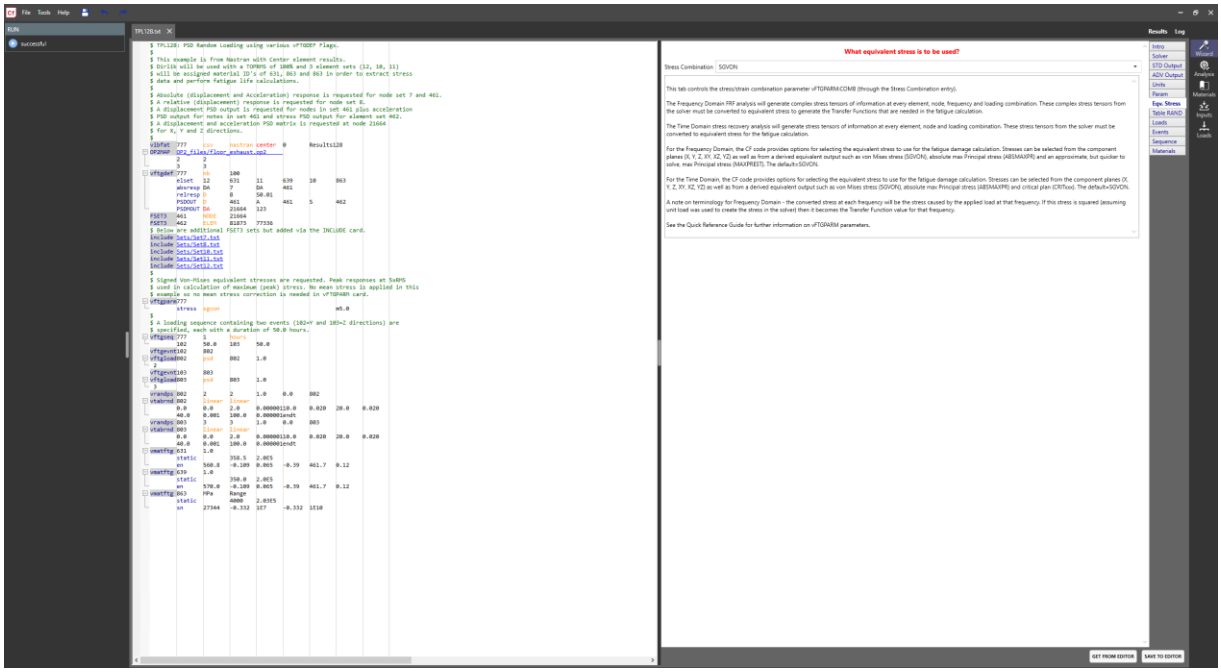
The **UNITS** tab establishes the units for incoming stresses. If MPa units are used in the model, then this tab is not needed because the default unit for stress in Cf is MPA. If the units for the model are not MPA then the User must enter the proper conversion to bring the units to MPA. Cf also assumes a load of UNIT load is used to create the transfer functions. If this is not the case, the User must specify the load magnitude used (example 10g instead of 1G).



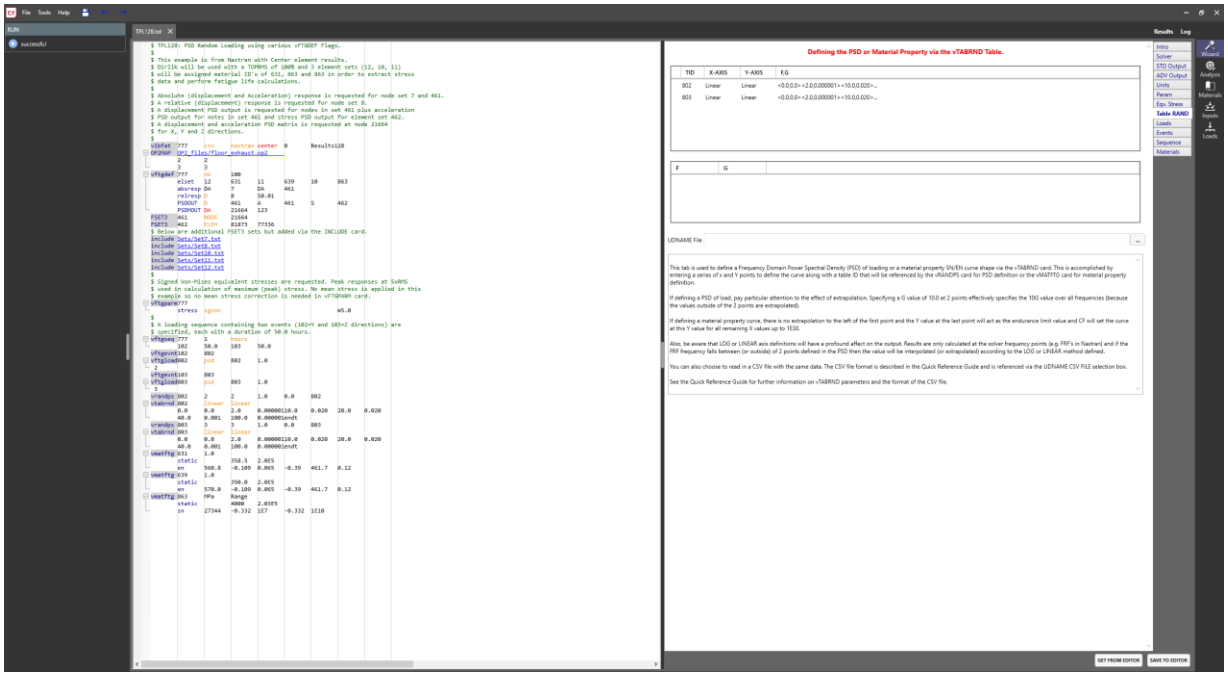
The **PARAM** tab controls many of the parameters associates with the fatigue calculation including mean stress corrections (needed when a static load is applied), bin position for calculating cycles to failure and debug options.



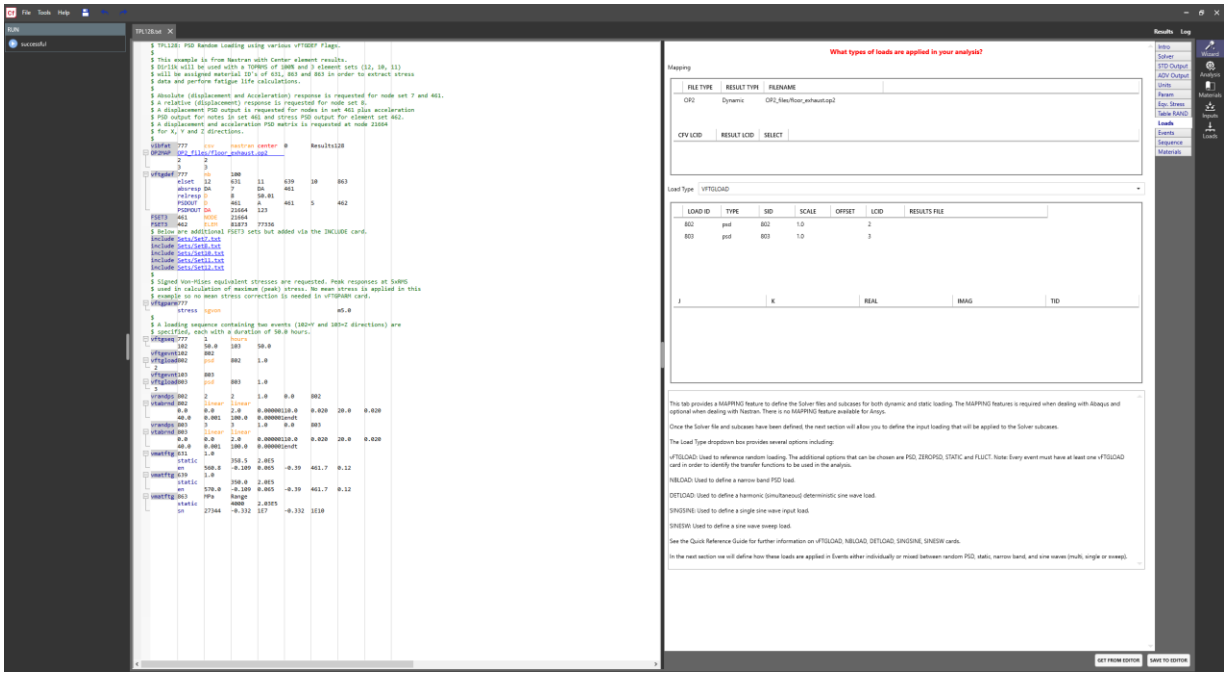
The **EQV.STRESS** tab controls which equivalent stress should be used (vFTGPARM: COMB entry).



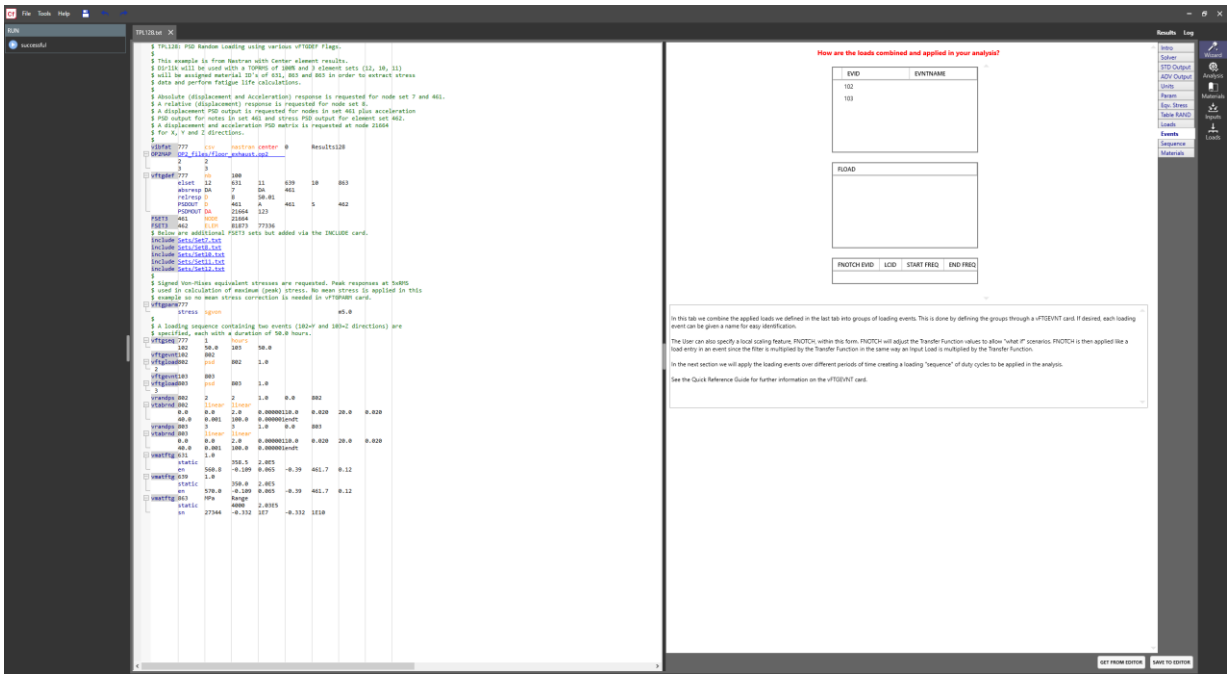
The **Table RAND** tab is used to make tables for material specification (type TABLE) or for input loading as defined in the next tab (i.e. loads tab).



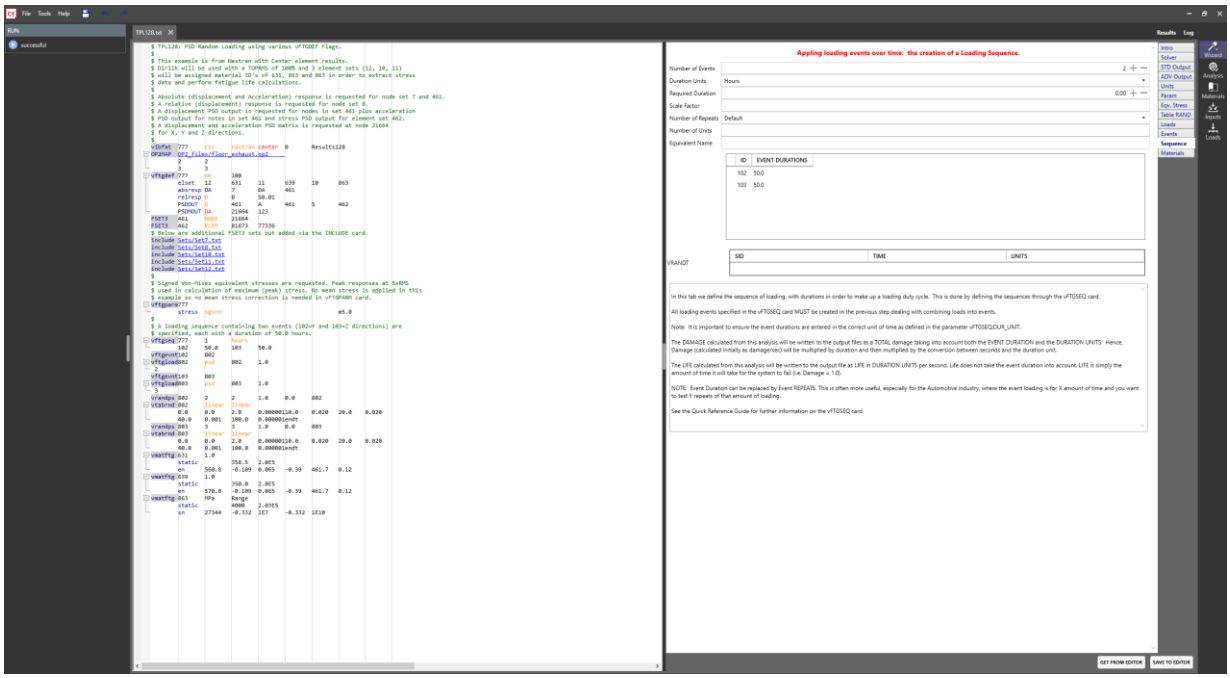
The **LOADS** tab is where the table (f, g) of loading (defined in the Table RAND tab) is connected with the stress results file and the appropriate load case ID.



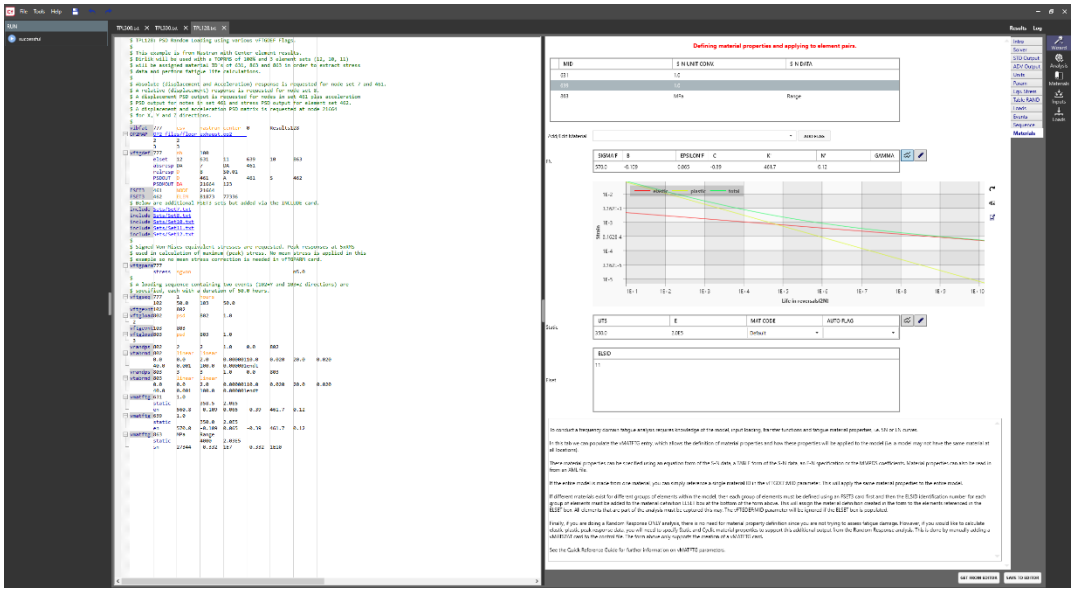
The **EVENTS** tab is where the Events are defined (i.e. which loads are added together in each Event).



The **SEQUENCE** tab is where the Events are added together to make a sequence (duty cycle). Here the durations for each event are defined. There are a number of parameters that can be added including event duration or number of repeats, duration units, scale factor, etc.



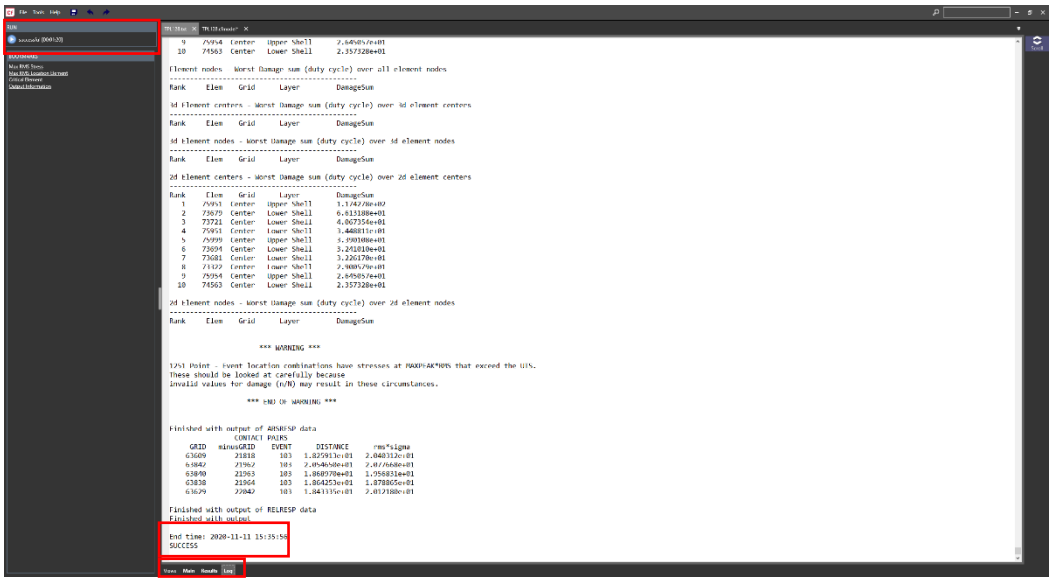
The **MATERIALS** tab is where the materials are defined. Multiple materials of different types can be defined here. When applicable, the lower part of the form shows which element groups (ELSETS) are assigned to each material. The User can also plot the material curves.



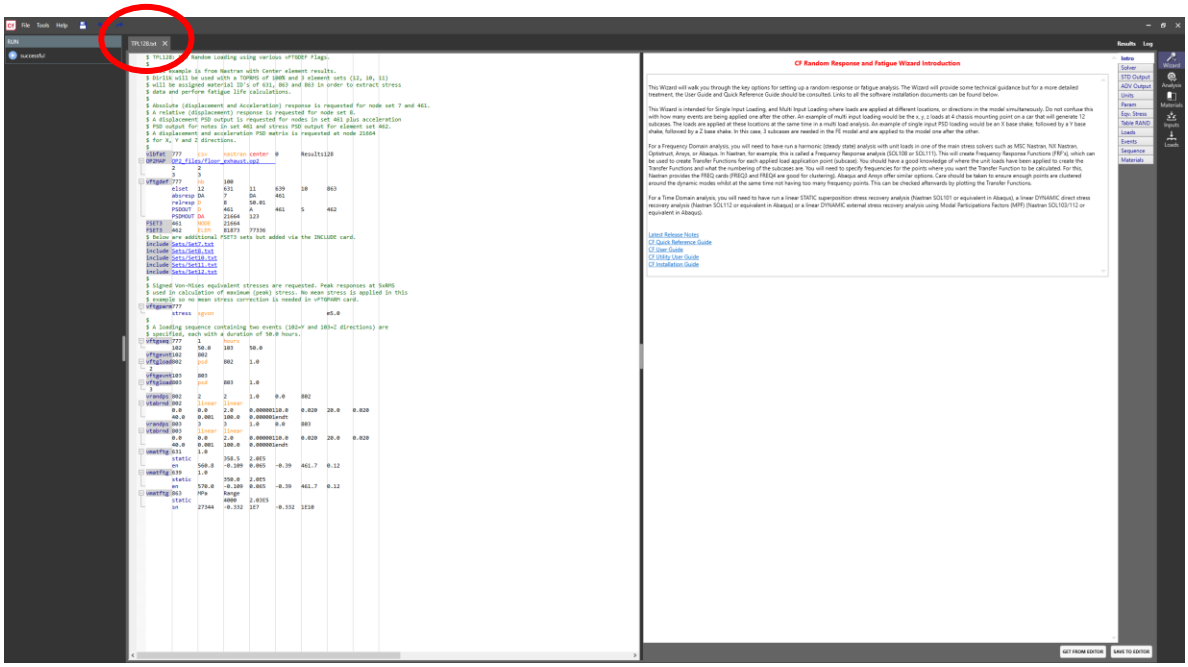
Job Execution – Run, Edit View, Log View and Console View

Once a job is defined, it must be saved (i.e., transferred from the Wizard toolbox (right side) to the editor / Control File (left side)) before pressing the **run** button to execute the analysis. Any new data entered into the Wizard form will be lost if not saved to the editor.

After selecting RUN, the **LOG** screen will display and is used to track the progress of the run. Once completed, the word “SUCCESS” will be displayed at the bottom of the LOG file. The LOG file is also written to the default directory.

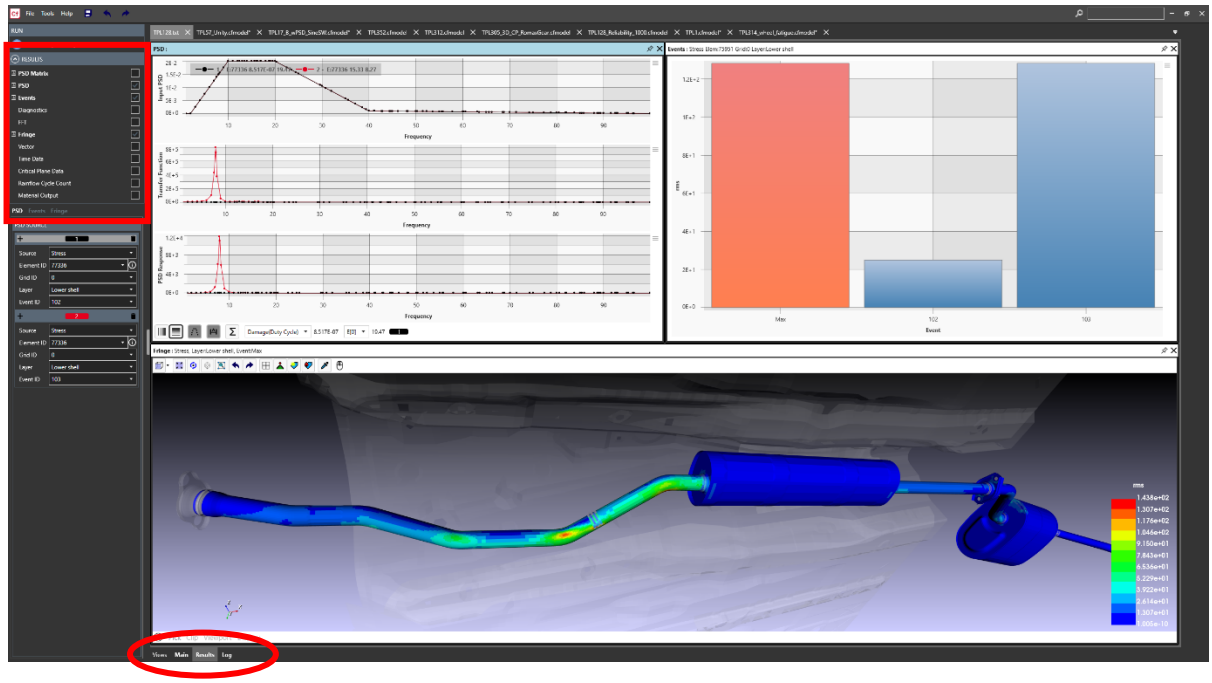


Selecting the Control File tab will return you to the EDIT screen where, if desired, additional edits can be made quickly to the control file itself (left side of the screen) and the job rerun.



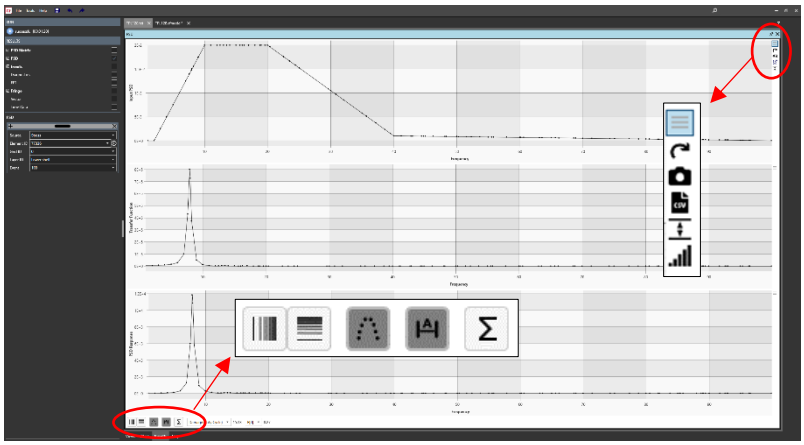
Viewing Results – Frequency Domain Analysis Outputs

Selecting the RESULTS tab will allow the User to post process the results from a successful analysis. Shown on the left of the GUI are a number of result output options available for this example. Depending on the features selected, several of the output plotter will provide results but some may not. For example, the PSD Matrix plotter will only have output results if “loads cascading” is requested in the analysis. Below is an example of some of the results for TPL128.txt. These plotters can be “unpinned” from the GUI and displayed as separate images.



PSD PLOTTER

Checking the **PSD** box will open this screen that will display PSD results for various locations. For this example, the LOGLVL was set to zero (0), which means only the critical stress location (i.e. element with the highest stress output) is available to view in the PSD viewer (i.e. element 77336). Note: the PSD Plotter gives the User several additional options for viewing the data.



To the right of each graph, the User is provided with 4 options.

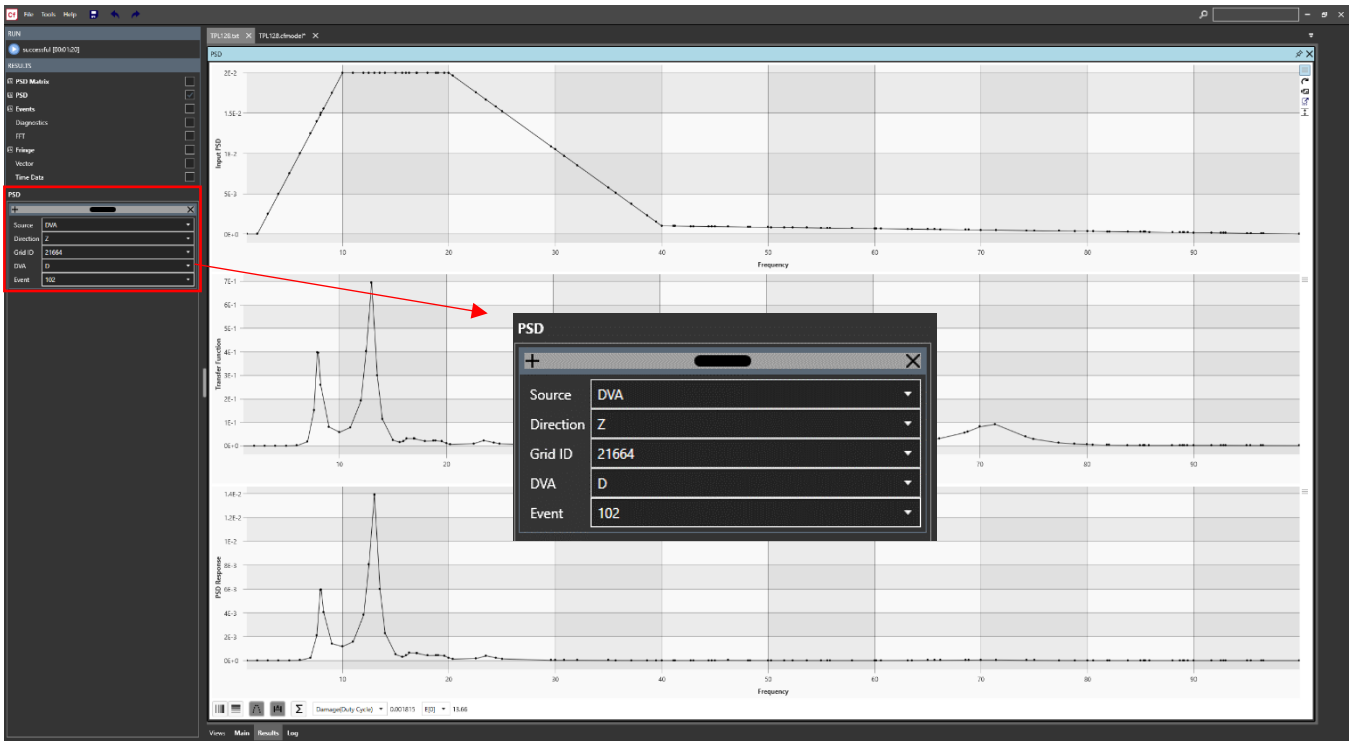
(1) Reset the original view of the graph. Useful after modifying the view with Auto Y-scale.
(2) Export the graph to a PNG file. Useful for reports.
(3) Export the graph data points to a CSV file. Useful for data manipulation in a spreadsheet.
(4) Auto Y-scale which will zoom in on the Y axis such that the top of the smallest Event bar becomes barely visible.
(5) Adjust the quality of the screen plot.

At the bottom left, the User is also provided the options to:

(1) Change the X or Y scale to LOG scale.
(2) Turn the marker dots on the plot on or off.
(3) Zoom into a section of the X scale.
(4) Show cumulative RMS response PSD plot (for Stress only).

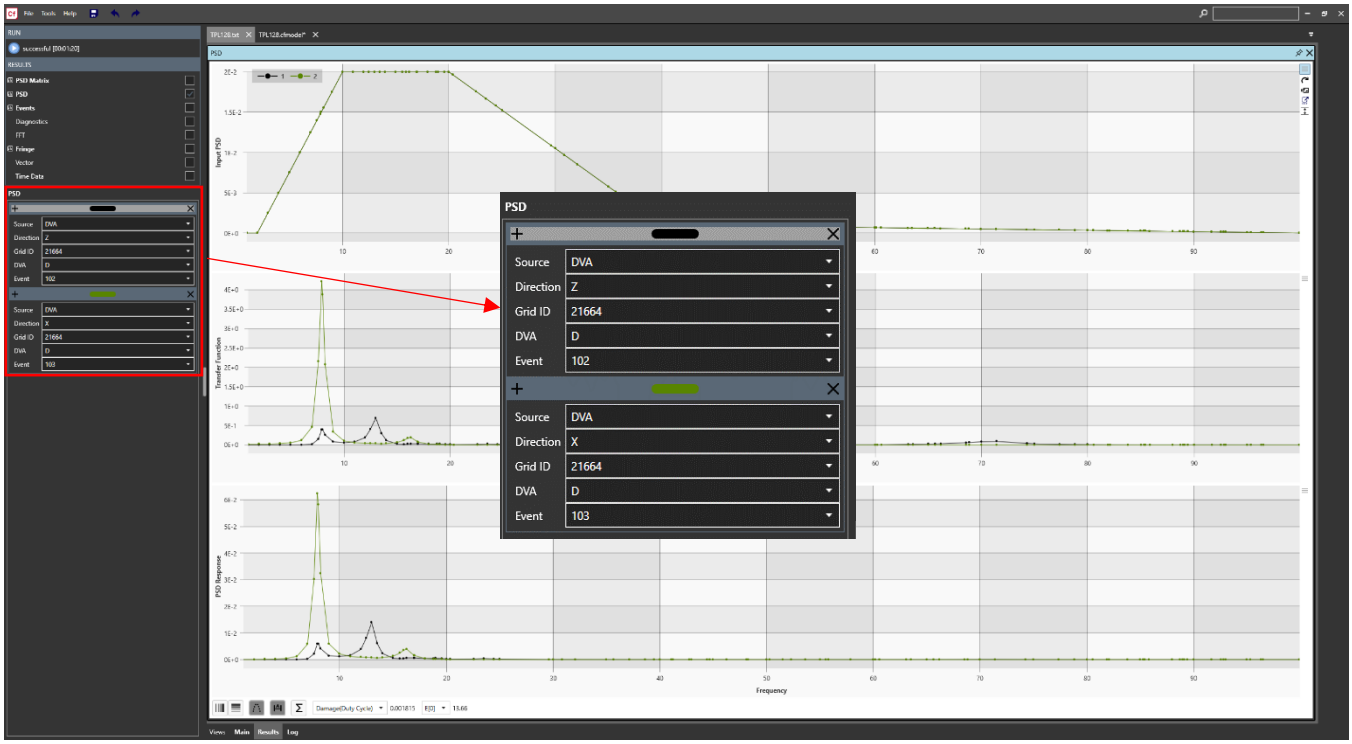
The options for “Type”, can be Stress, DVA (displacement, velocity, and acceleration) or Relative (displacement) assuming random and relative results were requested with ABSRESP and RELRESP in the Control File.

Below is an example of Z displacement (D) at grid ID 21863 caused by Event 102.



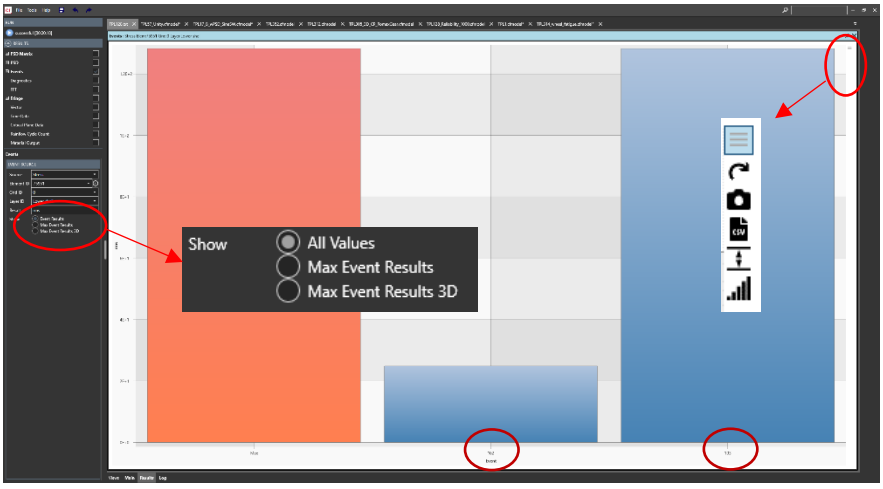
If relative response data is available (RELRESP), then “Type” can be changed to Relative to see relative displacement results between nodes.

The PSD Plotter also allows for multiple PSDs to be plotted on the same image. Below is the Z direction displacement for both Events 102 and 103 plotted on the same image.



EVENT PLOTTER

Clicking on the **Event Plotter** icon will bring up the window shown below. For this example, we see the MAX RMS stress result and individual Event results at Element 75951. Note: The Event Plotter provides the User with several additional options for viewing the Event data.



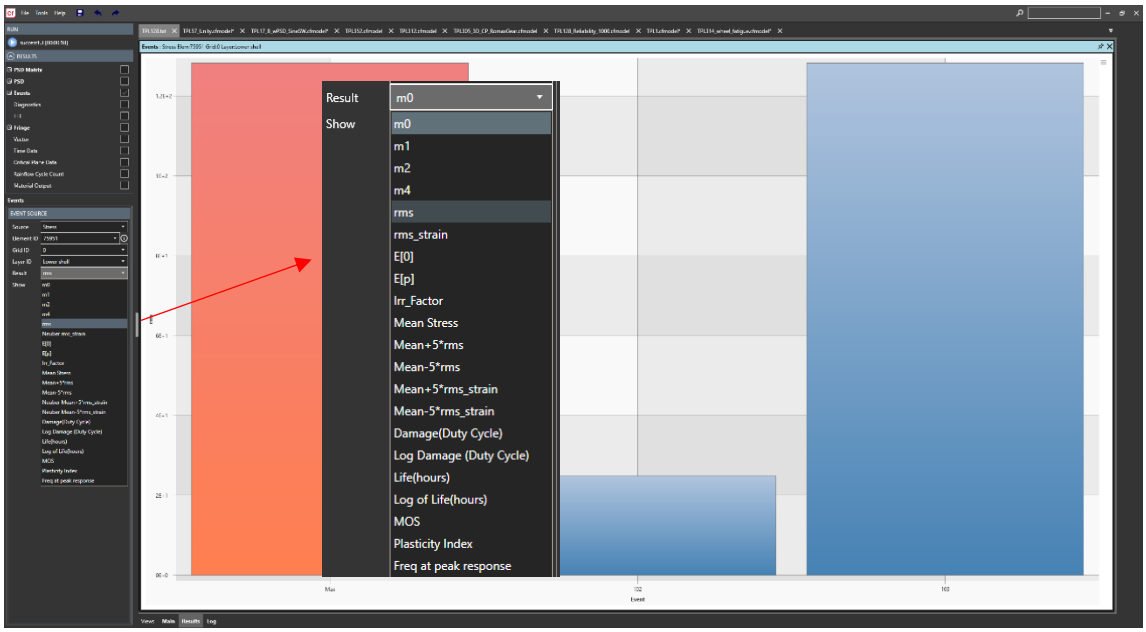
To the right of the graph, the User is provided with 5 options.

(1) Reset the original view of the graph. Useful after modifying the view with Auto Y-scale.
(2) Export the graph to a PNG file. Useful for reports.
(3) Export the graph data points to a CSV file. Useful for data manipulation in a spreadsheet.
(4) Auto Y-scale which will zoom in on the Y axis such that the top of the smallest Event bar becomes barely visible.
(5) Adjust the quality of the screen plot.

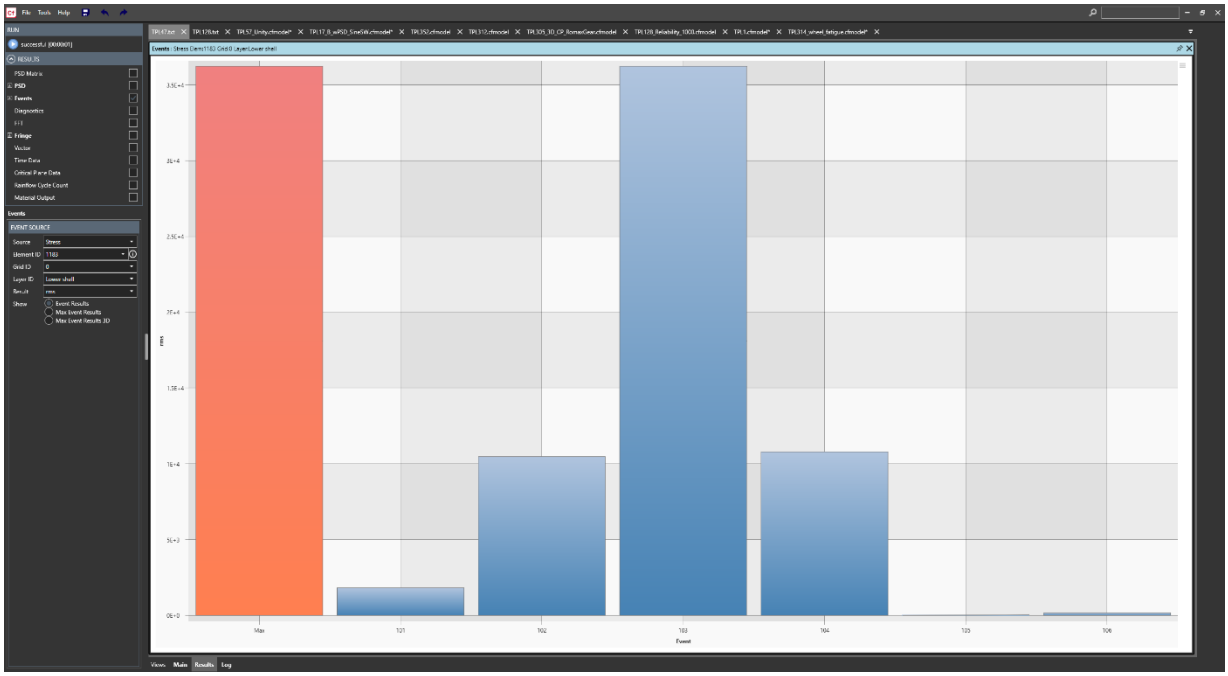
At the left of the graph, the User is provided with 3 options.

(1) View all results (default).
(2) View only MAX event results
(3) View only MAX results on 3D plot.

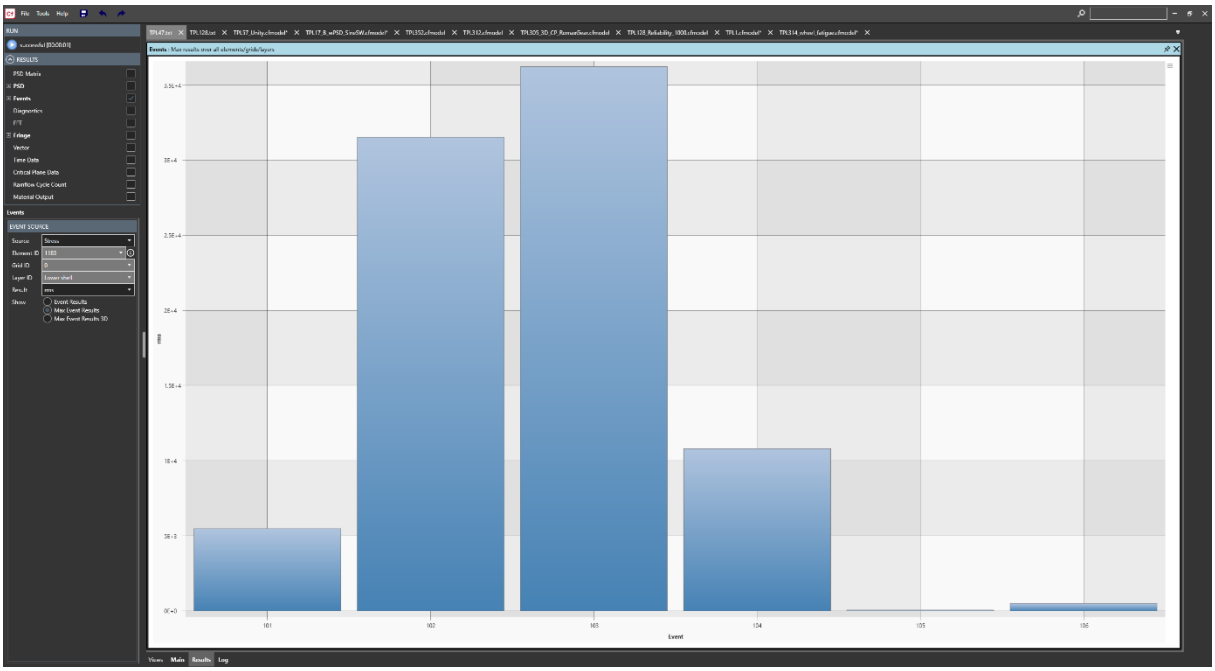
Note that Result type can be changed to several different options.



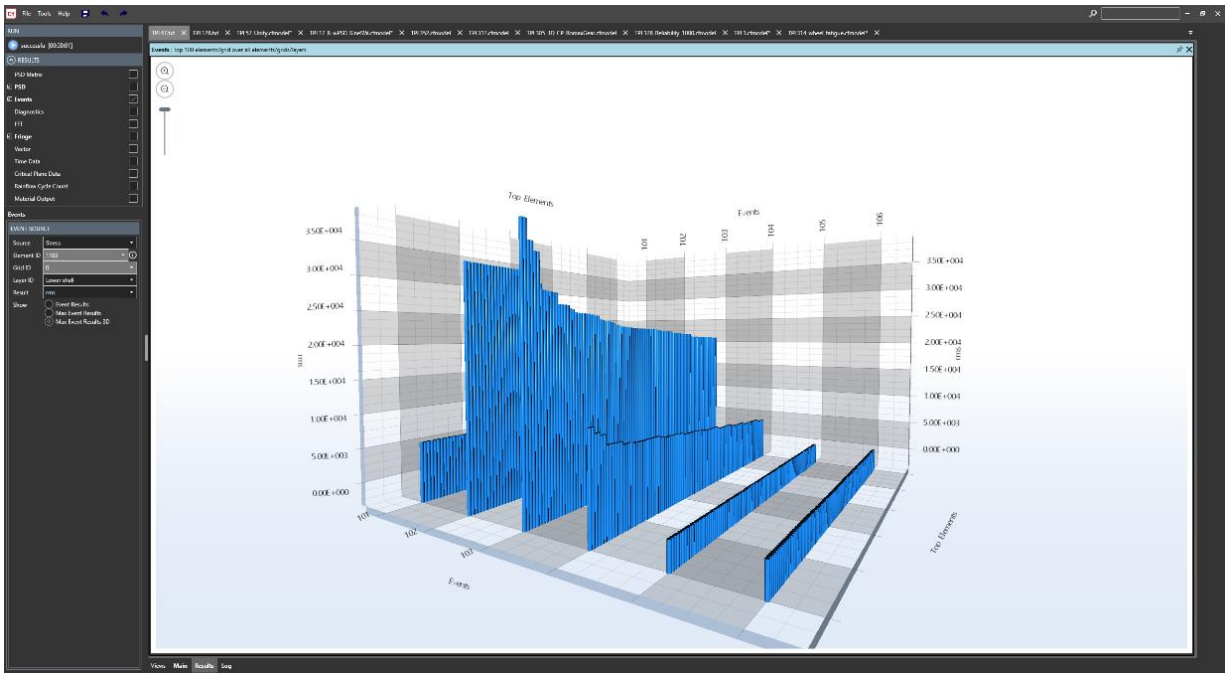
To illustrate the use of the left 3 options, let us look at TPL47 with vFTGSEQ: EVNTOUT=1. The default Event Plotter shows the worst output at a particular Element ID / Grid ID / Layer ID and Result type combination.



However, it is also sometimes useful to know the worst (or max) event value regardless of the Element ID / Grid ED / Layer ID combination. To show this result, the User can select the **MAX EVENT RESULTS** radio button, which will provide the worst (or max) result at every Event regardless of what combination of Element / Grid and Layer that result comes from. In the example below, it can be seen that Event 102 has a significant contribution to the rms stress that, perhaps, may not have been fully appreciated by looking at the default Event Plotter results above. NOTE: When using this option, several output rows are not available and are greyed out.

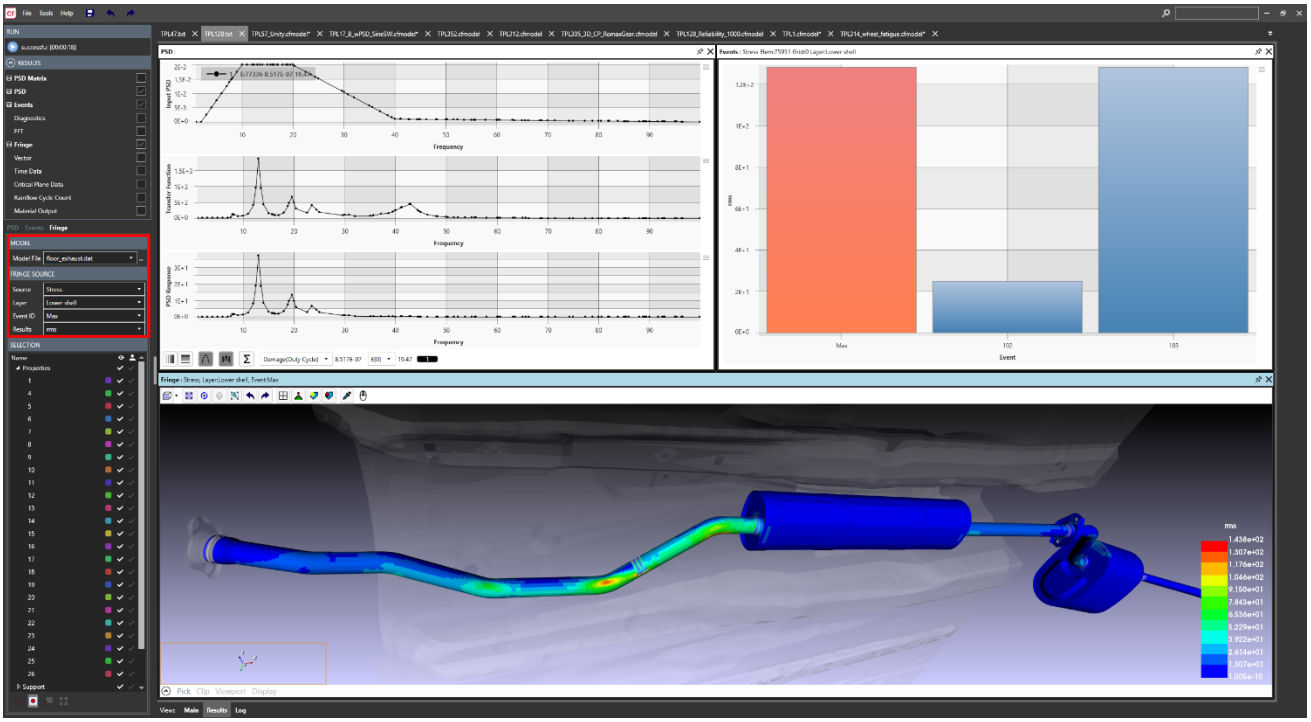


Finally, it may also be useful to know the MAX Event Results across all elements to see how the results are distributed. By selecting **MAX EVENT RESULTS 3D**, the User can see the Event results across all elements in that event. From the example below, it is clear to see that the MAX results occur in Event 103, but those MAX results only occur for a limited number of elements. Knowing the elements where the max results occur can help the User identify possible changes to the model in this region that could lower the max stress values. Note: this type of information can also be seen using the Fringe Plotter with the CAE model. NOTE: When using this option, several output rows are not available and are greyed out.



FRINGE VIEWER

Clicking the **Fringe View** box will begin the process of loading results of the analysis into memory. To see the results, you must also select a model (mesh) file, on which the results will be displayed (in this case the mesh file is floor_exhaust.dat). Displaying a Fringe Plot can be done alongside other Results options as shown below.

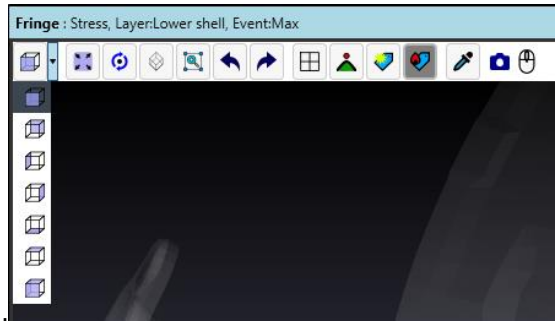


After selecting the Fringe View, you will have the choice of results to view superimposed on the model. Below is the RMS plot for the lower surface of the shell elements in the MSC Nastran model. Note:

For Damage or Life plots, **Event ID = SUMMED** means the summed results from all events.

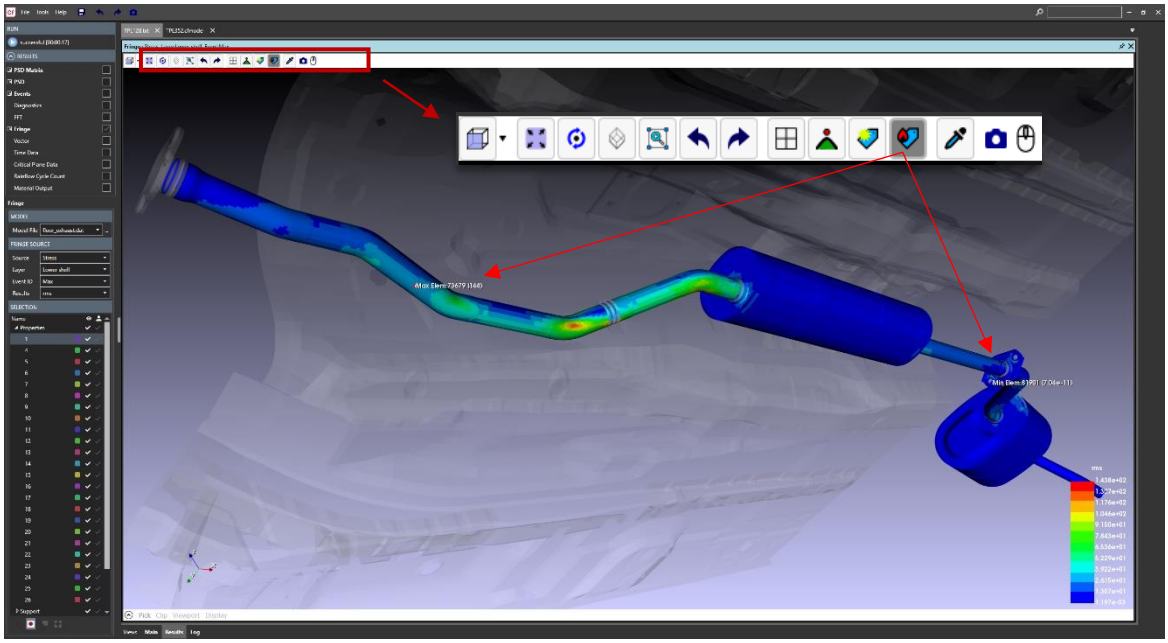
If all other outputs, **Event ID = MAX** means the max element / nodal output across all Events. Hence, the MAX plot may be a mixture of results from all Events in the analysis.

There are also a number of preset views / options available in the top bar of the plotter. From left to right, the icons are:

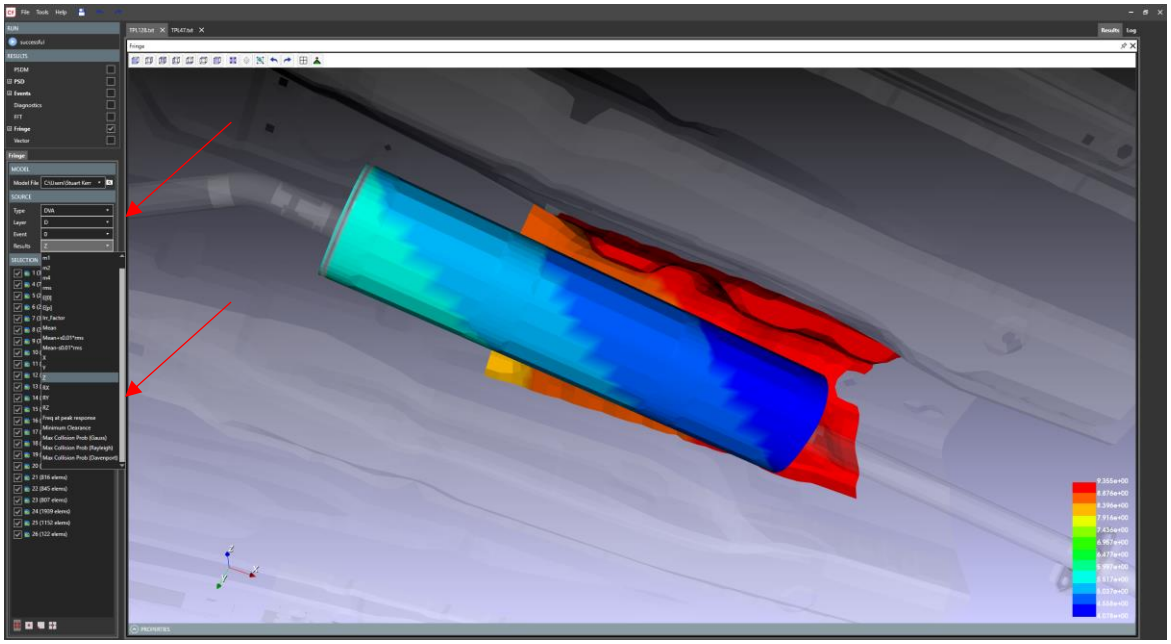


- A dropdown with 7 preset views
- Fit all content into the view
- Set a new rotational center for viewing
- Change to parallel projection
- Zoom using “rubber band” approach
- Undo and redo camera
- Show / hide element edges
- Show / hide hotspots (after using hotspot filter)
- Show / hide all marker labels
- Show / hide min / max results values
- Probe element for nodal information
- Take a screen shot of entire Fringe plot workspace
- How to use mouse controls

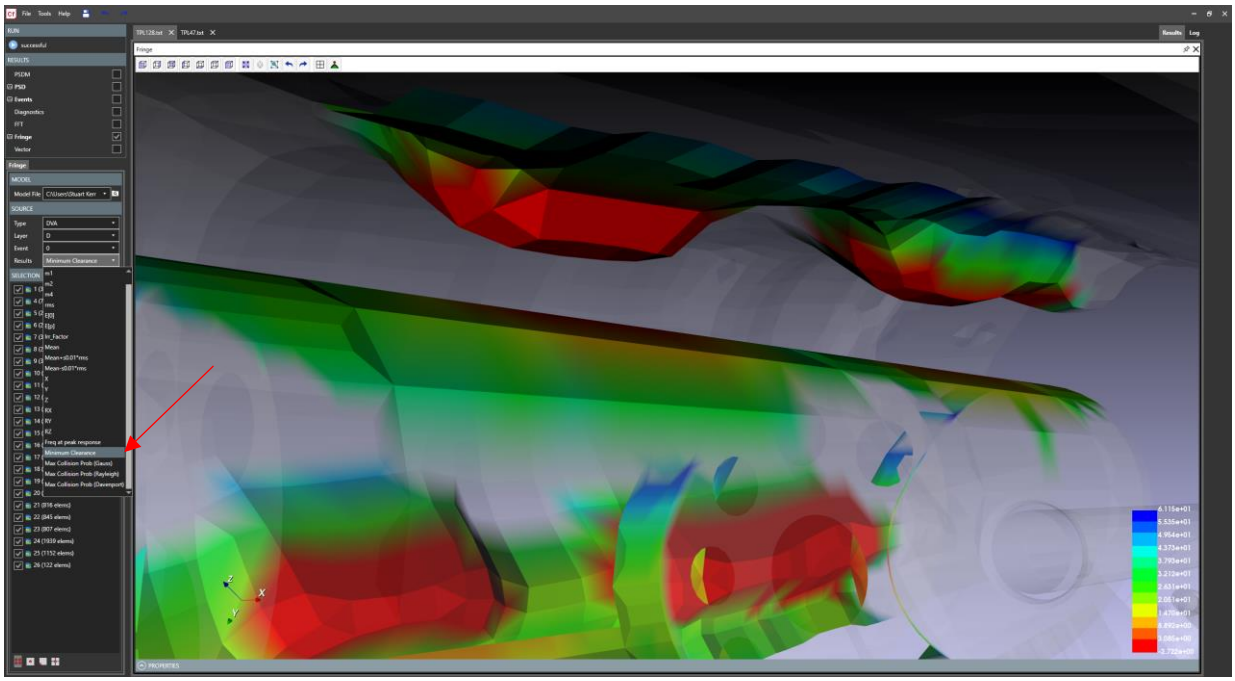
Example, the highlighted icon shows the Max / Min values of the result type shown across the entire model.



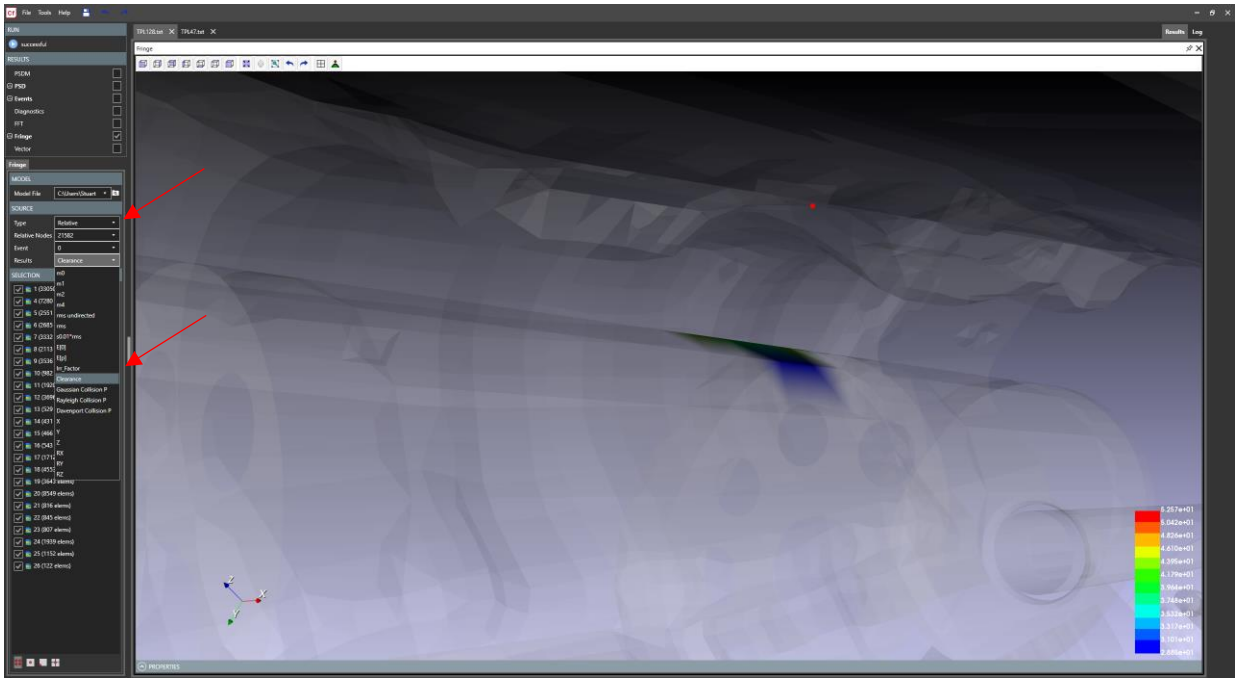
If Type is changed to DVA, then a number of additional results can be plotted. In this example, the Z direction displacements (i.e. Layer=0) are shown for the floor pan and the muffler.



In the following image, the Minimum Clearance between the floor pan and the muffler is shown as a fringe plot. This is the Minimum Clearance taking into account the phasing of the motion between the two parts.

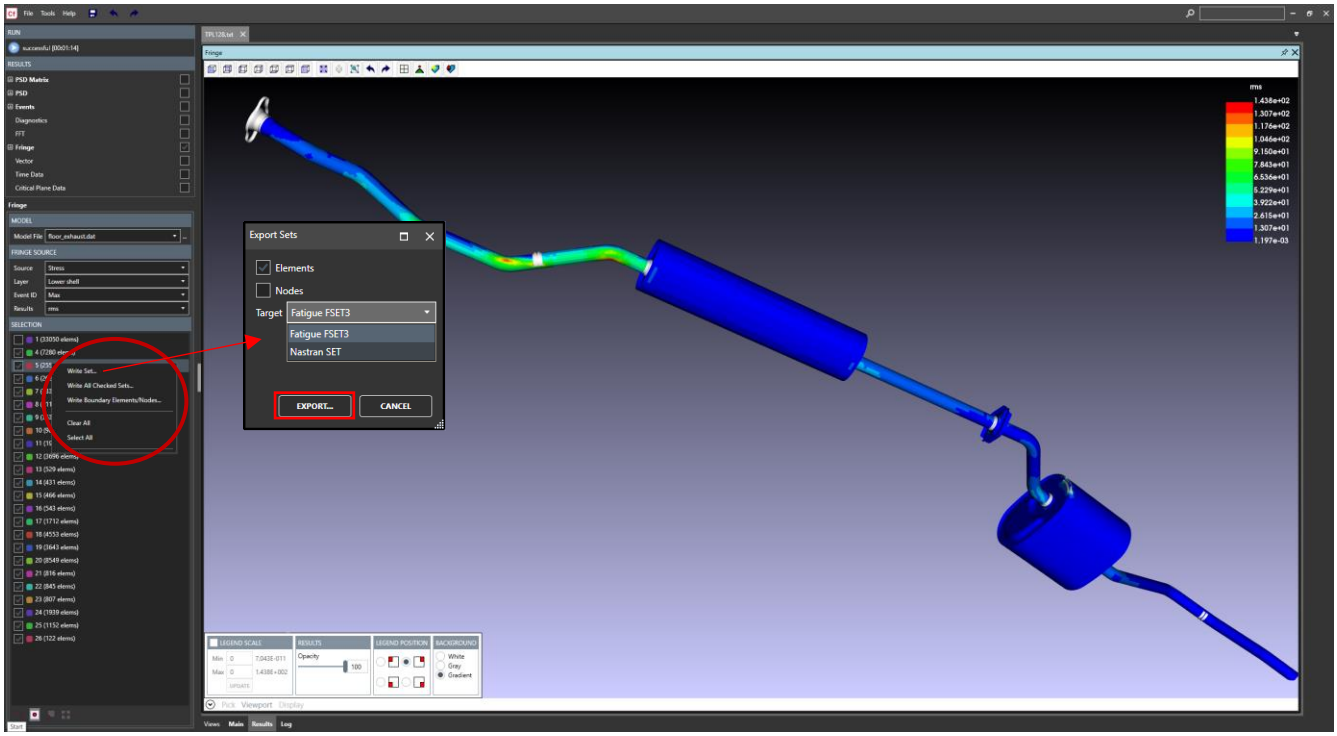


If Type is changed to Relative, then additional results can be plotted to display relative information between nodes. In this example, the relative displacement clearance can be shown between node 21582 (on the floor panel) and all other nodes in the ABSRESP node set (exhaust).



If the model contains property / element groups (sets) and these are required to be separate fatigue materials groups in the Cf input file, then the required sets can be viewed or turned off using the Property Sets selection option or by selecting the part in the model with Control+Left mouse click. For example: property set (1), the floor panel, is turned off in the image below.

You can also right click on any Property Set to write out a set containing element and/or nodes in an FSET3 format (used within CF) or SET format (used in Nastran). Note: The FSET3 will be written in Free Form format. See FSET3 entry for more information.



If you wish to have one FSET3 or SET text file for multiple property sets, simply select all the property sets of interest first, then right mouse click on the PROPERTY SETS main title bar (circled in the image above) and select “Write All Checked Sets”. To create one file that contains all the elements (or nodes or both) that make up the selected property sets. Select “Write Set” to create a file containing the elements and/or nodes for a single property set.

There is also an option to only write out the “Write Boundary Elements/Nodes” to an FSET3 or SET file. Boundary elements/nodes are those on the free face of the model i.e., any element or node that touches the outside surface of the model.

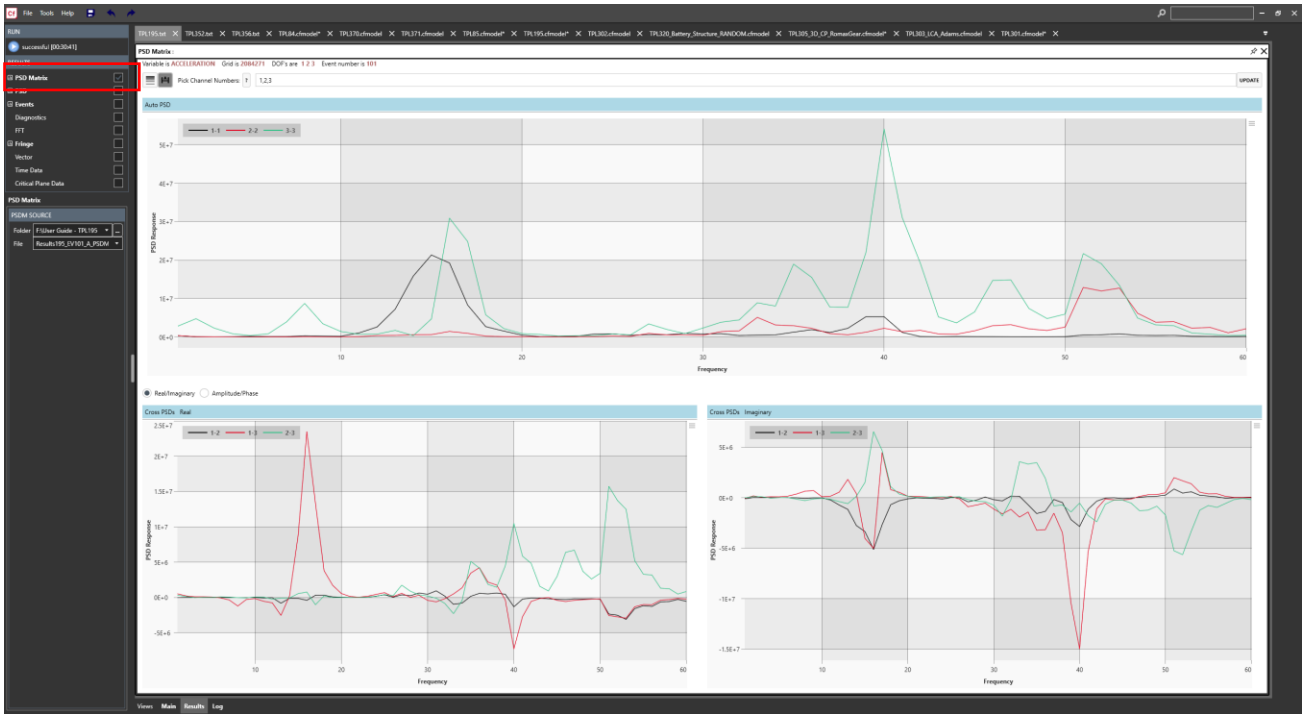
PSD MATRIX PLOTTER

TPL128 did not request a “Load Cascading” output so the PSD Matrix output has no results. However, if we look at another example (say, TPL195), we can see what the PSD Matrix output looks like.

Checking the **PSD Matrix** box will display a PSD Matrix file a User defined location. The PSD Matrix file is created when a User request a “loads cascading” output i.e., PSDMOUT, for a particular element and degrees of freedom.

Below is an example of a PSD Matrix output from TPL195 at a particular location for 3 degrees of freedom. The upper plot is the DIRECT PSDs, which are generated at the points of loading i.e., 3 degrees of freedom (X, Y, Z) means 3 Direct PSD profiles. The bottom images are the real and imaginary portions of the complex CROSS PSDs. There are 3 real Cross PSDs and 3 imaginary Cross PSDs.

See the section “***INTRODUCTION TO THE TIME2PSD CONTROL FILE VIEW***” to get more information on the PSD Matrix output.



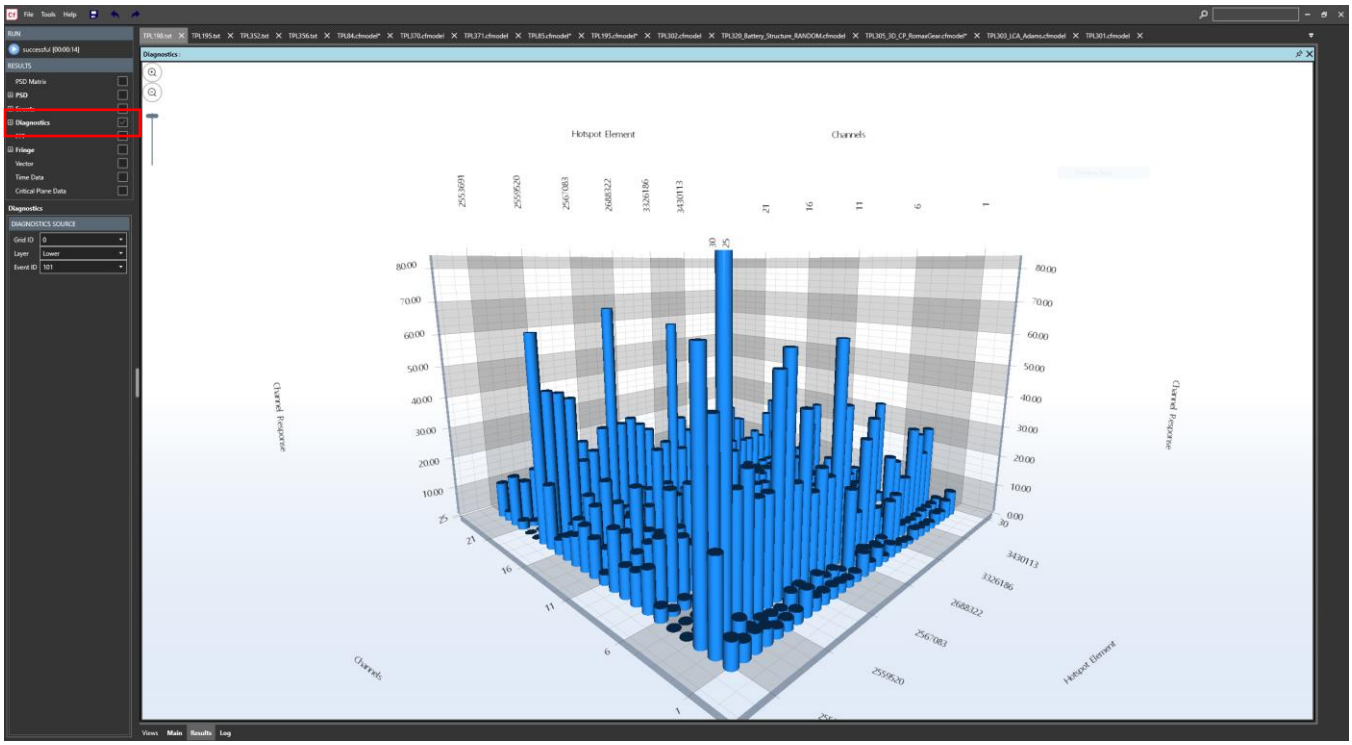
DIAGNOSTICS PLOTTER

TPL128 did not request a “Diagnostics” output so the Diagnostics output has no results. However, if we look at another example (say, TPL198), we can see what the Diagnostics output looks like.

Checking the **Diagnostics** box will display a channel contribution plot of RMS stress for a User defined event. This plot is very useful when doing an analysis with multi-channel, multi event loading.

Below is an example of a Diagnostics plot for TPL198. This analysis is a “random multiple input PSD” analysis. The loading consists of 19 Events with 24 channels per event.

The plot is generated by specifying a number of “hotspot” element locations. For each location, the software will generate the RMS stress contribution of each loading channel (i.e. 24 locations). On the left of the plot, the User can change the Event number to see the contributions with respect to the Events. If the nodal information is available, the User can look at element/node contributions as well.



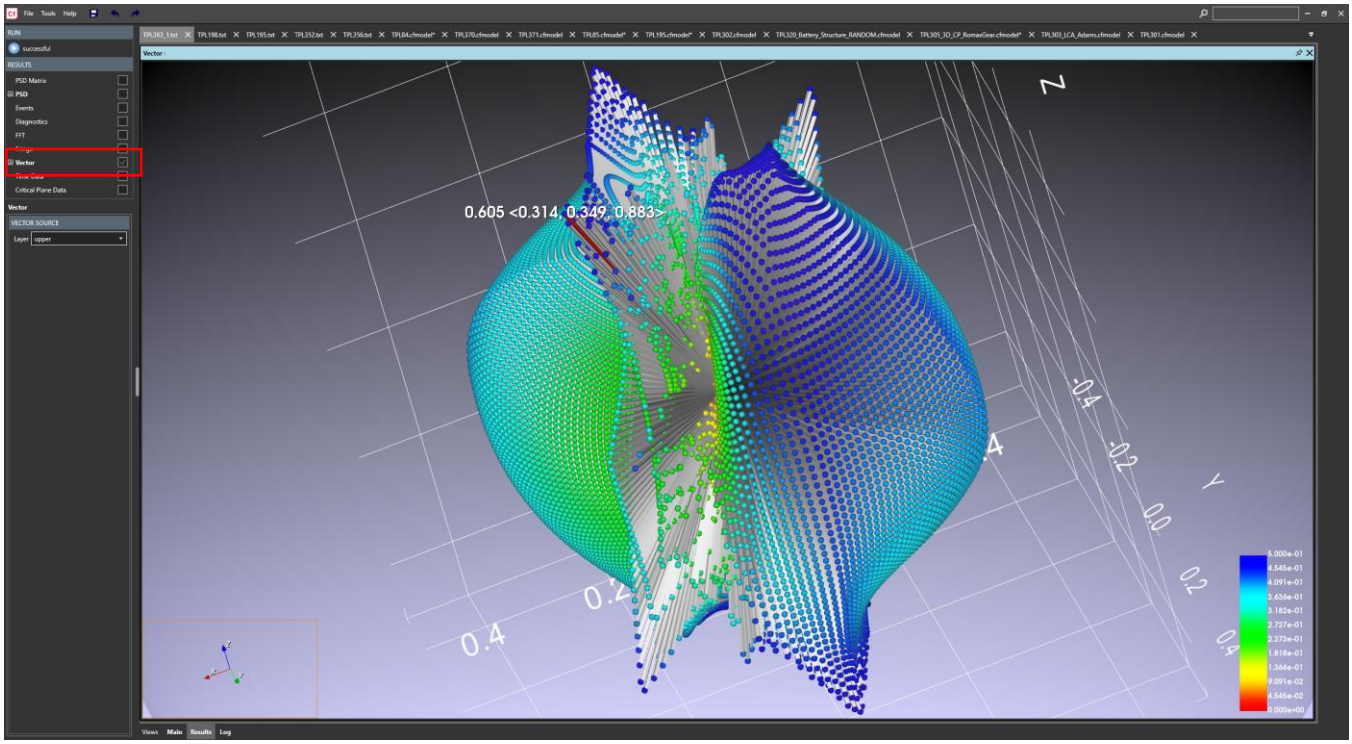
VECTOR PLOTTER

TPL128 did not request a “Vector” output so the Vector output has no results. However, if we look at another example (say, TPL363), we can see what the Vector output looks like.

Checking the **Vector** box will display a Vector plot displaying results for thousands of analyses. A VECTOR request is used in Loads Management. It will calculate the response of a system based on the loading being applied in a particular Vector direction i.e., not specifically in the X, Y or Z direction. In other words, if you have a part that you wish to test in a laboratory, you can use the VECTOR function to define an angle for application of the input loading. You can also let CAEfatigue figure out the best vector angle by combining the SGATE entry with VECTOR output.

If you conduct the SGATE / VECTOR analysis, the Vector plot will have (potentially) thousands of results plotted on a polar plot. The direction highlighted as RED will be the vector direction that produces the worst damage. All other cylinders within the plot are individual results of the same analysis. Hence, the software does the vector calculation thousands of times to find the worst damage angle.

Note: any cylinder with a dark blue tip is a good solution but the best solution (vector direction) will be highlighted in RED and identified with a Quality of Fit index number followed by the X, Y and Z components of the vector direction.

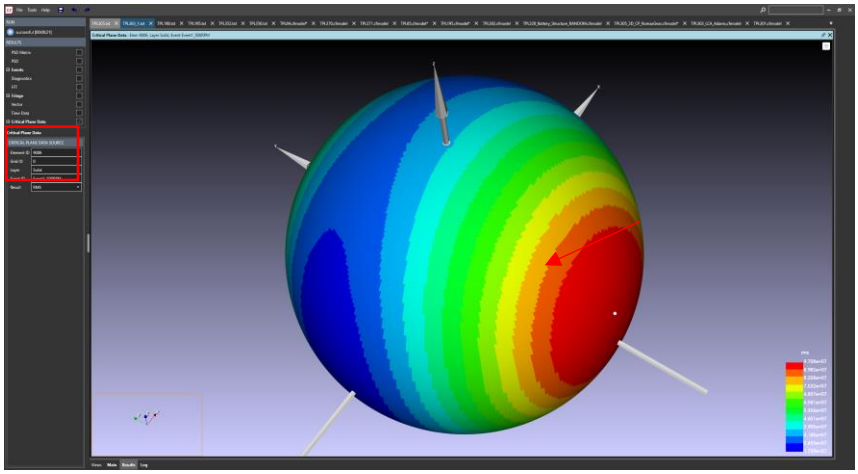


CRITICAL PLANE DATA PLOTTER

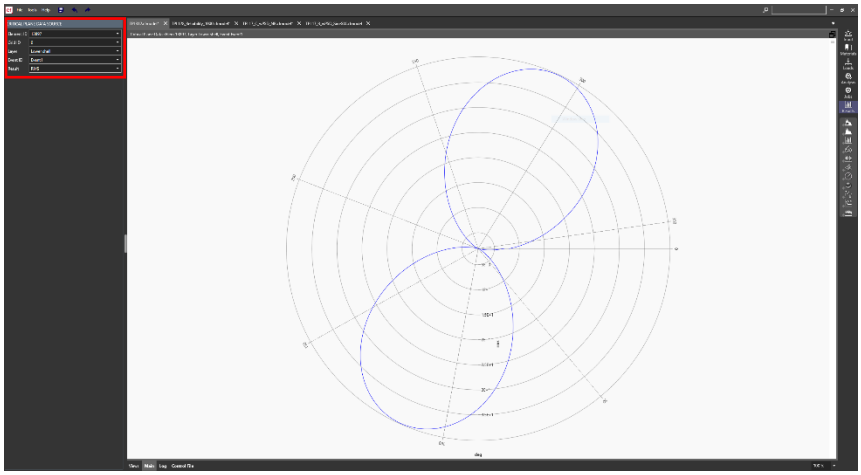
TPL128 did not use a Critical Plane analysis approach so the Critical Plane output has no results. However, if we look at another example (say, TPL305 or TPL312), we can see what the Critical Plane output looks like for 2D and 3D elements.

First, the elements of interest must be defined as those you wish to output for. This is done using the vFTGDEF> CRITOUT entry along with setting the Critical Plane analysis approach via vFTGPARM> COMB = CRITXXX. After the analysis, check the **CRITICAL PLANE DATA** box (if using the Control File interface) or selecting the **CRITICAL PLANE DATA** output in the Results box (if using the Process Flow interface) to see a visualization of the critical plan results.

For 3D elements, the display will show a critical plane sphere with a white dot referencing the normal direction to the Critical Plane for either a Damage result or RMS stress result. Below is an example of a Critical Plan Data output from TPL305 at a User defined element / node / layer / event location. The normal direction is represented by the white dot on the sphere. In this example, the normal is (almost) pointed down the negative Y axis. This means that the critical plane is (almost) along the X, Z plane. This is the plane along which the crack will grow.

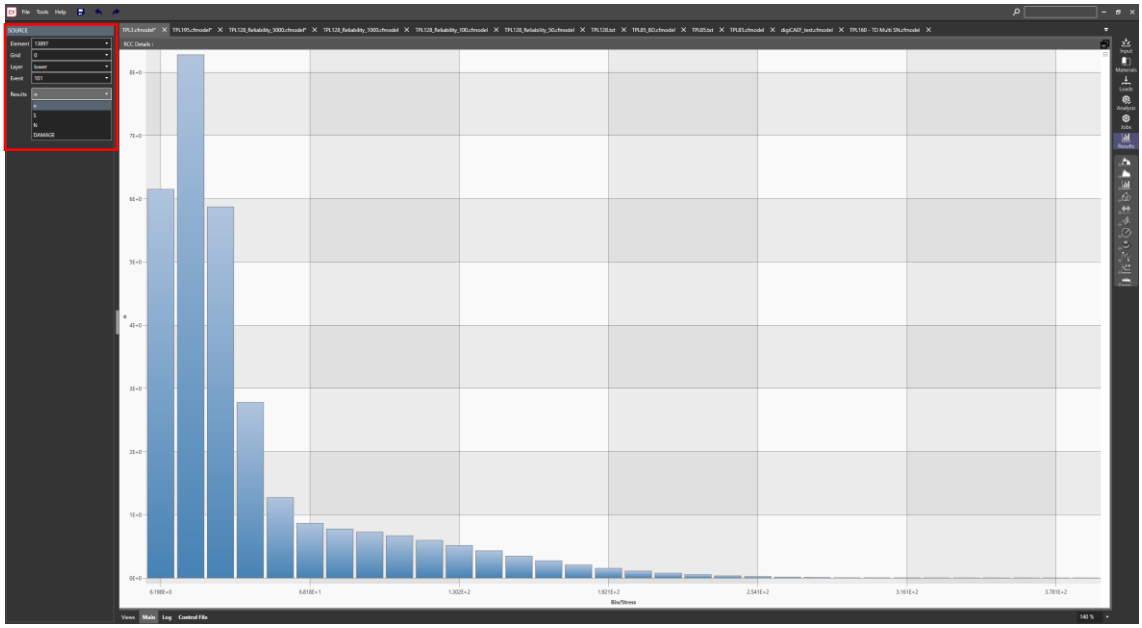


For 2D elements, the display will show a critical plane polar plot of Damage or RMS stress result. Below is an example of a Critical Plane polar plot for TPL312 at a User defined element / node / layer / event location. The normal direction radiates from the center to the outer most edge of the plot orbit, i.e., in this case ~110° / 290°.



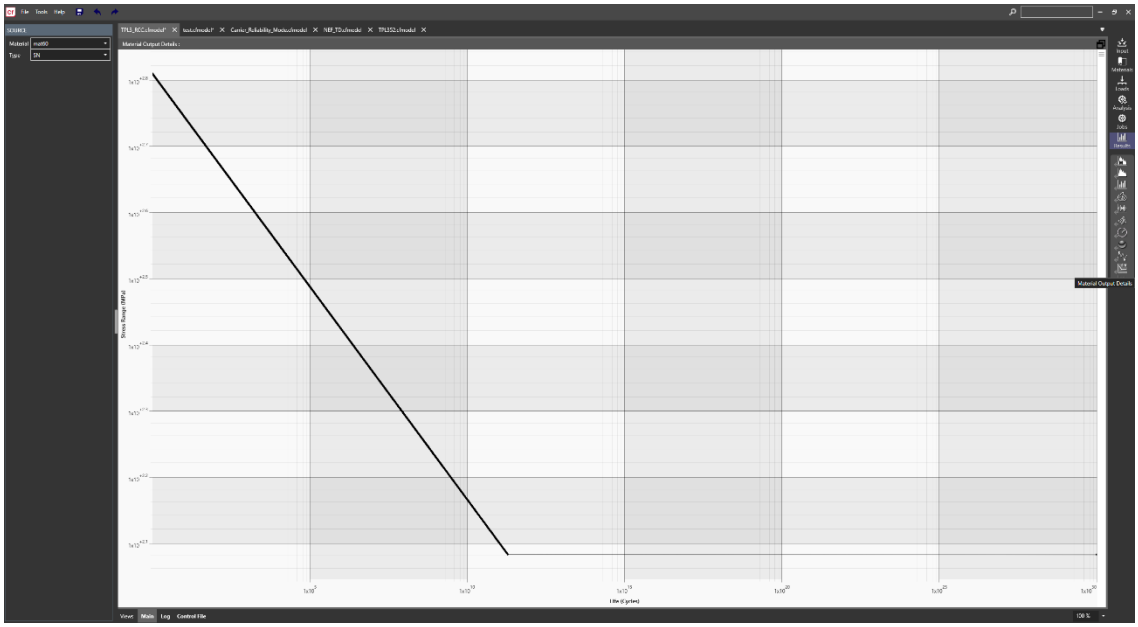
RAINFLOW CYCLE COUNT PLOTTER

CF provides the ability to view the RCC.csv data created in a frequency domain analysis when LOGLVL=2. The User can view that actual number of cycles (n), as show below, the stress bin values (S), the cycles to failure (N) and the Damage.



MATERIAL DATA PLOTTER

CF provides the ability to view the SN curve data created in a frequency domain analysis when vFTGPARM> OPTIONAL = MATOUT (material out). This is a plot of the SN data derived by Cf based on the SN or EN data provided in the vMATFTG entry.



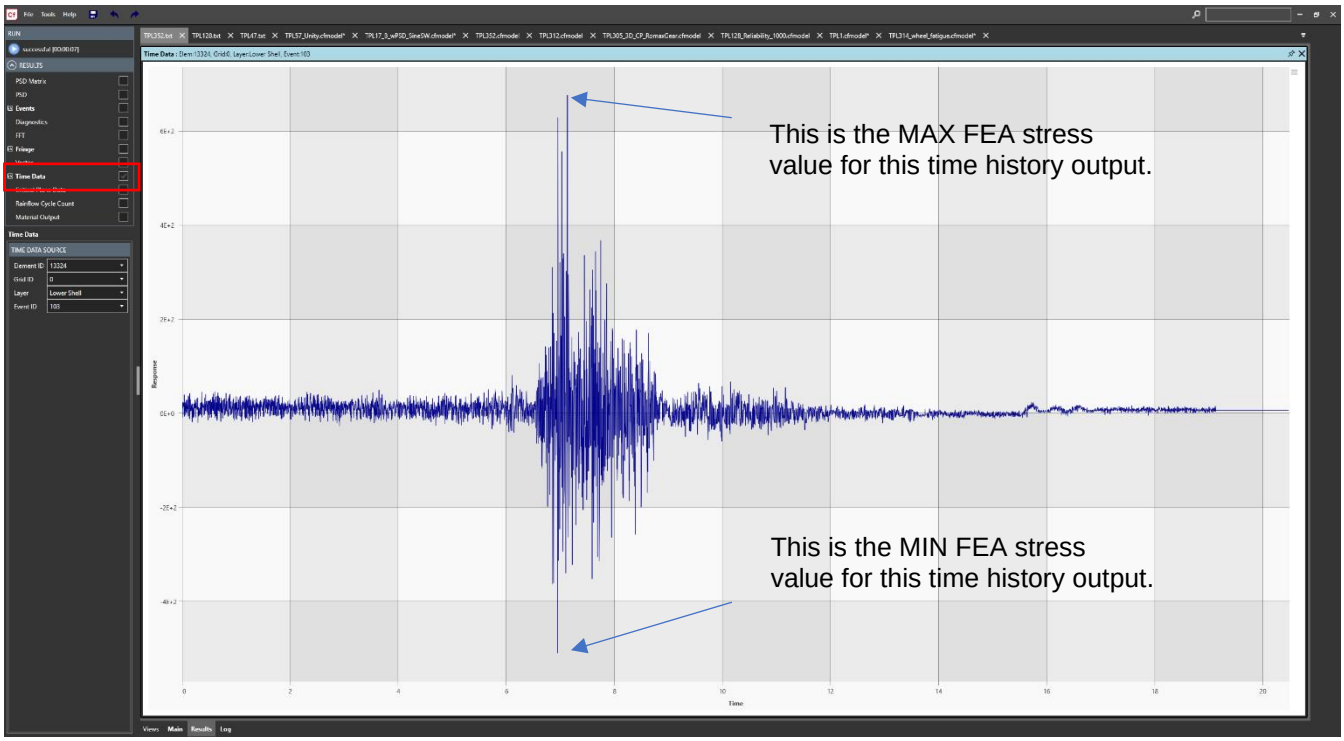
Viewing Results – Time Domain Analysis Outputs

Several of the existing plots discussed for the Frequency Domain, also work for the Time Domain. This list includes Event plotter, Fringe plotter and Critical Plane Data plotter. The **Time Data** plotter however is unique the time domain analysis.

TIME DATA PLOTTER

When doing a time domain analysis, the User will have access to the Time Data Plotter if requesting a TIMEOUT result. Note: this is very similar to the PSDOUT request used in the Frequency Domain. Checking the **TIME DATA** box will display the response time signal results at a User defined location. For TPL352 (LQSTATIC analysis), the plot below shows the **FEA** response stress at a User defined element.

Note: the Fringe Plotter will show **Neuber** Stress (not FEA stress) if the appropriate material properties are available. However, the Time Data output will show **FEA** stress. Keep this in mind when comparing Fringe plot Max or Min Neuber stress output to the Time Data output.



Viewing Results – Fringe Plotter Properties Box: Viewing Options

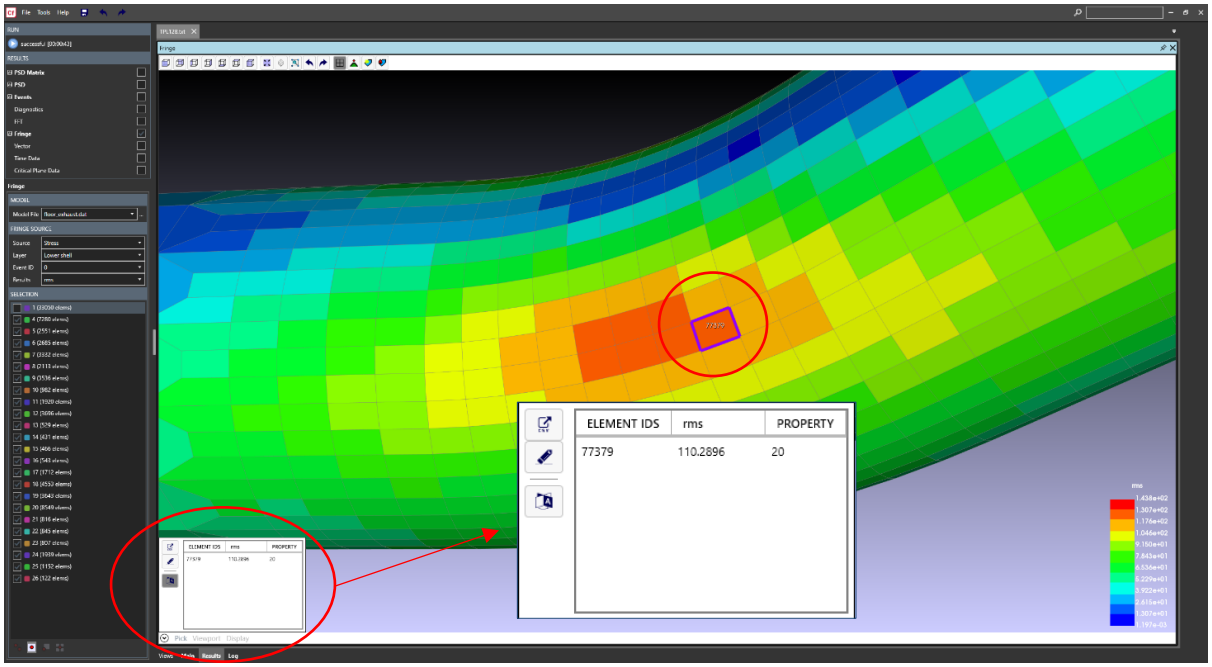
PICK OPTION

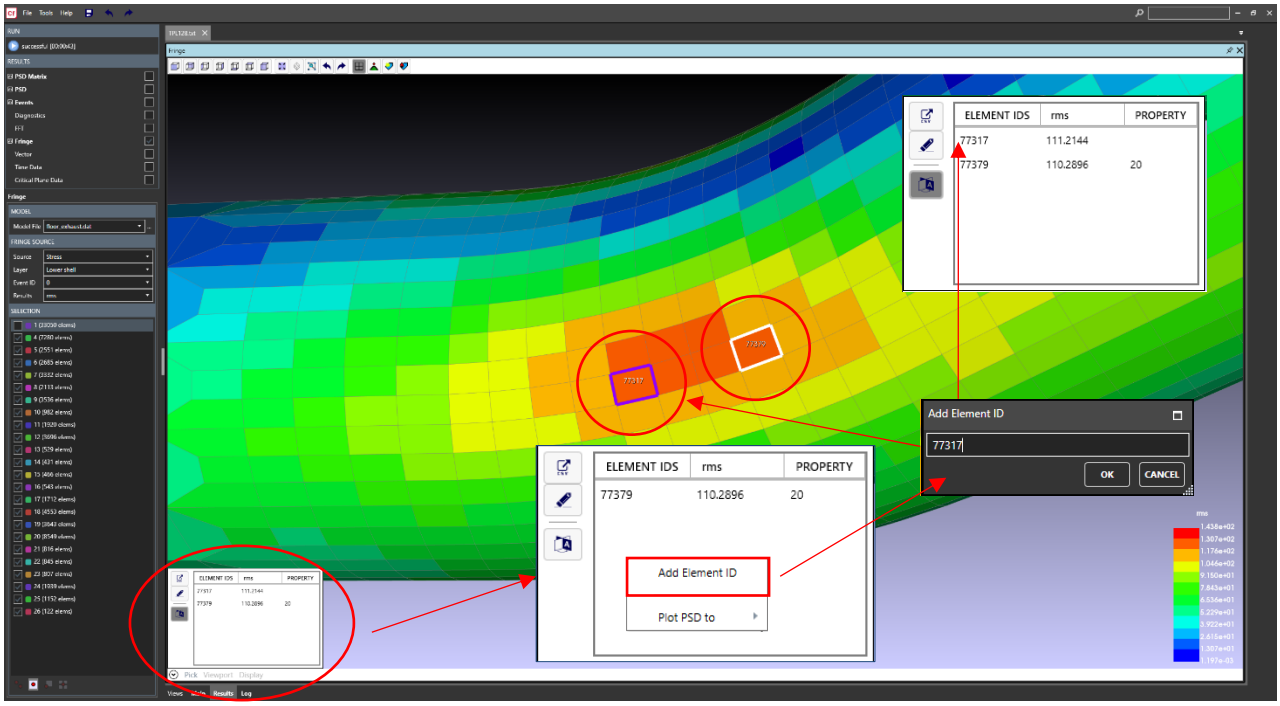
You can also choose to get precise information at the element / node level by opening the Properties Box at the bottom of the screen to select the PICK option. Once opened, you can select individual element center or node results (if available) from the fringe plot and display those results in the Pick box. The image below is showing the RMS stress results for Element 77379 with the “Show Label” icon selected ON, to place the element number on the plot. Additional information including the RMS Stress value and Property Set number are also shown.

NOTE: The **RMS** column in the PICK window will change when the **Results** output selection is changed, i.e. if the Results Output is changed to DAMAGE, then the RMS column will change to become a DAMAGE column and will show the DAMAGE results for the elements / nodes selected.

If viewing results for a specific element or node number is desire, the User can right mouse click within the PICK box and select "**ADD ELEMENT ID**" and enter the desired Element ID number (if nodes are available with the element, the data will appear for those as well. If working with DVA data, the selection will be "**ADD NODE ID**".

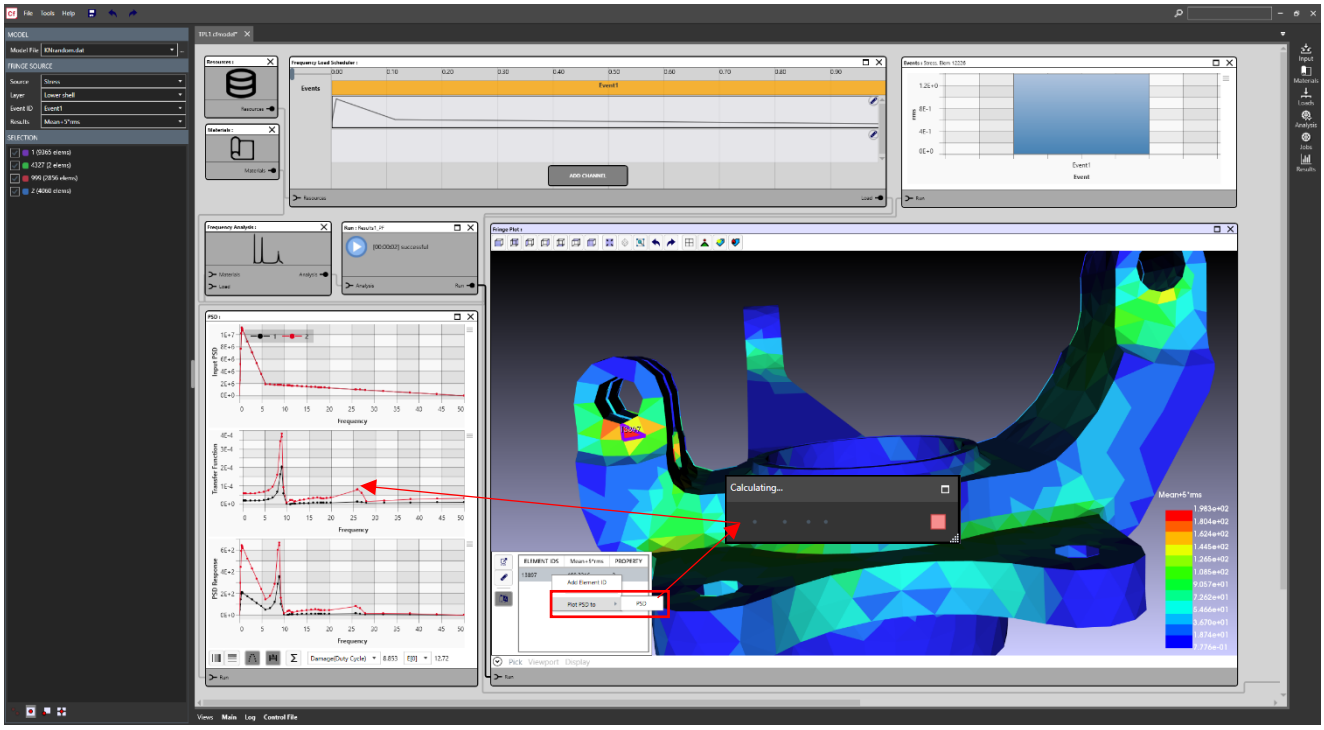
Note: Multiple IDs can be entered using a comma (,) and/or dash (-) between entries.



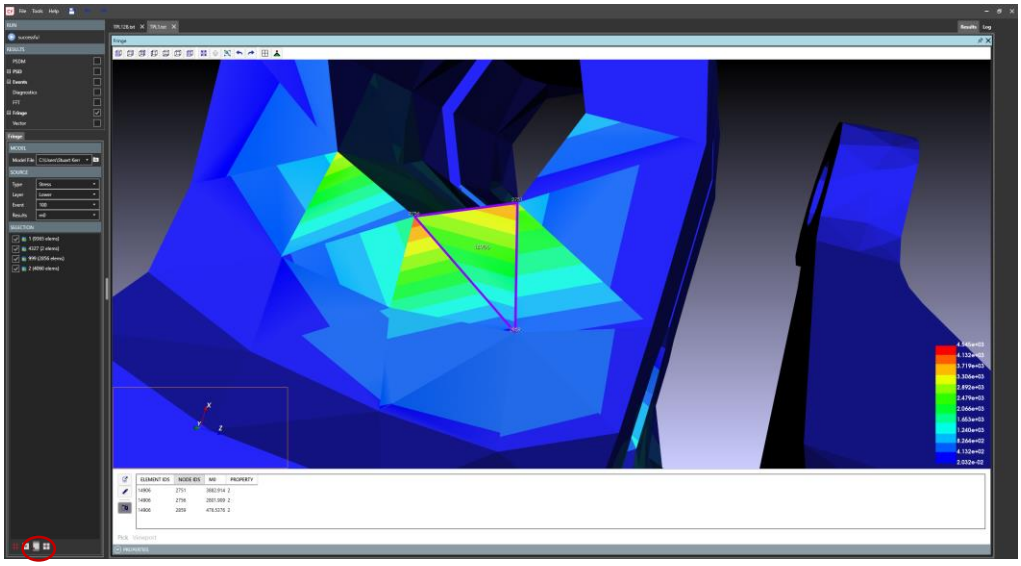


Within a Frequency Domain analysis, the same PICK box, the User can also plot the PSD information for the element direction to the PSD PLOTTER by selecting "**PLOT PSD TO**" and selecting the appropriate results output. NOTE: this currently works for Process Flow examples only.

In the example below (for TPL1), the User selected element 13897 and requested to "**PLOT PSD TO**" the existing **PSD** plot. Upon requesting the PSD, the software calculated the PSD information for the new element and promptly plots the data in the PSD Plotter as another set of data points (in RED for the example below).

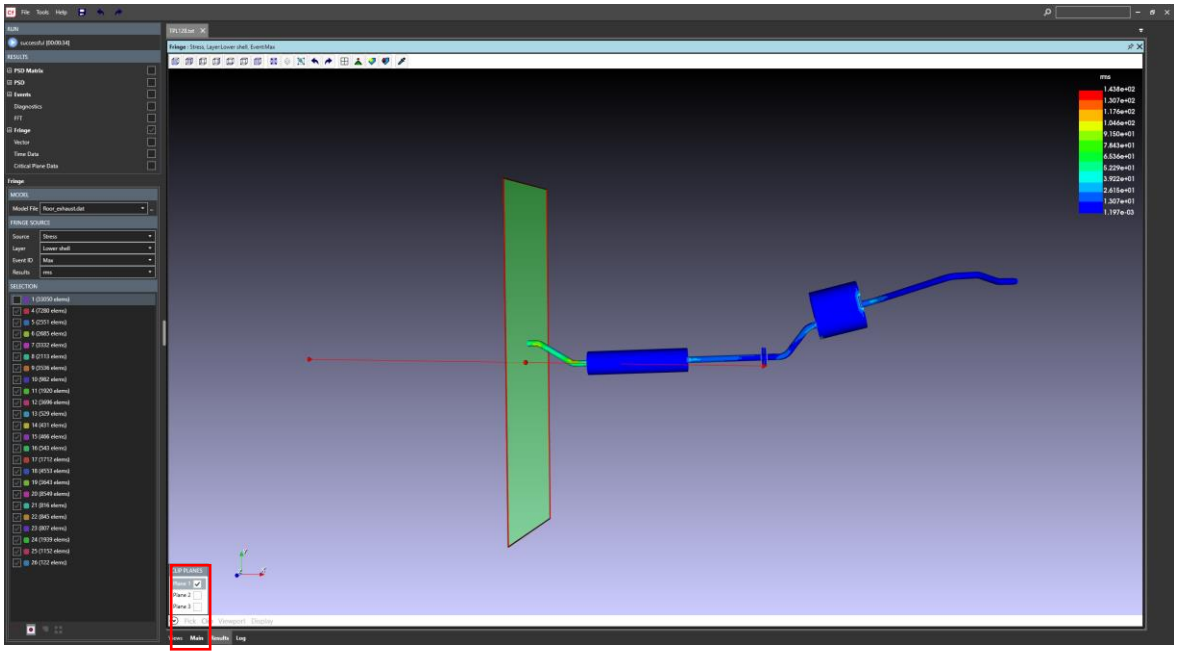


Another example below from TPL1, shows element and node results because the button "**Unaveraged Corner**" was selected. In this case, the element and node numbers are shown on the plot and the RMS Stress / Property Set information is shown in the PICK window. This example is from TPL1.txt.

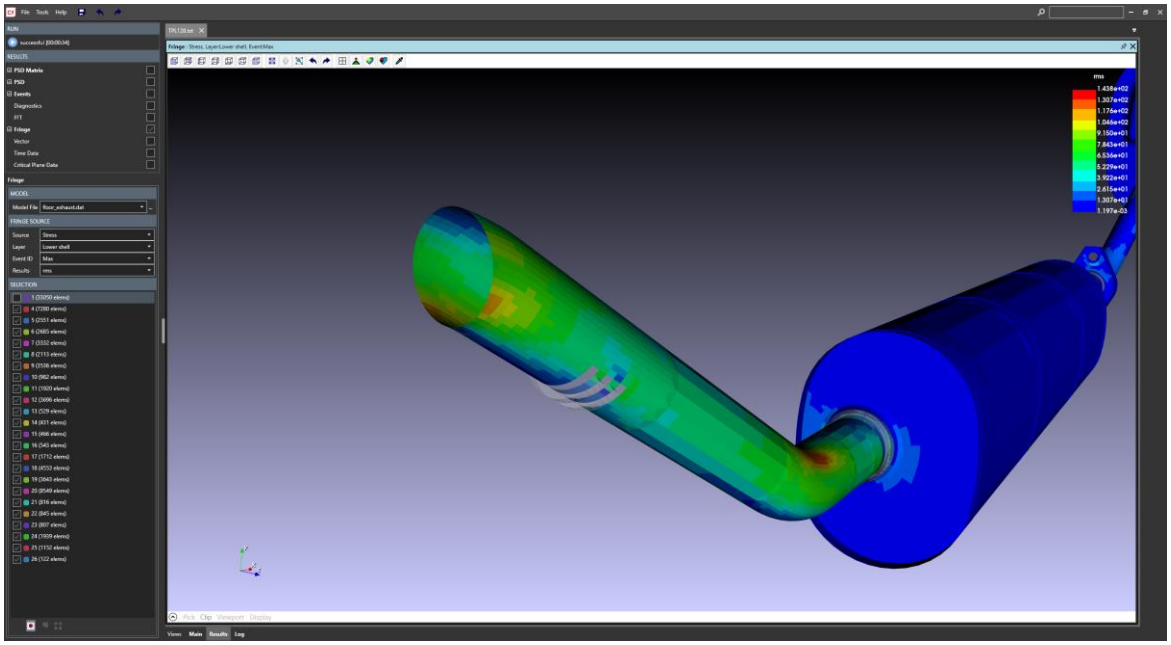


CLIP OPTION

By selecting the CLIP option, the user can set up to 3 different clipping planes (by default in the X, Y and Z directions) as desired. The planes are activated by checking on of the plane boxes. The plane can then be manipulated to the right position and orientation. Note: up to 3 planes can be used.

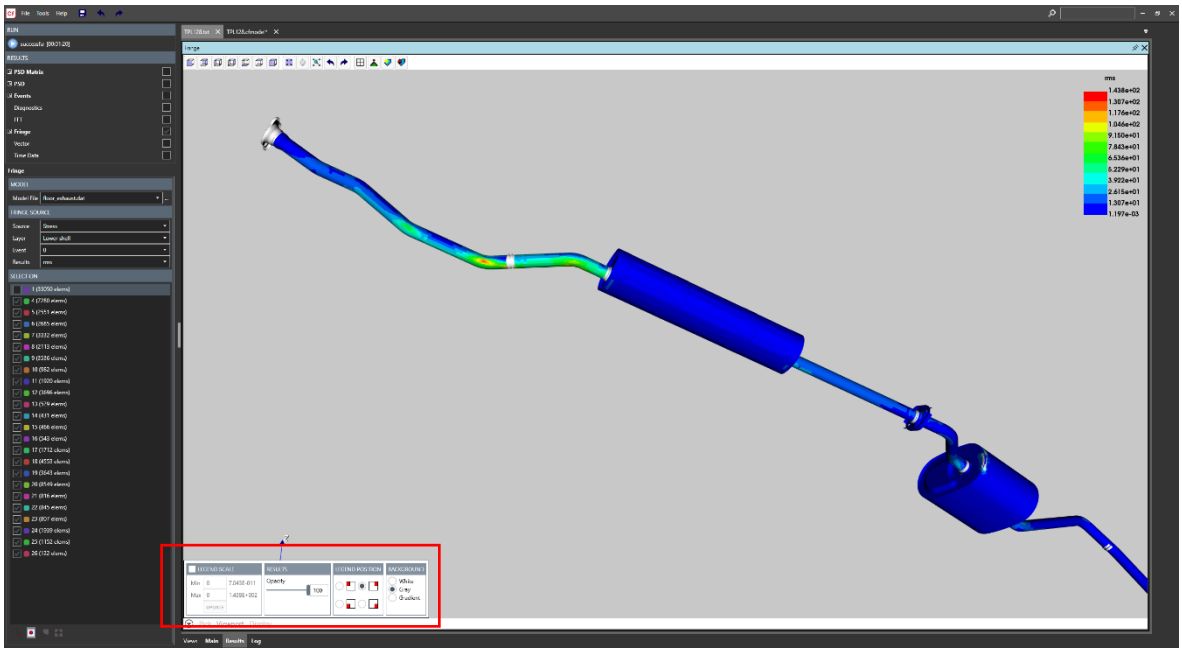


Once the plane(s) are set, the user simply needs to exit the CLIP option and the clipping plane that are checked, will remain in place to allow viewing. Below is the result of the clip plane show above.



VIEWPORT OPTION

By selecting the VIEWPORT option, the User can also change the background color, legend location, the opacity of elements that do not have results and the min/max scale values. The image below is showing the RMS stress results with everything below 20 MPa plotted as the lower scale color (deep blue) and everything above 100 MPa plotted at the upper scale color (red). The background of the plot has been changed to gray, the opacity increased to 100% (turns elements without results to solid white), and the scale legend has been moved to a bottom-left corner.

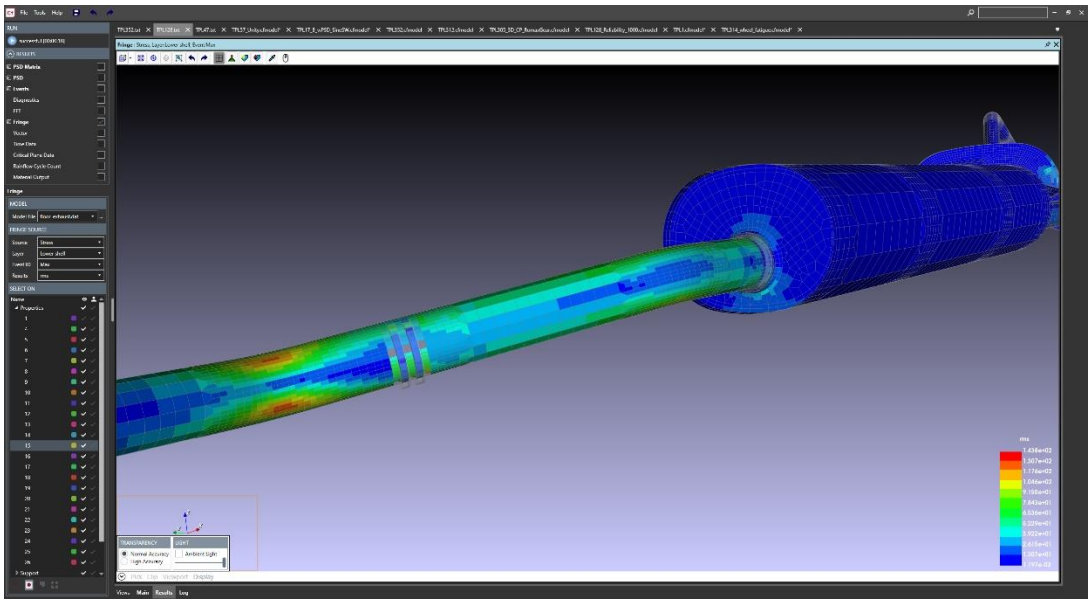


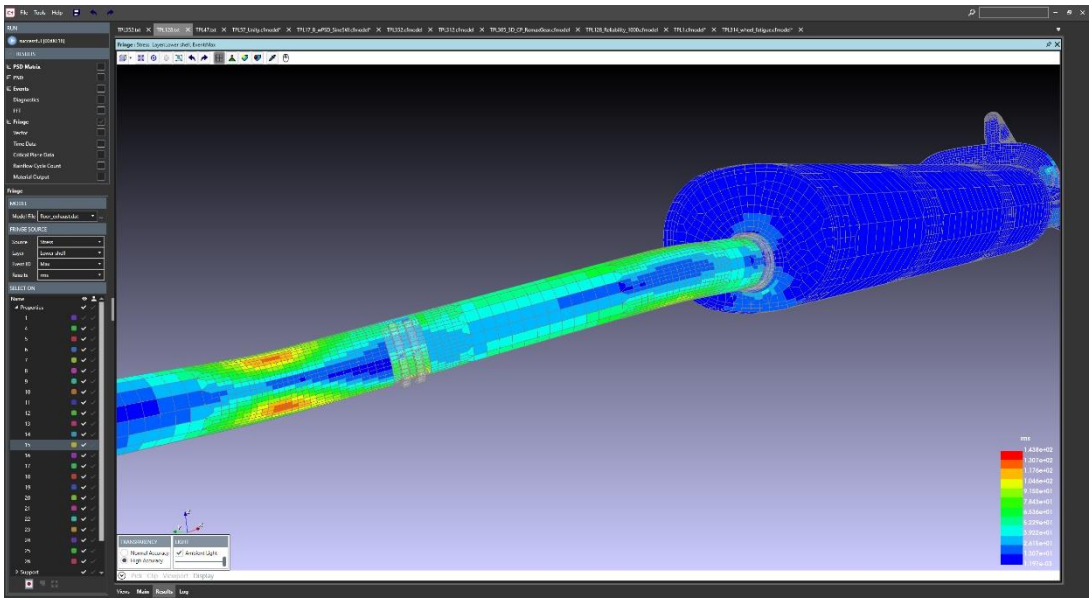
DISPLAY OPTIONS

By selecting the DISPLAY option, the User can adjust the lighting and transparency settings when viewing any model. These work best when the model has large areas of transparencies (i.e. area with no results).

The User can select HIGH accuracy in the plotting of the model edges for a very defined (sharp) image, when compared to the NORMAL accuracy, which is default within the plotter. The User can also change the lighting used to illuminate the model to AMBIENT light. Full ambient lighting will remove the shadows seen when using the default light source for the Fringe plotter.

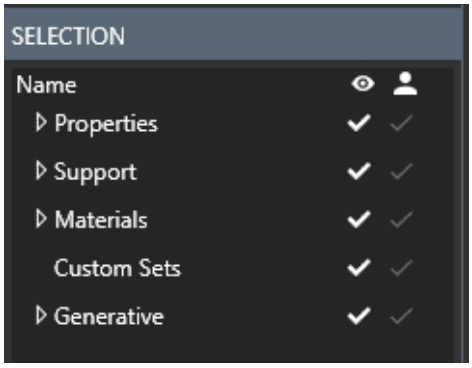
Below is an image with normal accuracy and no ambient lighting versus an image with High Accuracy and 100% Ambient Light (although the image captured for this document is far less impressive then what is viewed on the computer screen).





Viewing Results – Fringe Plotter Selection Menu

The left menu of the fringe plotter has a SELECTION box. Within the box are NAME sets consisting of the following.



- **Properties**: These are all the “property” sets defined in the solver. In Nastran, this would be the any “PSHELL” or “PSOLID” entry ID
- **Support**: These are all the sets defined in the solver. In Nastran, this would be any “SET” ID.
- **Material**: These are all the materials defined in the solver. Example: in MSC Nastran this would be the “MAT1” IDs.
- ***Not yet implemented: Custom Sets**:*
    - *Seam Weld: Allows the User to easily select a group of elements needed to define the element along the toe line of a weld.*
    - *User Sets: Allows the User to select any number of elements to create a group of elements. This can then be exported.*
- **Generative**:
    - Unassigned Elements: This will show all elements that do not have a result in the CSV output file.
    - Boundary Elements: This will show only the elements at the boundary of the mesh that have a result in the CSV output file.
    - Results Entries: This will show all elements that have a result in the CSV output file.

7 | INTRODUCTION TO THE FATIGUE PROCESS FLOW VIEW

NOTE: The images shown in this section reflect CAEfatigue v2021 and may change slightly with future releases.

CAEfatigue operates by executing a series of commands laid out in a “control file”. The actual control file can be defined within CAEfatigue via the Control File view. Alternatively, CAEfatigue allows the User to define the analysis via a graphical Process Flow view. This view will then create the control file when the User is ready to execute the job.

This chapter will explain the Process Flow view along with an example that shows the relationship between the Process Flow entries and the associated Control File entries and fields described in the Quick Reference Guide.

Current Limitations of the Process Flow in this Release

The Process Flow is a graphical interface for the User to run an analysis. However, there are a few entries / fields that are not yet available when compared to using the Control File (text based) interface.

Below is a list of features / capabilities not yet support by the Process Flow:

- The TIME2PSD toolset is not available in the Process Flow interface.
- When creating a Process Flow from Empty or from a Template, the following Control File entries / fields are not yet supported for the Frequency Domain
    - vIBFAT> OPTIONS> STRLIM (defaulted to 1 when ELSETS are present)
    - vFTGDEF>
        - CLIPLEV / MAXFREQ / MARSTR / STRBINS
        - SGATE
        - USERWHS
        - SEAMW>WTYPE
    - FE_UNIT> CONJCON
    - vFTGPARM>
        - STRESS> RCCT
        - SPOTW> ZCOMP
        - SEAMW> THICK
        - USERWHS
    - vFTGSEQ>
        - MOS_DUR
    - VECTOR
    - PERTURB

Overview and Introduction to the Process Flow Menu Items

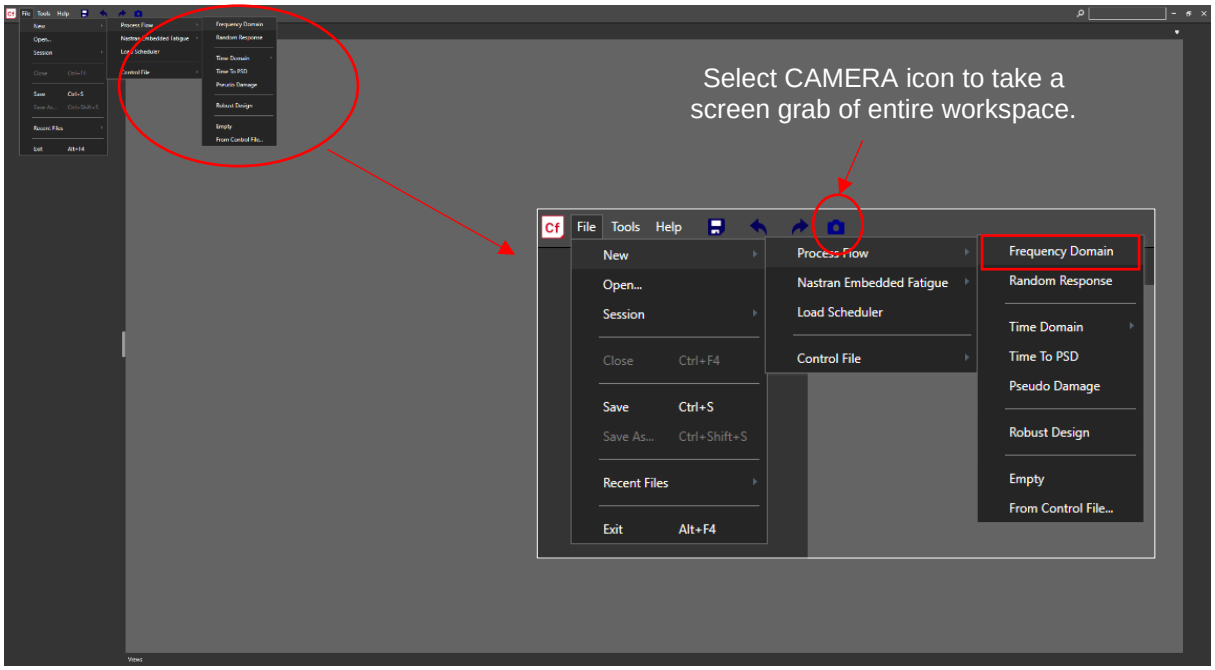
To start CF, click on the desktop Cf icon (see below) or find the ProcessApp.exe file for the program in the default installation directory “C:\Program Files\MSC.Software\CAEfatigue\<ver>\Windows\ProcessApp.exe.

After the application window opens, select **FILE>NEW> Process Flow** and select either Empty, From Control File, Random Response, Frequency Domain, Pseudo Damage or Time Domain. Empty will give you a new (blank) screen to start with. “From Control File” will allow the User to import an exist text file. Random, Frequency, Pseudo Damage or Time Domain will give you predefined templates to start.

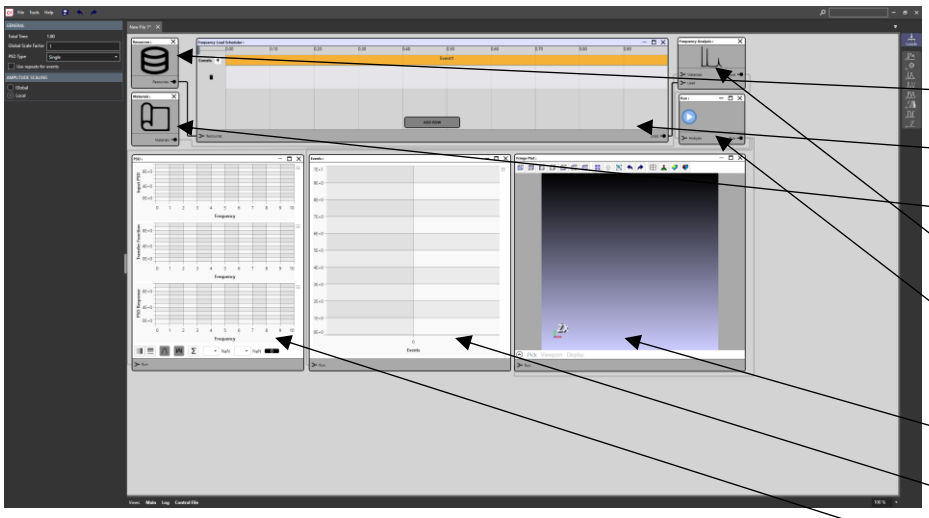
→

A Frequency Domain Process Flow Example – TPL1.cfmodel

To start this example, the User must select File>New>Process Flow>Frequency Domain to get a NEW Process Flow Frequency Domain template from the File menu on the main start page.



Below is the starting screen for a Frequency Domain analysis. We will use this template to recreate TPL1.



Input:

Resources

Frequency Load Scheduler

Materials

Frequency Analysis

Run (Execute Job)

Output (examples):

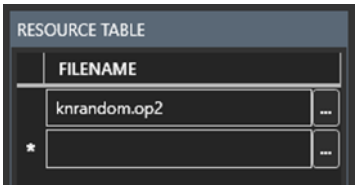
Fringe Plotter

Event Plotter

PSD Plotter

Each box in the Process Flow requires entries and fields to be populated. Below are the boxes (entries and fields) that are required to run TPL1.

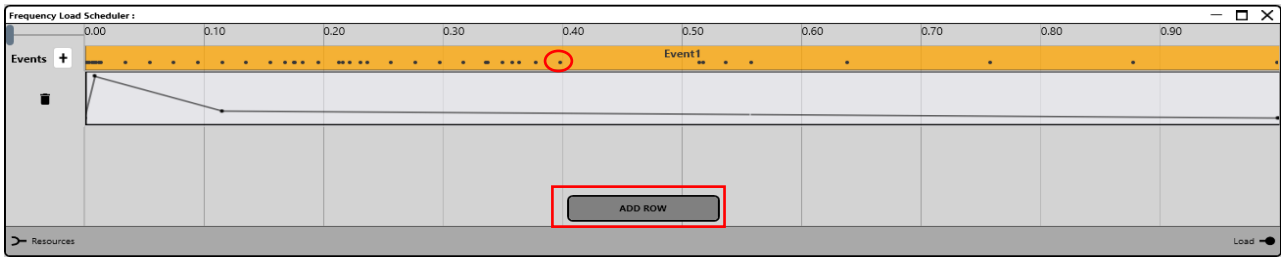
Resources:



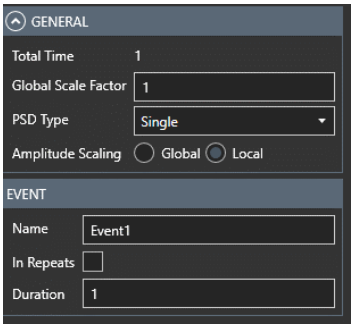
The **Resources** box is used to list the input files that are available for the fatigue analysis. This list would contain FE Solver results file, RSP files, PSDM files and Punch file (as necessary. For TPL1, the only resources file needed is the Nastran OP2 file knrandom.op2.

For other jobs these files could be Nastran OP2, Nastran HDF5, Ansys RST, Abaqus ODB, OptiStruct OP2, and RSP files (for TD static analysis).

Frequency Load Scheduler (select an Event box):



ADD ROW and dots in Event Header Bar: Use the ADD ROW button if additional loading is to be applied to the Event. For example, a sine sweep could be added to this event to do a “sine on random” analysis. Also, the “dots” shown in the **Yellow** Event 1 header bar, represent the FRF frequency points coming from the Solver file (OP2 file only). This is useful to see if your Random PSD is properly “**frequency matched**” to the Transfer Function.

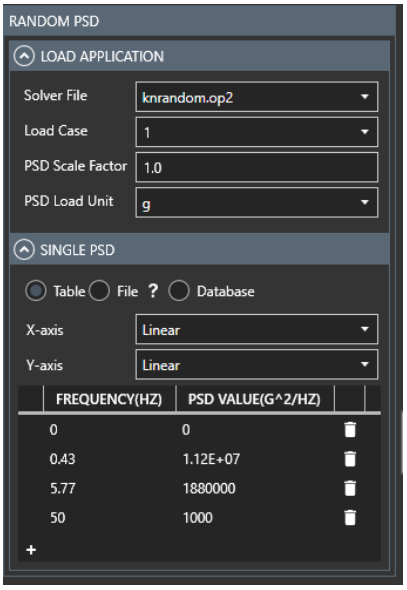


The Event box of the Frequency Load Scheduler is used to set the event duration timing.

Entries / Fields in Process Flow compared to Control File:

Global Scale Factor	vFTGSEQ>TSCALE
PSD Type	vFTGLOAD>TYPE (Note: The User has a choice to select Single, Multi or Multi Uncorrelated i.e., use diagonal PSDM terms only)
Use Repeats for Events	vFTGSEQ>NREP
Amplitude Scaling	n/a (choose to set vertical scale for all channels)
Event Name	vFTGEVNT>NAME>EVNTNAME
Event Duration	vFTGSEQ>EVT1

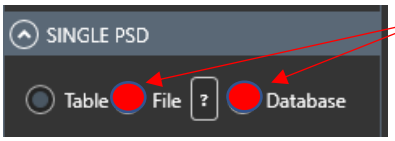
Frequency Load Scheduler (select a Random PSD box):



The Random PSD box of the Frequency Load Scheduler is used to enter loading information needed for the analysis. After selecting the Random PSD box, the User is presented with a number of entries to define the loading.

Entries / Fields in Process Flow compared to Control File:

Solver File (populated from Resource Table)	OP2MAP solver file name
Load Case ID (populated from Solver file)	OP2MAP LCID number
Load Scale Factor	vFTGLOAD>SCALE
PSD Load Unit	n/a (used as label in PSD Value title)
X-Axis	vTABRND: XAxis
Y-Axis	vTABRND: YAxis
PSD Table	vMATFTG> define PSD entry via a table definition



NOTE: The User can enter a PSD defined in a CSV or TXT file that has follows a vTABRND, vTABRND1 or TABRND1 format or a format as defined below. Select "?" to see this information.

#axis
Log,Log (or Linear, Linear)
#data
X1, Y1
X2, Y2
(continue)

The User can also select a PSD profile from the PSD loading database.

Understanding the Width of the Frequency Load Scheduler Box

The **Frequency Load Scheduler** has 2 scales to consider. The "width" of the **box** is determined by the time duration of the Event. In the case below, the event duration (EVT) is 1 second so you can see the time scale at the top showing 1 second of length.

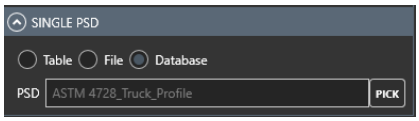
There is also a second scale to consider. Within the box itself, the display of a PSD or deterministic loading is based on the frequency range of the Solver FRF file. In the example above, the "dots" show the various FRF points in the Solver file and the range goes from 0 Hz to 50 Hz, hence, the PSD width range is from 0 Hz to 50 Hz. If the PSD or deterministic load, has the same frequency range as the FRF file than the PSD or Deterministic load will fill

the entire length of the box. This is the case for the PSD in the image above because it is defined from 0 Hz to 50 Hz. However, notice that the sine sweep, which only goes from 1 Hz to 5 Hz, is shown as only occupying a small portion of the total 50 Hz range of the window box.

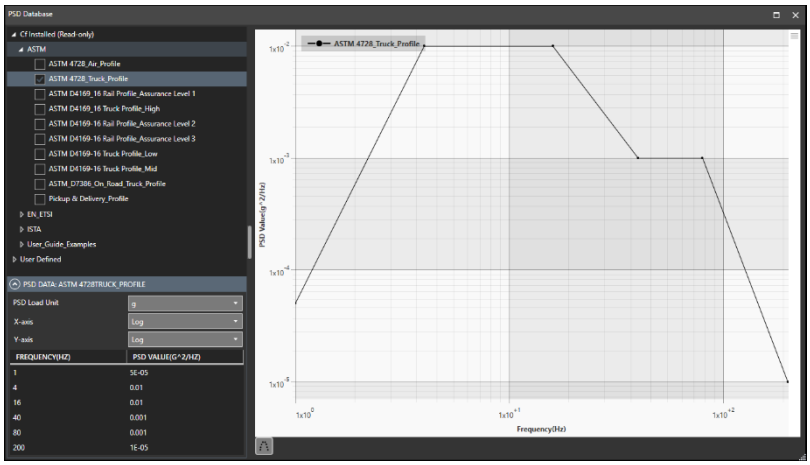
Brief Introduction to the PSD Database

As mentioned above, the user has the option to define a PSD provide using direction XY pair entries, using a PSD defined in an existing file or selecting a PSD via a PSD Database.

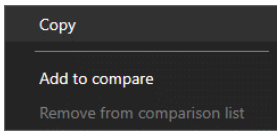
After selecting the “Database” radio button and selection PICK, the User will see the PSD Database interface.



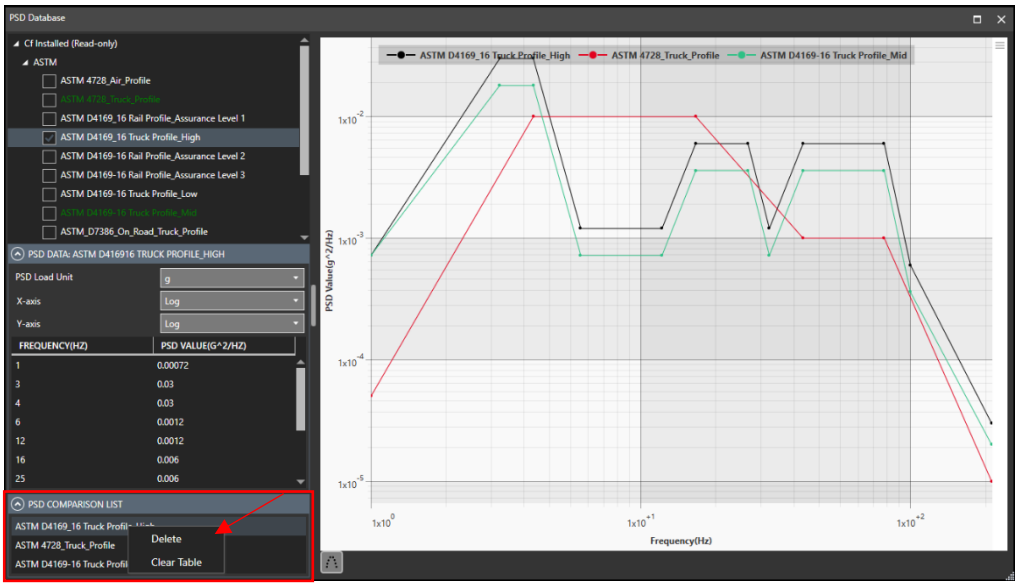
A material provide can be selection (check box) and review to see if it is the desired profile. If correct, the User simply close the interface (select X in upper right corner).



In some cases, the User may wish to compare several profiles prior to making a selection. To do this, the User can Right Mouse Click on any item and select “add to compare”. Any number of PSD profiles can be added to the comparison plot shown in the graph below.

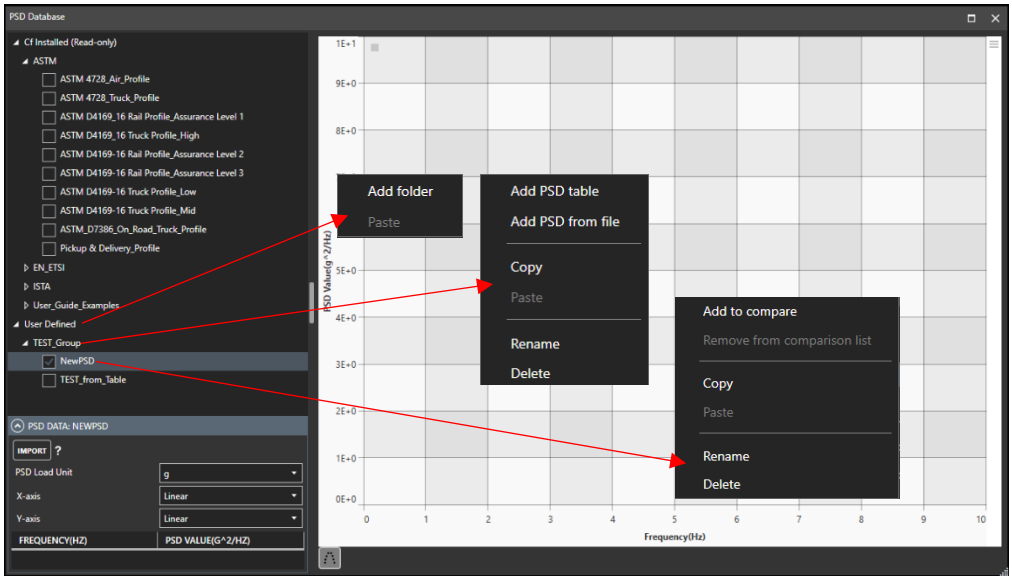


As profiles are added to the **PSD Compare List**, the profile names will turn green and the profile shapes added will be added to the plot. Once the desired profile shape is found, the User must clear the PSD Comparison List and then select the desired profile from the list to use in the analysis.



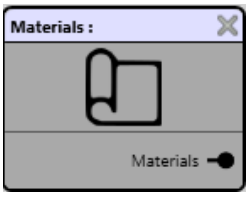
Users can also add their own profiles in a “folder” of profiles by right mouse clicking on the **User Defined** label. This will allow a folder name to be defined and then individual PSD profiles to be added to the folder group by right mouse clicking on the folder name. PSD profiles can be entered as XY pairs are read in from an existing file. Individual PSD profiles can be modified by right mouse clicking on the PSD profile name.

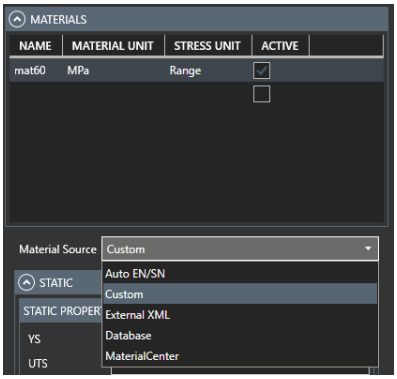
NOTE: The User can enter a PSD with table values entered directly into the PSD data table or import a PSD CSV or TXT file that has a complete vTABRND, vTABRND1 or TABRND1 or an XML definition.



.

Materials Definition:





The **Materials** box is used to enter the fatigue material properties. For TPL1, a simple table of properties are entered along with the Static material properties.

Note: the material information would be entered in several different ways, including the selection of material from a Material Database.

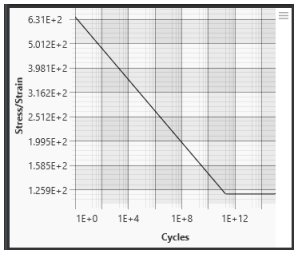
Entries / Fields in Process Flow compared to Control File:

Material Name	n/a
Material Unit	vMATFTG>CNVRTM
Stress Unit	vMATFTG>AMP_RNG
Material Source	Auto EN/SN – auto generated properties
	Custom – data entered by User
	External XML – User defined database
	Database – Cf internal database
	MaterialCenter – connect to MSC MC

Custom Example:

Static Properties	vMATFTG> define STATIC fields
EN / SN	vMATFTG> define EN or SN parameters
Surface Finish	vMATFTG> define KADJ parameters
DigimatSN	vMATFTG> select MAT and DOF files
Spotweld SN	vMATFTG> define SN0, SN1 and SN2 parameters
Seamweld SN	vMATFTG> define SNBR0 and SNBR1 parameters
MultiSN	vMATFTG> TABLE> import several SN curves via a multiple SN CSV file.
Table (EN / SN)	vMATFTG> define material entry via a table definition or XML Material file
Cycle	vMATFTG> define CYCLIC fields including K' and n'
Metallic (MMPDC)	vMATFTG> define MMPDS parameters

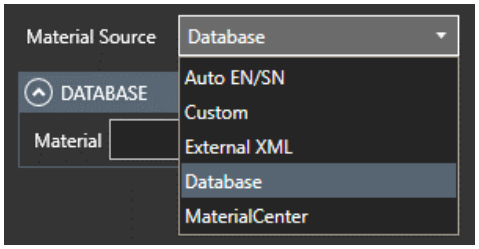
NOTE: The User can see a preview of the material curve by selecting the preview button show above.

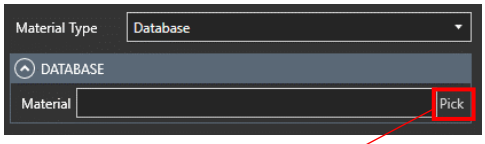


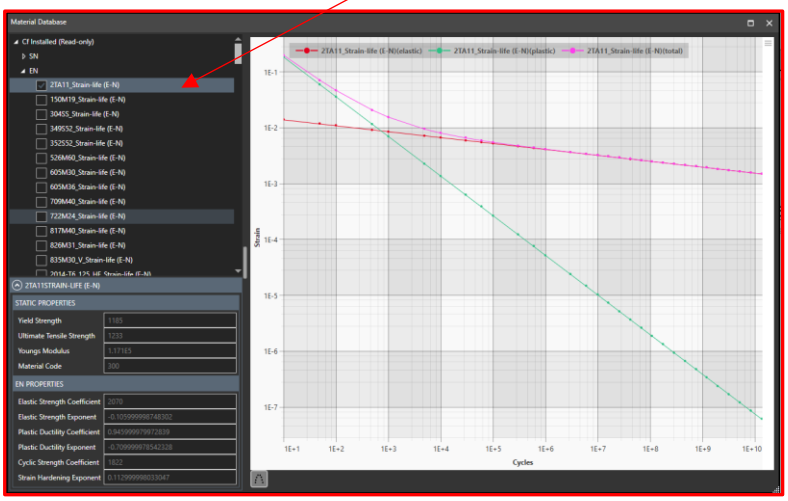
Brief Introduction to the Materials Database

As mentioned above, the user has the option to select a material from the Material Database. To see this option the User must select the “Database” option from the Material Type dropdown box.

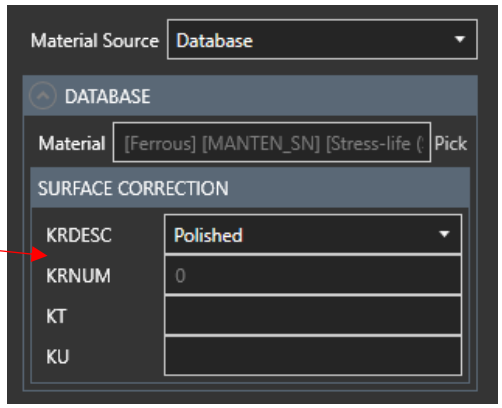
After selecting the “Database” option, the User must select PICK, select SN or EN materials list and then choose from the available materials list. Note that the data and graph for the chosen material will also be displayed.







NOTE: Once the material is picked, the User is given options to modify various Surface Correction parameters.

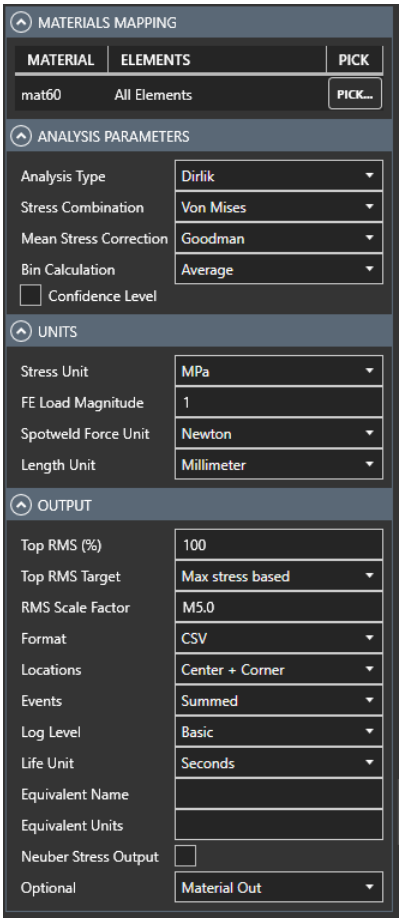


The Database is an XML database that can be edited with any text edition. The MASTER database is stored in the directory C:\Program Files\MSC.Software\CAEfatigue\<ver>\Windows\Resources\Materials > **cfv_materials_db.xml**. A copy of the file is also provided in the **Materials** folder within the TPL directory.

```
<Object ID="STEEL 1045 (2297)" ParentID="" Type="ENMatData" Category="Ferrous">
<property Name="Reference" Value="SAE J1099 - June 1998"/>
<property Name="E" Units="MPA" Value="205000"/>
<property Name="ef'" Value="1.379"/>
<property Name="K" Value=""/>
<property Name="K1C" Value=""/>
<property Name="K'" Units="MPA" Value="3083"/>
<property Name="Material Type" Value="99"/>
<property Name="Nc" Value="2.00E+08"/>
<property Name="SEc" Value="0"/>
<property Name="SEe" Value="0"/>
<property Name="SEp" Value="0"/>
<property Name="sf'" Units="MPA" Value="5813"/>
<property Name="UTS" Units="MPA" Value="2297"/>
<property Name="YS" Value=""/>
<property Name="b" Value="-0.154"/>
<property Name="c" Value="-1.0821"/>
<property Name="me" Value=""/>
<property Name="mp" Value=""/>
<property Name="n" Value=""/>
<property Name="n'" Value="0.075"/>
</Object>
```

Brief Introduction to the Analysis Options

Frequency Analysis Definition:



The **Frequency Analysis box** is used to setup the fatigue analysis including a number of analysis options found in the vFTGDEF entry.

Entries / Fields in **Materials** – Process Flow compared to Control File:

Use this section to apply material properties to the whole model or to various groups within the model. Similar to vFTGDEF>ELSET / FSET3 combination.

Entries / Fields in **Analysis Parameters** – Process Flow compared to Control File:

Analysis Type	vFTGDEF>ATYPE
Stress Combination	vFTGPARM>COMB
Mean Stress Correction	vFTGPARM>CORR
Bin Calculation	vFTGPARM>NCALC
Confidence Level	vFTGPARM>CERTSV

Entries / Fields in **Units** – Process Flow compared to Control File:

Solver Stress	FE_UNITS>CNVRTS
Spotweld Force Unit	FE_UNITS>CNVRTF
Length Unit	FE_UNITS>CNVRTL
FE Load Magnitude	FE_UNITS>FE_L_MAG

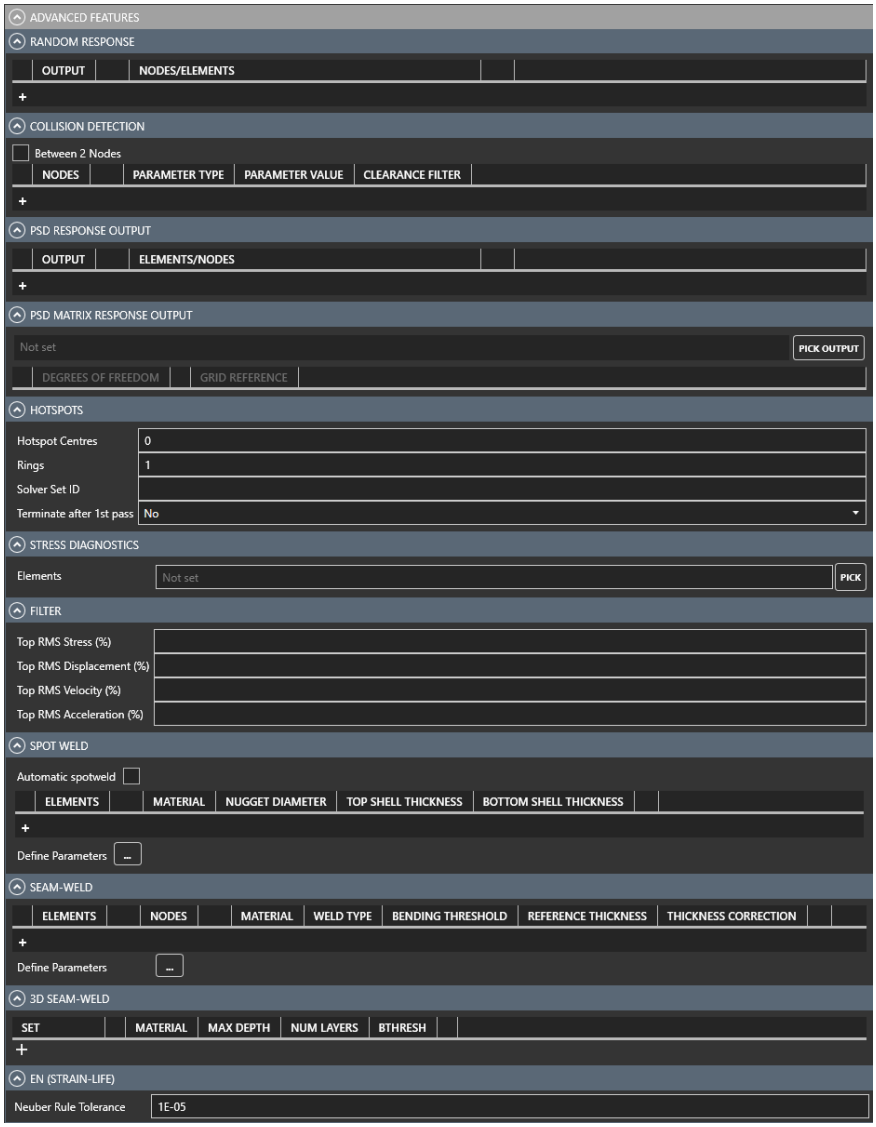
Entries / Fields in **OUTPUT** – Process Flow compared to Control File:

Top RMS	vFTGDEF>TOPRMS
Top RMS Target	Options: Max Stress Based and UTS Based
RMS Scale Factor	vFTGPARM>MAXPEAK
Format	vIBFAT>OFILTYPE
Layers	n/a
Locations	vIBFAT>GIDOUT
Events	vFTGSEQ>EVNTOUT
Log Level	vIBFAT>LOGLVL
Life Unit	vFTGSEQ>DURATION UNITS
Equivalent Name	vFTGSEQ>UNITS>EQNAME
Equivalent Units	vFTGSEQ>UNITS>EQUIV
Neuber Stress Output	vFTGPARM>CYCPROC>TRSTOUT
Optional	vFTGPARM> DEBUG

Note: **Neuber Stress Output** is turned off by default when doing a Frequency Domain analysis but is always calculated when doing a Time Domain analysis. If you wish to compare FD to TD analysis, please select the Neuber Stress Output checkbox for the FD analysis to ensure like-for-like comparison between the FD and TD.

Show Advanced Features

Both the Frequency Domain and Time Domain analysis offers additional “advanced features”, which are described below for Frequency Domain.



Random Response

Random Response provides output in the form of displacement, velocity, acceleration, force and Monitor Point force. This box is equivalent to the ABSRESP entry and contains the same parameters found in the Control File view.

Collision Detection

Random Response provides the probability of collision between 2 groups of nodes or 2 individual nodes. This is box is equivalent to the RELRESP and RELRESP2 entries and contains the same parameters found in the Control File view.

PSD Response Output

PSD Response Output provides PSDs at grids and/or elements in the form of displacement, velocity, acceleration, stress, force and Monitor Point force. This box is equivalent to the PSDOUT entry and contains the same parameters found in the Control File view.

PSD Matrix Response Output

PSD Matrix Response Output provides PSDs at grids and/or elements in the form of displacement, velocity, acceleration and Monitor Point force. This box is equivalent to the PSDMOUT entry and contains the same parameters found in the Control File view.

Hotspots

Hotspots provides the highest stress areas on the model. This box is equivalent to the HOTSPOTS entry and contains the same parameters found in the Control File view. Set Count=0 to disable this feature.

Stress Diagnostics

Stress Diagnostics provides the ability to review the influence of each channel within a multi-input loading analysis on any number of elements. This box is equivalent to the DIAGNOS entry and contains the same parameters found in the Control File view.

Filter

Filter provides the ability to adjust the RMS value that will be used to filter the analysis output results for stress, displacement, velocity and acceleration. This box is equivalent to the FILTER entry and contains the same parameters found in the Control File view.

Spot Weld – also available for Time Domain

Spot Weld provides the ability to calculate results for spot welds in the model. The User can select “Use default spot weld dimensions and materials”, which will automatically find all HEXA elements in the modal and apply the default spot weld material to those HEXA elements. The default spot weld material property is listed in the appendix defining material data. Note: The User may have HEXA elements in the model that are not spot welds. In this case, the User must define which HEXA element are spot welds. This box is equivalent to the SPOTWELD entry and contains the same parameters found in the Control File view.

Seam Weld – also available for Time Domain

Seam Weld provides the ability to calculate results at the seam weld TOE. This box is equivalent to the SEAMW entry and contains the same parameters found in the Control File view.

Not yet implemented: ***3D-Seam Weld – also available for Time Domain***

3D Seam Weld provides the ability to calculate results at the seam weld TOE across solid elements. This box is equivalent to the SEAMW3D entry and contains the same parameters found in the Control File view.

EN (Strain – Life)

This section is used to override the default setting for the tolerance use to do a Neuber correction to FEA stresses. (vFTGPARM> TOLER field). Current default setting is 1.0 e5.

Surrogate Loads

This section is used to conduct a Surrogate Loads analysis, using the stress target file from a previous Cf analysis (like done in TPL177) or by using a stress or acceleration target file obtained from a TIME2PSD analysis of time history responses from a strain gauge (converted to stress) or accelerometer on a test vehicle.

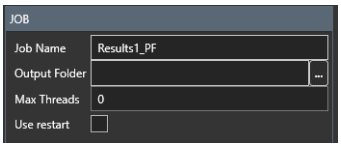
Stress Concentration Factors (Kt)

This section is used to apply KT Material Property scale and offset factors to a group / groups of elements. See KTDATA in the QRG.

Factor of Safety (FOS) – also available in Time Domain

This section is used to conduct a factor of safety evaluation using LIFE in repeats. The FOS results is a multiplier on stress that can be applied before the LIFE values reduce below the require value set as the Target.

RUN:



The **RUN** box is used to execute the analysis and to provide the output data with an optional file name.

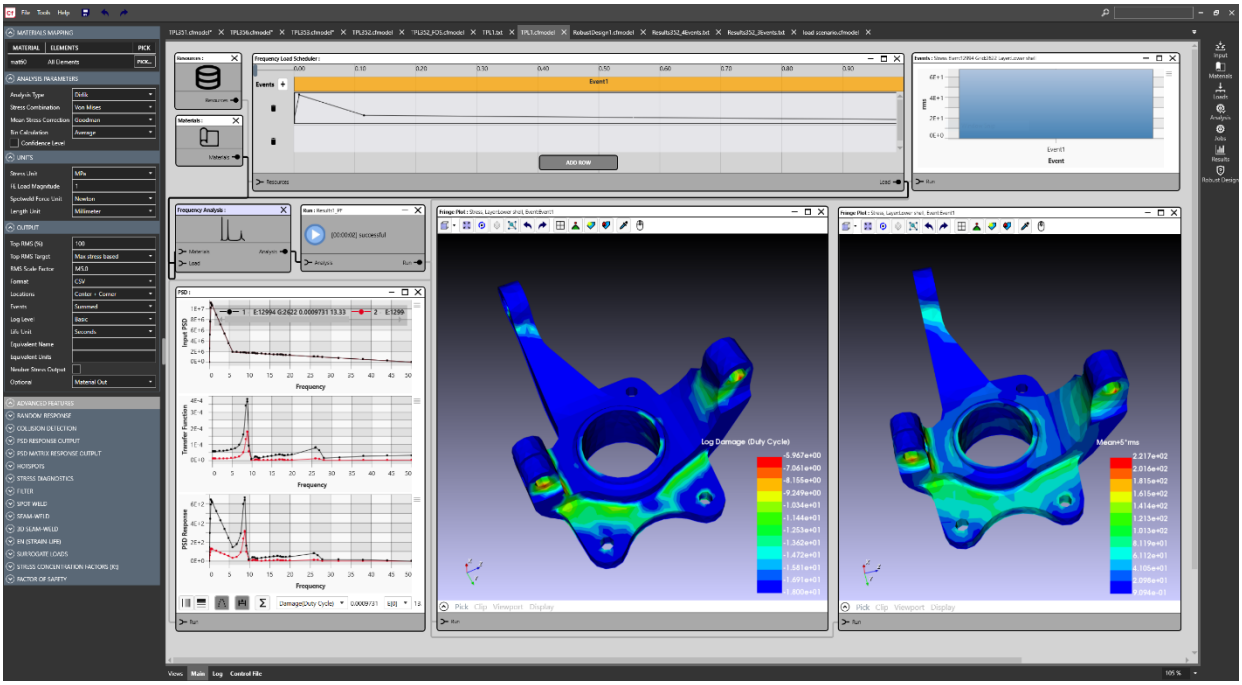
Entries / Fields in Process Flow compared to Control File:

Job Name	vIBFAT>JOBNAME
Output Folder	vIBFAT>OPTIONS>OUTDIR
Max Threads	PARALLEL
Use Restart	Restart

Viewing Results – PSD Plotter, Event Plotter and Fringe Viewer

The various result boxes available in the Process Flow are the same as the Result outputs available from in a Control File run. However, unlike the Control File run, the User can have duplicate windows of the same type of output box i.e., adding 2 Fringe Plot boxes within the same Process Flow run (see figure below).

Figure 7-3 TPL1 (Process Flow) with 2 Fringe Plot views



Additional Viewing Options

After the job has run, the User will have access to addition viewing option as show at the bottom of the image above.

Main – displays the Process Flow view. This can also be done by selecting the job tab at the top.

Log – displays the LOG file window that is identical to that seen in the Control File view.

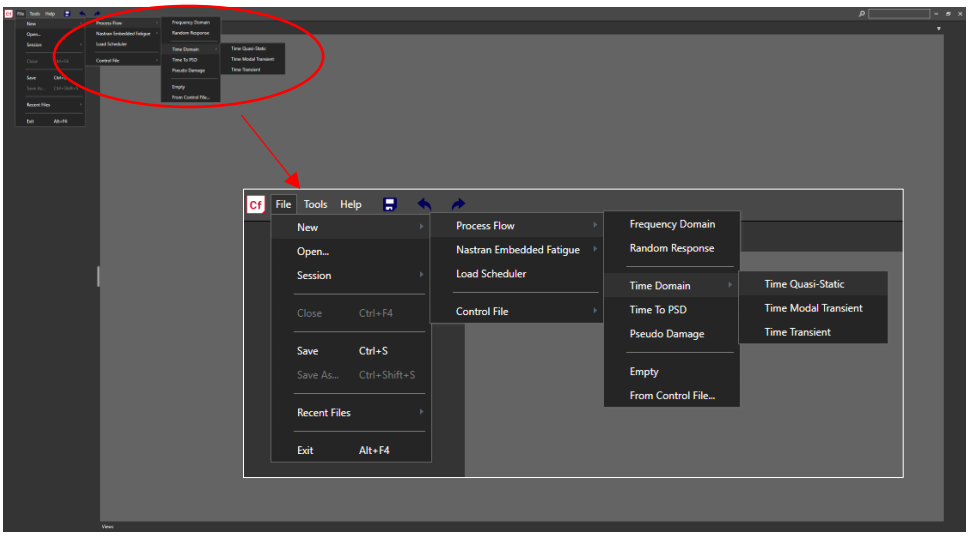
Control File – displays a READ ONLY view of the autogenerated Control File that is created from the Process Flow.

A Time Domain LQSTATIC, LMTRANS Analysis – TPL358 Example

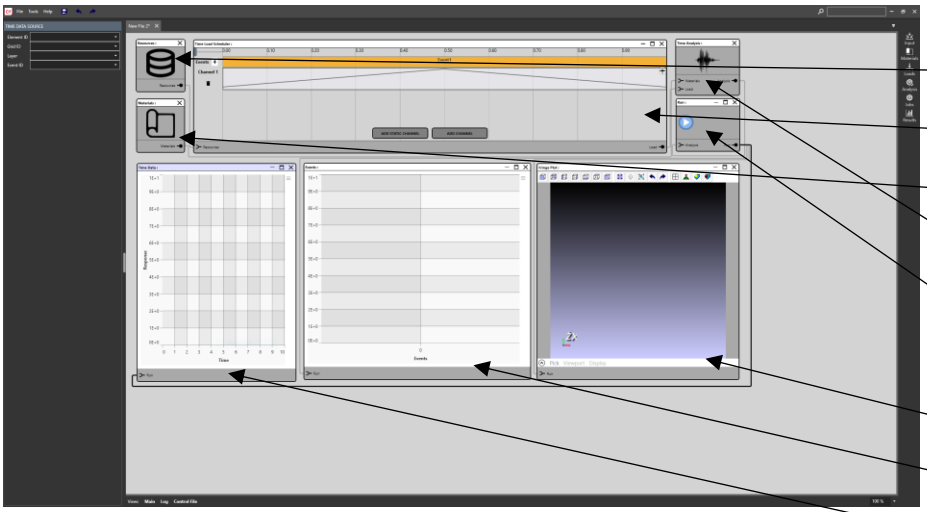
TPL358 in this User Guide provides a process flow example for a linear quasi-static time domain analysis, frequency domain PSD analysis and a Time Modal Transient time domain analysis. We will focus on the static and modal TD analyses.

To start either examples, the User must select **New> Process Flow> Time Domain** to get a NEW Process Flow Time Domain template for either the **Time Quasi-Static** analysis or **Time Modal Transient** analysis, from the File menu on the main start page.

Note: The Time Transient approach is also available for the Process Flow interface.



Below is the template screen for the **Time Quasi-Static** analysis.



Input:

Resources

Time Load Scheduler

Materials

Time Analysis

Run (Execute Job)

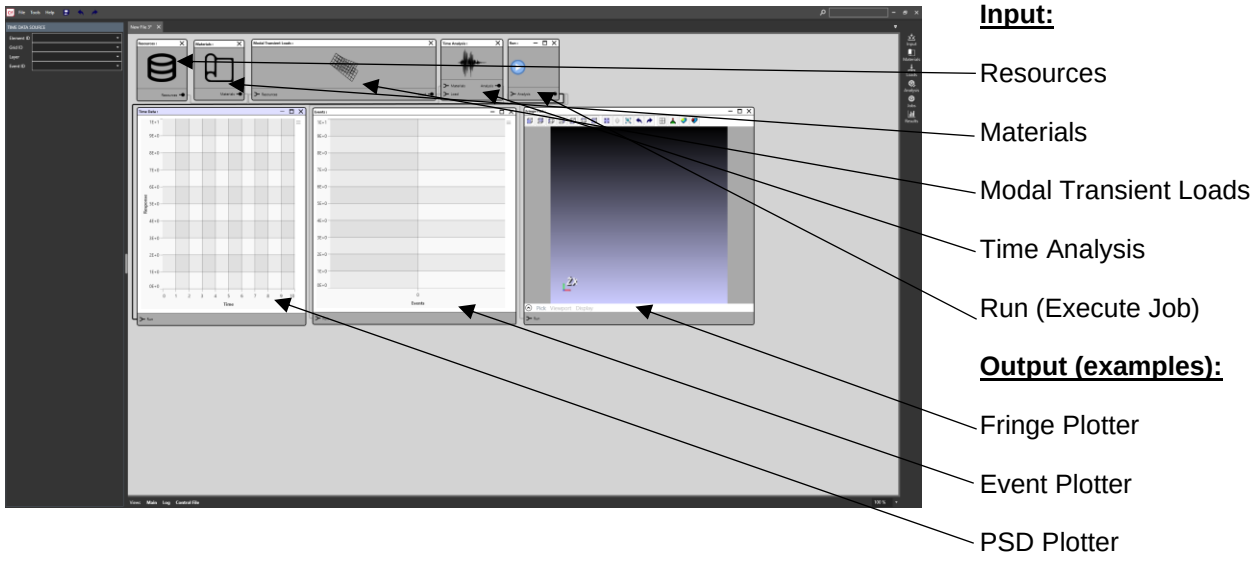
Output (examples):

Fringe Plotter

Event Plotter

PSD Plotter

Below is the template screen for the **Time Modal** analysis.



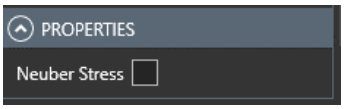
Each box in the Process Flow requires entries and fields to be populated. Below are the boxes (entries and fields) that are required to the **Time Quasi-Static** analysis in TPL358. See this example for how a **Time Modal** analysis is set up.

Resources:

The **Resources** box is used to list the input files that are available for the fatigue analysis. This list would contain Solver file, RSP files, PSDM files and Punch file (as necessary). If any of the solver files contain nonlinear stresses, the User should select the **NEUBER STRESS** check box (shown below), which will remove the Neuber Correction process step.

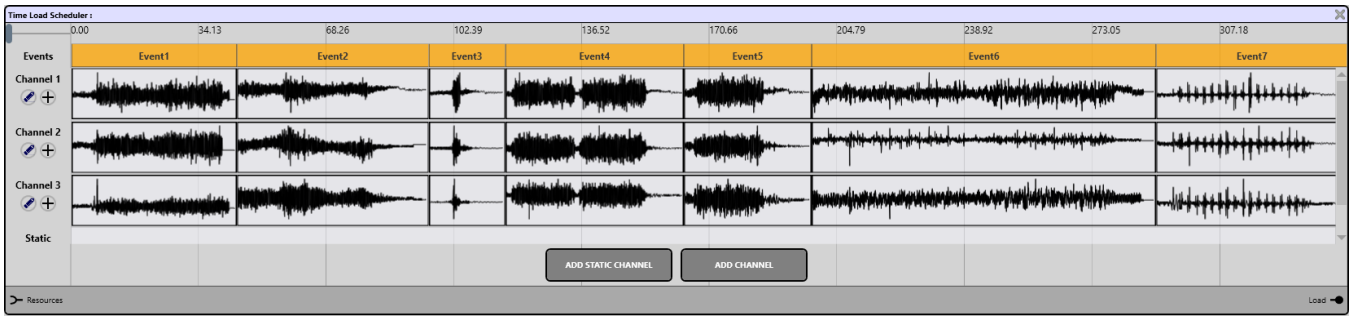
For the LQSTATIC analysis in TPL358, the only resource files needed is the Nastran OP2 file knrandom.op2 and BDF file knrandom.bdf.

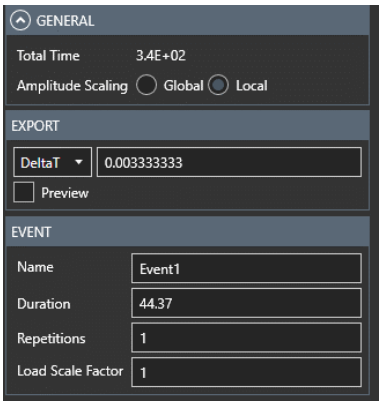
For other jobs these files could be Nastran OP2, Nastran HDF5, Ansys RST, Abaqus ODB, OptiStruct OP2, and PSDM files (for FD multiple input analysis).



If any of the Solver stress files contain non-linear stress information (i.e., already Neuber corrected stress) then the User should select Neuber Stress box to indicate that that solver stress is already a Neuber Stress i.e., not an FEA stress.

Time Load Scheduler (select an Event box):



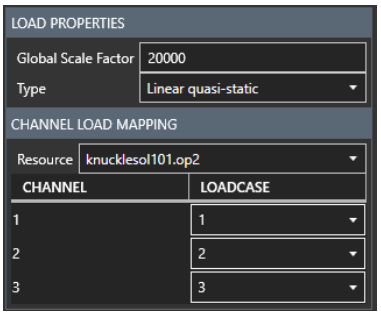


The loading time histories required for this example are loaded into the Time Load Scheduler. The import format can be RSP or CSV with a start time of 0 seconds.

Once the time histories are loaded, the Event box of the Time Load Scheduler is used to set export parameter as well as provided viewing options to see the time history before export or to see the time history using a global Y axis scale. See ***Introduction to the Load Scheduler*** in this document for more information on the Time Load Scheduler and importing time history files.

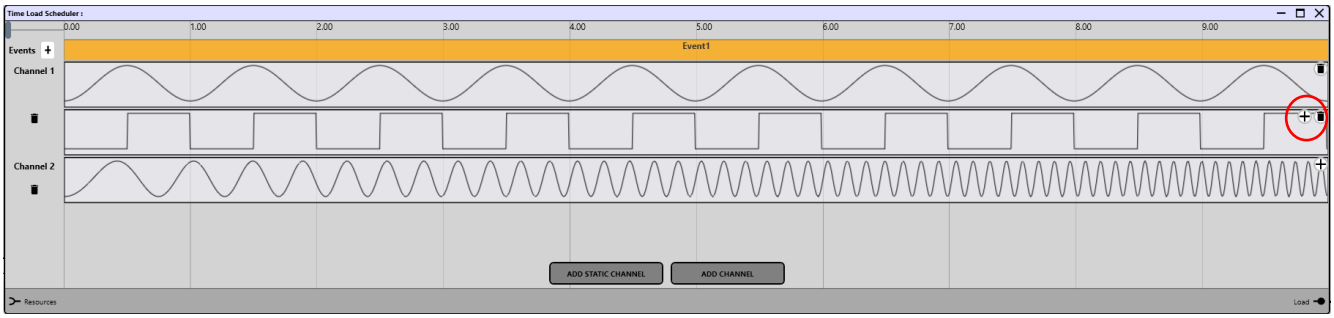
Entries / Fields in Process Flow compared to Control File:

Name	vFTGEVNT>NAME>EVNTNAME
Duration	vFTGSEQ>EVT1
Repetitions	vFTGSEQ>EVT1 (when “Use repeats for events” is selected i.e., NREP=YES)
Load Scale Factor	Loading on each Event – not individual channels



Global Scale Factor	vFTGSEQ>TSCALE
Type	vFTGLOAD>TYPE
Resource	OP2MAP>FILENAME
Channel / Loadcase	OP2MAP>CFLCID / OP2LCID

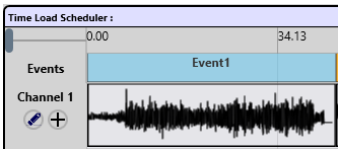
Special Case for adding SUB-CHANNELS

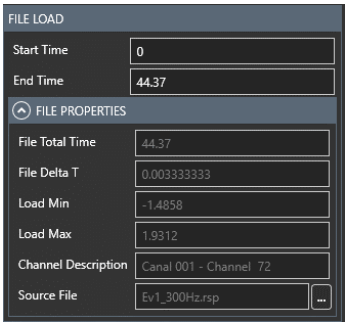


There may be situations where the User wants to “add together” several different time signals within 1 channel. Above is an example where Channel 1 contains 2 time signals i.e., one sine wave and one square wave). Channel 2 contains a sine sweep with no additional subchannels.

By selecting the “+” sign (on right side of time history), the User is able to create a SUBCHANNEL directly under the exiting Channel time signal. A second time signal can then be added to the subchannel. The 2 time signals in Channel 1 will be added together in the analysis to create a combined single time signal for Channel 1. Channel 2 will remain as a sine sweep because no additional subchannels were defined for this channel.

Time Load Scheduler (select an Event box):



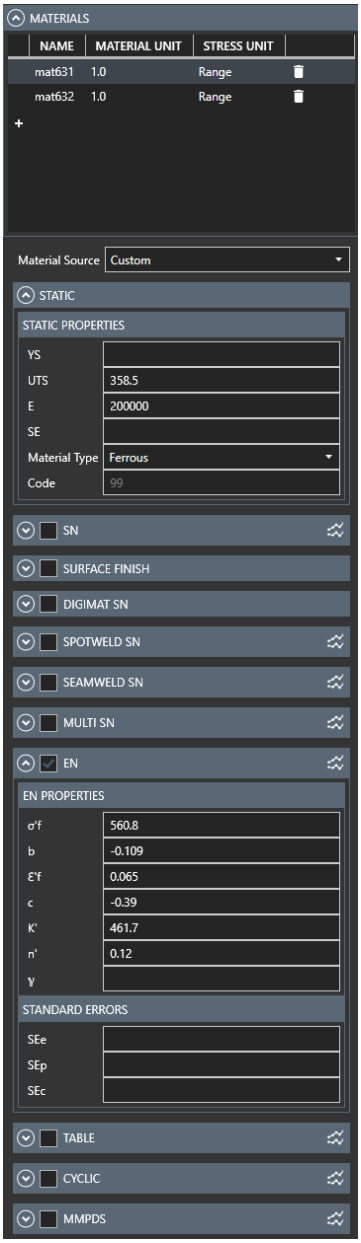


The Channel box within an Event is used to provide information about the RSP / RPC file that was loaded into the Channel. See ***Introduction to the Load Scheduler*** in this document for more information on importing files within the load scheduler.

The entries / fields in this box cannot be changed. It is for information only and provides data on the individual channel selected. In this case, Event 1 / Channel 1 was imported from the Ev1_300Hz.rsp file. This RSP file contains 3 channels; hence, 3 channels are show in Time Load Scheduler for Event 1.

The information for Channel 1 is shown to the left. This type of File Properties box is available for all event / channel combinations.

Materials:



The **Materials** box is used to enter the fatigue material properties. For this example, an EN description was entered along with the Static material properties.

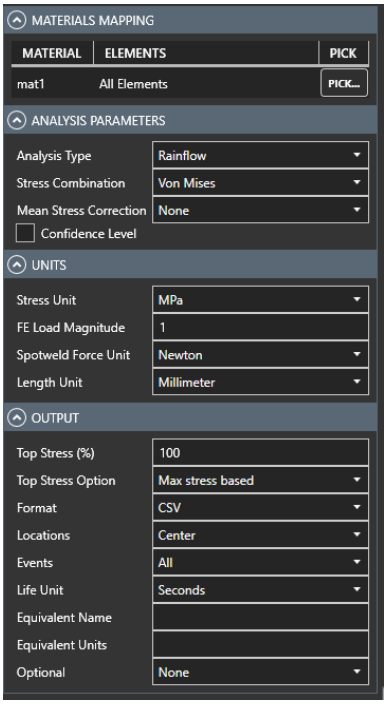
Note: the material information would be entered in several different ways, including the selection of material from a Material Database.

Entries / Fields in Process Flow compared to Control File:

Material Name	n/a
Material Unit	vMATFTG>CNVRTM
Stress Unit	vMATFTG>AMP_RNG
Material Type	Define an SN/EN material, select AutoSN or AutoEN, select internal or external XML database
Material Property:	Define any of the items below.
Static	vMATFTG> define Static parameters
Cycle	vMATFTG> define CYCLIC parameters including K' and n'
EN / SN	vMATFTG> define EN or SN parameters
Surface Finish	vMATFTG> defines surface finish parameters
Digimat SN	n/a (link to Digimat material information)
Spotweld SN	vMATFTG> define SN0, SN1 and SN2 parameters
Seamweld SN	vMATFTG> define SNBR0 and SNBR1 parameters
Multi SN	vMATFTG> define SN parameters
Table (EN / SN)	vMATFTG> define material entry via a table definition or XML Material file
Cyclic	vMATFTG> define K' and n' parameters
Metallic (MMPDS)	vMATFTG> define MMPDS parameters

NOTE: The User can enter table values (as shown) or enter a CSV or TXT file that has a complete vTABRND, vTABRND1 or TABRND1 or an XML definition. See "Quick Reference Guide" for more information.

Time Analysis:



The **Time Analysis box** is used to setup the fatigue analysis including a number of analysis options found in the vFTGDEF entry.

Entries / Fields in MATERIAL MAPPING

Use this section to apply material properties to the whole model or element sets within the model (similar to ELSET/FSET3).

Entries / Fields in **Analysis Parameters** – Process Flow compared to Control File:

Analysis Type	vFTGDEF>ATYPE
Stress Combination	vFTGPARM>COMB
Mean Stress Correction	vFTGPARM>CORR
Confidence Level	vFTGPARM>CERTNTY

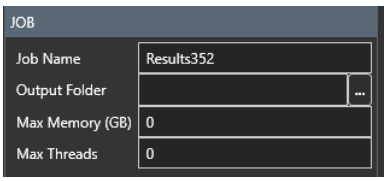
Entries / Fields in **Units** – Process Flow compared to Control File:

Solver Stress	FE_UNITS>CNVRTS
Spotweld Force Unit	FE_UNITS>CONVRTF
Length Unit	FE_UNITS>CONVRTL
FE Load Magnitude	FE_UNITS>FE_L_MAG

Entries / Fields in **OUTPUT** – Process Flow compared to Control File:

Top Stress (%)	vFTGDEF>TOPSTR
Top Stress Option	Options: Max Stress Based, M, S and U
Format	vIBFAT>OFILTYPE
Locations	vIBFAT>GIDOUT
Events	vFTGSEQ>EVNTOUT
Life Unit	vFTGSEQ>DURATION UNITS
Equivalent Name	vFTGSEQ>UNITS>EQNAME
Equivalent Units	vFTGSEQ>UNITS>EQUIV
Optional	vFTGPARM> DEBUG

RUN:



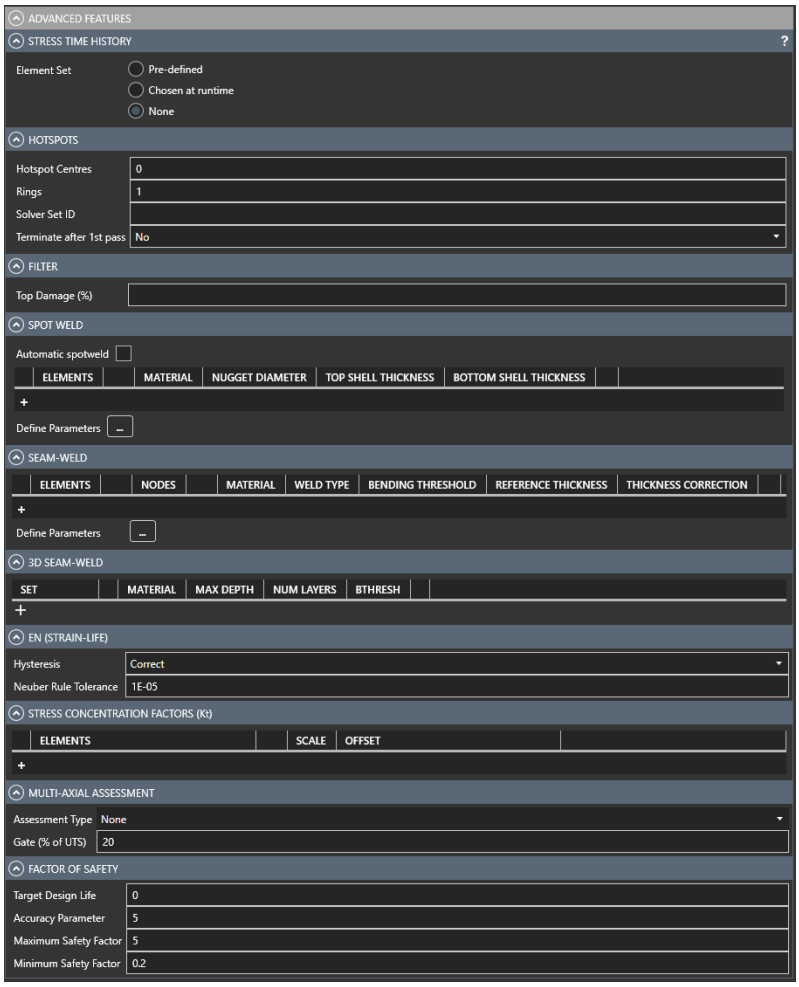
The **RUN** box is used execute the analysis and to provide the output data with an optional file name.

Entries / Fields in Process Flow compared to Control File:

Job Name	vIBFAT>JOBNAME
Output Folder	vIBFAT>OPTIONS>OUTDIR
Max Memory (GB)	MEMORY
Max Threads	PARALLEL

Show Advanced Features

Both the Frequency Domain and Time Domain analysis offers additional “advanced features”, which are described below for the Time Domain.

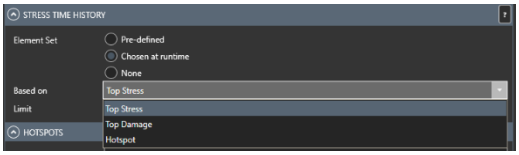


Stress Time History

This feature is called TIMEOUT in the Control File view and was previously in the Output Section. TIMEOUT has been expanded to allow the user to either pre-define element(s) for output or to let CAEfatigue choose the element for output based 3 different filters.

Top Stress: same as defined in the Output section. By default, it is based on elements falling within 100% of Max Stress.

Top Damage / Hotspot: same as defined in the FILTER and HOTSPOT sections. By default, a Top Damage filter or a Hotspot filter must be set up as part of the TIMEOUT feature.



Hotspots

Hotspots provides the highest stress areas on the model. This box is equivalent to the HOTSPOTS entry and contains the same parameters found in the Control File view. Set Count=0 to disable this feature.

Filter

Filter provides the ability to select a damage value that will be used to filter the analysis output results. This box is equivalent to the Time Domain FILTER entry found in the Control File view.

Spot Weld – also available for Frequency Domain

Spot Weld provides the ability to calculate results for spot welds in the model. The User can select “Use default spot weld dimensions and materials”, which will automatically find all HEXA elements in the modal and apply the default spot weld material to those HEXA elements. The default spot weld material property is listed in the appendix defining material data. Note: The User may have HEXA elements in the model that are not spot welds. In this case, the User must define which HEXA element are spot welds. This box is equivalent to the SPOTWELD entry and contains the same parameters found in the Control File view.

Seam Weld – also available for Frequency Domain

Seam Weld provides the ability to calculate results at the seam weld TOE. This box is equivalent to the SEAMW entry and contains the same parameters found in the Control File view.

Not yet implemented: ***3D-Seam Weld – also available for Time Domain***

3D Seam Weld provides the ability to calculate results at the seam weld TOE across solid elements. This box is equivalent to the SEAMW3D entry and contains the same parameters found in the Control File view.

EN (Strain – Life)

This section is used to override the default setting for the way hysteresis loops are placed in the rainflow cycle calculation (vFTGPARM> CYCPRO> HYST). The current default setting is CORR. This section is also used to override the tolerance use to do a Neuber correction to FEA stresses. (vFTGPARM> TOLER field). The current default setting is 1.0 e5.

Stress Concentration Factors (Kt)

This section is used to enter KT material parameters for applying a scale factor and offset factor to any group or groups of elements.

Factor of Safety (FOS) – also available in Time Domain

This section is used to conduct a factor of safety evaluation using LIFE in repeats. The FOS results is a multiplier on stress that can be applied before the LIFE values reduce below the require value set as the Target.

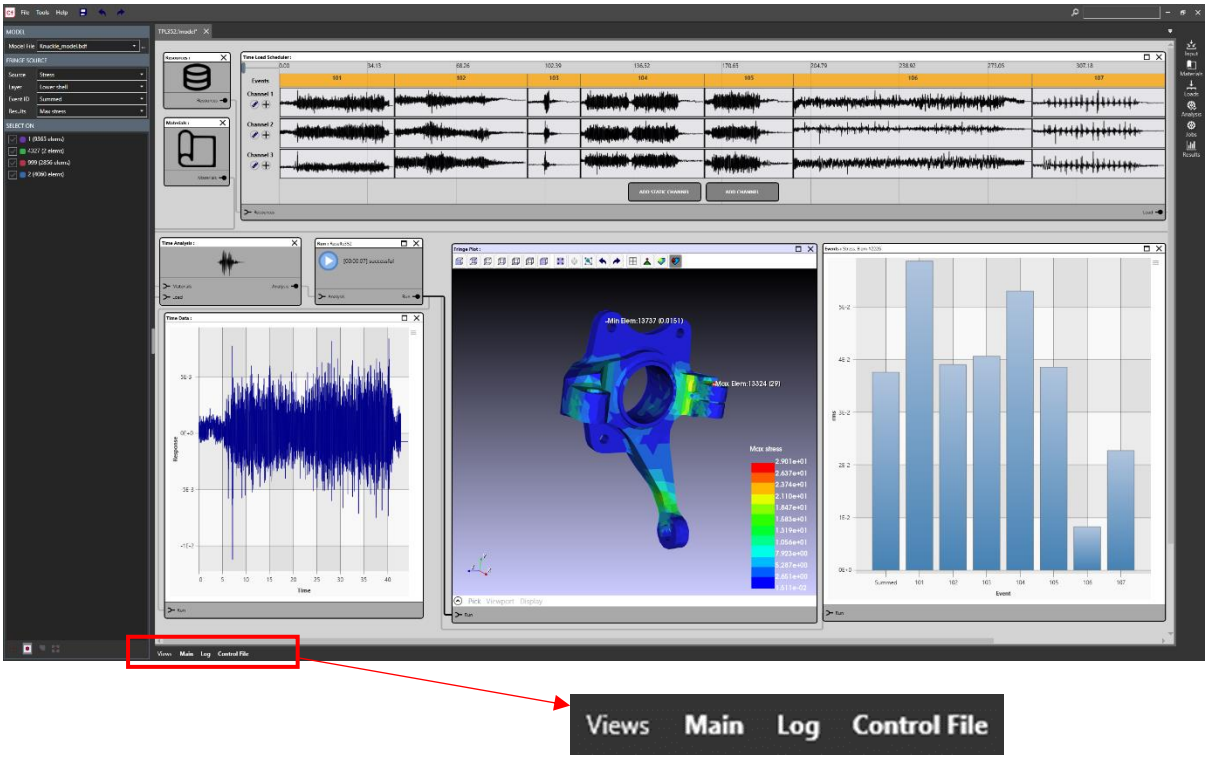
Viewing Results – Same as Control File view

The various result boxes available in the Process Flow are the same as the outputs boxes available from within a fatigue Control File run. See the **Fatigue Control File – Viewing Results** sections for more detailed information.

However, unlike the Control File run, the User can have duplicate windows of the same type of output box. Below is the standard output set of results that are usually shown i.e., Response Time Signal output, Fringe Plotter and Event Plotter for a time domain analysis. Any number of plotters can be added to the workspace as desired by the User.

NOTE: The Time Data Plotter provide FEA stress information only. However, the remaining output boxes will provide Neuber stress data provided EN material is provided (or SN with CYCLIC data).

Figure 7-4 TPL352 (Process Flow) with standard output views



Additional Viewing Options

After the job has run, the User will have access to addition viewing option as show at the bottom of the image above.

Main – displays the Process Flow view. This can also be done by selecting the job tab at the top.

Log – displays the LOG file window that is identical to that seen in the Control File view.

Control File – displays a READ ONLY view of the auto-generated Control File that is created from the Process Flow.

8 | INTRODUCTION TO THE TIME2PSD CONTROL FILE VIEW

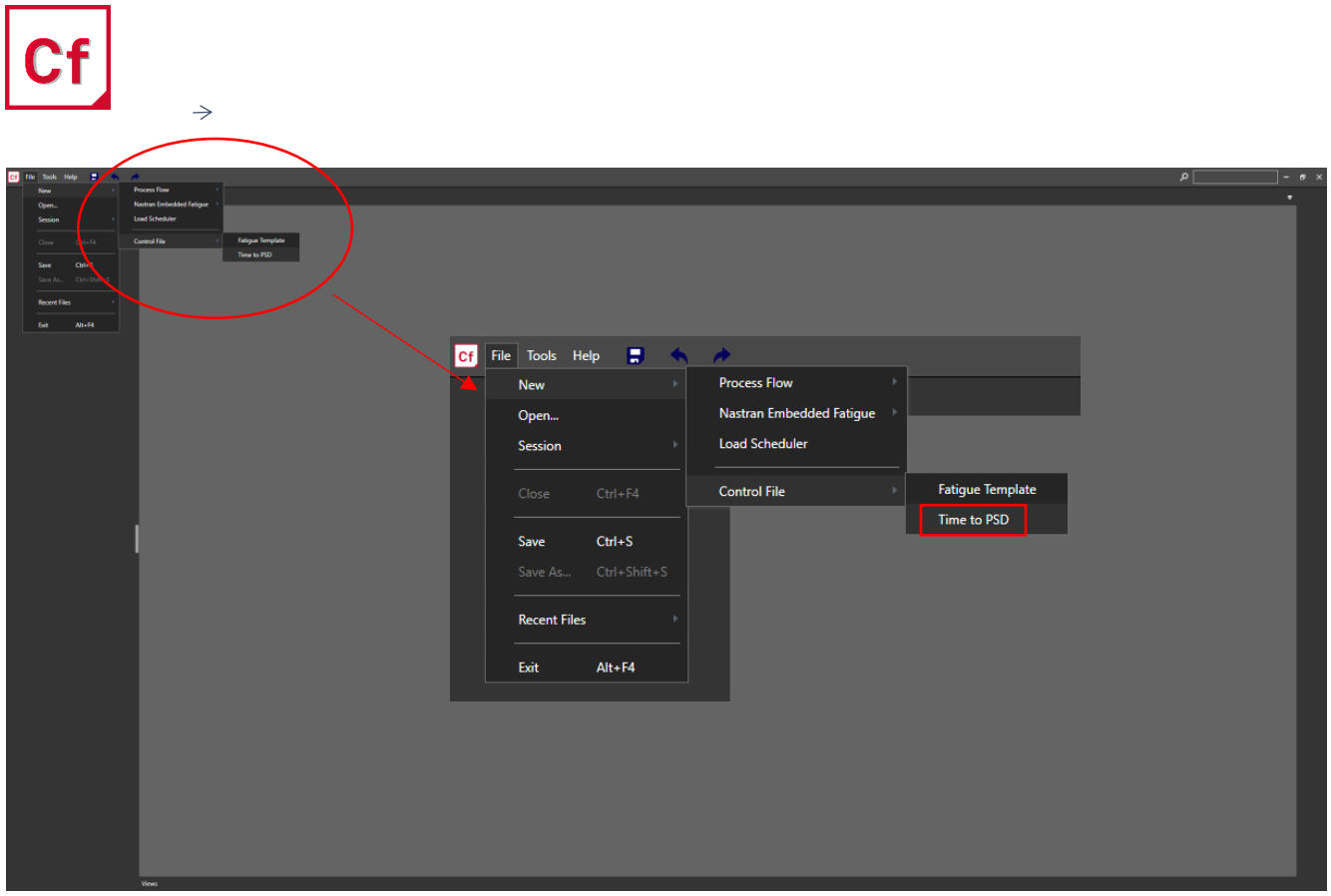
NOTE: The images shown in this section reflect CAEfatigue v2020 and may change slightly with future releases.

CF includes a graphical user interface application for executing the program. For the interface to run successfully, ".NET 4.5" (or later – see latest installation guide) must be installed on the system (this should normally be the case).

Overview and Introduction to the TIME2PSD Wizard

To start CF, click on the desktop Cf icon (see below) or find the ProcessApp.exe file for the program in the default installation directory "C:\Program Files\MSC.Software\CAEfatigue\<ver>\Windows\".

This will open Cf in interface application window as shown below. To highlight the different features of the TIME2PSD toolset we will open and run a job. To do this we can either create a NEW TIME2PSD file as shown below to generate the initial base structure of our TIME2PSD file; or, we can select ""FILE>OPEN" to pick an existing TIME2PSD control file that was already created, which can then be edited to match the analysis we are trying to accomplish. In the image below, we selected OPEN > TPL85.txt. This will open the test problem in the editor window.

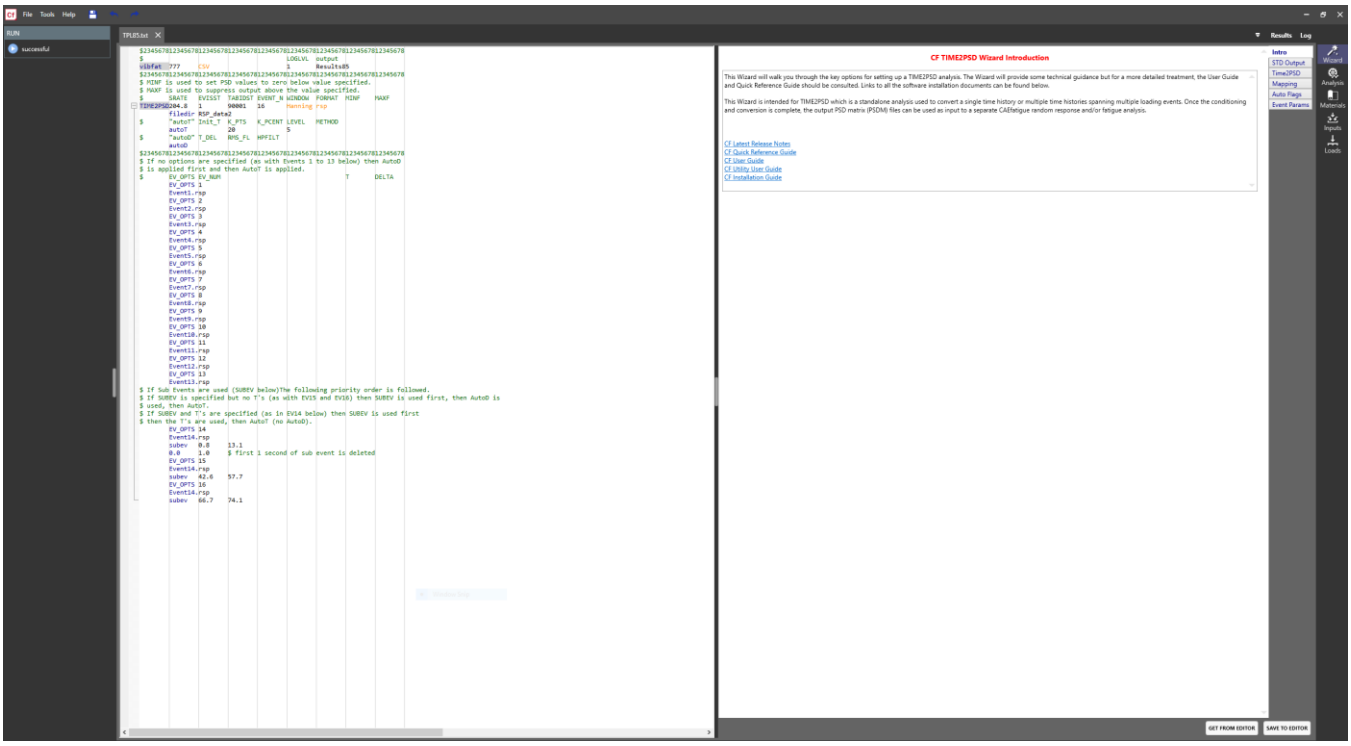


TIME2PSD Wizard

Select the **Wizard** icon to open the Wizard toolbox as shown on the right-hand side of the window below. We will use the Wizard toolbox to examine the parameters chosen for TPL85 as well as explain many of these parameters and their interaction.

NOTE: If the User wishes to bypass the Wizard, they can select the ANALYSIS menu from below the Wizard to get access to ENTRY / FIELD combinations for direct drag and drop insertion into the Control File editor window (left). For a TIME2PSD analysis, only the vIBFAT and TIME2PSD entries are needed.

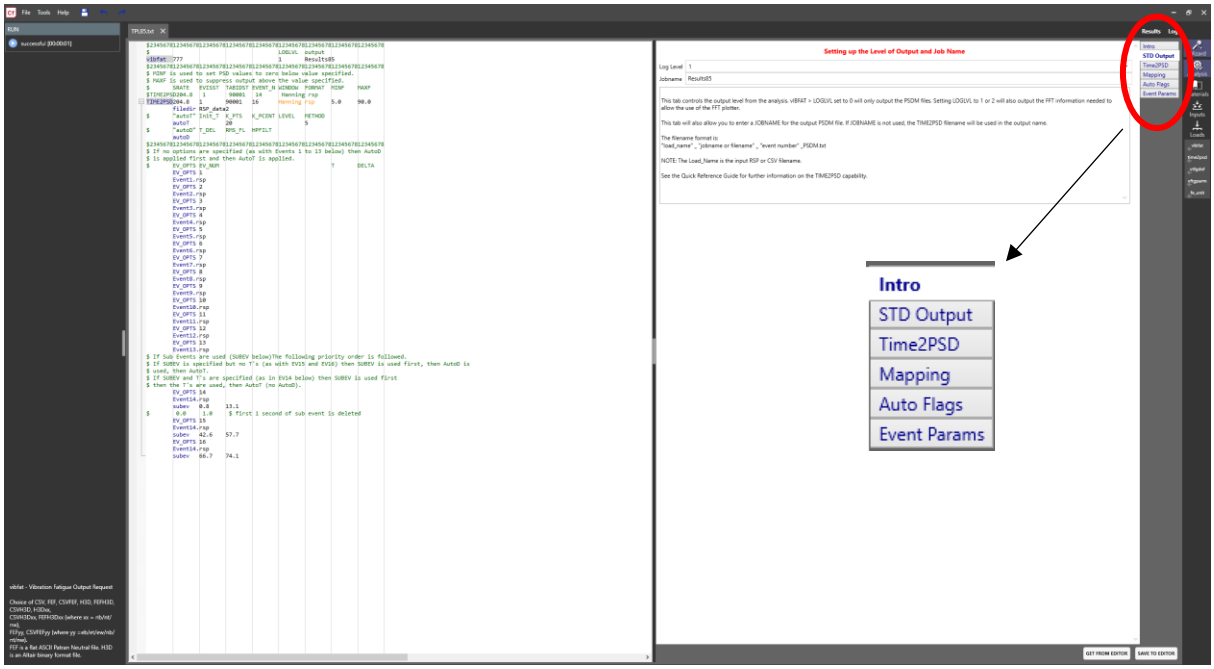
For an experienced user, this can be faster than typing in the entries and fields manually. Also, when available, the User should consider starting with an existing Test Problem as a template that closely matches the type of analysis the User wishes to do. Often existing jobs created by the User become the templates for future work. This is what we are doing for this example by starting with TPL85.



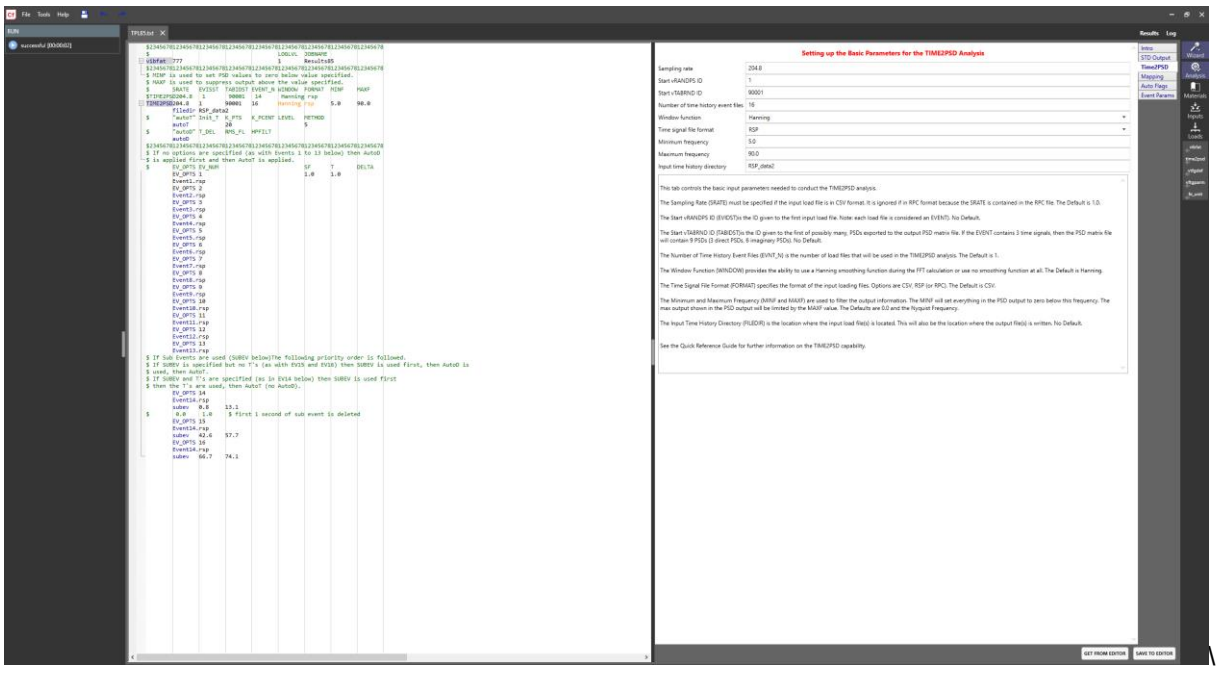
The Wizard contains tabs that are used to specify the key input parameters for executing a new job or changing an existing job. The Wizard also provides some information on the key parameters to help with the understanding. Note: The toolbox makes use of horizontal and vertical scroll bars for some tabs.

The **INTRO 0** tab (as shown above) has no parameters to specify but there are links to documentation including the User Guide, Quick Reference Guide and the Latest Release Notes.

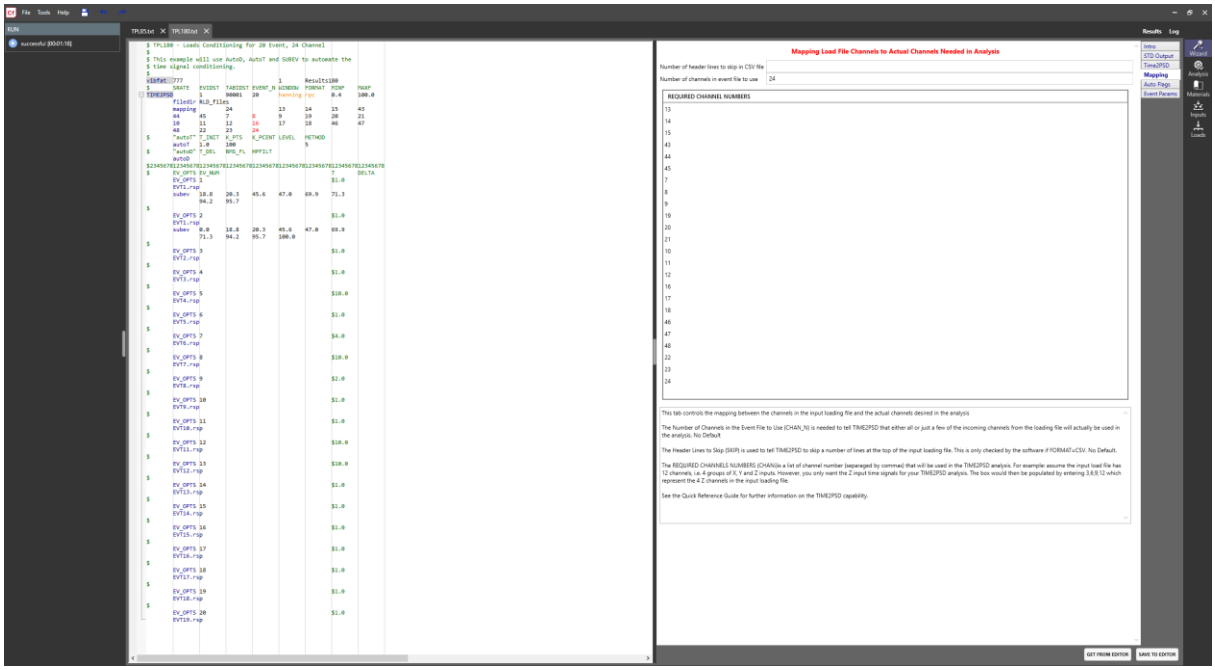
The **STD OUTPUT** tab controls the analysis output. LOGLVL=0 will output just the final PSDM files whereas LOGLVL=1 will also provide output that can be seen in the FFT Plotter, which will be shown later. The JOBNAME field allows the User to give a unique name to the output files. If JOBNAME is left blank, the TPL name will be used for the result files.



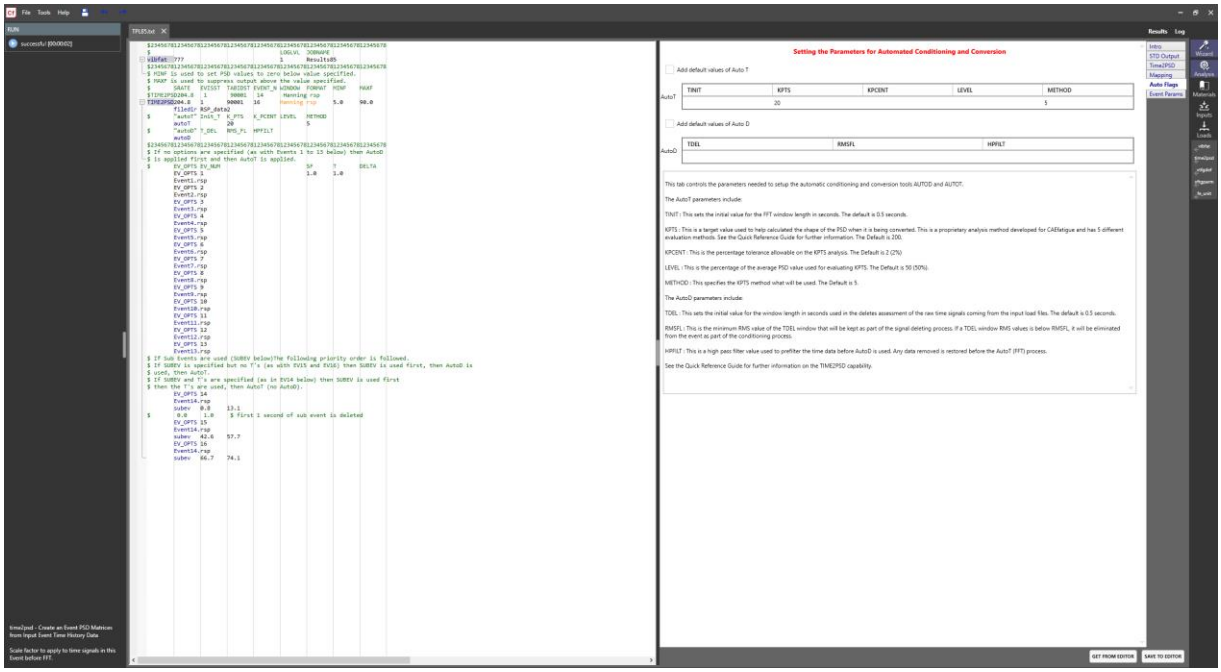
The **TIME2PSD** tab sets up most of the parameters needed to do the analysis including the number of events, the input file type and the filters that will be applied to the output PSD..



The **MAPPING** tab is used to uniquely select channel from an event that will be used in the analysis. For example, if the CSF or RSP file contains 10 channels, the User can choose to only use Channel 1, 2 and 3 instead of all 10 and the User can also list the channels in a specific order. Below is TPL180 for example.

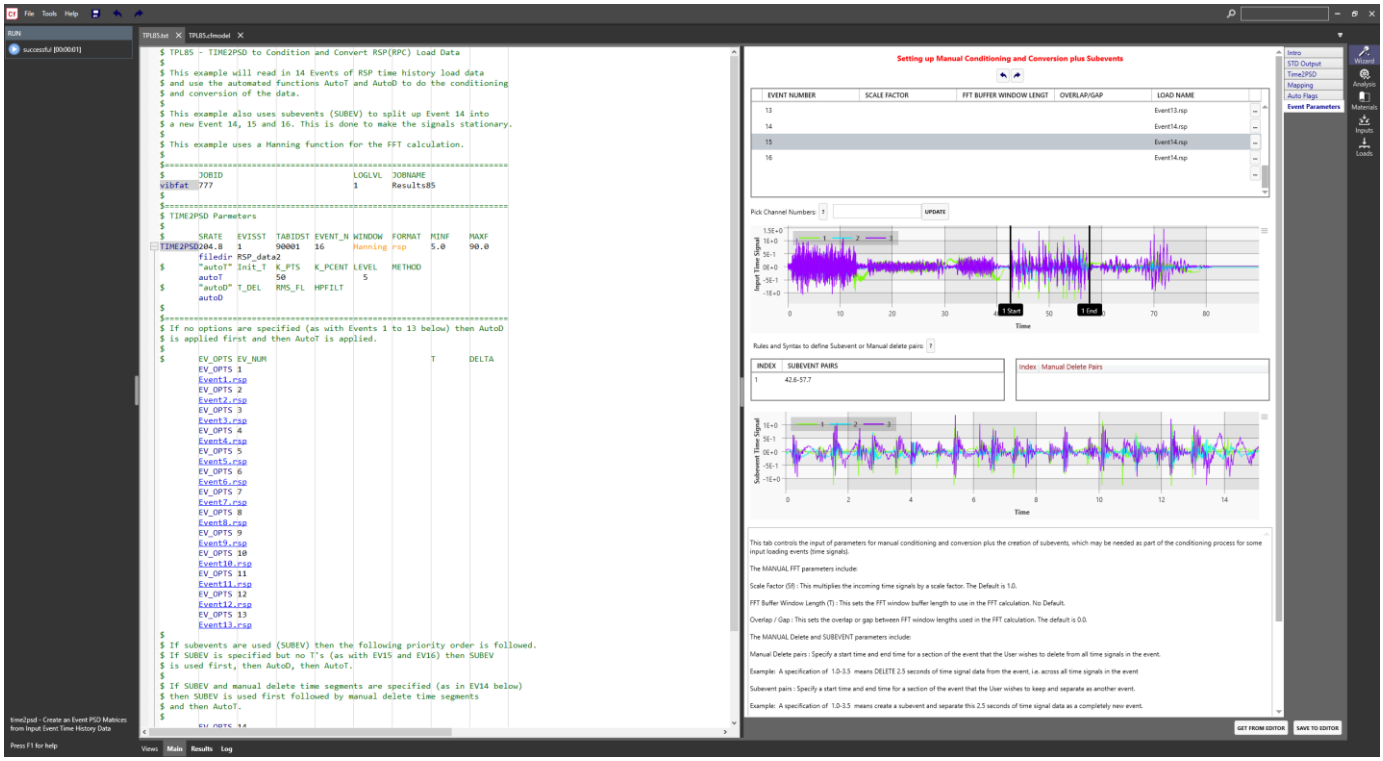


The **AUTO FLAGS** tab controls the instructions needed to use the automatic filters AUTOD and AUTOT. These filters are used to automatically remove low intensity parts of the signal (so resolve some stationarity issues) and to convert the conditioned signal into a PSD using a proprietary KPOINTS method.



The **Event Params** tab controls the manual conditioning and conversion method should the User not wish to use the AUTOD and AUTOT features. It also includes the graphical setting of the subevent (SUBEV) feature. Subevents are used to separate (split) the single time signal event into multiple events. This is often needed to resolve issues with stationarity.

In the example below, a section from Event 14 was identified as unique and nonstationary. In this case we use a subevent to separate this section from Event 14 and add it to a new Event 15.



More Information about SUBEV, AutoD and AutoT.

When working in the Frequency Domain, Cf is a random response and fatigue solver that uses an input Power Spectral Density (PSD) together with the stress transfer functions and material properties to deliver response statistics and fatigue damage. However, some Users only have time domain data to use for input. In these cases, Cf offers a unique capability called TIME2PSD that helps the User convert a time domain input file into an accurate PSD through a semi-automated process.

TIME2PSD is a pre-processing module that is run separately from Cf and will accurately convert input time histories into a PSD matrix without User input needed to "clean" time histories containing dead spots, pothole spikes, etc. Once the process is complete, the PSD is brought into Cf to be process into response and damage output.

When used in the Control File interface, the TIME2PSD feature conditions and converts the time signals into PSDs using 3 processes:

1. **Subevents**: Subevents is a pre-filtering step that allows the User to separate (split) an Event, containing any number of time histories into a number of parts that can be grouped together or, into additional events. This is highly recommended when the time history contains sections with significantly different frequency content. This type of occurrence in a time history violates the key assumption that the time history is Stationary throughout the signal. Hence, the User can split (and group) different sections of the Event or split off sections of the event into multiple new Events where the content within each new event adhere to the Stationary assumption.

2. **Deletes**: Segments of the time history data that do not significantly contribute to the damage must be removed so they are not carried into the PSD. The TIME2PSD module does this using pairs of time values that define sections of the time data that will be deleted from the original load file. These pairs of time values must be entered by the User. In the case of multiple time histories (example: multiple channels in an RPC file), channel correlation must be maintained so any deletes specified by the User for one channel must be apply simultaneously to all channels. This specification for multiple channels can be a laborious and challenging task; therefore, TIME2PSD incorporates an automatic method (*AutoD*).

3. **Definition of window length (T) and overlap (δ).** This step involves a more difficult decision for the User, which is the choice of window length and overlap for the FFT algorithm. These items are used in a "PWELCH" type calculation where multiple estimates are averaged to create the final PSD values. These values can be specified manually for each event, but this can again be challenging since too short a window length can result in a PSD with insufficient frequency resolution and too long a window length will result in poor statistical accuracy (too much scatter). TIME2PSD provides an automatic way to calculate the T and δ values (*AutoT*), which uses a proprietary evaluation process using K Points.

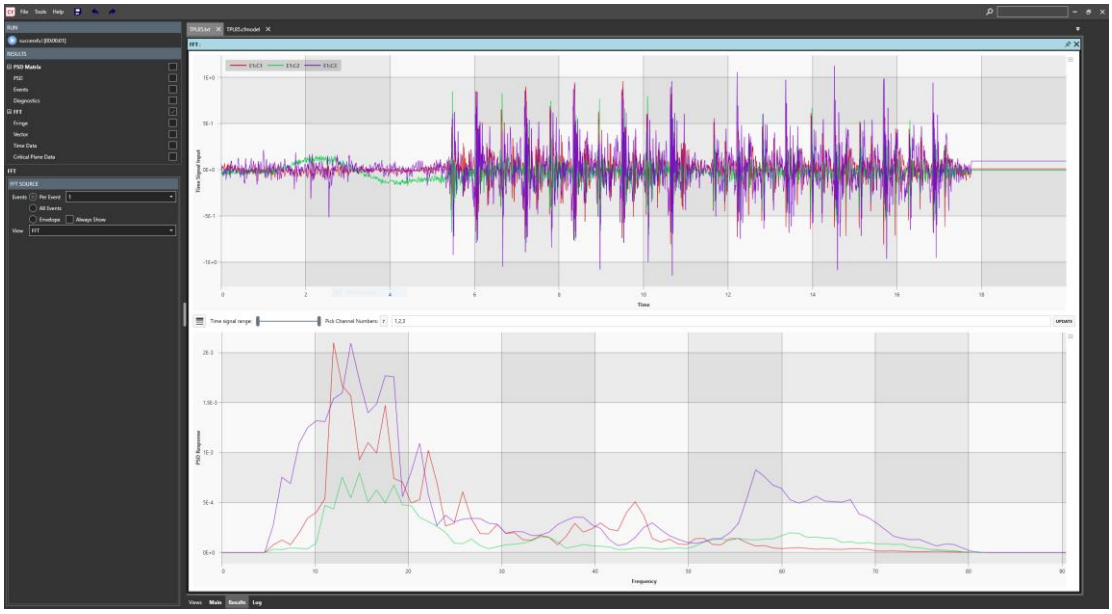
Viewing Results via the FFT Plotter and PSDM Plotter

FFT Plotter

Below is an FFT Plotter image of the output results. Note: this image can only be obtained if LOGLVL is set to 1 or 2.

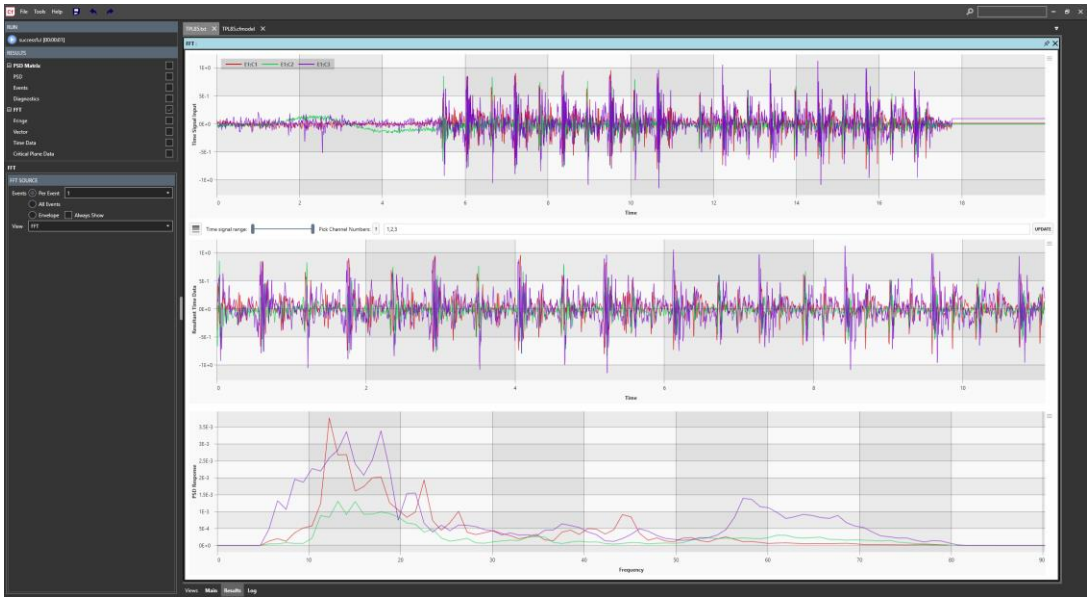
The FFT Plotter image will (typically) have 3 windows. The upper window is the initial time history event file (i.e. RPC/CSV file). The middle window image is the "conditioned" time signal image. The lower window shows the converted PSD(s) profiles.

However, if AutoD or "Deletes" is not used than no conditioning is done, and the middle window is not shown (i.e. no deletes were used to remove the flat parts of the signal). See below

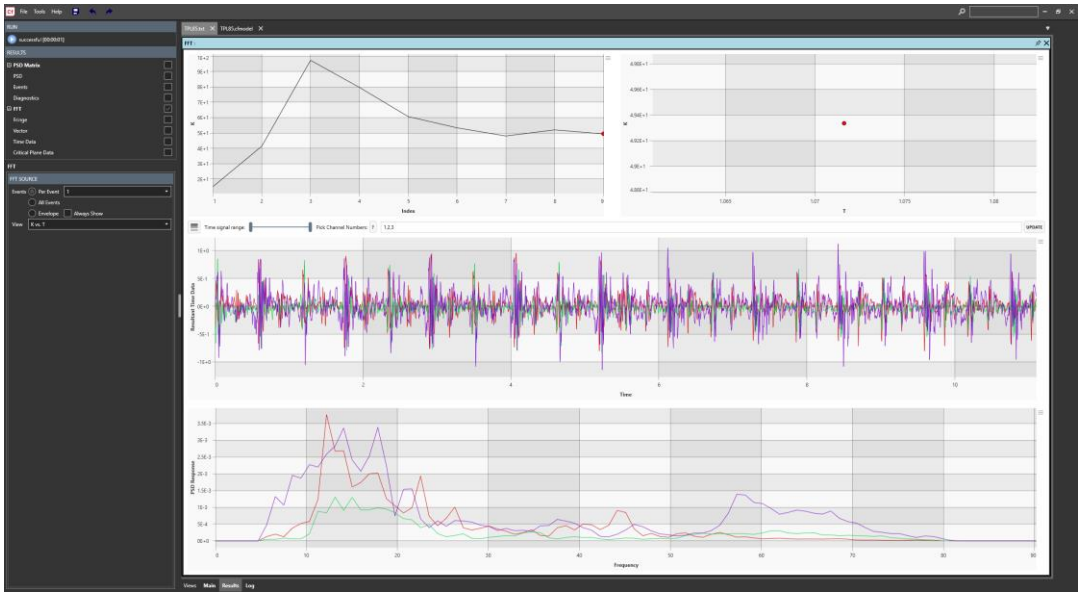


Results obtained without proper conditioning can be very problematic for doing accurate damage calculations because the non-stationarity in the raw time signal event has not been removed i.e., the low intensity "flat" parts of the signal.

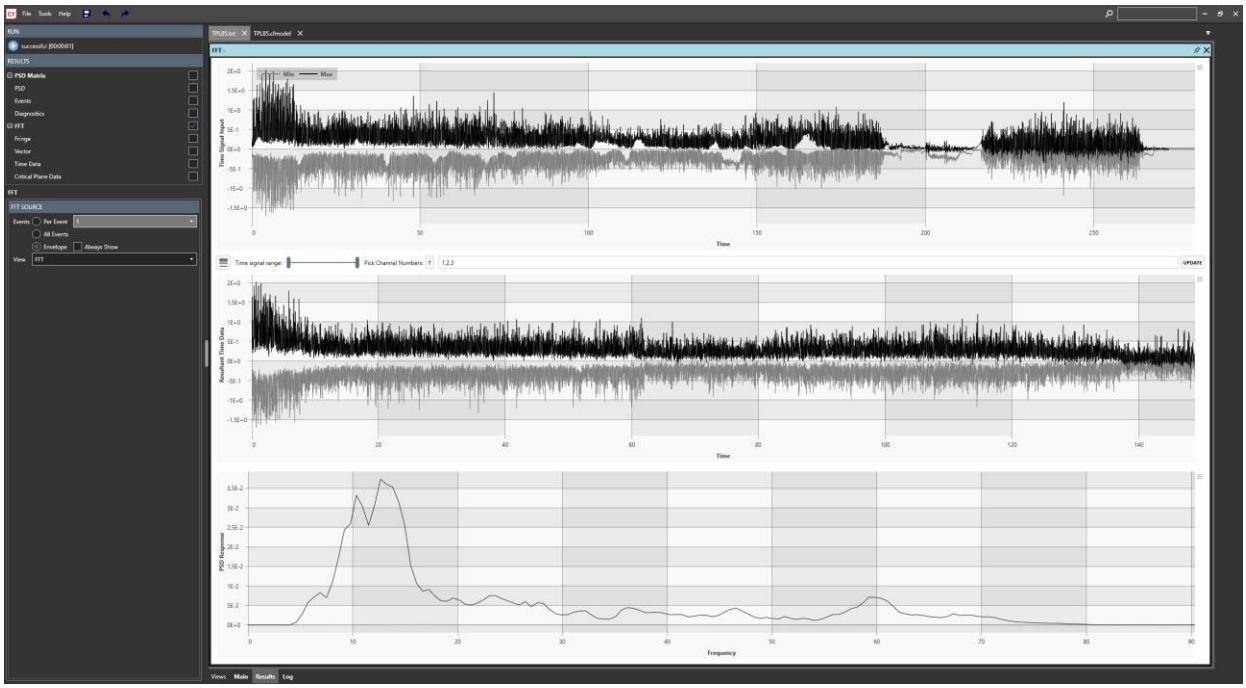
The following plot shows the original time history followed by the conditioned time history after it was processed through TIME2PSD using the AutoD process. The conditioning of the time history produces an “accurate” representation of the PSD frequency content i.e., note the differences near the first peak of response between the PSD plot below (max = 0.0038) and the PSD plot above (max = 0.0021).



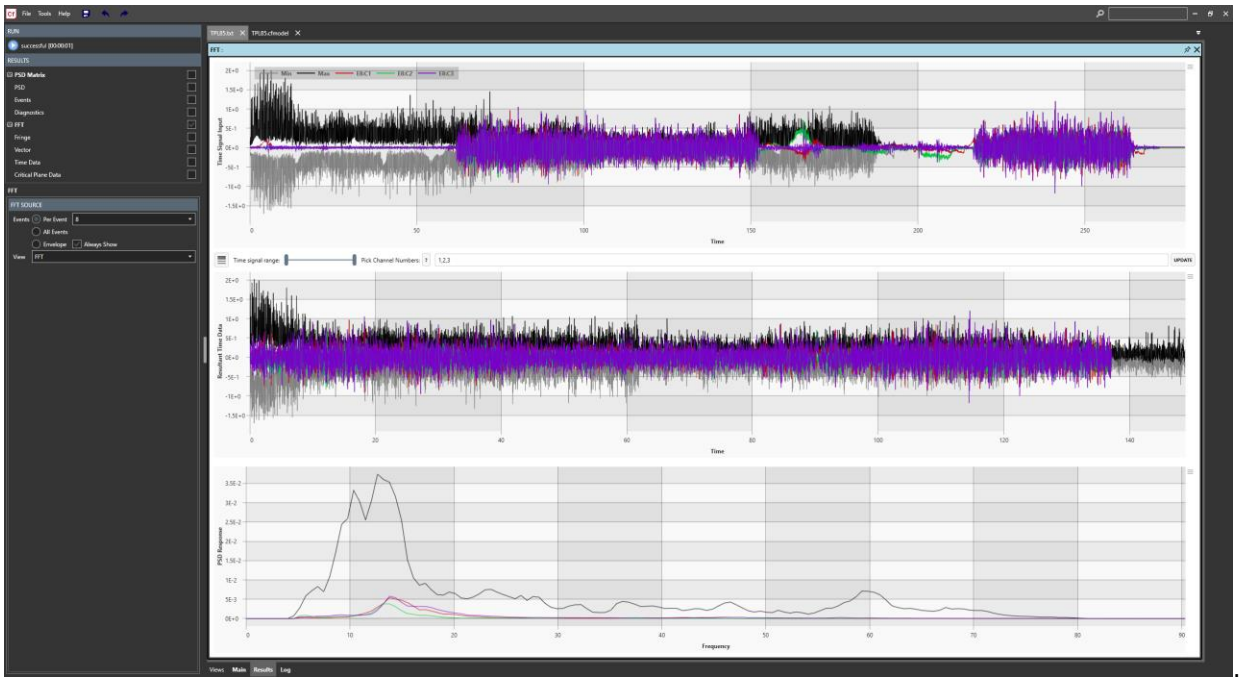
The shape of the PSD is controlled by the frequency content, window length and overlap. When using the automated feature AutoT, the Cf program controls the shape of the PSD using K Points. Below is a plot of the K Point values used by AutoT to generate the PSD shapes above. The K Points method is priority to the Cf software.



The FFT plot also permits the User to plot an “envelope” that encompasses the data from all channels for all events. Below is a plot showing the envelope only.



The User can also plot the data for a single Event with an overlay of the “all channels / all events” envelope. The plot below shows Event 8 with an overlay of the envelope. Note how Event 8 is not a major contributor to the overall envelope shape i.e., looking at the amount of PSD shape for Event 8 under the PSD envelope shape.



PSDM Plotter

The FFT plotter provides images of the raw time signal event, the conditioned time signal even and the converted PSDs. However, it would be more appropriate to say that the converted PSDs are the DIRECT converted PSDs. There are also a number of cross PSDs that are generated to represent the “phasing” responses between time signals within an event.

For Event 1 of this example, there are 3 time signals (channels). This results in 3 Direct PSDs (example: the response in the X direction due to an input in the X direction) and 3 cross PSDs that are complex and have real / imaginary components. Hence, if views as a matrix, the resulting number of PSDs would be a 3 x 3 PSD matrix or PSDM.

X Direct (1-1)	Cross PSD X to Y (img) (1-2)	Cross PSD X to Z (img) (1-3)
Cross PSD X to Y (real) (1-2)	**Y Direct (2-2)**	Cross PSD Y to Z (img) (2-3)
Cross PSD X to Z (real) (1-3)	Cross PSD Y to Z (real) (2-3)	**Z Direct (3-3)**

Adding the Cross PSDs into the evaluation of Damage can be vital and is often misunderstood by the User. To help the User visualize this added content, the Cf program also offers a PSDM view.

Note that the lower image of the FFT plotter is the same as the upper image of the PSDM plotter i.e., a plot of the DIRECT PSDs. The PSDM potter also provide a view of the complex cross PSDs that are also part of the response (this can be displayed as real/imaginary components or amplitude/phase components). All 9 PSDs are contained in the PSDM file, which is the output from the TIME2PSD toolset. This PSDM file is then read into a Cf fatigue calculation (control file or process flow) as a multi input PSD analysis.

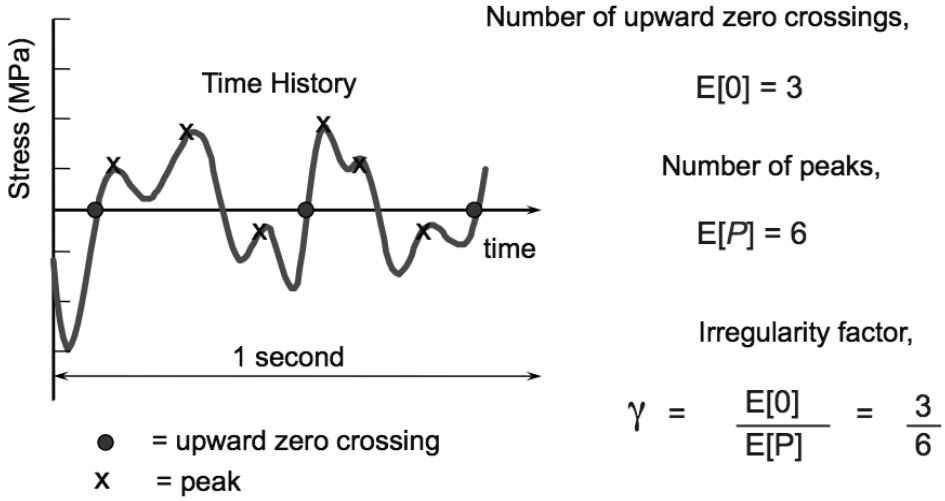
RMS, E[0], E[P] and Irregularity Factor Statics Available in the FFT Plotter

When converting the raw time signals to a PSD it is often useful to also understand how the signal statistics are changing. These statistics include rms, expected zero crossings – E[0], expected peaks – E[P] and irregularity factor – γ.

To calculate these statistics in the time domain has been well established in literature. For example, the RMS of a signal is the "square root of the arithmetic mean of the squares of the values" or in other words

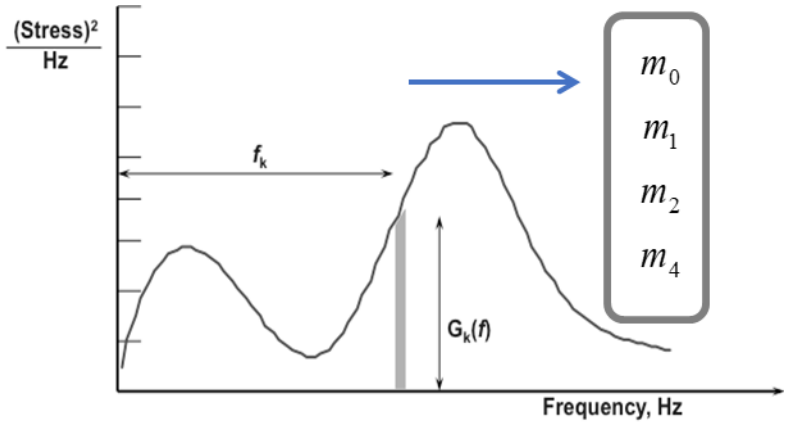
$$X_{RMS}=\sqrt{\frac{1}{N}\left(X_1^2+X_2^2+\cdots+X_N^2\right)}$$

The E[0] and E[P] values are also simple to generate and can be graphically understood from the image below. The expected zero crossing value is simply the number of times the signal crosses the mean average. In the image below E[0] = 3. The expected number of peaks value is the number of peaks in the signal. In the image below E[P] = 6. The irregularity factor value is simply E[0] / E[P].



To calculate these same statistics in the frequency domain requires knowledge of the ***spectral moments***, which are calculated from the response PSD using the formula below. The moments M0, M1, M2 and M4 are all that is needed to generate the needed statistics.

$$m_n = \int_0^\infty f^n\, G(f).df = \sum_1^i f_i^n G(f_i)\,\delta f$$

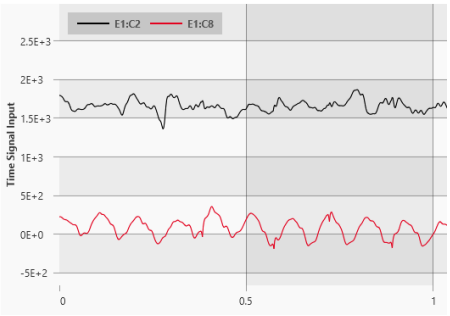


Below are the formulas that are used to calculate the statistic from the spectral moments.

$$RMS = \sqrt{M_0} \qquad E[0] = \sqrt{\frac{M_2}{M_0}} \qquad E[P] = \sqrt{\frac{M_4}{M_2}} \qquad \gamma = \frac{E[0]}{E[P]} \vee \sqrt{\frac{M_2^2}{M_0 M_4}}$$

Dealing with Signal Offsets: RMS versus Standard Deviation

The fluctuations of a time signals typically vary about the signal's mean value. Most times that mean value is zero but sometimes that value is not zero (see 2 examples below). When the signal mean value is zero, the **RMS** of the time signal is equal to the **Standard Deviation** of the time signal. This is because the Standard Deviation of a time signal ignores any mean offset in the signal, but the RMS calculation will incorporate the mean offset in the result. When we calculate the RMS from a PSD, there is never a mean offset, so the RMS equals the Standard Deviation when dealing with PSDs.

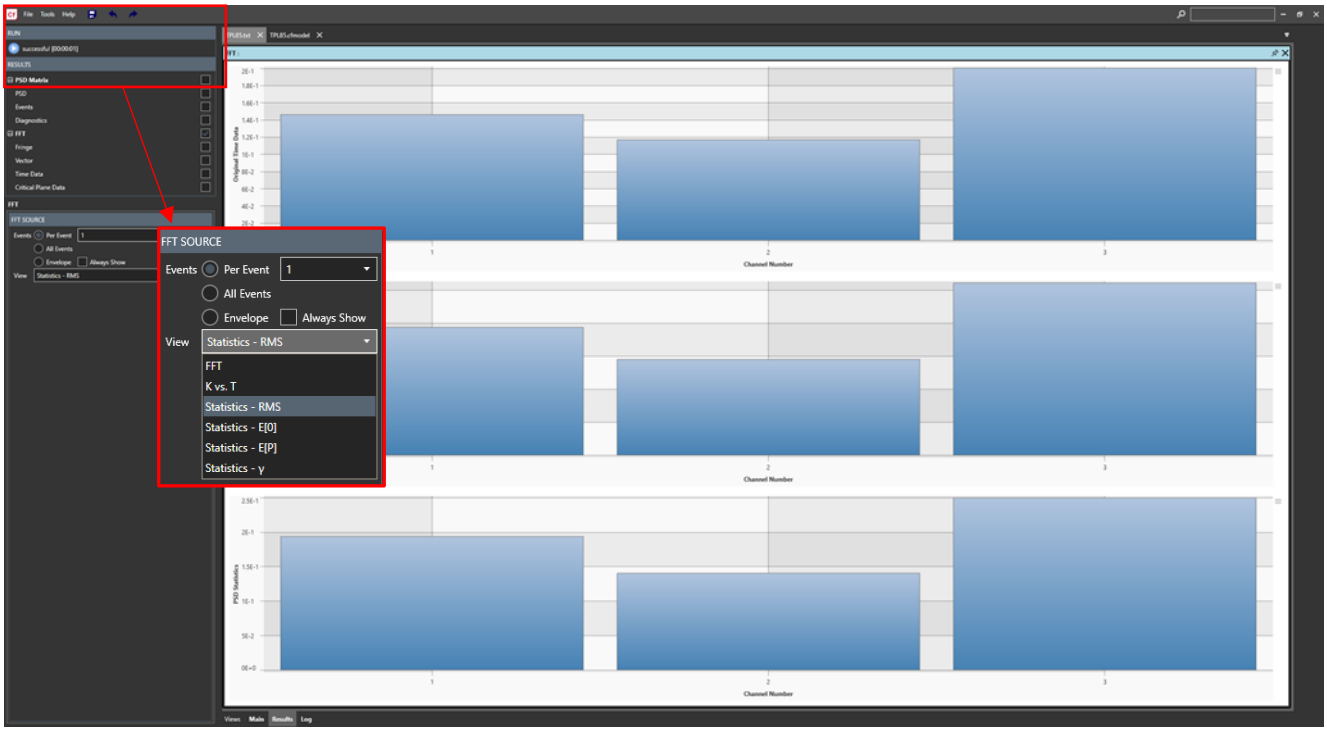


Non-zero offset means RMS **≠** Standard Deviation

Zero offset means RMS **=** Standard Deviation

Therefore … technically speaking … when we want to compare the RMS values from time signals and PSDs, we really should be looking at Standard Deviation values for the time signals (so that all mean values are removed) and the RMS (Standard Deviation) for the PSDs.

This is exactly what the FFT Plotter provides in a separate statistical view that can be seen by selecting the option from the menu as shown below. The label says RMS but it should be understood that any non-zero mean values are removed from the time signals before this RMS value is calculated; so in essence, it is a Standard Deviation plot for the time signals and an RMS plot for the PSD signals.


FFT SOURCE
Events
Per Event
1
All Events
Envelope
Always Show
View
Statistics - RMS
FFT
K vs. T
Statistics - RMS
Statistics - E[0]
Statistics - E[P]
Statistics - γ

9 | INTRODUCTION TO THE TIME2PSD PROCESS FLOW VIEW

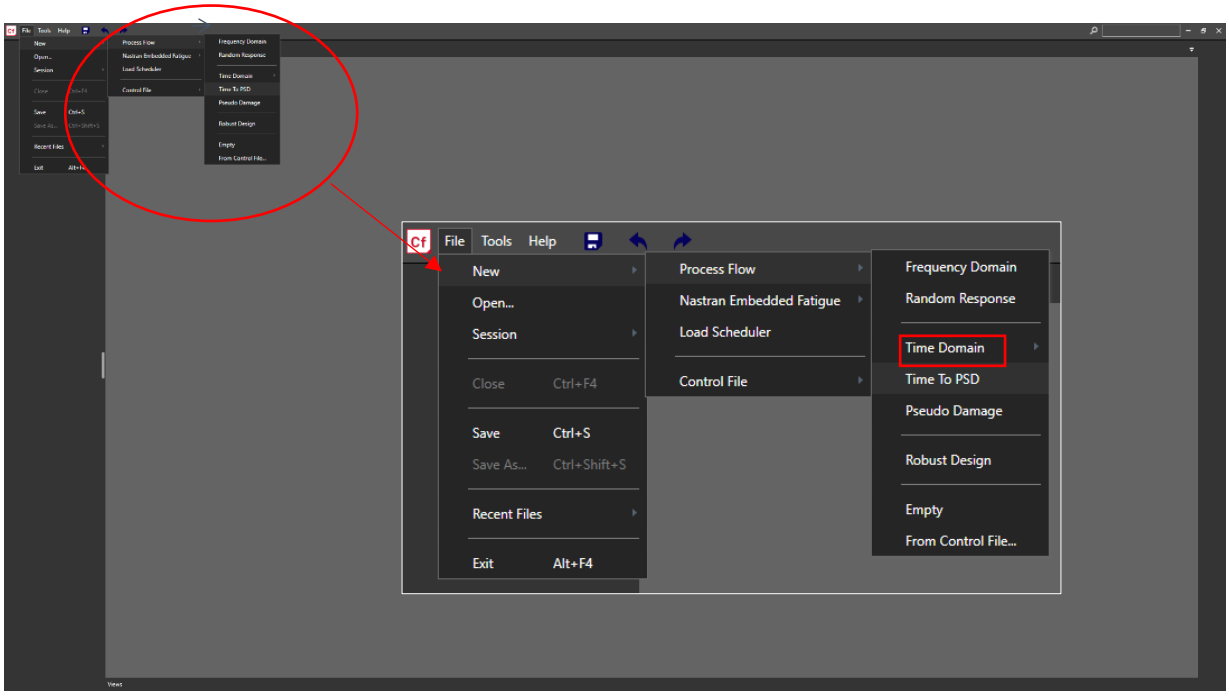
NOTE: The images shown in this section reflect CAEfatigue v2021 and may change slightly with future releases.

CF includes a graphical user interface application for executing the program. For the interface to run successfully, ".NET 4.5" (or later – see latest installation guide) must be installed on the system (this should normally be the case).

Overview and Introduction to the TIME2PSD Template File

To start CF, click on the desktop Cf icon (see below) or find the ProcessApp.exe file for the program in the default installation directory "C:\Program Files\MSC.Software\CAEfatigue\<ver>\Windows\".

This will open Cf in interface application window as shown below. To do this we can either create a NEW TIME2PSD process flow template file or we can select ""FILE>OPEN" to pick an existing TIME2PSD process flow file. In the image below, we selected to open the TIME2PSD template file. The User can also open the TPL85.cfmodel file to see the use of the process flow TIME2PSD capability.

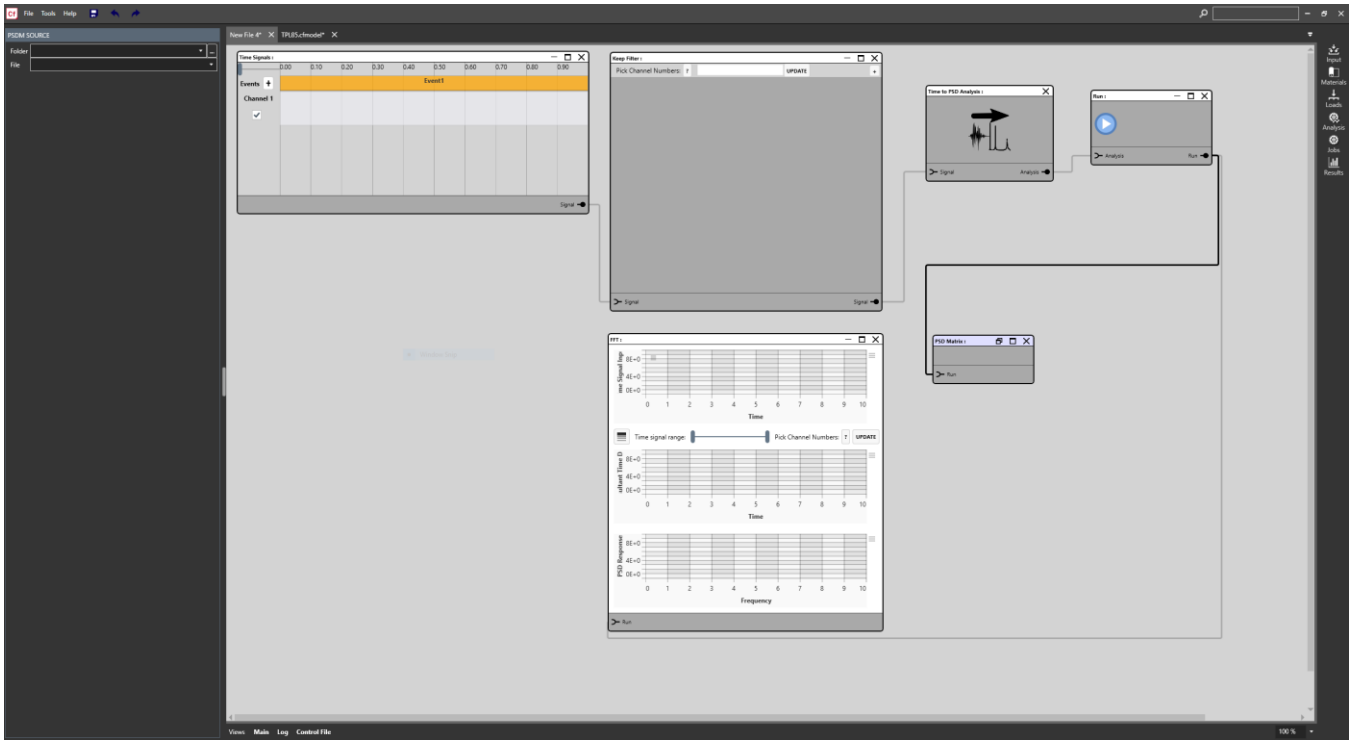


TIME2PSD Process Flow Template

When you select the **TIME to PSD** process flow template you will see the layout shown below. The workspace consists of 6 boxes.

- Time Signals: This box is used to define the loading files in RSP or CSV/TXT format.
- Keep Filter: This box is used to graphically define "deletes" and "subevents"

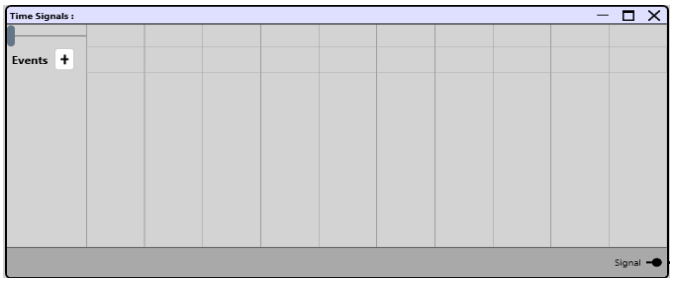
- FFT Plotter: Same as the plotter in the control file view
- PSDM Plotter: Same as the plotter used in the control file view. Note: by default, the plotter is minimized.
- Time to PSD Analysis: This box is used to set the analysis parameters.
- Run : This box is used to run the analysis.



Below are the individual entries that are used to do a TIME2PSD analysis. For clarity, we are using **TPL85.cfmodel** as an example for the boxes below.

Time Signal Box

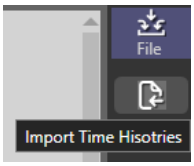
The Time Signal box is used to upload the input time histories of loading. These loadings can be in RSP or CSV/TXT format. For TPL85.cfmodel, the loading files are defined by 14 RSP files. We suggest deleting the single Event provided in the template file and then uploading the 14 RSP files (see steps below).



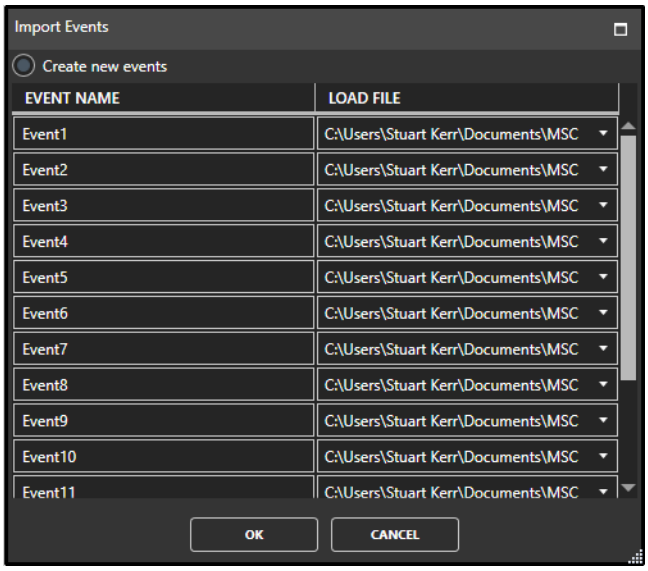
1) Delete the single Event

NOTE: **CSV files** must have a first column with TIME interval data (x data) followed by one or more columns of time history data (Y data). This TIME interval column is needed to determine the SAMPLING RATE (SWRATE).

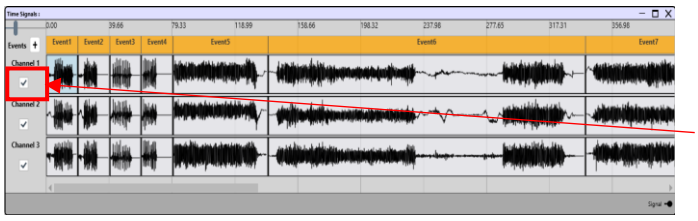
In a later revision, the User will be able to specify SWRATE like you can currently do in the Control File version of TIME2PSD.



2) Use the FILE> Import Time Histories to upload the 14 RSP files associated with TPL85.cfmodel



3) CAEfatigue will create new events for the 14 RSP files.



4) The 14 RSP files will come in as 14 new Events.

Note the check boxes under each channel number. These check boxes are used to keep or remove the channel from the analysis. This is analogous to the Mapping feature used in the control file.

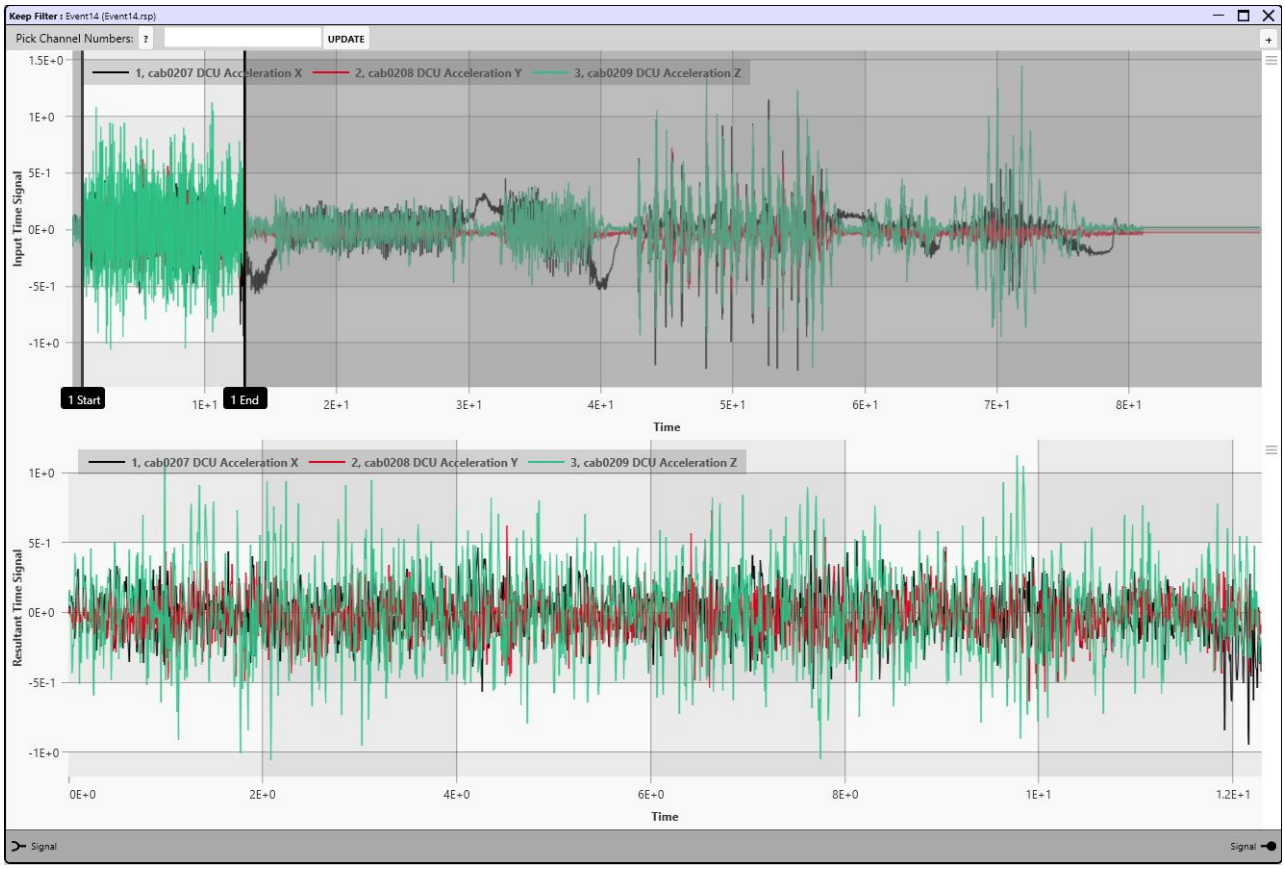
Each Event / Channel can be selected to view the time signal properties.

To the left is the file properties for Channel 1 of Event 1.

Note: When the loading files are imported, the sampling rate is automatically calculated. Hence, unlike the Control File view, it is no longer necessary to enter the sampling rate when a CSV/TXT file is imported.

Keep Filter Box

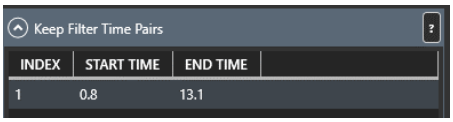
The KEEP FILTER box is used to keep sections of an event while ignoring unwanted sections. Select the Event bar to activate the KEEP FILTER box for that event. In the example below, the area between 1 Start and 1 End in the **Input Time Signal** window is kept in the analysis and the remaining grey area is ignored. The result of this selection is shown in the **Resultant Time Signal** window (bottom image).



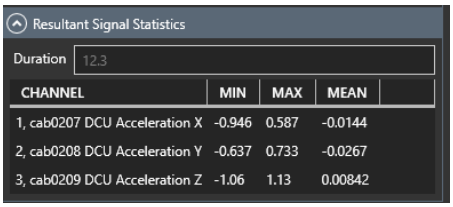
The START and END indicators are generated by right mouse clicking within the Input Time Signal window to open the CREATE option.

By selecting CREATE, you automatically generate a KEEP FILTER window with START/END indicators. Multiple KEEP sections can be generated by selecting the CREATE option as needed. All KEEP windows will be compiled into a single time signal within the Resultant Time Signal window. All grey areas (unselected areas) will be ignored.

The locations of the START and END markers can be graphically selected (by selecting and moving the markers) or they can be entered into the Keep Filter Time Pairs box (see below). The Resultant Signal Statistics box will also give you some information about the new time signals generated from the KEEP FILTER selection.



Any number of START/END markers can be specified in the **Keep Filter Time Pairs** box. For TPL85.cfmodel, the values 0.8 to 13.1 were entered. To delete the marker lines, simple delete the appropriate row from the Pairs box.



Note: Unlike the control file example, the process flow also gives you some statistics on the new time signal that was created from the KEEP FILTER selection.

There is no ability to add “deletes” when doing a TIME2PSD analysis within a Process Flow. The KEEP FILTER serves the purpose of defining both areas to keep (SUBEV) and areas to delete (Deletes).

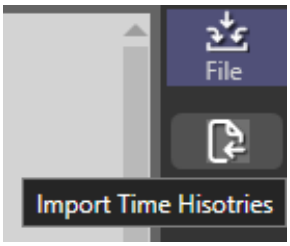
How to create Events 15 and Events 16 as done in TPL85.txt example

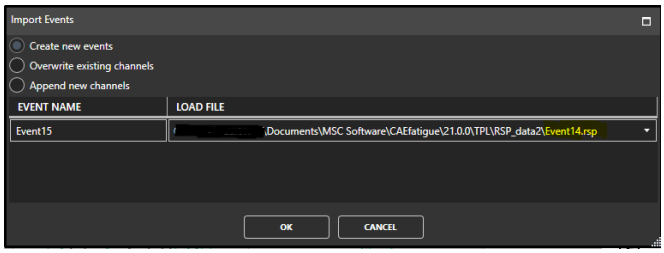
The Control File version of this example (TPL85.txt) uses the SUBEV feature to break up Event 14 into 3 separate events. This was done to make the data in Event 14 Stationary. The result of SUBEV produced a modified Event 14 and 2 additional events (15 and 16) added into the analysis (see below).

```
$ If Sub Events are used (SUBEV below) than the following priority is
$ followed: Subevents first, followed by AutoD and then AutoT.
$ However, if manual deletes are specified (as in EV14), then the
$ order is subevents, manual delete then AutoT (AutoD is not done).
        EV_OPTS 14
        Event14.rsp
        subev   0.8     13.1
        0.0     1.0     $ first 1 second of sub event is deleted
        EV_OPTS 15
        Event14.rsp
        subev   42.6    57.7
        EV_OPTS 16
        Event14.rsp
        subev   66.7    74.1
```

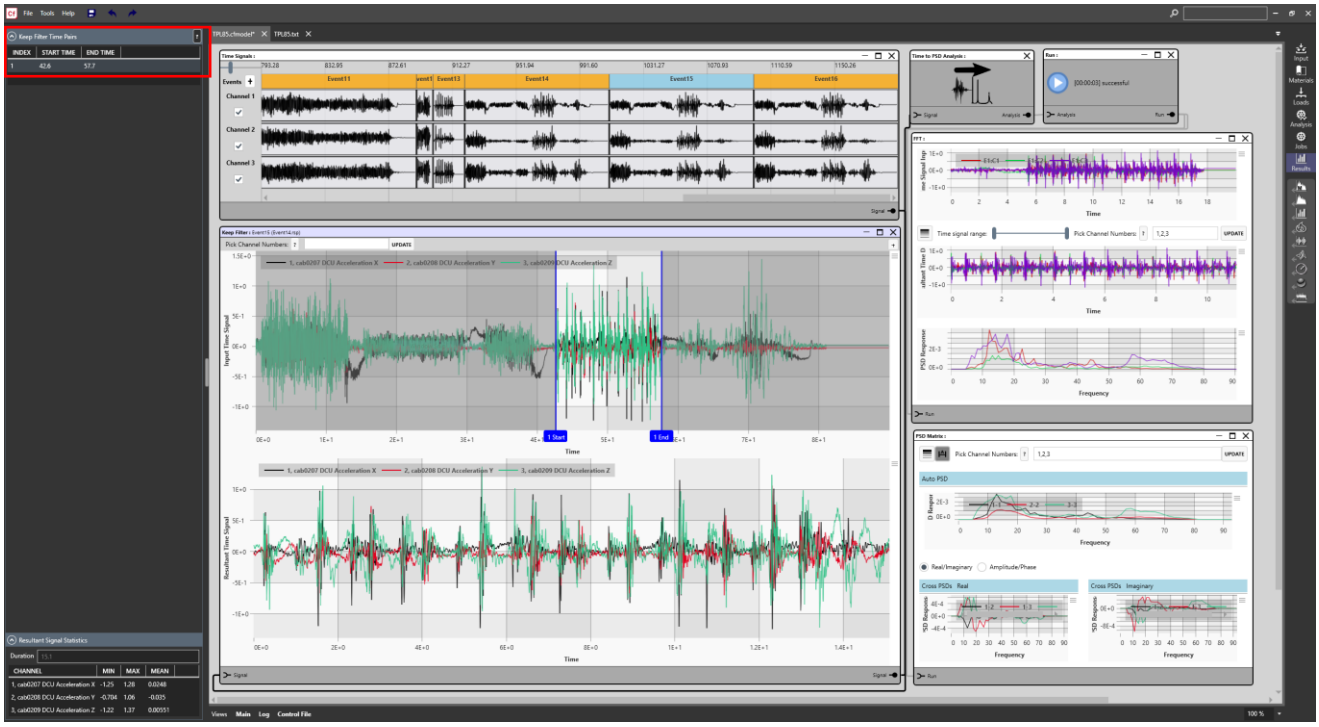
To duplicate this within the Process Flow, we must import the Event 14 loading file into the TIME SIGNALS box 2 additional times to create Event 15 and Event 16. We then need to add a KEEP FILTER to each event (14, 15 and 16) to mimic the SUBEV definitions show above.

To import Event 14 again, simply go to FILE> IMPORT TIME HISTORIES and navigate to the Event 14 location. Once chosen, select “Create New Events” from the Import Events box. This action will create a new EVENT 15 using the Event 14 loading file. Repeat the same step again to create a new EVENT 16.



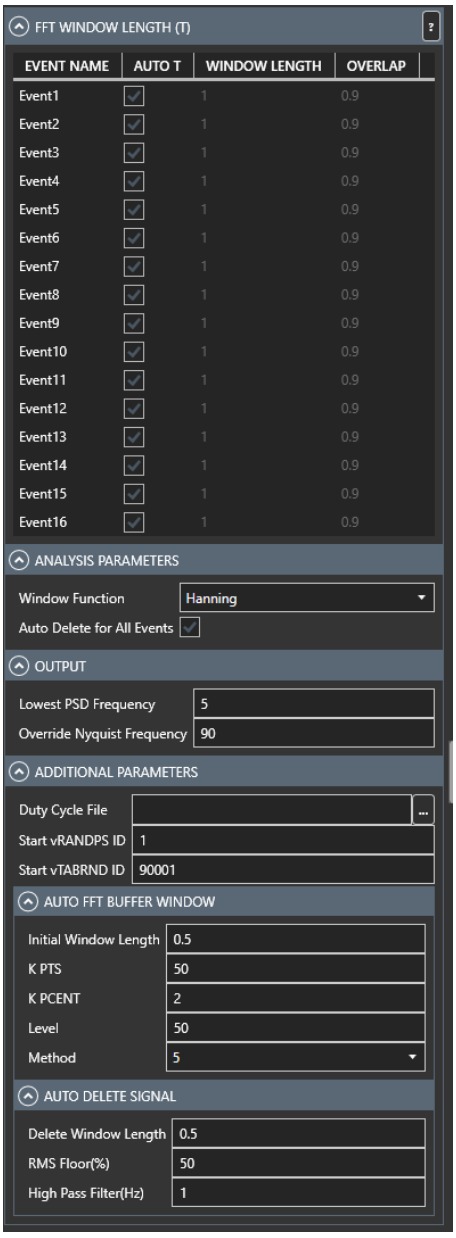


Once Event 15 and Event 16 are added, select the Event 15 header bar to activate the KEEP FILTER box. Right mouse click within the **Input Time Signal** window to open the CREATE button and select it. This action will place START/END markers on the image. Enter the Start and End times (42.6 and 57.7, respectively) within the Keep Filter Time Pairs box (shown in red below) or roughly select the same area by dragging the marker bars directly within the Input Time Signal window. Remember: AutoD will be used on this event so there is no need to be super accurate in your graphical selection.



Repeat the same actions for Event 16 using the Start / End times of 66.7 and 74.1 (or close to these number if using graphical selection).

Time to PSD Analysis Box:



The TIME2PSD analysis is controlled within this box.

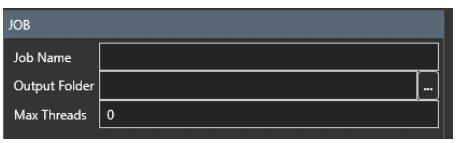
For this example, we are running both AutoT and AutoD for all events. If you wish to override AutoT for any event, you can simply deselect the AutoT checkbox for the event and enter the Window Length and Overlap. This will stop AutoT from running for the chosen event and the software will use the manual settings.

Like the previous control file example, we are using a Hanning window for the Fourier transform and limiting the PSDM output to between 5 Hz and 90 Hz.

The AutoT defaults are used except for the K PTS variable which has been set to 50 from the default value of 200.

The AutoD defaults are used as is.

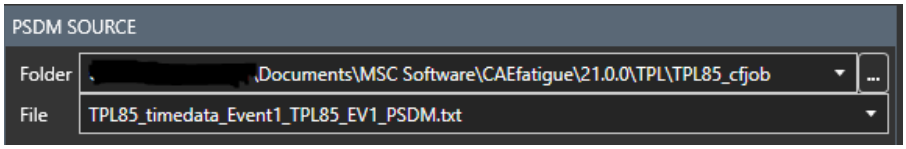
RUN Box



The TIME2PSD analysis is controlled within this box. The RUN box is identical to the RUN box used for a process flow Frequency Domain analysis

Output – PSDM Plotter and FFT Plotter

The PSDM Plotter and FFT Plotter are identical to those found in the previous control file example. Note: by default, the PSDM Plotter is minimized in the template file since it is not always required by the User To see the plotter, simply select the minimize button which will allow viewing of the PSDM files.



More Information about KEEP FILTER, AutoD and AutoT.

When working in the Frequency Domain, Cf is a random response and fatigue solver that uses an input Power Spectral Density (PSD) together with the stress transfer functions and material properties to deliver response statistics and fatigue damage. However, some Users only have time domain data to use for input. In these cases, Cf offers a unique capability called TIME2PSD that helps the User convert a time domain input file into an accurate PSD through a semi-automated process.

TIME2PSD is a pre-processing module that is run separately from Cf and will accurately convert input time histories into a PSD matrix without User input needed to "clean" time histories containing dead spots, pothole spikes, etc. Once the process is complete, the PSD is brought into Cf to be process into response and damage output.

When used in the Process Flow interface, the TIME2PSD feature conditions and converts the time signals into PSDs using 2 processes:

1. **KEEP FILTER**: **Separating nonstationary section from the Events.** The Keep Filter box is a pre-filtering step that allows the User to separate (split) an Event, containing any number of time histories into a number of parts that can be grouped together or, into additional events. This is highly recommended when the time history contains sections with significantly different frequency content. This type of occurrence in a time history violates the key assumption that the time history is Stationary throughout the signal. Hence, the User can split (and group) different sections of the Event or split off sections of the event into multiple new Events where the content within each new event adhere to the Stationary assumption.

2. **AutoD**: **Filtering nondamaging sections from the Events**. Nonstationary segments of the time history data that do not significantly contribute to the damage must be removed so they are not carried into the PSD. The TIME2PSD module offer an automated approach, called AutoD, which reads through the event in small increments of time and analyzes the RMS intensities of small increments as it reads through the entire event. Smaller sections with low RMS intensity are removed from the signal during the first pass through the event and then the process is repeated. Eventually, (mostly) all low RMS sections of the event are removed leaving only the parts of the event that cause the most damage. Note: this filtering takes into account all time histories within the event and when low intensity sections of the event are removed, they are removed from all channels.

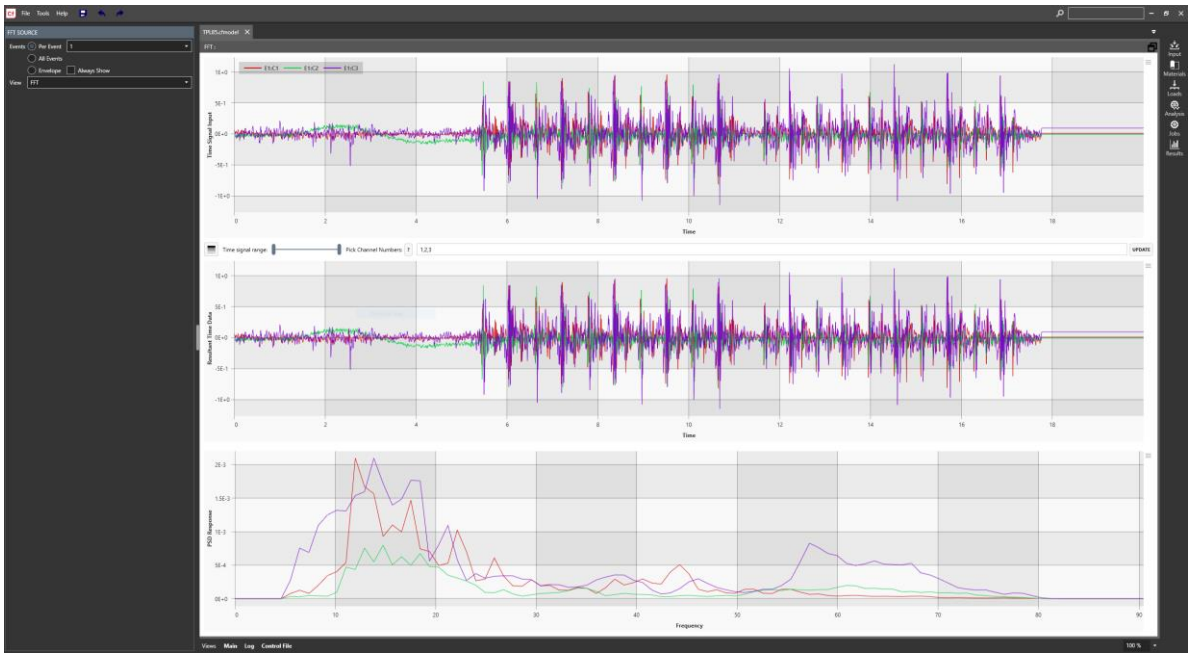
3. **AutoT: Definition of window length (T) and overlap (δ).** This step involves a more difficult decision for the User, which is the choice of window length and overlap for the FFT algorithm. These items are used in a "PWELCH" type calculation where multiple estimates are averaged to create the final PSD values. These values can be specified manually for each event, but this can again be challenging since too short a window length can result in a PSD with insufficient frequency resolution and too long a window length will result in poor statistical accuracy (too much scatter). TIME2PSD provides an automatic way to calculate the T and δ values (*AutoT*), which uses a proprietary evaluation process using K Points. The K-PTS value can be adjusted by the User to control the "smoothness" of the PSD profile(s) created in the FFT conversion process. The User should evaluate the PSD shape and adjust the K-PTS values until desired results are achieved.

Viewing Results via the FFT Plotter and PSDM Plotter

FFT Plotter

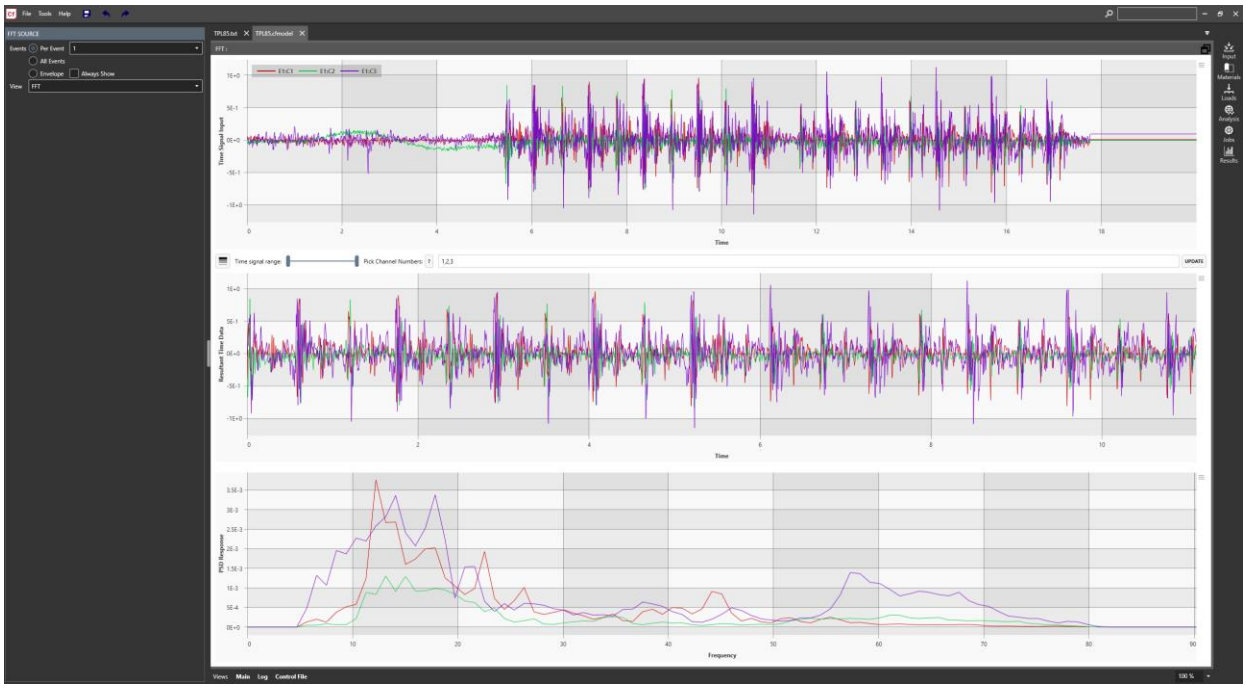
Below is an FFT Plotter image from TPL85.cfmodel for Event 1.

The image has 3 windows. The upper window is the initial time history event file (i.e. RPC/CSV file). The middle window image is the “conditioned” time signal image after low intensity RMS sections are removed from the initial time histories. In this case, no conditioning was requested in the analysis, so the upper and middle plot are identical (i.e. AutoD is unchecked and not used to remove the flat parts of the signal). The lower window is the converted PSDs.

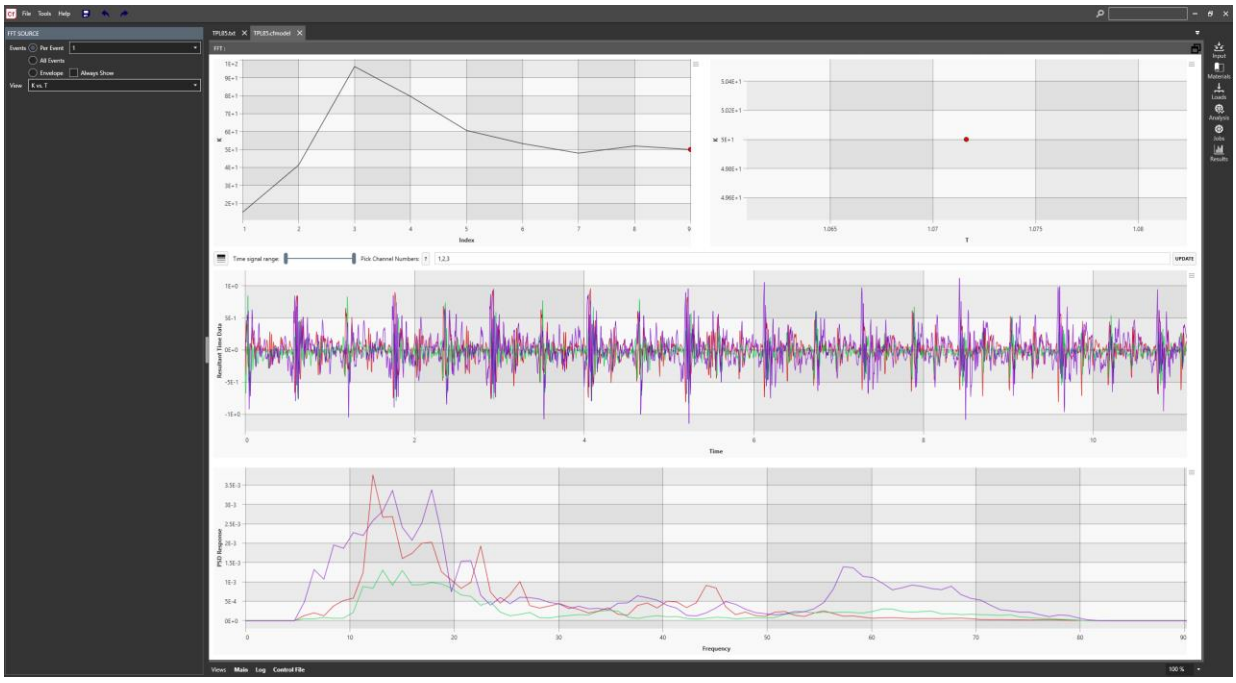


Results obtained without proper conditioning can be very problematic for doing accurate damage calculations because the non-stationarity in the raw time signal event has not been removed i.e., the low intensity “flat” parts of the signal.

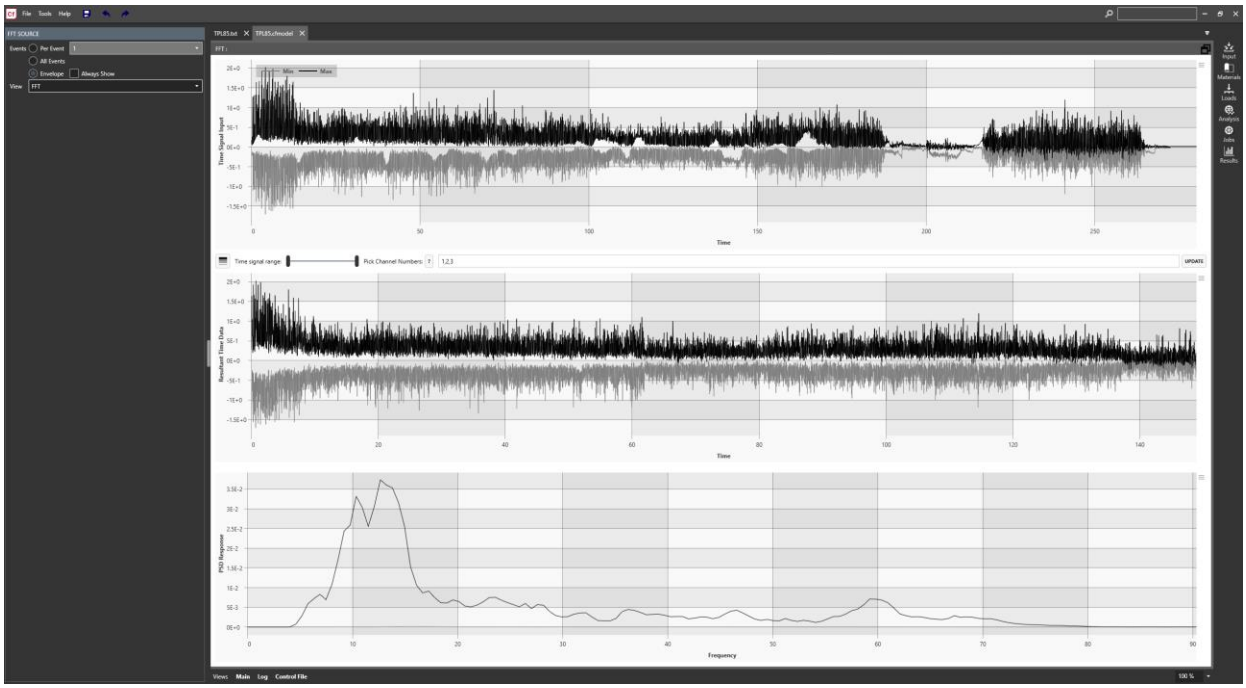
The following plot shows the original time history followed by the conditioned time history after it was processed through TIME2PSD using the AutoD process. The conditioning of the time history produces an “accurate” representation of the PSD frequency content i.e., note the differences near the first peak of response between the PSD plot below (max = 0.0038) and the PSD plot above (max = 0.0021).



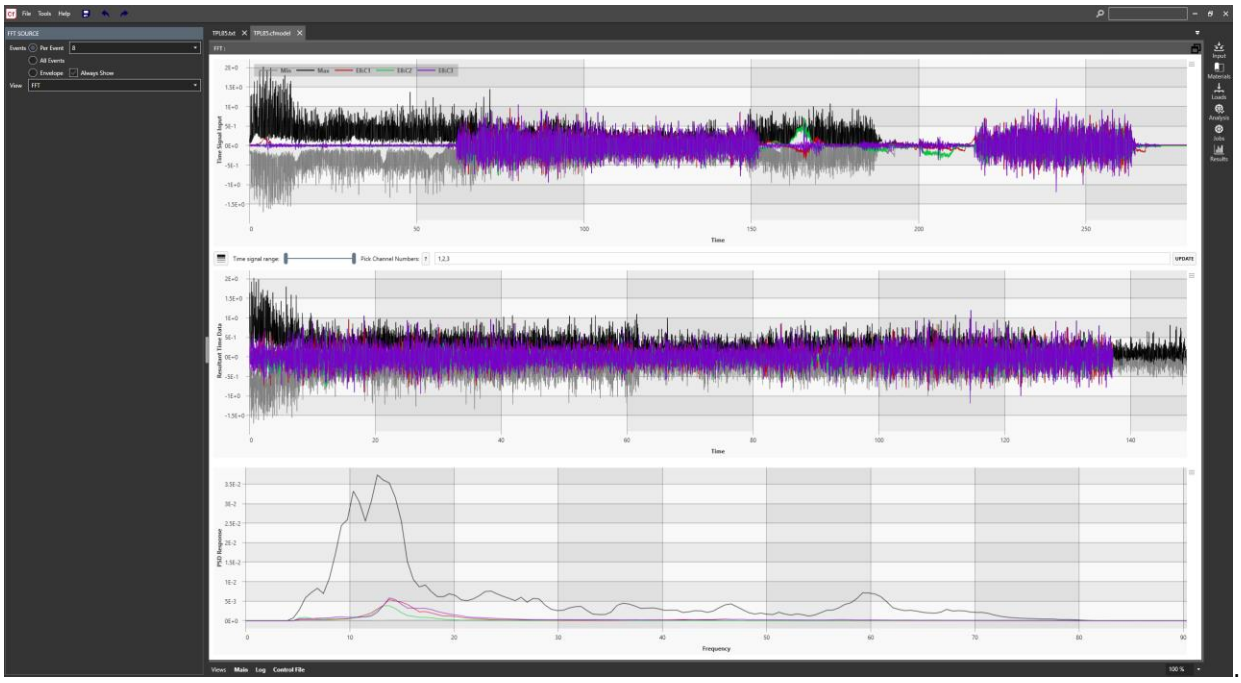
The shape of the PSD is controlled by the frequency content, window length and overlap. When using the automated feature AutoT, the Cf program controls the shape of the PSD using K Points. Below is a plot of the K Point values used by AutoT to generate the PSD shapes above. The K Points method is priority to the Cf software.



The FFT plot also permits the User to plot an “envelop” that encompasses the data from all channels for all events. Below is a plot showing the envelope only.



The User can also plot the data for a single Event with an overlay of the “all channels / all events” envelope. The plot below shows Event 8 with an overlay of the envelope. Note how Event 8 is not a major contributor to the overall envelope shape i.e., looking at the amount of PSD shape for Event 8 under the PSD envelope shape.



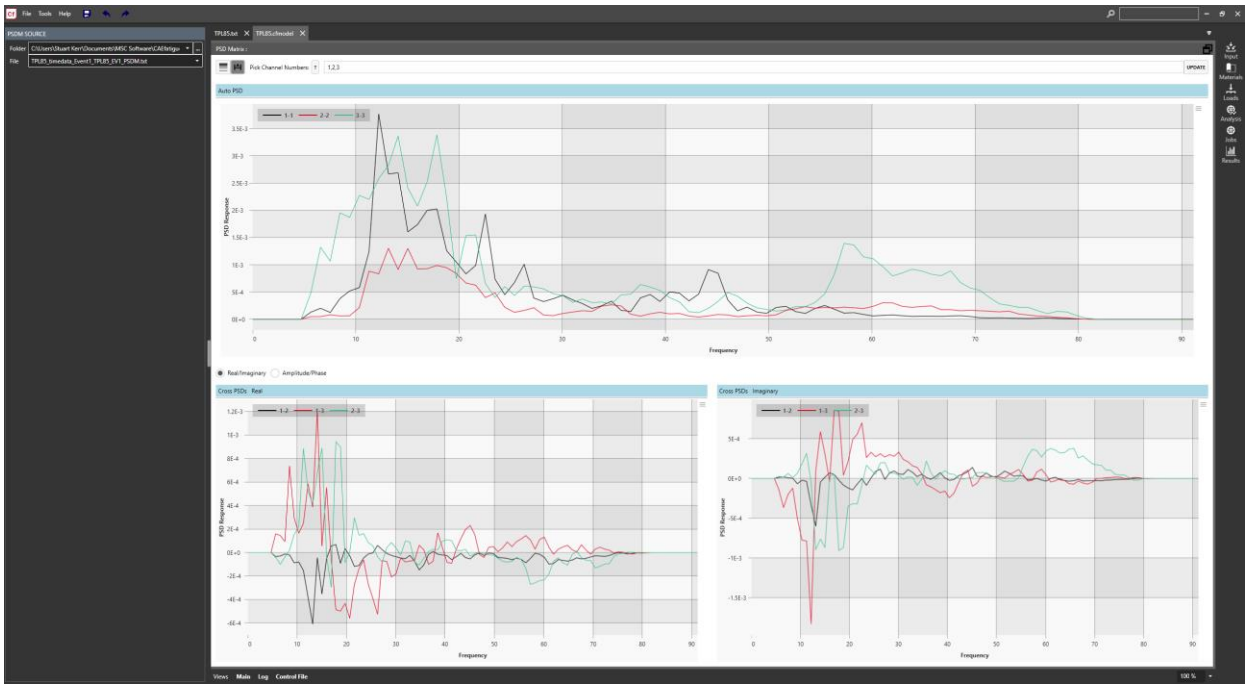
PSDM Plotter

The FFT plotter provides images of the raw time signal event, the conditioned time signal even and the converted PSDs. However, it would be more appropriate to say that the converted PSDs are the DIRECT converted PSDs. There are also a number of cross PSDs that are generated to represent the “phasing” responses between time signals within an event.

For Event 1 of this example, there are 3 time signals (channels). This results in 3 Direct PSDs (example: the response in the X direction due to an input in the X direction) and 3 cross PSDs that are complex and have real / imaginary components. Hence, if views as a matrix, the resulting number of PSDs would be a 3 x 3 PSD matrix or PSDM.

X Direct (1-1)	Cross PSD X to Y (img) (1-2)	Cross PSD X to Z (img) (1-3)
Cross PSD X to Y (real) (1-2)	**Y Direct (2-2)**	Cross PSD Y to Z (img) (2-3)
Cross PSD X to Z (real) (1-3)	Cross PSD Y to Z (real) (2-3)	**Z Direct (3-3)**

Adding the Cross PSDs into the evaluation of Damage can be vital and is often misunderstood by the User. To help the User visualize this added content, the Cf program also offers a PSDM view.



10 | INTRODUCTION TO THE TIME DOMAIN LOADS SCHEDULER

NOTE: The images shown in this section reflect CAEfatigue v2020 and may change slightly with future releases.

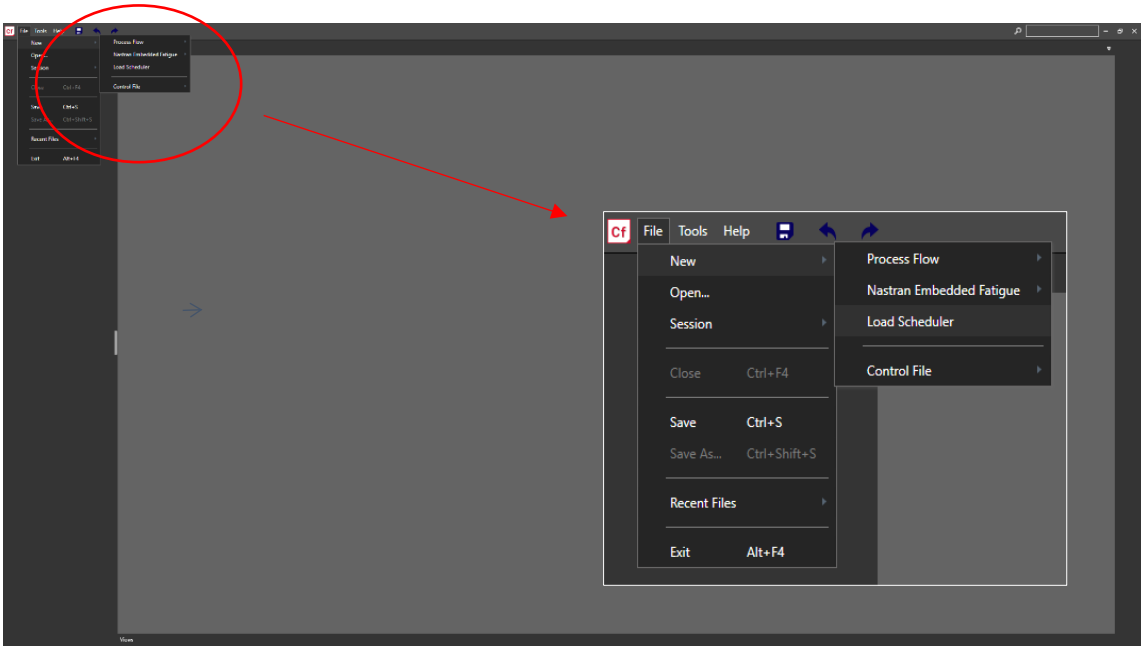
CF includes a graphical user interface application for executing the program and plotting the PSD response for critical locations (see below). For the interface to run successfully, ".NET 4.5" (or later – see latest installation guide) must be installed on the system (this should normally be the case).

Overview and Introduction to the Load Scheduler Menu Items

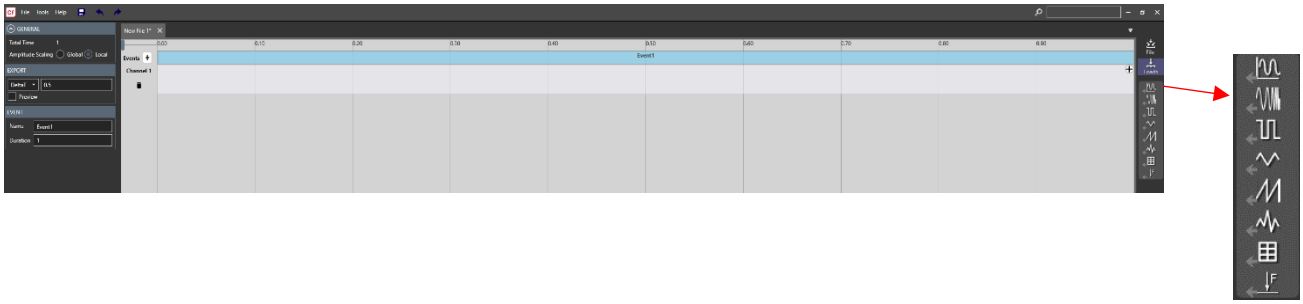
To start CF, click on the desktop Cf icon (see below) or find the ProcessApp.exe file for the program in the default installation directory "C:\Program Files\MSC.Software\CAEfatigue\<ver>\Windows\".

This will open CF in the CFG application window as shown below (Cf remembers the last file you were working on). If can choose to open a new Load Scheduler project by selecting **FILE>NEW>Loads Scheduler.**

Figure 5 Opening the LOADS SCHEDULER and selecting FILE or LOADS



The LOAD SCHEDULER allows for the creation of time domain loading that can be used in a random response or fatigue analysis. Once selected the user is provided with choice on the right side of the GUI to import/export a FILE or to begin selecting / creating LOADS.

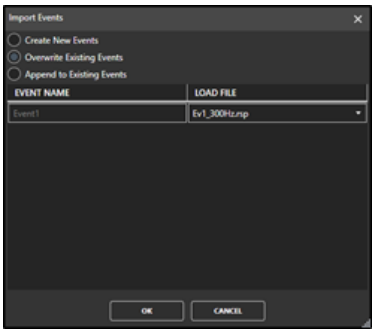


File Import of Existing Time Signals

The FILE IMPORT option allows the user to import existing time domain signal data from an RSP, RPC, CSV, TXT or DAC format. This data may represent a single time history or many time signals.

Note: the CSV / TXT file must contain no header rows, the first column must be the X data and remaining column(s) must be Y data.

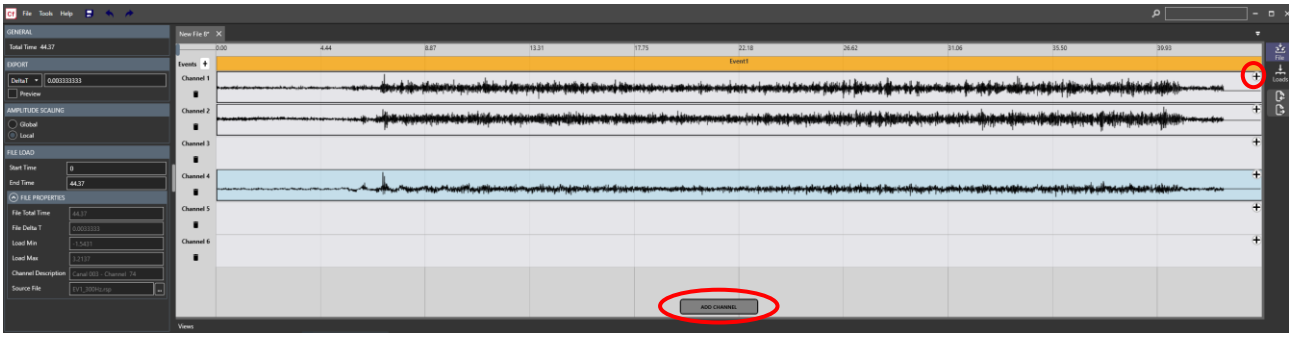
To import, simple select the EVENT you wish to import the file into and then select the IMPORT FILE icon to select the data you wish to import. You must choose to overwrite any existing data in the Event, append (add to) the existing Event or simply create a new Event with the imported data.



To modify the time signals that has been imported, the User can select the individual time signals and delete or move the time signal to other channels or other events. Below is an example of several time signals that have been imported from an RSP file. Event 1, Channel 4 was created by clicking the ADD CHANNEL button and then dragging the former Channel 3 down to Channel 4. This creates a ZERO amplitude signal Channel 3 and zero amplitude signals for the added Channels 5 and 6.

NOTE: The same number of channels will be used for all events in the LOAD SCHEDULER. Hence, if a second Event is created in the example below, it would automatically have 6 channels. If another channel is added (by selecting the + sign), then it will be applied to both events. Also, any channel can have a subchannel by selecting the + sign on the Channel itself. This may be desired if the User wants to apply multiple sinewaves (for example) to the same channel. In this case, all subchannels would be added together to create a new signal for the channel.

Figure 10-6 Imported RPC files into Event 1.

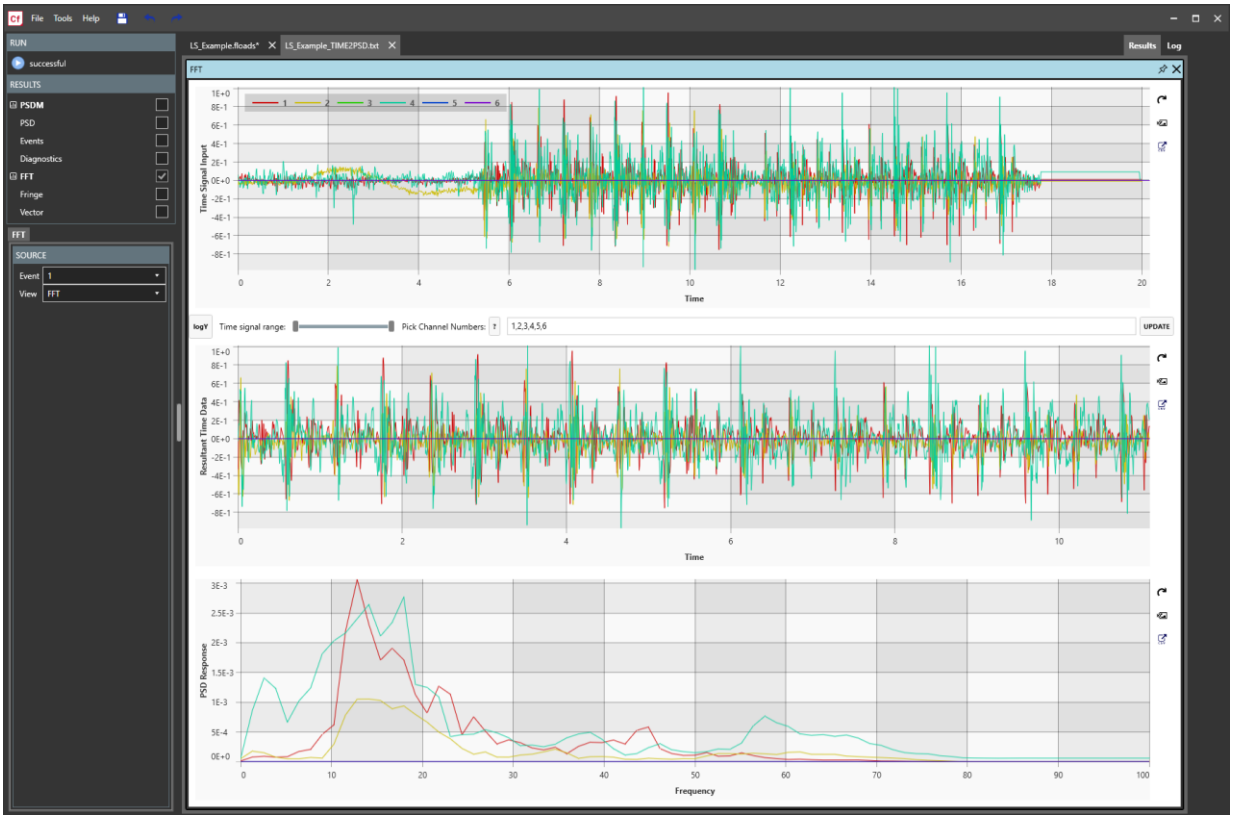


File Export of Channels and Events

Once complete, the User can export the scheduler content to an **RPC / RSP / CSV / TXT or TABLED1 (BDF) format**. We can then use the exported RPC/RSP/CSV/TXT file formats as a loading input for a Time Domain fatigue analysis or in the TIME2PSD tool to convert the signals into PSDs for a Frequency Domain fatigue analysis. The BDF format can be used directly by MSC Nastran for a linear dynamic analysis.

On the following page is the TIME2PSD FFT Plotter output generated from the conversion of the loading schedule layout shown previously. Note that the RAW DATA in the upper image matches the time signals in the Loads Scheduler including the zero amplitude Channels 3, 5 and 6.

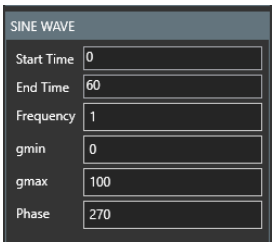
Figure 10-7 Raw data plot (upper image) representation of the Load Scheduler signals.

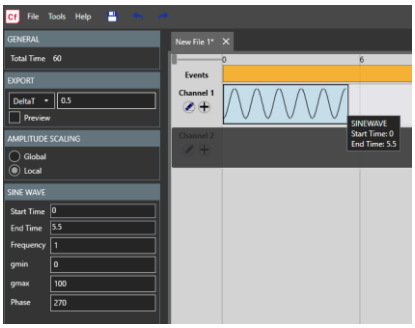


Generation of Unique Time Signals

The LOAD SCHEDULER also has the capability to add additional time signals as channels into events. The following options are available. These additional signals can be added in the same channel, as a subchannel to allow overlapping of signals or in a separate channel by themselves.

- ⇨ Create new event
- ⇨ Add sine wave signal
- ⇨ Add sine sweep signal
- ⇨ Add square wave signal
- ⇨ Add triangle wave signal
- ⇨ Add sawtooth signal
- ⇨ Add Block Loading signal
- ⇨ Add X, Y pair signal
- ⇨ Add static load signal





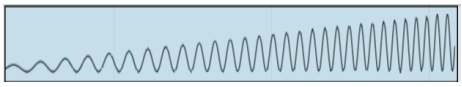
Sine Wave, Square Wave, Triangle Wave, Sawtooth Wave

By default, the signal length when dropped into a channel, will equal the Event length.

The User can then graphically change the length by clicking on the left or right blue box border and pulling the border to the right or left (but only within the boundaries of the Event box).

The box edge will move in increments determined by the DeltaT value. The User can also enter the start / end times for the signal in the dialog box.

Within the dialog box, the User can also change the frequency of the signal, the Min and Max heights of the signal and (for sine wave only) the starting Phase of the signal. The blue signal box will change as most entries are modified.



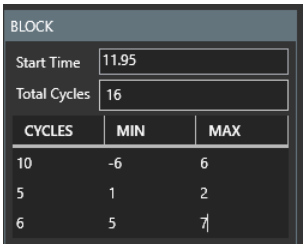
Sine Sweep

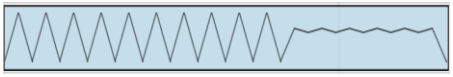
By default, the signal length when dropped into a channel, will equal the Event length.

The User can then graphically change the length by clicking on the left or right blue box border and pulling the border to the right or left (but only within the boundaries of the Event box).

The box edge will move in increments determined by the DeltaT value. The User can also enter the start / end times for the signal in the dialog box.

Within the dialog box, the User can also change the start and end frequency of the signal, the Min and Max heights of the signal and the starting Phase of the signal. The blue signal box will change as most entries are modified.



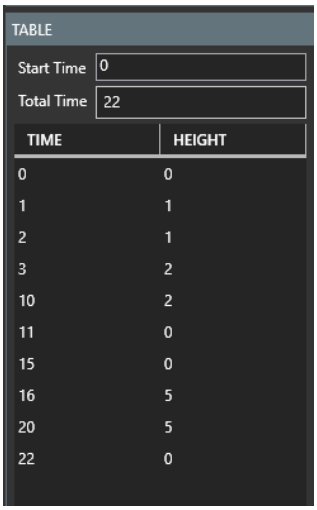


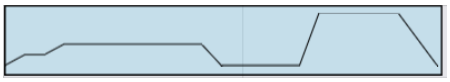
Block Loading

Block Loading permit the User to “build” a triangle wave of loading where each cycle is 1 second in duration.

The User cannot graphically change the size of this box. The length is controlled by the inputs in the dialog box.

Within the dialog box, the User must enter rows with Cycles / Min / Max values. If the User enters an incorrect value i.e., too many cycles for the length of the Event or a min value greater than the max value, the entry row will light up RED signifying a mistake that needs to be corrected.





X, Y Pair Loading

By default, the X Y Pair Loading box, when dropped into a channel, will equal the Event length, however it will collapse to 0 width when you first select the dialog box to start entering number pairs.

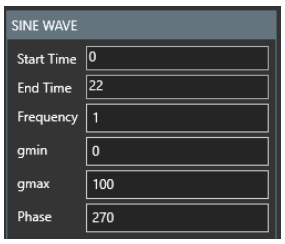
The User cannot graphically change the size of blue box. The length is controlled by the inputs in the dialog box.

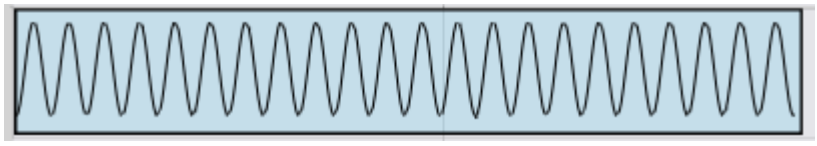
Within the dialog box, the User must enter rows of Time / Height values. The TIME inputs must be ascending in value. If the User enters an incorrect value, the value will change to RED signifying a mistake that needs to be corrected.

Potential Issues with Screen Resolution for Display

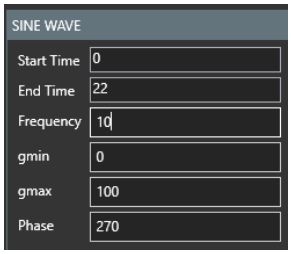
The BLUE box thumbnail provide on the screen is displayed at the available resolution of the screen. It is not uncommon for higher frequency signals to appear distorted due to the low available resolution of the screen.

Below is a thumbnail image of a 1 Hz sine wave as seen on the screen

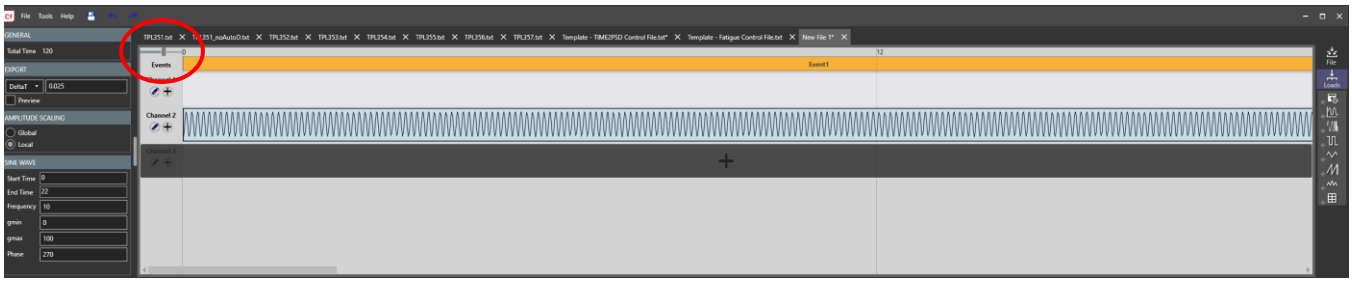




Below is a thumbnail image of a 10 Hz sine wave as seen on the screen. There is nothing wrong with the wave creation, but the computer screen resolution is unable to plot the wave correctly.



If we use the ZOOM slider (circled below), we can see that the 10 Hz sine wave has been accurately created. “Zooming in” allows the computer screen to apply more pixels to displaying the image correctly.

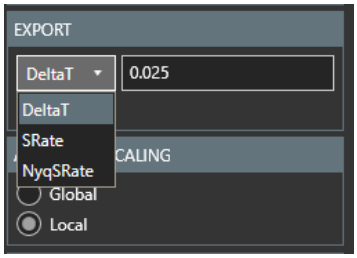


Therefore, if you see display issues, they are likely caused by the screen resolution.

Potential Issues with Signal Digitizing for Output

This is a very important section and should be clearly understood by the User.

When outputting channel data to a file, the Load Scheduler must “digitize” all the channels and subchannels within an event at the same sampling rate. This is REQUIRED when signals are being added together to make a single event of time history loading. Within the software, this sampling rate is specified by the User in the EXPORT section as **DeltaT** (Delta Time), **SRate** (Sampling Rate) or **NyqSrate** (Nyquest Sample Rate).



WHAT VALUE SHOULD I USE FOR SAMPLING RATE IN THE LOAD SCHEDULER?

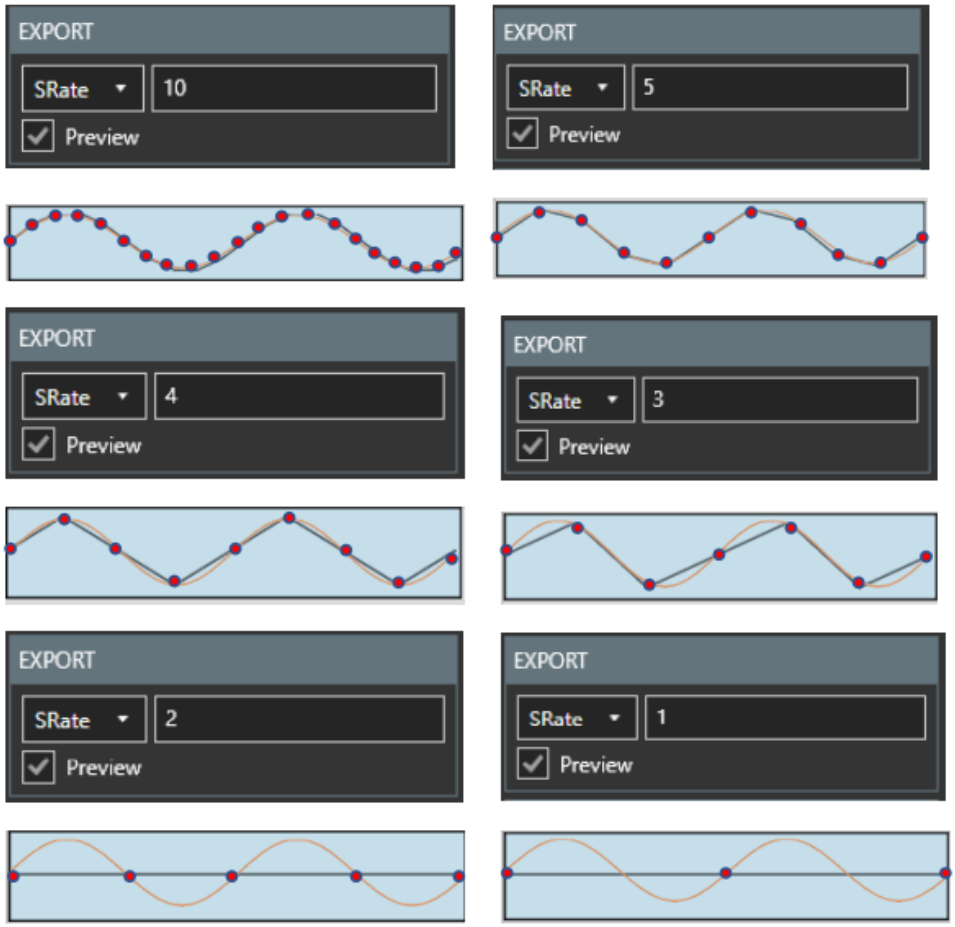
To properly digitize a signal and capture both signal frequency and signal amplitude, the User must allow for **20 points per cycle**. So, if the User wants to capture a Sine Wave of 1 Hz, they must select a sampling rate of 1 x 20 or 20 Hz. For a 40 Hz sine wave, the SRate would be 40 x 20 or SRate=800 or DeltaT=1/800=0.025. This will ensure that the discretized output signal will accurately reflect the correct frequency and amplitude of the intended signal.

Below are thumbnail images of the 1 Hz sine wave using a SRate = 20. The image on the left is the **SCREEN** thumbnail which is not affected by the SRate value. Regardless of the SRate, the sine wave will always look correct on the screen, taking into account the potential screen resolution issues discussed previously.

The image on the right uses the **PREVIEW** check box which shows how the digitized signal will look when **exported**. As can be seen, the two images are the same because we have used the proper 20 times (or more) SRate to capture the proper shape and amplitude of the sine wave for output.

What happens as we drop the sampling rate i.e., select less points to define the sine wave for export?

Below are thumbnail images of the 1 Hz sine wave using SRate = 10, 5, 4, 3, 2 and finally 1. The images use the PREVIEW check box to show how the digitized signal will look when exported. As can be seen, the quality of the sine wave shape deteriorates as the SRate decreases below the 20x rule, until the sine wave is represented by a flat line. This is because the sampling rate is too low to accurately digitize the shape and amplitude of the sine wave as it was intended for output by the User.



The discretizing issue noted above is applicable to all loads created in the Loads Scheduler tool. Some short cuts can be used if the signals are just square waves or just triangle wave (for example) because the shapes have flat sides and can be accurately defined (in some cases) with less points. However, if several waves overlap or are offset slightly than the higher resolution of 20 points per cycle will be needed. Therefore, it is recommended that the User always use a setting of 20 x highest frequency as the initial export SRate.

Example of Adding Additional Signals to Events

The Load Scheduler is capable of adding simple formats like sine waves, sine sweeps, square waves, block loading, etc. to a channel in an existing or new event. In the example on the next page, the original LOAD SCHEDULE example using an RPC file input has been modified to add:

Channel 3 – 3 Hz sine wave with an amplitude of 1.0

Channel 5 – 1 Hz square tooth wave with an amplitude of 2.0

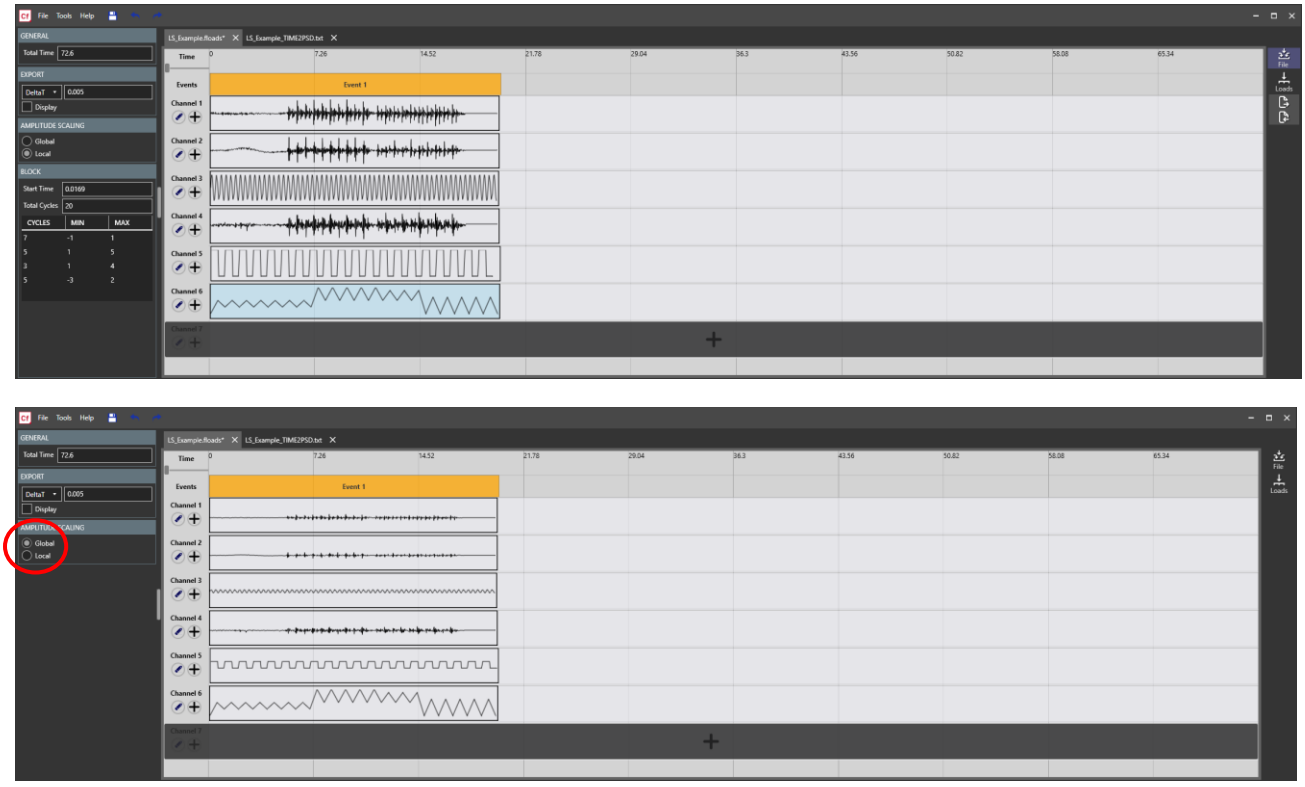
Channel 6 – 1 Hz block loading comprising various amplitude (up to 5) and block sections

This additional content varies significantly in amplitude compared to the original time signals brought in from the RSP file. This can be an issue for the User if this was not the intention. In the figures to follow, the LOAD SCHEDULER signals are show in LOCAL scaling, meaning each signal is plotted to fill the box, and shown in GLOBAL scaling, meaning each signal is plotted using the same scaling across all boxes.

GLOBAL scaling can give the User and immediate check to ensure the amplitudes entered for the loading is appropriate to what was intended.

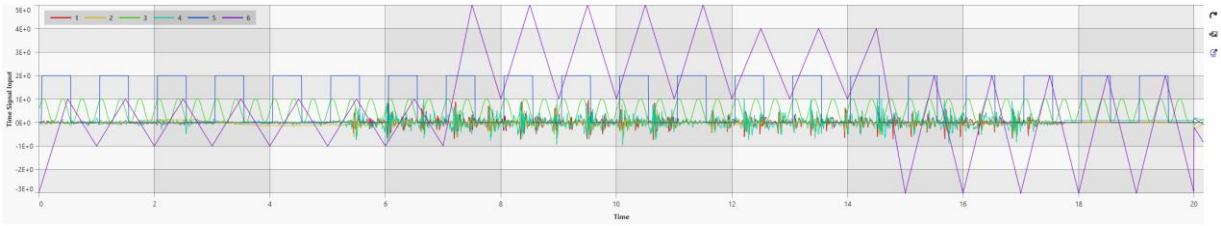
Figure 10-8 Load Scheduler with added content. TOP: Showing all signals at a LOCAL scale. BOTTOM: Showing all signals using same GLOBAL scale.

Note: BLOCK LOADING is much larger than the other loads and will dominate the event.



Below is a raw data image from the FFT Plotter what shows the time signals from the Loads Scheduler combined into one event. You can see from this plot that the BLOCK LOADING clearly dominates the event as depicted in the global view of the Loads Scheduler above.

Figure 10-9 FFT Plotter – Raw Data image representation of the Load Scheduler file above.



Please note: It would not be appropriate to convert this data into the Frequency Domain because several of the signals are not RANDOM. All signals in an event must be Stationary, Random and Gaussian in order to be properly converted into the Frequency Domain. The above event however can be used in a Time Domain analysis without issue.

11 | INTRODUCTION TO THE TIME DOMAIN LOAD SCENARIO CREATER

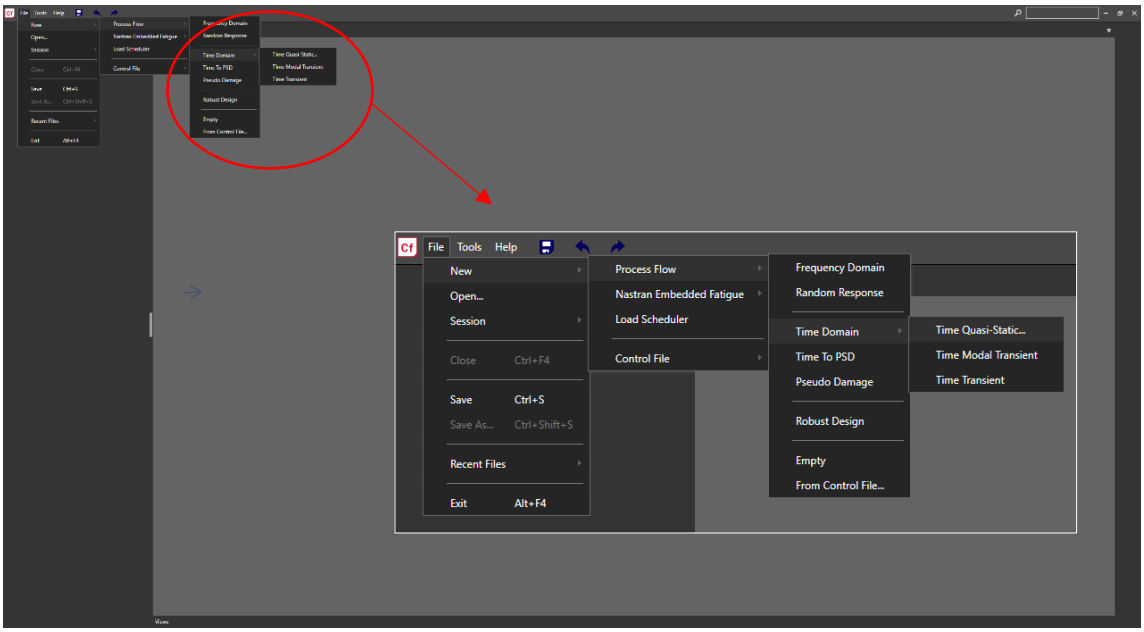
NOTE: The images shown in this section reflect CAEfatigue v2022 and may change slightly with future releases.

Overview and Introduction to the Load Scenario Creator for Time Domain Linear or Nonlinear Static Analysis

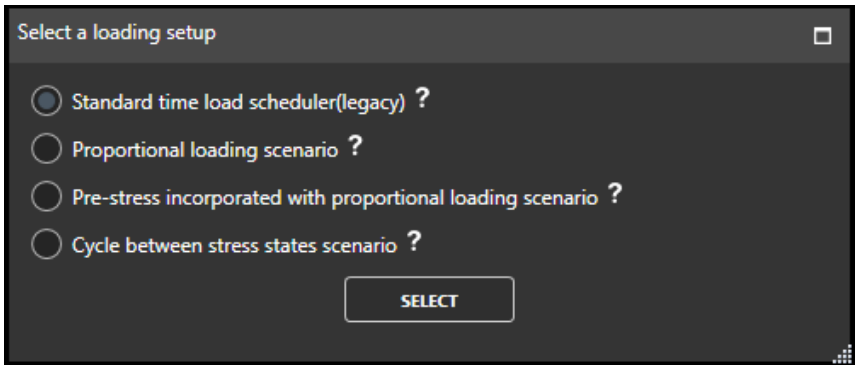
To start CF, click on the desktop Cf icon (see below) or find the ProcessApp.exe file for the program in the default installation directory “C:\Program Files\MSC.Software\CAEfatigue\<ver>\Windows\”.

This will open CF in the CFG application window as shown below (Cf remembers the last file you were working on). If can choose to open a new Load Scheduler project by selecting **FILE>NEW>PROCESS FLOW>TIME DOMAIN>TIME QUASI-STATIC.**

Figure 10 Opening a Loads Scenario template from a Quasi-Static request.



After selecting Time Quasi-Static, the User is presented with several template options. The standard Time Load Scheduler (legacy) is the original quasi-static template with a simple on/off triangle loading (R-ratio = 0). The remaining templates provide the User with the ability to automatically create loading time histories to support their analyses. Below are the template options available

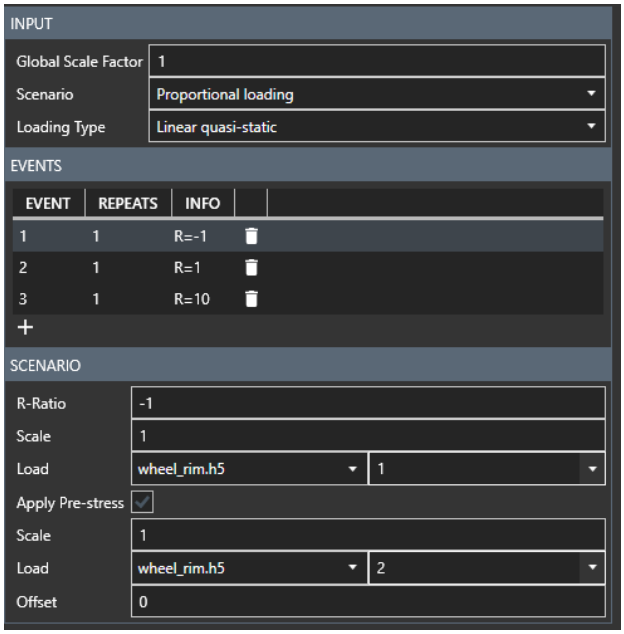


Proportional Loading Scenario

This load scenario provides the ability to create multiple events with a single time history of loading created per event. The time history of loading will be created to satisfy a User specified **R-ratio**. For example, an R-ratio of -1 means that the negative stress peak has the same absolute value as the positive stress peak. Hence, the loading is fully reversed between tension and compression.

Below are some examples of R-ratio values, where R-ratios is the minimum peak stress is divided by the maximum peak stress while retaining the sign. A positive value ($0.0 < \text{value} \leq +\infty$, excluding 1.0) indicates a **tensile** stress whereas a negative value ($-\infty \leq \text{value} \leq 0.0$) indicates **compressive** stress.

An example of a linear quasi-static setup is shown below. It allows the User to adjust several parameters while created the proportional loading time histories.



Global Scale Factor: Adjust the scale factor for all loads via the vFTGSEQ>TSCALE entry.

EVENTS>REPEAT: Adjust the number of times the 1 second time history is applied. This sets the event duration.

EVENT>INFO: Adjust the shape of the generated time history loading to match a User specified R-ratio as explained earlier.

SCENARIO>R-Ratio: Adjust the r-ratio to use in the generation of the loading time history. R-ratio can be any value except 1.

SCENARIO>SCALE: Adjust the event scale factor via vFTGLOAD>SCALE.

SCENARIO> LOAD: Select the MSC Nastran load case that applies to the loading being created.

Apply Pre-Stress: A checkbox that allows the addition of a prestress loading with the ability to adjust the prestress Scale and Offset.

For the example above, the CSV file created for Event = 1 with R = -1 looks like below.

	A	B
1	Time	Channel 1
2	0	-1
3	0.5	1
4	1	-1

The min and max values for Channel 1 have the same absolute magnitude, which is the expected result when R = -1.

The duration of the signal has been set to 1 second, which allows an easy use of REPEATS to set an appropriate duty cycle duration. For example, we can set REPEATS=3600 to create a duty cycle of 1 hour.

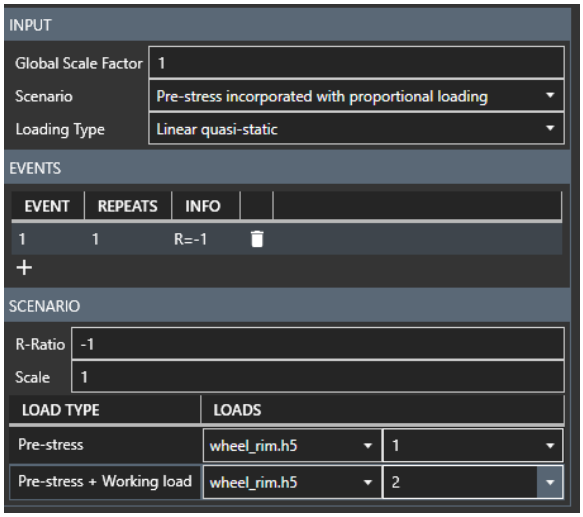
Pre-stress Incorporated with Proportional Loading Scenario

Sometimes a User does not have simple proportional (working) load and prestress load as two separate solver files that can be used as shown in the previous loading scenario. For some solvers such as MSC Nastran SOL400, Abaqus, Simufact, Marc and others, the User has a prestress applied in the first subcase (or also called “step 1” load case), followed by a second subcase / (or “step 2” load case) where a same prestress is applied **plus** a maximum working load is also applied.

Ultimately, the User is trying to calculate damage if he/she oscillates between the prestress + max working load and the prestress + min working load, but the solver stress results do not readily permit this. Hence, CAEfatigue has developed a loading scenario that manipulates the stress results to give us the desired loading effect.

This load scenario is similar to Proportional Loading in that it provides the ability to create multiple with time histories that are created to satisfy a User specified **R-ratio**.

An example of a linear quasi-static setup is shown below. It allows the User to adjust several parameters like we did in the previous scenario. However, in this scenario, the Load Types are a **Prestress** load and a **Prestress + Working Load (WL)**. Below is an example of what the setup might look like using a dummy H5 file.



To get the desired loading cycle, i.e. oscillating between prestress + max WL and prestress + min WL, we create a load time histories for each load type. For the example above, the CSV file created for Event = 1 with R = -1 looks like below. Channel 1 is applied to the “Pre-stress” loading and Channel 2 is applied to the “Pre-stress + Working load” loading, which is really prestress + maximum working load.

	A	B	C
1	Time	Channel 1	Channel 2
2	0	0	1
3	0.5	2	-1
4	1	0	1

The values for the channels are calculated as follows:

Channel 1 time history goes from “0” to “1 - R value” to “0”, where R value = -1.
Channel 2 time history goes from “1” to “R value” to “1”, where R value = -1.

At TIME 0, the first prestress loading is OFF but the second prestress+WL is ON. This means we only have the desired condition “**prestress + max WL**”, when we initially kick off the analysis. At time step 0.5, we turn ON the first prestress loading but double the loading while completely removing / subtracting the second prestress+WL loading. This results in only a single prestress being applied along with a minimum working load, hence the desired condition of “**prestress + min WL**” is created. At the final time step 1.0, we return to the desired condition “**prestress + max WL**”. This completes the desired cycle the User was trying to achieve.

Cycle Between Stress States (ON/OFF) Scenario

This loading scenario is very common and somewhat tedious to set up manually. It involves a series of load cases from a single solver file that are applied in sequence, i.e. one load is turned OFF as the next load is turned ON. There is no need to adjust the amplitude of the time history as it should only go from OFF/ON/OFF or 0/1/0 in amplitude.

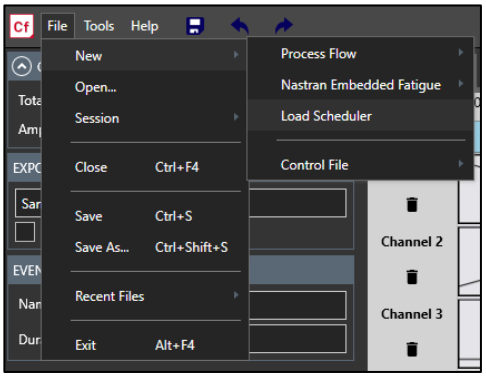
An example of a linear quasi-static setup is shown below. It allows the User to adjust several parameters like we did in the previous scenarios. However, in this scenario, the LOADS are supplied from a single solver file with multiple subcases. Below is an example of what the setup might look like using a dummy H5 file. The H5 file has been created with 12 subcases representing different stress results (starting at 0 degrees) of a wheel as it rotates in 30 degree increments. The #12 subcase represents the wheel at 330 degrees. The next position (360 degrees) is the same as the first position (#1) and will be automatically accounted for in the CSV loading file.

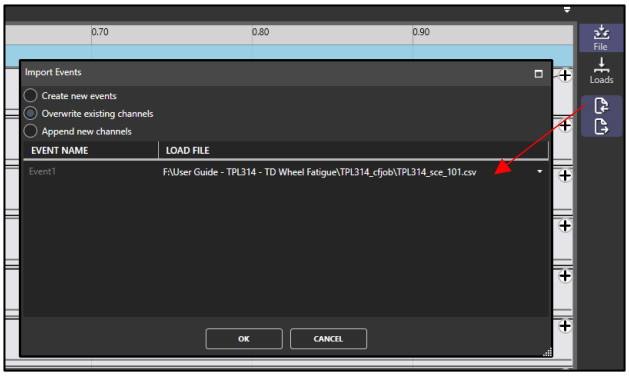
The CSV file below represents what was generated from this loading scenario. As can be seen, at TIME 0 the first subcase is already ON (i.e., the value is 1.0). At the next time step 0.083, Channel 1 is turned OFF while Channel 2 is turned ON. This pattern continues to time step 1.0 where Channel 12 is turned OFF and Channel 1 is turned back ON, representing closing loop that Channel 1 was missing (see explanation below).

	A	B	C	D	E	F	G	H	I	J	K	L	M
1	Time	Channel 1	Channel 2	Channel 3	Channel 4	Channel 5	Channel 6	Channel 7	Channel 8	Channel 9	Channel 10	Channel 11	Channel 12
2	0	1	0	0	0	0	0	0	0	0	0	0	0
3	0.083333	0	1	0	0	0	0	0	0	0	0	0	0
4	0.166667	0	0	1	0	0	0	0	0	0	0	0	0
5	0.25	0	0	0	1	0	0	0	0	0	0	0	0
6	0.333333	0	0	0	0	1	0	0	0	0	0	0	0
7	0.416667	0	0	0	0	0	1	0	0	0	0	0	0
8	0.5	0	0	0	0	0	0	1	0	0	0	0	0
9	0.583333	0	0	0	0	0	0	0	1	0	0	0	0
10	0.666667	0	0	0	0	0	0	0	0	1	0	0	0
11	0.75	0	0	0	0	0	0	0	0	0	1	0	0
12	0.833333	0	0	0	0	0	0	0	0	0	0	1	0
13	0.916667	0	0	0	0	0	0	0	0	0	0	0	1
14	1	1	0	0	0	0	0	0	0	0	0	0	0

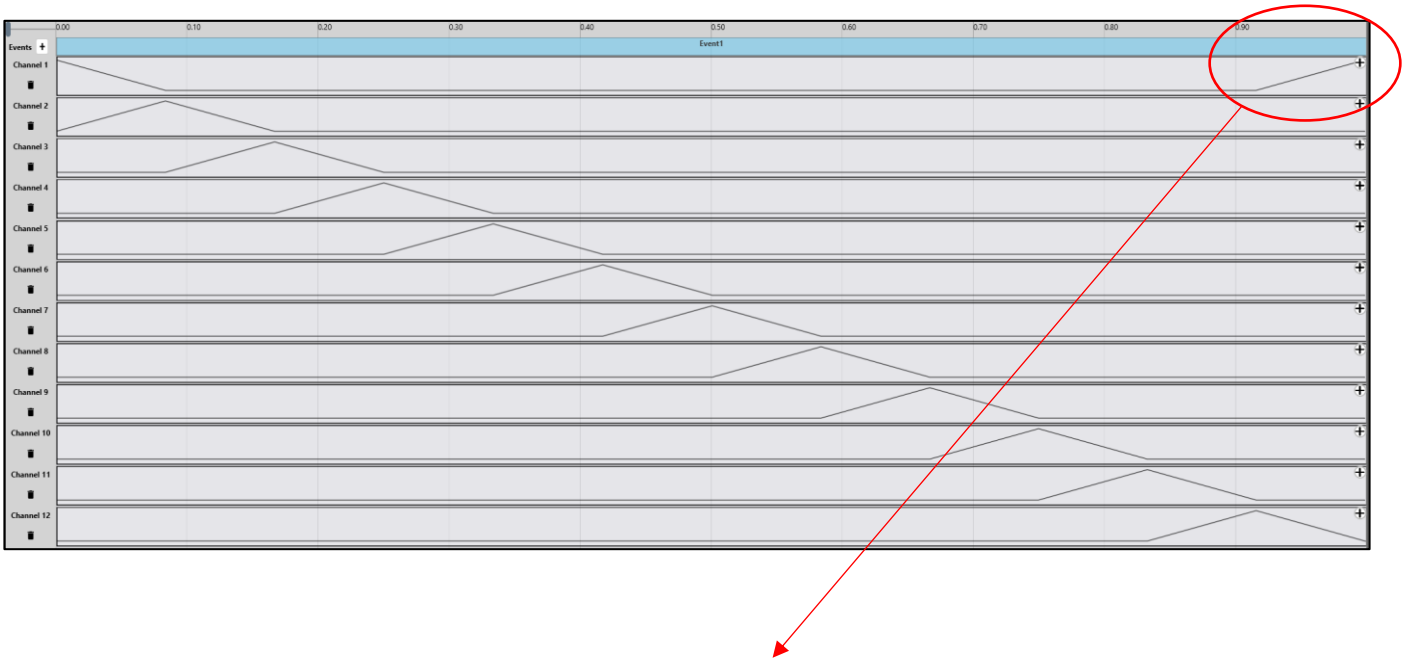
What does the loading look like?

If we take the autogenerated CSV file into the Cf Load Scheduler, we can see the “triangle” ON/OFF loading that CAEfatigue is applying in this example.





Below is the loading profile represented in the CSV file.



The Load Scenario adds the closing half cycle of loading by returning the status of Channel 1 back to ON at the end of the loading cycle. By doing this we ensure each subcase has a complete OFF/ON/OFF cycle.

IMPORTANT TO NOTE

CAEfatigue expects that the load cases provided by the solver go from 0 degrees (starting position) to an ending position that is one rotation increment short of completing a full 360 degree revolution. CAEfatigue then completes the loading cycle as show above.

12 | INTRODUCTION TO THE RELIABILITY MODE FOR RUNNING MULTIPLE JOBS WITH VARYING INPUTS

NOTE: The images shown in this section reflect CAEfatigue v2021 and may change slightly with future releases.

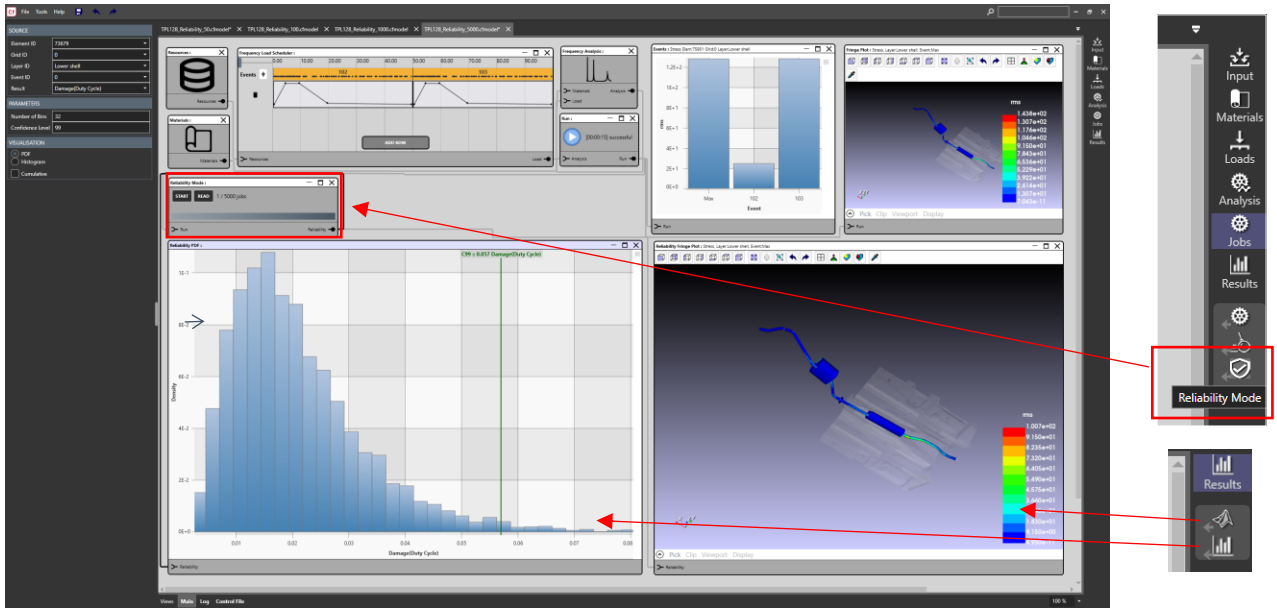
Overview and Introduction to the Reliability Mode

Reliability Mode is a capability that can be added to both time domain and frequency domain runs within CAEfatigue. The capability permits the user to vary parameters using User defined probability distributions and Cf will do the analysis repeatedly and provide a distribution of selected output results.

To use the **Reliability Mode**, simply add the Reliability Mode feature to any Cf analysis. For example, below is the Reliability Mode feature added to TPL128. The Reliability Mode box is brought into the Process Flow workspace along with the Reliability PDF Viewer and Reliability Fringe Viewer.

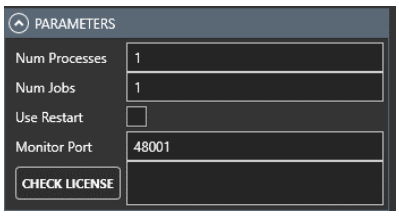
Note: The Reliability Mode PDF and Fringe viewers are unique to the Reliability Mode and can only be accessed when the User selects the Reliability Mode box.

Figure 11 Adding the Reliability Mode feature with PDF and Fringe viewers to TPL128.



Review of Reliability Mode Menus from within the Process Flow

When the Reliability Box is selected, a left side menu appears to allow input of various parameters for input and output for the Reliability analysis.



PARAMETERS

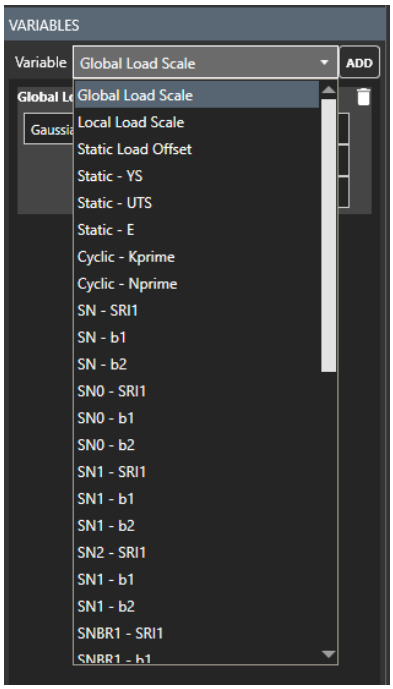
Num Processes: Select the number of simultaneous jobs you want the run. You must have this value of available licenses plus one to run the GUI, or the program will not run for lack of licenses. Note: once started the Reliability Mode will continue to run even if the software is closed (to free up the GUI license).

Num Jobs: Select the number of CF jobs you want to run as part of the Reliability analysis. The default is 1, but CF will run a maximum of 4 simultaneous jobs if the licenses are available.

Use Restart: CF will use the restart files (*.cfvdata) to speed up the analysis process. **This means that an initial analysis must be run first prior to using the Reliability Mode**, in order to generate the RESTART files.

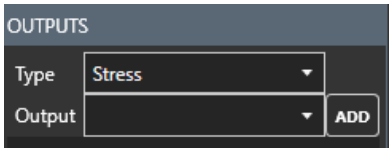
Monitor Port: CF use a Dispatcher program to interrogate the Reliability Mode process as it is running. CF monitors the process through a server port and arbitrarily selects port 48001 as default for this process. This port can be change if the CF program reports that port 48001 is in use.

Check License: Allow a quick check of available licenses so the User can determine how many simultaneous jobs to run, while keeping one license for the open GUI.



VARIABLE

Variable: The User can select from a large number of variables to modify as part of the analysis using various distributions.



OUTPUTS

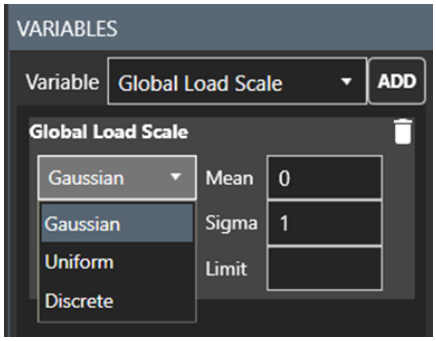
Type: Only STRESS type output is available at this time.

Output: Select / Add the types of output you want to see from the analysis. See below for more information on Output.

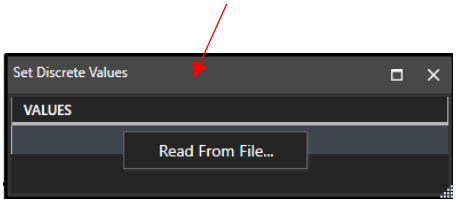
Options for Adjusting Variables and Selecting Outputs

CF allows the User to adjust a number of variables as part of the analysis. Below is an example of Global Load Scale being adjusting by a small amount over numerous runs. If desired, more than one input variable can be adjusted in the same Reliability Mode analysis. The input variables are adjusted by following a known Gaussian, Uniform or Discrete distribution. The User is required to pick the distribution type and add the parameters to follow. These parameters define how tightly you want to change the variable.

If at least one of the variables follows a Discrete distribution, the parameters from all variables will be the combined space of all combinations of the discrete values.



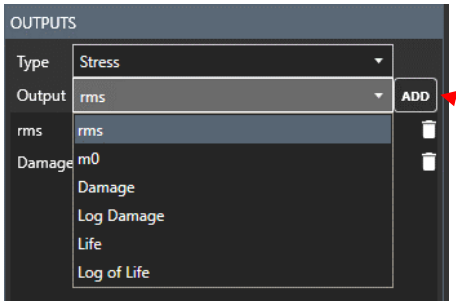
The User can enter Discrete variables using direct entry of values or, with a right mouse click on the Values title, can enter the variables via a CSV/TXT file of values, each entered on a separate line like below.



Format of Discrete Variables file

0.9
1.0
1.1

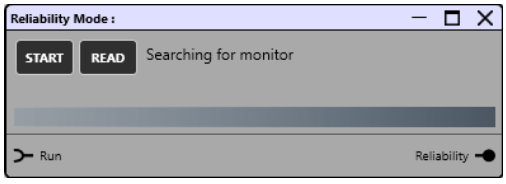
The Output options from a Reliability analysis are listed via the Output box in the OUTPUTS section. The options are RMS stress, spectral moment (M_0), Damage and Life. The User must select a type of output and then select the ADD button. More than one output can be chosen.



The User must select at least 1 output and use the ADD button to add it to the list of output items.

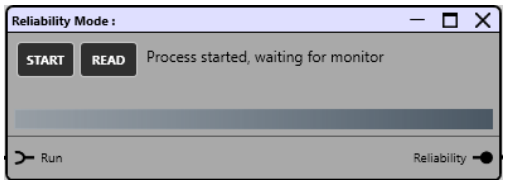
Review of Reliability Mode Progress Box and Output Plots

After the Reliability analysis is set up, the User must select START to initiate the analysis. READ can be used to view the analysis results when the analysis is complete or anytime during the analysis, as desired.

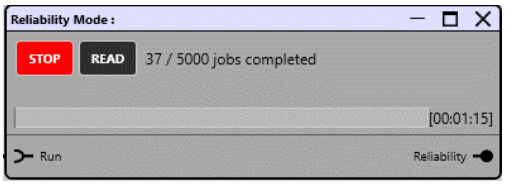


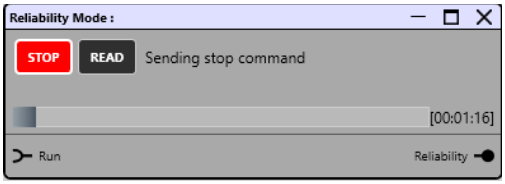
Note:

"Searching for Monitor" means the program is ready to communicate with the Reliability Mode analysis (through the Monitor Port) to read/display the results as the job is being run.



Once START is selected, the Reliability Mode program will open the Monitor Port and initiate communications. This may take a minute to complete before the job begins.





Once the port is open, the Reliability Mode job will start running and show the number of jobs completed. The User can select READ at any time to see the results of the jobs that have completed. The first jobs that run are at the limits for each input variables (n). This results in 2^n jobs that are for the limits only, followed by the remaining jobs requested.

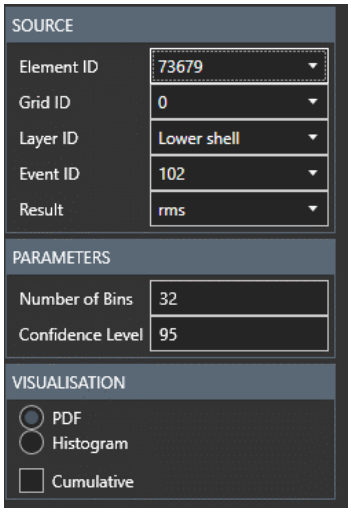
If desired, the User can select the STOP button to terminate the Reliability Mode analysis. If pressed, the Dispatcher program will send a signal to the program to stop. This may take a minute to complete.

See **TPL128_Reliability** for a full example using this feature.

Reliability PDF Plotter and Fringe Plotter

Both the Reliability PDF and Fringe plotter are unique to the Reliability Mode and can only be accessed in the right side menu when the Reliability Box is selected.

The Reliability PSD plotter is similar to the Cf Event Plotter but with additional features.



Similar to the Event Plotter, the SOURCE section is the same.

PARAMETERS

Number of Bins: This is the number of bins that will be used in the probability density function (PDF) and histogram plots.

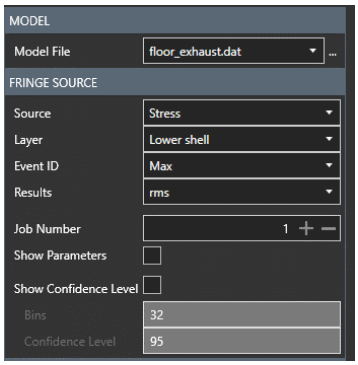
Confidence Level: This is a confidence level the User can set and will show a value on the PDF and histogram plots. Example, 95% means that there is a 95% confidence that the output will not exceed this value.

VISUALISATION

PDF / Histogram: These are the 2 types of plots you can show.

Cumulative: This changes the plot to show cumulative values.

The Reliability Fringe plotter is similar to the Cf Fringe Plotter but with additional features.



Compared to the CF Fringe Plotter, the MODEL and SOURCE section are very similar with the following additions:

Job Number: This allows you to plot results for any job that was run in the analysis.

Show Parameters: This will add a small box on the Fringe plot that contains the input variable information that was used for a particular job.

Show Confidence Level: This will show a Fringe plot for the Bins and Confidence Level values chosen by the User

Files Required and Generated by the Reliability Mode

The Reliability Mode feature can be added to any new or existing CAEfatigue analysis hence, the same inputs for any Cf analysis are also required for a Reliability Mode analysis. The Reliability Mode can also take advantage of the RESTART files, <filename>.cfvdata. Hence, prior to doing a Reliability Mode analysis, ensure that the original Cf job is run once to create the Restart files.

Once selecting START, the Reliability Mode generates a number of files, which are stored in the working directory.

<filename>_reliability.txt	This is a copy of the original Cf control file and is referred to as the **template file**. It is used by the Reliability Mode to generate the _1, _2, etc. TXT files needed to complete the requested reliability analysis.
<filename>_1.txt through to the number of simultaneous jobs requested	These are the control files that are run up to the number of jobs requested.
<filename>_1. XXX through to the number of simultaneous jobs requested	These are the standard Cf output files that are created for each job that is run. These file types can be CSV, LOG, PSD, d_rel, _dva, etc.
<filename>_dispatcher.log	This is the Reliability Mode LOG file that is generated when the Reliability Mode is run. Any issues with the Reliability Mode analysis will be reported in this file.
<filename>_reliability.json	This is the **configuration file** that is needed for BATCH operations within Linux of Windows. It defines all the Parameters, Input Variables and Outputs required.
<filename>_reliability.h5	This is the main output file that contains all the Reliability Mode results. The User needs an HDF5 viewer program to see the contents of this file.

Contents of the JSON Configuration File

The Reliability Mode requires a configuration file to run either via the Windows GUI or in batch mode for Windows and Linux. The file is most easily created by setting up the job within the Process Flow GUI, which creates the JSON file automatically. Otherwise, the User must create the file manually. Below is the JSON file used in **TPL128_Reliability_100.cfmodel**.

NOTE: To run this job outside the Process Flow GUI, the User must also create a template file (control file) <filename>.TXT that defines the job to be done. The Process Flow GUI will also create this file automatically if the Reliability Mode is defined within the Process Flow. The automatically created JSON and TXT files can then be used in BATCH mode.

Example of a JSON configuration file:

```
{
  "Parameters": {
    "JobName": "TPL128_Reliability_100",
    "TemplateFile": "TPL128_Reliability_100_reliability.txt",
    "OutputDirectory": "",
    "RestartJobName": "TPL128_Reliability_100",
    "OutputFile": "TPL128_Reliability_100_reliability.h5",
    "NumProcesses": 10,
    "NumRuns": 100,
    "MonitorPort": 48001,
```

```
    "OutputCSV": false,
    "ResultType": "stress"
  },
  "Variables": [
    {
      "Name": "v1",
      "Description": "Global Load Scale",
      "Distribution": {
        "Type": "gaussian",
        "Mean": 1.0,
        "Sigma": 0.1,
        "Limit": 3.0
      }
    }
  ],
  "Outputs": [
    {
      "Name": "",
      "Column": "rms"
    },
    {
      "Name": "",
      "Column": "Damage"
    }
  ]
}
```

Example of a template file used with above JSON file:

```
$ Generated by CAEFatigue (2021.3.1 6087)
$
vibfat  777     csv     Nastran center  0       TPL128_Reliability_100
        OPTIONS         1
vmatftg 60      1.0     range
        static          358.5   200000  99
        en      560.8   -0.109  0.065   -0.39   461.7   0.12
vmatftg 61      1.0     range
        static          350     200000  99
        en      570     -0.109  0.065   -0.39   461.7   0.12
vmatftg 62      MPa     range
        static          4000    203000  99
        sn      27344   -0.332  1e+07   -0.332  1e+10
fe_units777     MPA     1       B       N       mm
vftgdef 777     nb      100
        elset   12      60      11      61      10      62
        absresp DA      7
        relresp D       8       S1.0
        psdout  D       10001   A       10002   S       10003
        psdmout DA      21664   123
include F:\User Guide - TPL128_Reliability\Set\Set12.txt
include F:\User Guide - TPL128_Reliability\Set\Set11.txt
include F:\User Guide - TPL128_Reliability\Set\Set10.txt
include F:\User Guide - TPL128_Reliability\Set\Set7.txt
include F:\User Guide - TPL128_Reliability\Set\Set8.txt
fset3   10001   NODE
        21664
fset3   10002   NODE
        21664
fset3   10003   ELEM
        81873   77336
vftgparm777
        stress  SGVON   None    naver   None            m5.0
vftgseq 777     1       hours           {v1}    no
        101     50      102     50
vftgevnt101     401
        name    102
vrandps 501     1       1       1       0       60
vftgevnt102     402
        name    103
```

```
vrandps 502     2       2       1       0       61
op2map  F:\User Guide - TPL128_Reliability\Solver\floor_exhaust.op2
        1       2
        2       3
vftgload401     psd     501     1
 1
vftgload402     psd     502     1
 2
vtabrnd 60      linear  linear
        0       0       2       1E-06   10      0.02    20      0.02
        40      0.001   100     1E-06   endt
vtabrnd 61      linear  linear
        0       0       2       1E-06   10      0.02    20      0.02
        40      0.001   100     1E-06   endt
```

The JSON format has 3 sections: (1) PARAMETERS, (2) VARIABLES, and (3) OUTPUTS.

Parameter Section: (Parameters are not case sensitive)

JobName	Job name, all generated control files will be named after it
TemplateFile	File path to the template file to use
WorkingDirectory	Working directory for each simulation, can be left out
OutputDirectory	Where results from each run is stored
RestartJobName	In frequency domain, set the name of the job to restart from
OutputFile	File path to output database file, stored in HDF5 format
NumProcesses	Number of concurrent processes to run
NumRuns	Total number of runs to execute. For random distributions, the larger the number the more reliable the results.
LicensePerJob	Flag to limit the number of jobs allowed per run in Linux. Default is false but if true, the command line option “-max-jobs=<integer>” will be used in the Linux analysis.
MonitorPort	Start the monitor at this port so GUI can monitor progress. Default 48001. If set to 0 then
InstallationFolder	Path to your installation of CAEfatigue, e.g. /msc/CAEfatigue/2021_2
ResultType	What result type to include in the output. Only “stress” is supported in this release.
SaveFailingJobs	If a job fails, both the control and log file is saved with the job number prepended. Default false.
SeedValue	Set the seed value for random parameters. If set to 0, it’s automatically picked. Default 0.
BatchFilename	If set, the reliability dispatcher will call this batch file instead of CAEfatigue, which can be useful if a different queue system is used in e.g. HPC environments.

Variable Section

Name	This name should be short <6 characters and is used in the template file
Description	A description of the variable, will be stored in the output file.
Distribution	See below for different distribution types

Gaussian (normal)

Type	“gaussian”
Mean	Mean value of distribution
Sigma	Standard deviation of distribution
Limit	Limit given in a factor of standard deviations; all numbers will be limited to be within (Limit)*(standard deviation)

Uniform

Type	“uniform”
Min	Minimum value
Max	Maximum value

Discrete

Type	“discrete”
Values	List of double values entered directly or through CSV/TXT file as separate rows, each with one discrete value.

Outputs Section

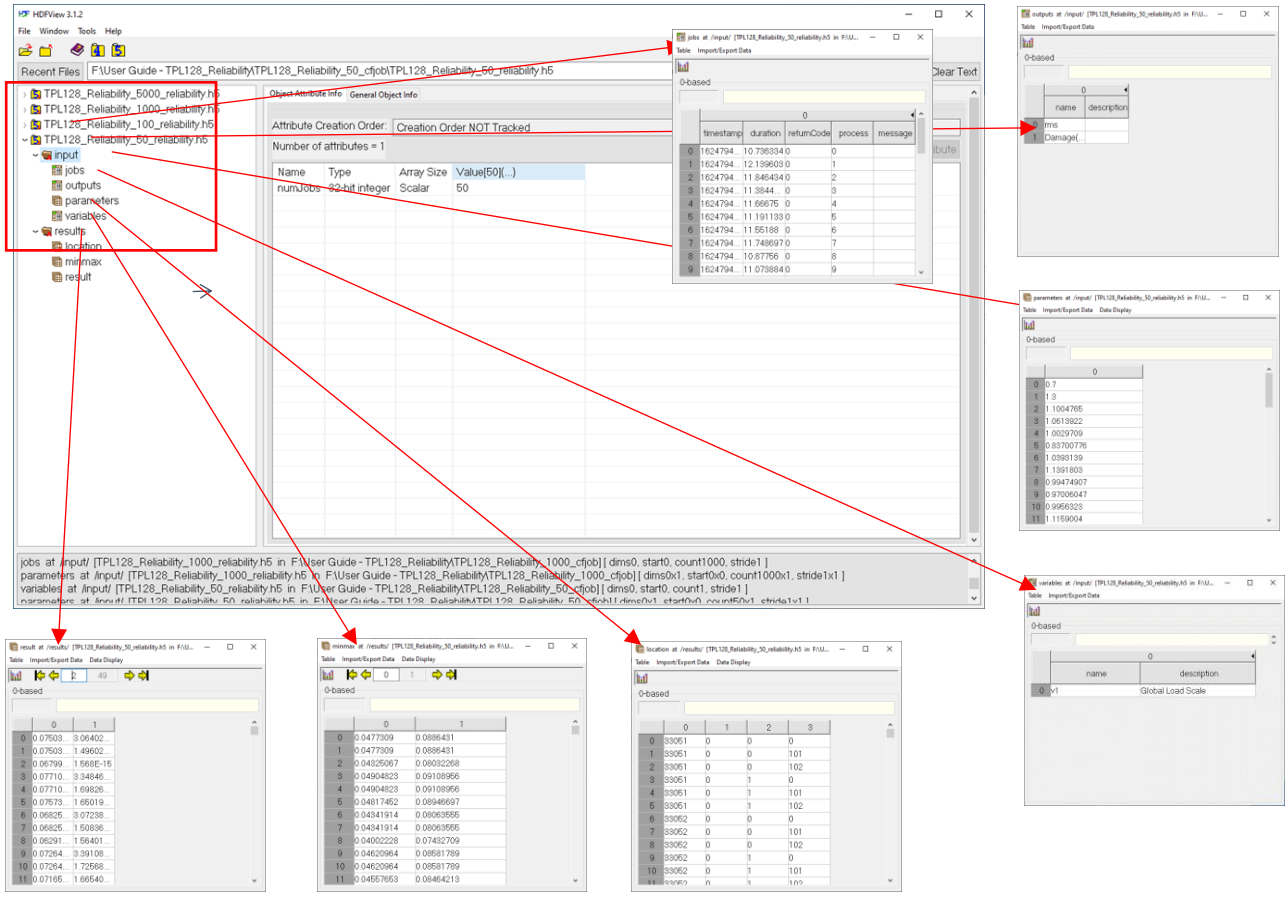
Name	Name of output (can be blank)
Column	Column name in a job result file. It’s sufficient to have a partial name, e.g. “Damage” instead of the full “Damage(Duty Cycle)”

Viewing and Plotting Results from the HDF5 file.

The Reliability Mode puts all results into an HDF5 format. To view these results, the User needs to have an HDF5 viewer. Below is the HDF5 results as seen using **HDFView** by the **HDF® View** company. The User can select Input values or Result values to view by selecting the appropriate sub-menu. Examples are show below.

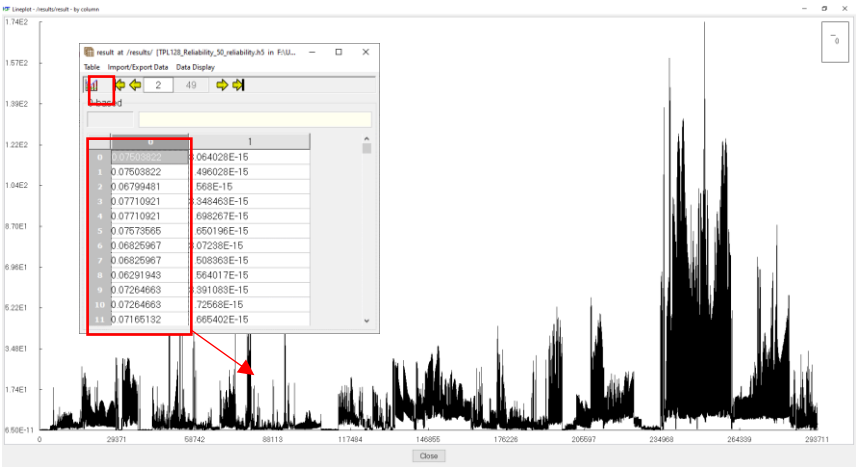
Input - Jobs	Provides a list of all jobs including the duration of each run and the simultaneous job number (process number) of each job.
Input - Output	Provides a list of the requested output types.
Input - Parameters	Provides a list of the input variable values used for each analysis.
Input - Variables	Provides a list of the input variable names.
Results - Location	Provides the element, grid, layer and event ID where the data will be analyzed.
Results - MinMax	Provides the smallest and largest results for all selected Outputs, across all locations after all jobs are complete. There is a complete data set for the smallest value and a complete data set for the largest value.
Results - Result	Provides all result values for all output types requested. There is a complete data set for each job. User the arrows to select which job to review.

Figure 12 Viewing the HDF5 data with HDFView (3rd party viewer).



For the MinMax and Result data tables, the User can also plot the information shown in the table by selecting a column (or row) of interest and selecting the plot button.

Figure 13 How to plot data from the HDFView data tables.



To the left is the **Results** data table.

Column 0 (RMS output) is plotted for JOB ID=2, which is the upper sigma limit value analysis.

Individual data points are plotted for every element, grid, layer and event location in the analysis.

Running Reliability Mode in BATCH within Windows or Linux

The Reliability Mode can be run in Batch mode within Windows or Linux. Simply use the command line entry:

```
caefatigue -reliability-mode <configuration file>
```

Where the configuration file is the JSON file described earlier and within the JSON file is a needed Template file.

13 | INTRODUCTION TO ROBUST DESIGN FOR RUNNING MULTIPLE MSC Nastran JOBS WITH VARYING INPUTS

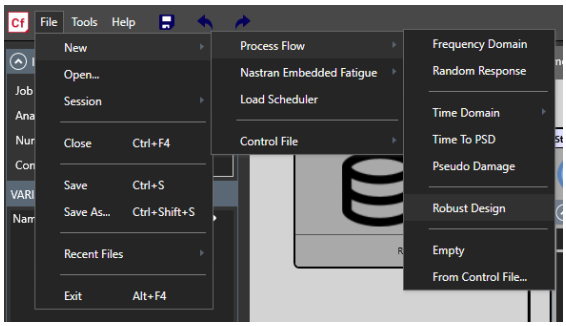
NOTE: The images shown in this section reflect CAEfatigue v2022 and may change slightly with future releases.

Overview and Introduction to the Robust Design

Note: This is the initial release of Robust Design and therefore, has some limitations with respect to the Nastran analyses it will interact with and the output parameters it will support. This release will be significantly improved with subsequent releases, and we invite feedback from our customers as to what is needed to improve this capability further.

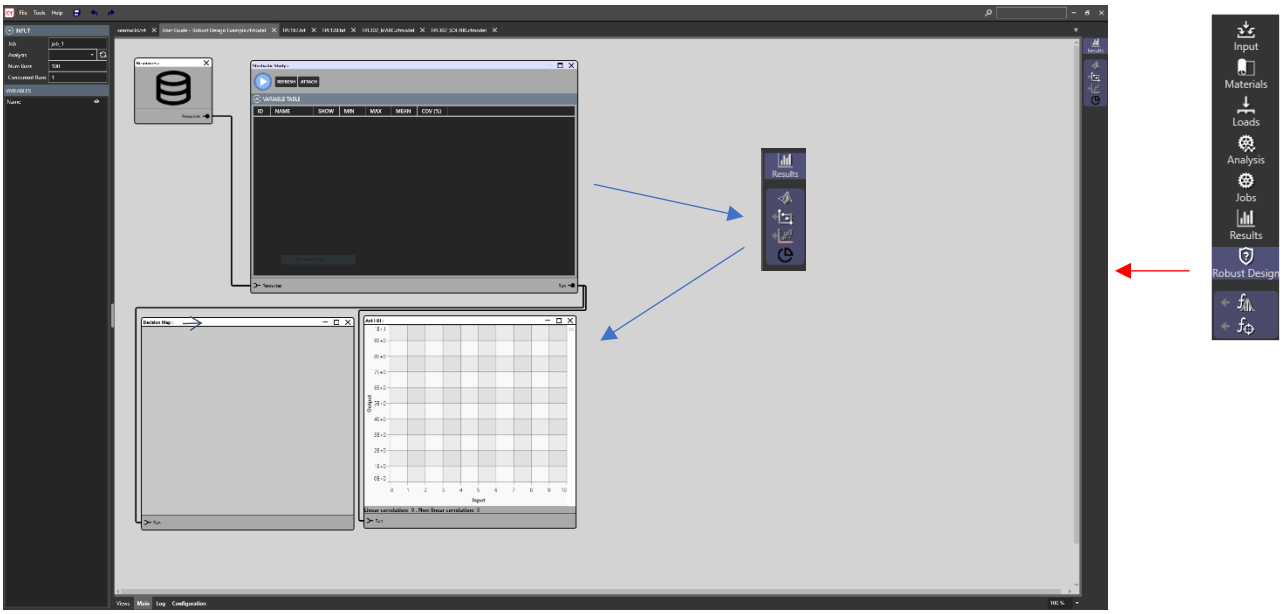
Robust Design can be used to include uncertainty into the inputs for a MSC Nastran **SOL101** and **SOL103** analysis in order to perform stochastic simulation. This is done by specifying the tolerances and scatter on any or all of the input variables used in the model i.e., variables like thickness, material properties, spring stiffness, beam cross-sections, loads, imposed displacement, etc.

To use the **Robust Design**, simply select the Robust Design template from the FILE menu system shown below. A User can also create the template from scratch by using the menu system on the right of the software.



Below is the Robust Design template that will be generated from the menu.

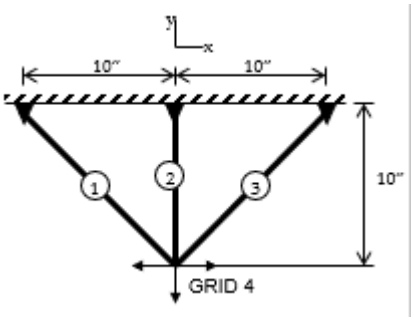
Figure 14 The Robust Design template or selected from scratch from right menu.

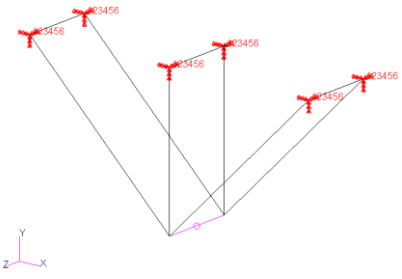


Review of Robust Design Menus from within the Process Flow

Within the Robust Design template, there are several boxes that provide the input required for the analysis. Below is an explanation of the boxes and menus using a very simple 3 shell element model. Within the Robust Design template, there are several boxes that provide the input required for the analysis. Below is an explanation of the boxes and menus using a very simple 3 shell element model. The nominal properties of the MSC Nastran™ model prior to any optimization are as follows.

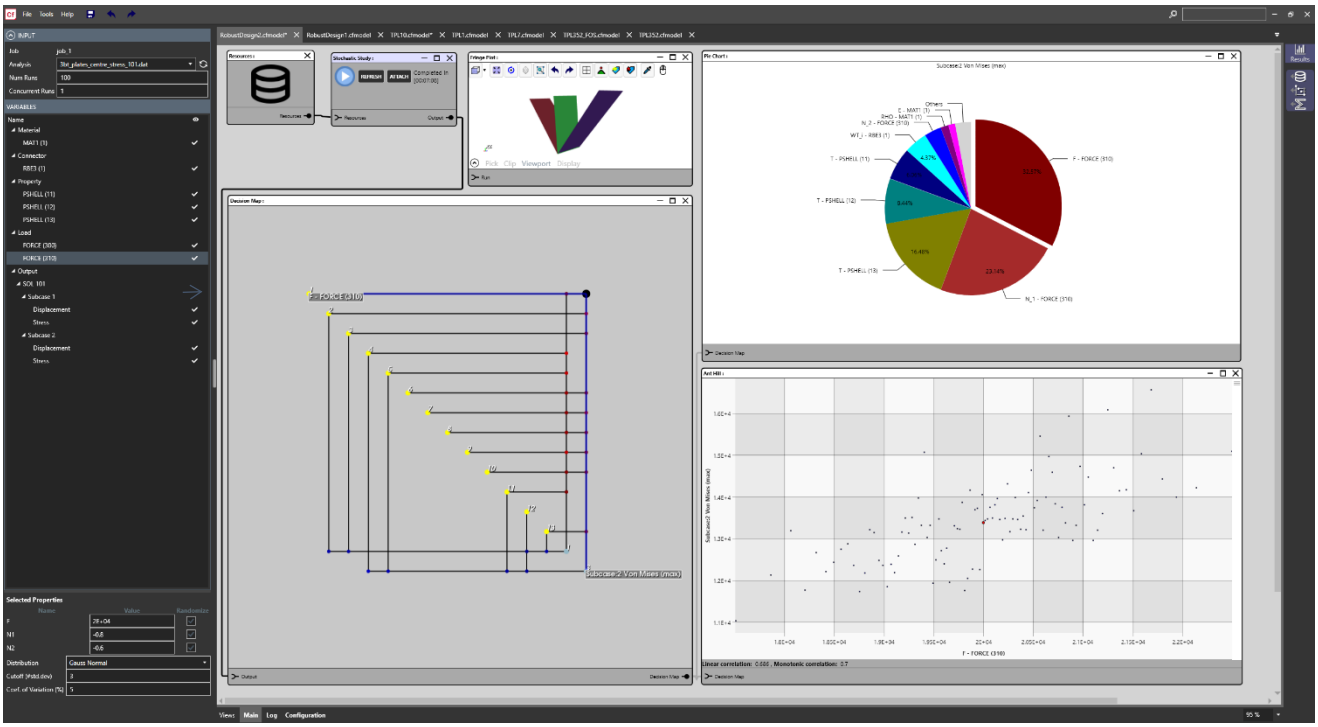
- The model is meshed with 3 CQUAD4 elements.
- A PSHELL property entry is defined for each element.
- The thickness of the elements are defined at 0.2 inches with a depth dimension of 5.0 inches.
- This defines an area=1.0 and retains reasonable element edge length.
- The material definitions are:
    - Young's modulus E = 10E6
    - Poisson Ratio = 0.33
    - Density = 0.1
- The boundary conditions for the 6 upper GRID points are fixed in all 6 directions
- The 2 lowest GRID points are linked at the centre location of the model with an RBE3 constraint. None of the lower GRID points are restrained.
- An X and Y direction loading of 20,000 lbsf is applied at the RBE3 centre location.
- The design is constrained as follows:
    - Maximum allowable tensile stress is restricted to 20,000 psi.
    - Maximum allowable compressive stress is restricted to 15,000 psi.
    - Maximum allowable movement of grid in the X and Y direction is restricted to +/- 0.2 inches.



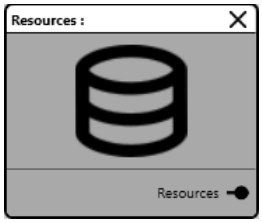


Although a prior optimization step is not necessary to use Robust Design, it is interesting to observe the behaviour of the optimized model by adding variability to the properties using Robust Design. Any optimization method may have been used, and here MSC Nastran SOL 200 was used to minimize the weight of the structure by changing the thickness of the CQUAD4 elements. The SOL 200 analysis suggests that the weight may be reduced from 3.83 lbs to 2.69 lbs while not exceeding the design constraints. The weight reduction comes from a thinner structure and the resulting thicknesses of the CQUAD4 elements were recovered and used to define the input file for the Robust Design analysis.

Figure 15 The Robust Design process flow for a 3 element example. Fringe Plotter only used to visualize the model and is not normally part of the Robust Design output viewers.

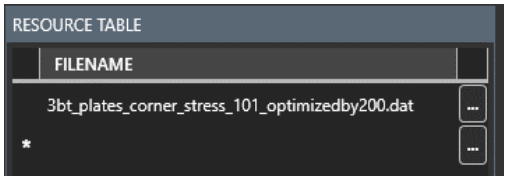


RESOURCES



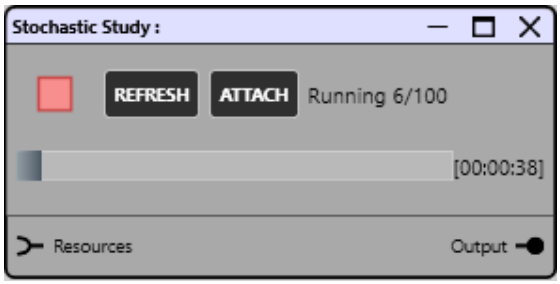
Resource Table

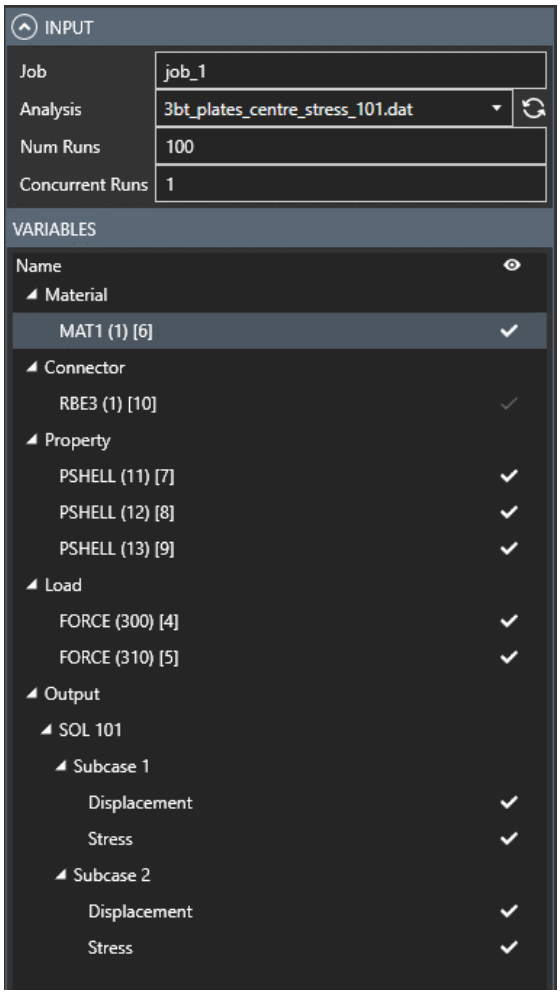
This is the area where you may add one or more MSC Nastran™ BDF/DAT input files to be used by the Robust Design process. Currently Robust Design supports any SOL analysis that provides STRESS and/or DISPLACEMENT output or normal mode frequencies from SOL 103.



STOCHASTIC STUDY

This is the heart of the Robust Design tool. From this box and menus, the User can set up the variables to modify, the number of MSC Nastranruns to use in the study and view the summary data from the runs within the Variable Table of the Stochastic Study box. The box will be populated with values only when some stochastic results are available.





To the left is a view of the Stochastic Study box (during an job execution) and the left side menus. Each variable definition is controlled by one or more of the variables displayed in the RANDOM VARIABLES box at the bottom left of the screen.

Job: User specified name to be given to the HDF5 output file that stores the stochastic results.

Analysis: Select the BDF/DAT file to use in the analysis.

Num Runs: Select the number of MSC Nastran™ runs to use for the stochastic analysis. Each MSC Nastran™ run, or sample, will select a unique combination of variables from those defined as random variables. A sufficient number of analyses are required to make the correlations meaningful. **Default = 100 ***

Concurrent Runs: Allows the User to run concurrent MSC Nastran jobs.

Each input and output parameter found in the MSC Nastran BDF/DAT file is listed under the various Name titles as shown below.

Material: Contains material definitions.

Connector: Contains information on connectors used within the model.

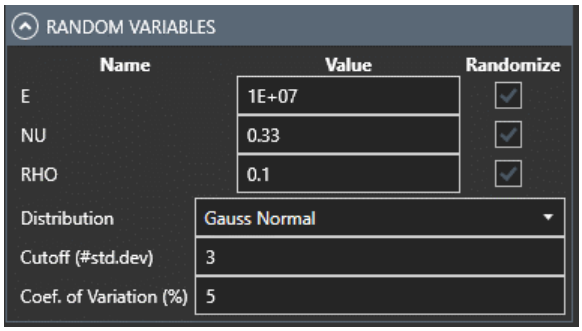
Property: Contains property information for elements within the model.

Load: Contains the loading information.

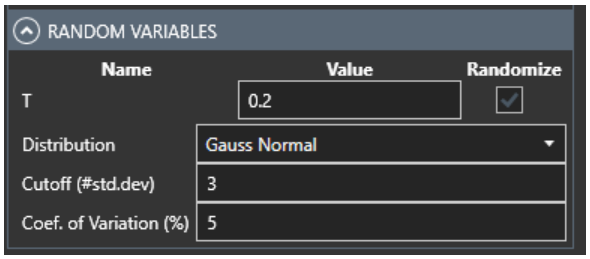
Output: Contains information on the output requests within the model. Note: Robust Design currently supports STRESS, MASS and DISPLACEMENT outputs for SOL 101 and FREQUENCY output for SOL 103.

* Robust Design uses the Latin Hypercube sampling method, which is a clever near random sampling technique that provides a good representation of the output distribution with as little as 100 samples.

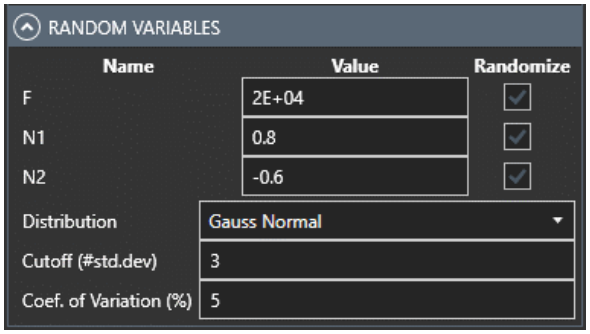
Material Variables



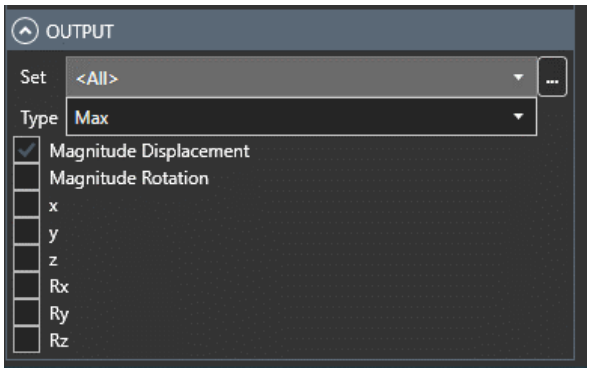
Property Variables



Load Variables



Displacement Output Variables



Name, Value and Randomize

The nominal values for the various parameters in each of the boxes shown to the left are populated from the MSC Nastran BDF/DAT file. Each parameter is selected to be randomized by default.

The randomization properties are:

Distribution: Which distribution do you want to use. Options are Gaussian Normal, Uniform and Weibull. Default = Gauss Normal.

Cutoff (#std.dev) : Default = +/- 3 standard deviations.

Coef. of Variation (%): How much to you want to vary the nominal values as a % of the nominal value. Default = 5%.

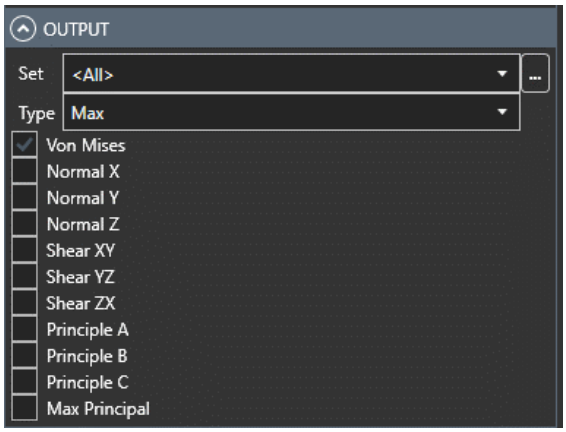
Output Options currently available in Robust Design

Robust Design support the review of Displacement and Stress correlations to the input parameters. To the left are the available options for displacement and stress.

Set: Allows the choice of any sets if defined in the BDF/DAT file. If no sets are defined, Default = ALL.

Type: Allows the choice of output results to do correlation on. Options are the actual **Value**, **Min** result and **Max** result. Default = MAX.

Stress Output Variables

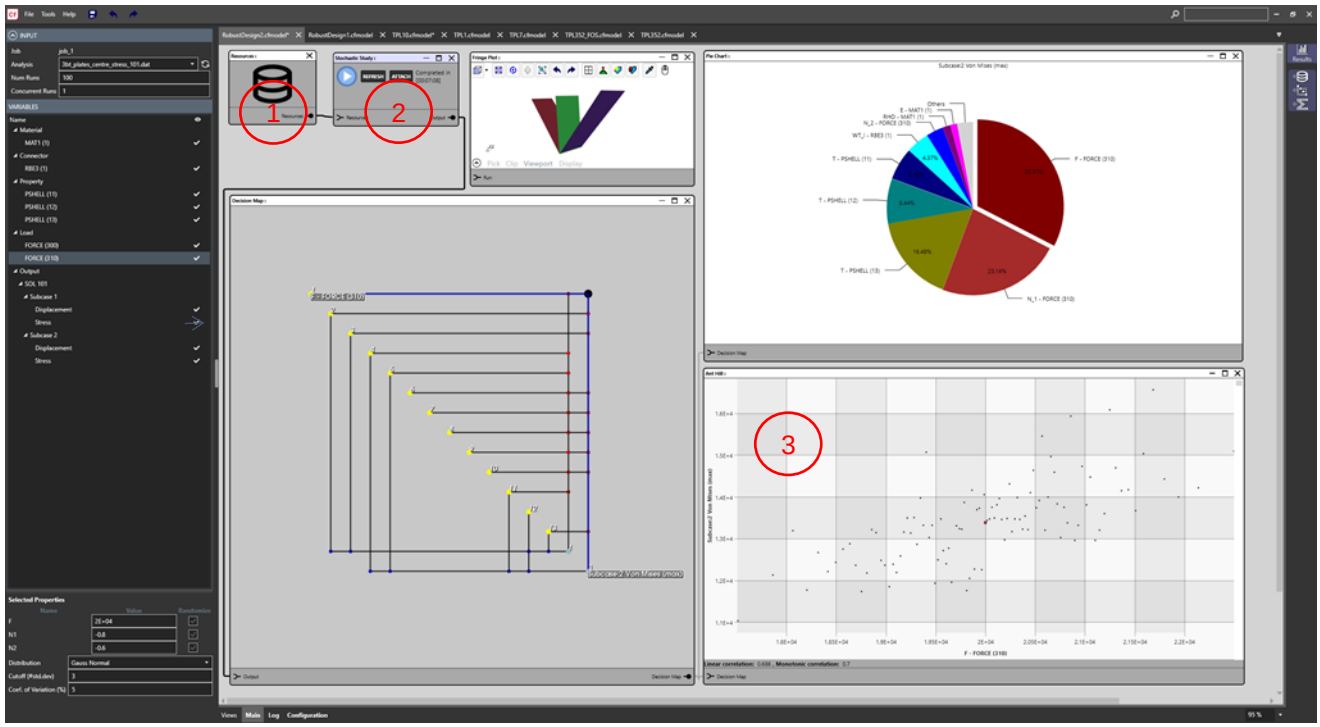


Running Robust Design in the Process Flow

Step 1

Assuming the Robust Design template has been opened, the first step in running a Robust Design analysis is to provide the MSC Nastran BDF/DAT file, which will be rerun dozens of times with varying parameters. For this example, the linear static superposition (SOL101) MSC Nastran file ***3bt_plates_centre_stress_101.dat*** is provided as the baseline from which to run Robust Design. The model for this analysis has been previously described.

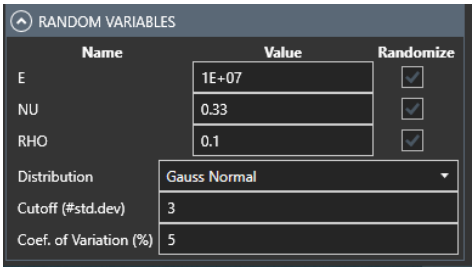
Figure 16 The Robust Design order of operation.



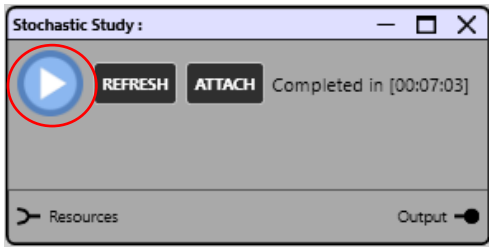
Step 2

Next, we click on the **Stochastic Study** box to setup and run the Robust Design study. At the top of the left menu, we must select the ***3bt_plates_centre_stress_101.dat***, if not already shown, which will automatically populate the VARIABLES section of the left menu. Please note that the User can also change the Job, Num Runs and Concurrent Runs if the defaults are not acceptable.

At this point we must select the check marks next to the input variables that we want to use in the assessment as well as the output variables which will be used to assess correlation to the input variables. Selection of the variable will make the corresponding Random Variables box available, which is prepopulated with the nominal values from the MSC Nastran™ file along **with** the default randomization parameters. Example of the MATERIAL input variable is below.

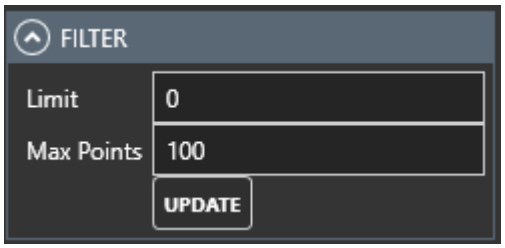


Once we have selected all the input and output variables to use in the assessment, the User must select the RUN button on the Stochastic Study box. The variable table will be updated periodically as the results become available and when the run is complete. If the User desires, the REFRESH button may be selected to update the results of the analysis.



Step 3

After the runs are complete and the REFRESH button is selected, the various correlations between output and input can be analyzed. For this, we click on the DECISION MAP viewer, which will also show the FILTER menu to the left side.



The **FILTER** menu allows the User to control the number correlation relationships.

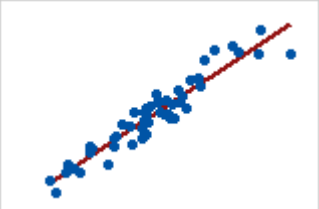
Limit: Sets the minimum correlation value that a relationship must have to view it on the Decision Map. Any correlation over +/- 0.5 is sufficient to review. Default = +/- 0.5. For this example, we have set the LIMIT to 0 to see all results.

Max Points: Sets the top number of correlation relationships to view (these are the red and blue dots in the Decision Map). Default = 100.

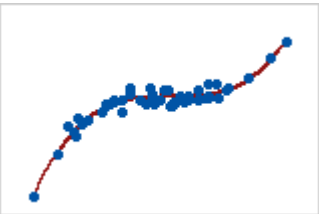
What is the Linear and Monotonic Correlation Value?

The purpose of Robust Design is to inform the User of the relationships between input and output variables. The stronger the relationship, the higher the "correlation" value. CAEfatigue provides 2 correlation values: Linear Correlation and Monotonic Correlation

A **Linear** correlation is a trend in the data that can be modeled by a straight line with either a positive or negative slope with data typically moving at a constant rate. To determine the strength of the correlation, we do a **Pearson Correlation** test where the values vary between -1.0 and 1.0. A Pearson rank correlation is considered as weak below +/- 0.2, moderate between +/-0.2 to +/-0.4), and strong above +/-0.4.

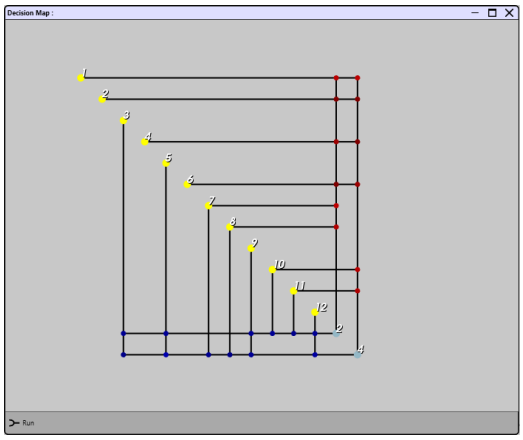


A **Monotonic** correlation is a trend in the data where variable A and variable B typically both increase or decrease together but not necessarily at a constant rate throughout the data. To determine the strength of the correlation, we do a **Spearman Rank-Order Correlation** test where the values vary between -1.0 and 1.0. A Spearman rank correlation is considered as very weak (0 to +/- 0.19), weak (+/-0.2 to +/-0.39), moderate (+/-0.4 to +/-0.59), strong (+/-0.6 to +/-0.79) and very strong (+/-0.8 to +/-1).



CAEfatigue chooses a LIMIT of 0.5 by default which means we are presenting moderate to strong correlation results. If there is a marked difference in the Linear and Monotonic correlation values, there is likely some issues present in the ant hill plot that should be investigated.

Decision Map with Lower Correlation LIMIT set to 0.0 (all correlation relationships)

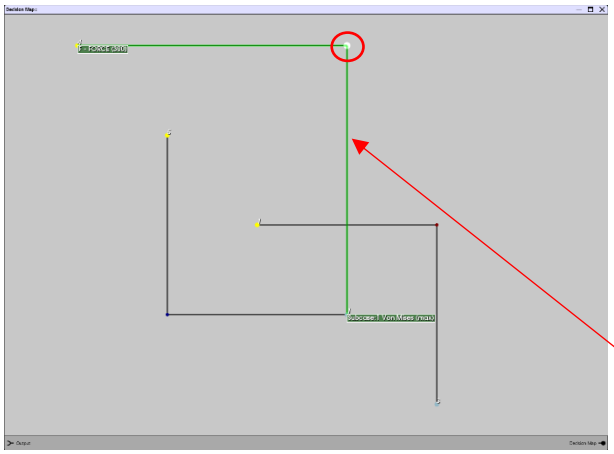


The **Decision Map** provides a quick view of all the relevant correlation relationships between the input variables (shown as yellow dots) and the output variables (shown as light blue dots).

Limit: Sets the minimum correlation value that a relationship must have to view it on the Decision Map. Any correlation over +/- 0.5 is sufficient to review. Default = +/- 0.5. For this example we have set the LIMIT to 0 to see all results.

Max Points: Sets the top number of correlation relationships to view (these are the red and blue dots in the Decision Map).

Decision Map with Lower Correlation LIMIT set to 0.5 (only moderate to strong correlation relationships)



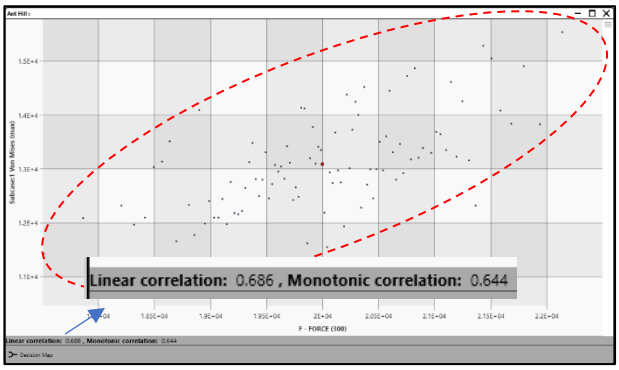
The **Decision Map** to the left shows only the correlation relationships that have a value of +/-0.5 or better.

In this case, only 3 inputs have a moderate to strong correlation relationship with the output variables. Those input variables are:

Example:

The input variable Force (300) has a linear and monotonic correlation of ~0.650 with the Subcase 1 von Mises stress. We can see the data for this relationship by selecting the RED dot connecting the variables and then viewing the ANT HILL plot.

Notice that as you hover the mouse pointer over the diagonal inputs of the decision map, the links to all the correlated outputs are highlighted allowing you to see the degree of interconnection between an input and one or more outputs. Clicking on any of the off-diagonal **red** or **blue** dots shows the anthill plot for the corresponding input-output relationship. Clicking anywhere in the grey area of the Decision Map will clear the current selection.



The **ANT HILL** viewer to the left shows the correlation points generated from the MSC Nastran runs.

The Latin Hypercube sampling method has provided sufficient data from the 100 MSC Nastran runs to see an envelope of results. Increasing the number of runs would simply fill in the envelope with additional data points which would not provide further information about the correlation. Hence, this is why CAEfatigue sets the default MSC Nastran runs to 100.

Understanding the Correlation Values

A Robust Design analysis is looking at **ALL variables at the same time**. This is vitally important to understand since software programs designed for "sensitivity studies" give you information about a single input variable to a single output variable and not a **holistic** view of how all variables interact together. It is common to see data behave very differently as an independent variable verses a variable influenced by other interactions.

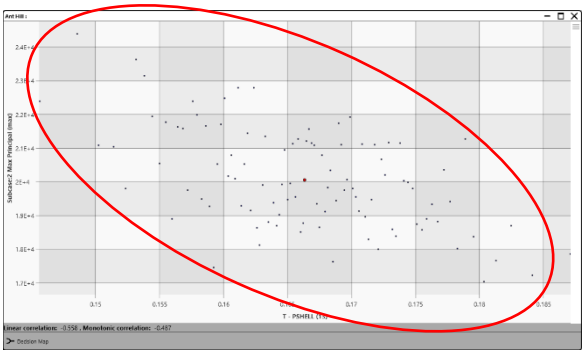
The Linear and Monotonic correlation values shown at the bottom of the **Ant Hill** plot are used when assessing how variables are correlated and whether the associated ant hill plot exhibits possible issues with the data. An issue is indicated by more than one cloud of points, or a form of cloud that is not more or less elliptical. Elliptical clouds, like a cigar shape, are desirable as they show predictable behavior of the inputs to output relationship. Long thin cigar shapes are an indication of high correlation and predictability, while shorter, fatter cigar shapes although still useful demonstrate lower correlation. A departure from an elliptical shape illustrates that there is some nonlinearity in the correlations and the Linear and Monotonic correlation values will diverge.

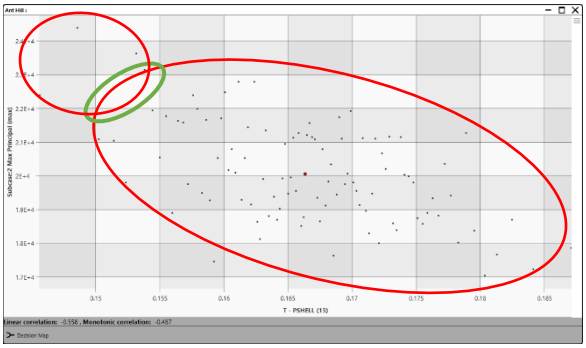
Marked departure from an elliptical shape (i.e., where the shape has concave portions along its perimeter and is therefore not convex), such as clouds with 2 or more clearly defined regions of correlation or even 2 or more separated clouds of points are indications of some issues in the model which makes outputs difficult to predict. In other words, small changes in the inputs may make the output response values move from one significantly different region of the plot to another. Sometimes the issue is an inherent characteristic of the problem, like a critical buckling load caused by initial imperfections or the frequencies of 2 closely spaced modes, where the frequencies may switch for small variations in the inputs. In other cases, the reason for the issue needs to be understood and preferably the design changed such that they do not occur.

Correlations between inputs, and between an output and more than one input is also useful information. If the inputs are to be chosen as design variables in order to move the design behavior to a different performance region, then design constraints should not be imposed on dependent inputs or outputs; if this is attempted, it often leads to an over-constrained problem. Identifying which inputs and outputs are correlated allows only one of the inputs to be assigned as a design variable or one of the outputs to be constrained; the other correlated inputs/outputs will naturally follow.

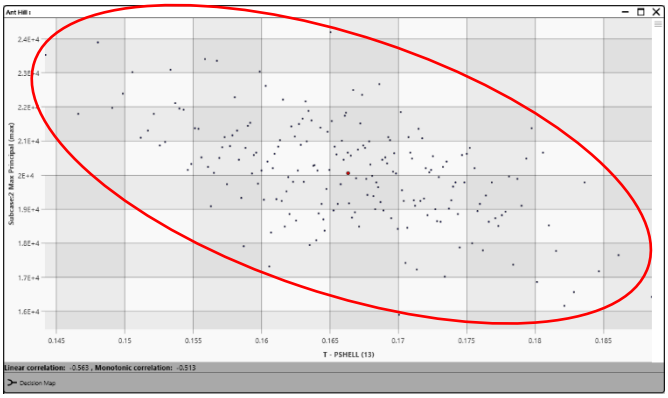
For example, the data below was generated using the input DAT file, ***3bt_plates_corner_stress_101_optimizedby200.dat*** that is a SOL200 optimized analysis. The results can either be a from a single population (first image) of from 2 populations (second image), meaning that Shell 13 Thickness may have a very different influence on Subcase 2: max Principal Stress when the shell thickness is lower than the threshold value between the 2 bubbles.

We can test this by increasing the number of MSC Nastran runs to see if sample selections fill in the area between the 2 red bubbles (green bubble) or if the trend of 2 separate data groups continue. If the trend continues, then we know that we should be aware that the thickness may influence the stress differently at lower values.





For closure on this question, the Robust Design analysis was rerun using 200 samples. This will help fill in the Green area if, indeed, the results are from a single population.

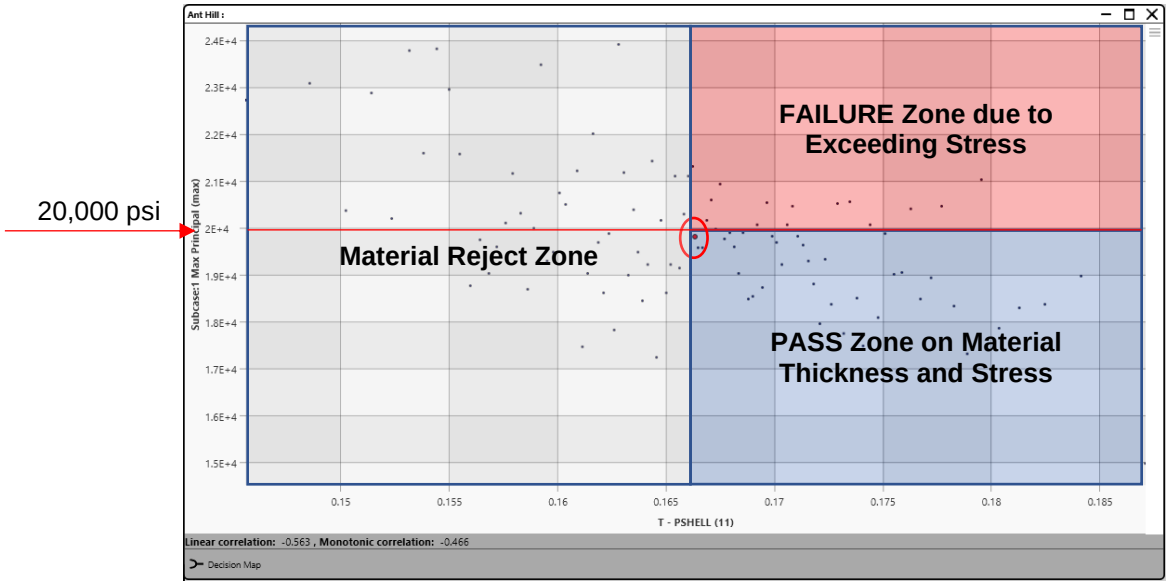


From the results on the left, it appears that some of the green area has been filled in suggesting that this data is indeed from a single population.

Also, as shown in another plot below, the ANT HILL viewer can quickly tell you that your design variable may fail. For example, had we just looked at a single variable, say Shell 11 Thickness, and its effect on Subcase 1 max Principal Stress for a single analysis at the nominal value (shown as RED dot within the plot), we would have seen a max stress results of ~ 20,000 PSI. This value is the max design constraint we set for the MSC Nastran SOL200 analysis, so we would have likely supported the use of this thickness (0.166 inches) as the thinness material thickness we can use without exceeding the max stress threshold (Pass Zone).

However, the **Robust Design** analysis shows via the Ant Hill viewer that many points above the 0.166 inch thickness will still result in stress values well above 20,000 PSI (Failure Zone). This is because we are analyzing the influence of all variables on how the shell thickness and max stress correlate with each other. So, even if we reject all sheet metal with a material thickness of less than 0.166 inches, we can still fail our design.

Hence, a responsible Engineer would look at the data and determine that a lower max stress design limit may be needed in the MSC Nastran analysis to ensure that the possibility of exceeding 20,000 PSI is minimized after considering how all input variables react with each other to generated stresses.



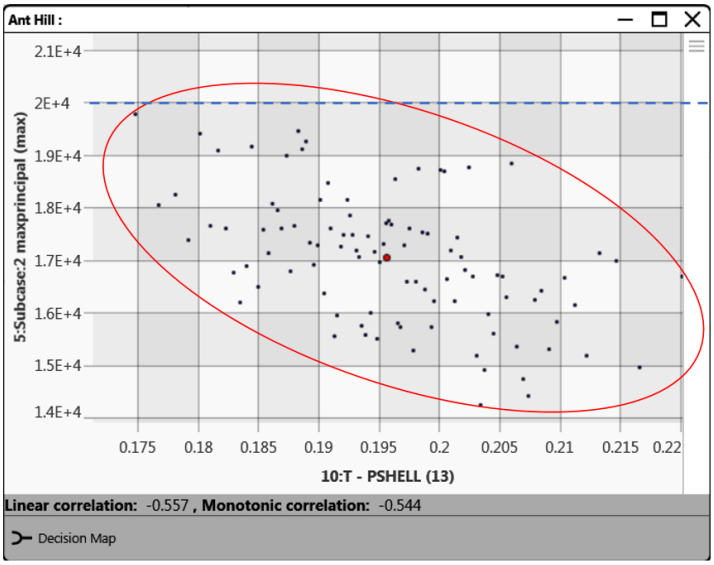
Understand Quality and Robustness from the Ant Hill Plot

We have seen that elliptical ant hill plots are desirable as they allow confidence in predicting an output when input variables change. If the cloud of points forming the ellipse are always above or below some critical response value that determines a failure level, we can say that the design is robust, even if there is poor correlation. For example, as shown above, the maximum allowable stress level of 20,000 psi is imposed by the strength of the material of a component. If the cloud of points shows the output stress responses are elliptical in shape and always appear below 20,000 psi, we know with a high degree of confidence that the design will not be over-stressed even with a random selection of input variables that place the output closer to its allowable limit.

If the ellipse encompassing the perimeter of the cloud of points is large in both major and minor diameters, this illustrates low quality in one or more of the input variables. If the output responses are always below the maximum allowable, this is not a problem, and we conclude the design is of low quality but is still robust.

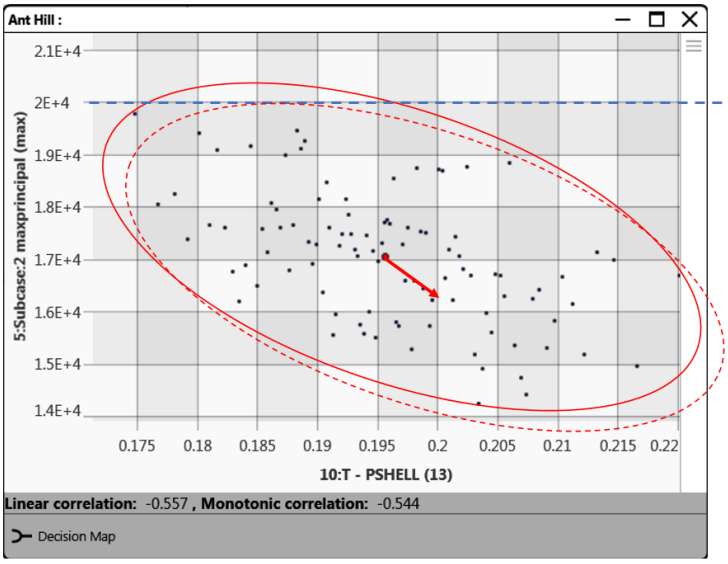
The subject of quality can be useful in both situations where an output response is either close to or far away from some allowable value. In the following anthill plot, we can see that none of the 100 runs exhibits a stress level over 20,000 psi. There is a high degree of convexity in the plot illustrated by the encompassing ellipse but notice a small part of the ellipse encroaches the region over 20,000 psi. As an engineer, this might give you some cause for

concern and you would like to reduce the stress level a little to be further away from the 20,000 psi allowable value, illustrated with the dotted line.



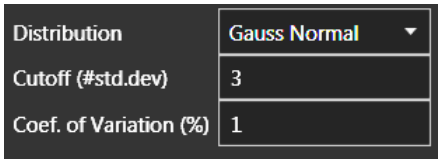
20,000 psi, max allowable stress

There are 2 things we could do to this design. The moderate correlation values (~0.55) are suggesting that if we increased the plate thickness, the stress would reduce. This is like trying to move the cloud of points down and to the right, but it would increase the weight. The dotted red line is only an idea of what the new ellipse may look like so we should be aware that the shape may change with the change to the input variable.



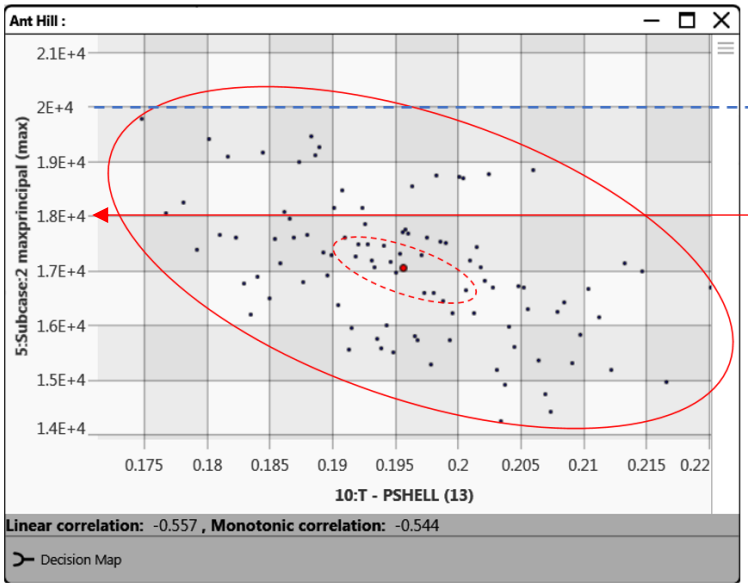
20,000 psi, max allowable stress

Alternatively, we could tighten the tolerance on the thickness of the plate elements in property 13. This would reduce the major and minor diameters of the ellipse. We can do this by defining a smaller % variation for the stochastic variable PSHELL 13. Instead of using 5% coefficient of variation, let's try 1% variation:



With a 5% coefficient of variation, the plots show the thickness was able to vary between ~0.175" and ~0.22". Setting 1% as the coefficient of variation means the thickness will only be able to vary between ~0.19" and ~0.21"; this won't result in a nominal increase in weight, but it will make the plate **more expensive** to manufacture.

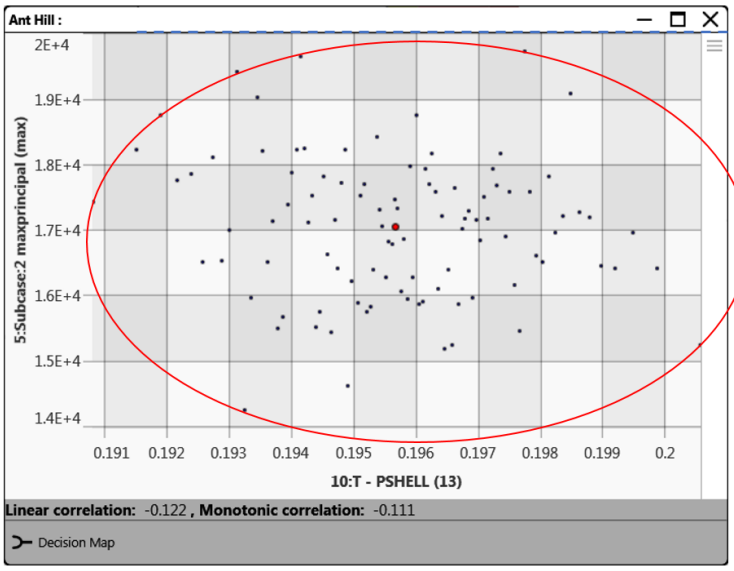
Looking at the plot, one may think that the ellipse will be much smaller than it was previously as show by the red dotted ellipse between the values of ~0.19" and ~0.21".



20,000 psi, max allowable stress

POTENTIAL ceiling of ellipse is now 18,000 psi, using reduced Coefficient of Variation (1%) that results in a material thickness that varies between ~0.19" and ~0.21"

Does this mean we can expect the stress levels to drop to less than 18,000 psi? Unfortunately, no because there is variability in **all** the other parameters, and this is another reason why you should define variability in anything that could exhibits it. The only way to know how the model performs with the tightened tolerance is to rerun the stochastic simulation. If we do this analysis, we see the results below.



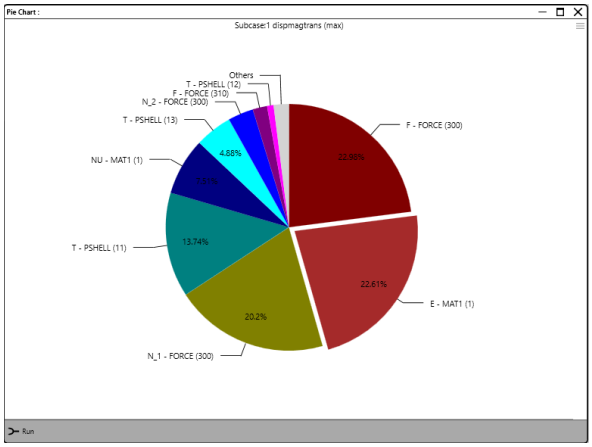
20,000 psi, max allowable stress

The shape of the cloud has changed (the ellipse has a lower aspect ratio and different orientation), and the correlation between plate thickness and stress is rather poor now. This explains why the results are only just below the 20,000 psi allow stress limit, but they likely mean we have a robust design.

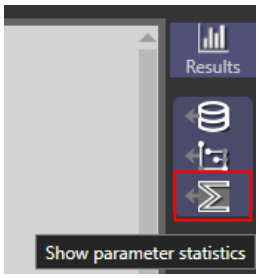
Now consider the opposite scenario. The design above defines a high tolerance on the thickness of the plate with PSHELL property 13, which makes it expensive. What happens if the decision is taken to manufacture the plate from a material with a higher allowable stress, say 30,000 psi? The plot above shows low correlation between the plate thickness and the stress, and the stress developed in the plate is now very far away from the allowable of 30,000 psi. In this case, we could relax the tolerance on the plate, which will make it less expensive but that will increase the scatter in the output (reduce its quality). However, it should still be a robust design because the stress should still be less than the max allowable.

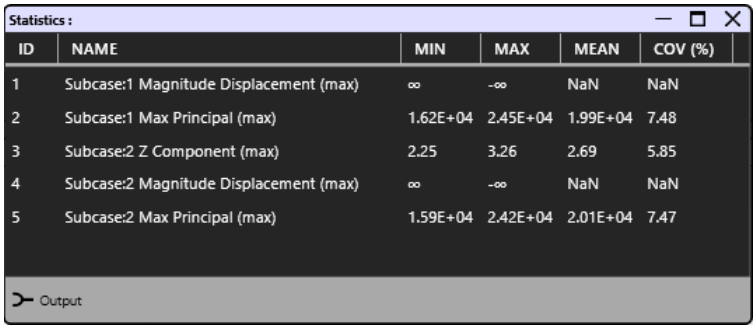
The **Pie Chart** display is useful in determining two aspects of the design. For a given output, the pie chart shows which of the input variables are the most influential in changing the output (response) value. There is little point in requesting an input be used to change the selected output if it does not have any significant effect on that output.

On the other hand, any variables that are identified by the pie chart as having a low degree of influence on an output value are variables for which tolerances may be relaxed. Relaxing the tolerance on a component may easily make that component much cheaper to manufacture.



The **Statistics** display, which is not part of the default template, can be selected from the right side menu when the Stochastic Study box is selected. The Statistics are those of the output parameters chosen for the analysis. It provides the Min, Max, Mean and COV (%). If an infinity sign is shown, it means that Cf is having difficult reading this output information from the Nastran DAT/BDF file and this should be investigated further.





ID	NAME	MIN	MAX	MEAN	COV (%)
1	Subcase:1 Magnitude Displacement (max)	∞	-∞	NaN	NaN
2	Subcase:1 Max Principal (max)	1.62E+04	2.45E+04	1.99E+04	7.48
3	Subcase:2 Z Component (max)	2.25	3.26	2.69	5.85
4	Subcase:2 Magnitude Displacement (max)	∞	-∞	NaN	NaN
5	Subcase:2 Max Principal (max)	1.59E+04	2.42E+04	2.01E+04	7.47

So why is the Robust Design data important?

Stochastic simulations in Robust Design only make observations of the natural behavior of a system defined with variability. From these natural variations, decisions may be made on how to improve the design. The important aspect of Robust Design is that the variations in the parameters yield correlations where all variables are considered simultaneously. In contrast, gradient based methods will compute sensitivities, which are indicators of the individual variation of a parameter while all other parameters remain fixed. Response surface methods require you to impose smooth variations on the behavior of the inputs and outputs and choose from a myriad of algorithms to solve for interpolated responses. This often leads to the masking of pathological behavior.

Notice that the only information that you need to supply to a Robust Design analysis is the range and type of distribution of the input properties, i.e. the physical way in which the inputs, all of them, vary. If you don't know the type of distribution, assign a uniform one (i.e. all values are equally likely to occur in a given range). Most of the time, the properties being configured have some quality control associated with them, like material density, Young's modulus or thickness of a sheet material.

The quality control procedures mean the distribution of the property under consideration is often a Gaussian one, as the manufacturer tries to achieve repeatability when producing components. With the range of possible values (i.e. the lower and upper limits), and if Gaussian, the standard deviation from the mean, as many inputs as possible should be assigned a distribution of the property in question; it is, after all, present in the physical case. As a consequence, there is no need to decide on which variables are "important" ones; in contrast, with Robust Design this aspect will be a result of the analysis.

With Robust Design, there is no need to decide on which algorithm or solver to impose on your system. There is no algorithm or solver, only observation of the natural consequence of variability in all the inputs. With 100 or so analysis runs, each with randomly selected different inputs, the natural behavior of the model may be assessed for the trends it exhibits. If the system has a high degree of correlation, it may be possible to use fewer analyses, but experience shows 100 is sufficient.

What's Next?

This initial release of Robust Design will be further improved in later versions of CAEfatigue and will also assist the User in understand the correlation data to the point where CAEfatigue will make the recommendations on how to achieve a design with improved performance over a nominal MSC Nastran analysis, i.e., where we should set the input values ensure design specification are not exceeded.

Also, a future version of Robust Design will amalgamate the features of RELIABILITY MODE so that the user can vary both the MSC Nastran input variable and CAEfatigue input variables.

Running Robust Design in BATCH within Windows or Linux

Robust Design can be run in Batch mode within Windows or Linux in a very similar fashion to what was described for Reliability Mode. The User can use the command line entry:

```
caefatigue -robust-design <configuration file>
```

Where the configuration file is the JSON file that contains the configuration information needed to run the job. Below is the JSON file content for the example used in this section.

```
{
  "Parameters": {
    "SolverType": "nastran",
    "NumProcesses": 100,
    "NumRuns": 1,
    "MonitorPort": 0,
    "SolverPath": "C:\\Program
Files\\MSC.Software\\MSC_Nastran\\2021.2\\bin\\nastran.exe",
    "JobName": "job_1",
    "TemplateFile": "job_1_variables.template"
  },
  "Inputs": [],
  "Results": [
    {
      "AnalysisType": null,
      "LoadCase": "1",
      "Step": "0",
      "LoadCaseID": 1,
      "StepID": 0,
      "Outputs": [
        {
          "Type": "displacement",
          "Groups": [
            {
              "SetName": "<All>",
              "Components": [],
              "Operation": 2
            }
          ]
        }
      ]
    }
  ],
  "Sets": []
}
```

Below is the content of the **job_1_variables.template** file referenced in the JSON file. The template file is the MSC Nastran BDF/DAT file that is modified each time the MSC Nastran solver is run using new variable values as selected by Robust Design.

```
SOL 101
CEND
TITLE = SYMMETRIC THREE PLATE FRAME DESIGN OPTIMIZATION
ECHO   = NONE
SPC    = 100
DISP   = ALL
STRESS = ALL
SUBCASE 1
   LABEL = LOAD CONDITION 1
   LOAD  = 300
SUBCASE 2
   LABEL = LOAD CONDITION 2
   LOAD  = 310
BEGIN BULK
MDLPRM,HDF5,1
$------------------------------------------------------------------------
$ ANALYSIS MODEL
$------------------------------------------------------------------------
$
SPC,    100,    1,      123456, ,       2,      123456
SPC,    100,    3,      123456
SPC,    100,   11,      123456, ,      12,      123456
SPC,    100,   13,      123456
FORCE,  300,  414,      ,       20000.,  0.8,   -0.6
FORCE,  310,  414,      ,       20000., -0.8,   -0.6
$
GRID,   1,      ,       -10.0 ,   0.0,  0.0
GRID,   2,      ,         0.0 ,   0.0,  0.0
GRID,   3,      ,        10.0 ,   0.0,  0.0
GRID,   4,      ,         0.0 , -10.0,  0.0
GRID,  11,      ,       -10.0 ,   0.0,  5.0
GRID,  12,      ,         0.0 ,   0.0,  5.0
GRID,  13,      ,        10.0 ,   0.0,  5.0
GRID,  14,      ,         0.0 , -10.0,  5.0
GRID, 414,      ,         0.0 , -10.0,  2.5
MAT1,   1,      1.0E+7, ,       0.33,   0.1
PSHELL, 11, 1, .2, 1,, 1
PSHELL, 12, 1, .2, 1,, 1
PSHELL, 13, 1, .2, 1,, 1
CQUAD4, 11, 11, 1, 4, 14, 11
CQUAD4, 12, 12, 2, 4, 14, 12
CQUAD4, 13, 13, 3, 4, 14, 13
$
RBE3, 101, , 414, 123456, 1., 1236, 4, 14
$
ENDDATA
```

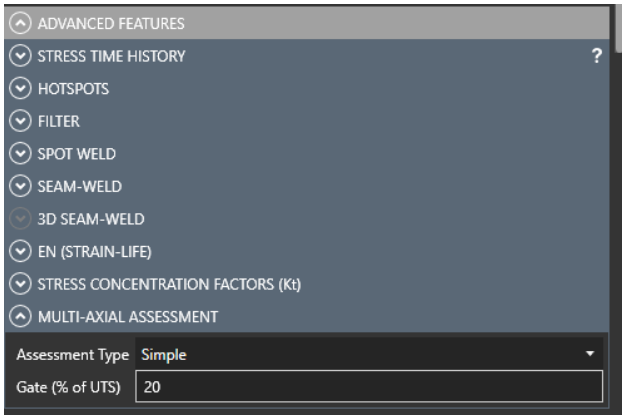
14 | INTRODUCTION TO THE TIME DOMAIN MULTIAXIAL ASSESSMENT

NOTE: The images shown in this section reflect CAEfatigue v2022 and may change slightly with future releases.

Overview and Introduction to the Time Domain Multiaxial Assessment Feature

Multiaxial assessment provides information about how the stress state varies throughout the multiaxial loading history. In essence, it tells us whether the analysis we have caried out is appropriate to the states of stress occurring in the component.

Within the Process Flow, this feature is accessible through the MULTI-AXIAL ASSESSMENT menu under Advanced Features. When an **Assessment Type** other than NONE is selected, the new entry vFTGPARM> MULTIAX is added to the Control File as seen below. See the **Quick Reference Guide** for more information about this feature and the options available. See TPL314 for an example of using his feature.




```
vftgparm777
        stress   crit180 goodman naver    none     1E-05
        multiax  simple  20
```

What is multiaxial loading?

This table below describes what is meant by Proportional Loading and Non-proportional Loading. “Loading” means the local stress state variations, not the global loading environment. Fortunately, we very often find that although the global loading environment has a complex set of out-of-phase loads, the local stress-state variations in the critical locations are much simpler. This is often dictated by geometry - for instance the stress state at the edge of a thin metal sheet will always be uniaxial.

	ϕ_p	a_e	Rarity/Difficulty
Uniaxial	constant	0	Most common and easily dealt with. Only one principal stress exists, which is σ_1. Standard analysis methods are OK, i.e. von Mises, etc.
Proportional Loading	constant	$-1 \le a_e \le 1$	Ratio is near the same value over time. Less common but easily dealt with by knowing a_e to correct from a uniaxial case.
Non-proportional Loading	may vary	may vary	Ratio my vary over time. Most rare and tricky to deal with.

A biaxiality analysis accomplishes the following:

1. First, we are concerned with stresses and strains at free surfaces, where a state of plane stress exists, i.e., the stress state is 2D in the plane of the element free surface. We therefore simplify our attempts to understand the free surface by transforming the stress results to local coordinate systems at each location where the x-y plane is the plane of the surface, i.e., surface resolved stresses. Thus, one of the reasons for skinning 3D models with shell elements.

2. The principal stresses are reordered from the conventional order where σ_z is the surface normal stress (and should be 0) and σ_1 and σ_2 are ordered in magnitude. σ_1 is the largest in-plane principal (in absolute value) and σ_2 is the other in-plane stress.

3. The biaxiality ratio (a_e) is calculated for every location at every time point: $a_e = \sigma_2/\sigma_1$. The angle, φ_p, that σ_1 makes with the local x-axis is also retained for each location at every time point.

4. The surface stress state is therefore described completely by σ_1, a_e and the angle, φ_p.

5. a_e and the angle, φ_p, become unstable when the stresses are small, so when we calculate statistics of these parameters a **gate** is applied to filter out these small stresses.

6. The biaxiality ratio can take on any number between -1 and +1. It will be zero (0) for uniaxial loading, nearly a constant value for proportional loading and may vary over time for non-proportional loading.

There are two reasons that we are interested in biaxiality.

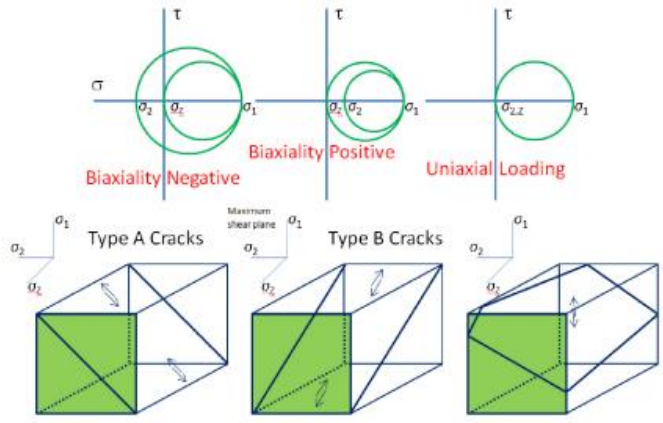
- We need to know what the biaxiality is to calculate the stress-strain response correctly from what would be the case if the loading was uniaxial.

- Biaxiality can significantly affect severity of fatigue damage.

For example: When the biaxiality analysis is negative (as indicated by the Mohr's circles of stress below), the maximum shear plane where cracks tend to initiate is oriented as shown in the diagram below. In the early stages of crack initiation, Type A cracks grow mainly along the surface in Mode 2 (shear) before transition to Mode 1, normal to the maximum principal stress.

When the biaxiality is positive however, the cracks tend to be driven more through the thickness (Type B). These are therefore more damaging for the same levels of shear strain.

Uniaxial loading is a special case where biaxiality ratio equals zero (a_e=0.0), i.e. only the σ_1 stress exist and standard analysis methods such as von Mises can be used.



Besides NONE, there are **three** multiaxial assessment options available in CAEfatigue. The choice of assessment method is set via the **vFTGPARM>MULTIAX** entry. See Quick Reference Guide for further information.

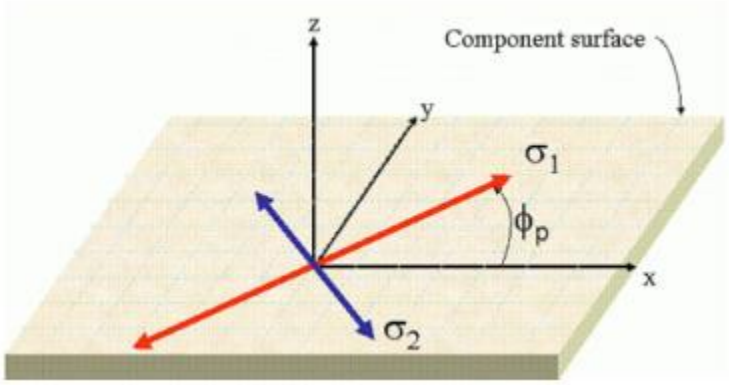
Simple Biaxiality Assessment

This option provides information about the behavior of the stress tensor in terms of the biaxiality ratio and the orientation of the absolute maximum principal stress. The input to the calculation is a 2-D or surface-resolved stress tensor history $\sigma_{ij}(n)$ where n is the point number from the time history or peak-valley sequence. The zz, xz and yz components are assumed to be zero, i.e.,

$$\sigma_{ij} = \begin{bmatrix} \sigma_{xx} & \sigma_{xy} & 0 \\ \sigma_{xy} & \sigma_{yy} & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

At each point (n), the principal stresses are determined. For the purposes of this calculation the principal stresses are σ_1, σ_2 and σ_z, where $\sigma_z = \sigma_{zz} = 0$ (the surface normal stress) and $|\sigma_1| \geq |\sigma_2|$ (the in-plane principal stresses).

So σ_1 is the Absolute Maximum Principal Stress and σ_2 is the other in-plane principal. At the same time, the orientation ϕ_p of σ_1 relative to the local x-axis is determined.



The biaxiality ratio is defined as:

$$a_e = \frac{\sigma_2}{\sigma_1}$$

And the direction is defined as:

ϕ_p

In order to reduce the impact of data points (n) where the stresses are very small, the number of points that contribute to the statistics output are reduced through the use of a **Biaxiality Gate** setting, which is defined as a percentage of the material UTS (typically 20%). Points will be excluded from the biaxiality analysis based on the following expression:

$$|\sigma_1(n)| \leq \frac{Biaxiality\ Gate}{100} x\ UTS$$

Statistical Output provided in CSV File for Stresses above the Biaxiality Gate

The following quantities are added to the CSV file when vFTGPARM>MULTIAX>SIMPLE is added to the control file:

- **Mean Biaxiality Ratio** - The mean value of $a_e(n)$ during the loading sequence.
- **Standard Deviation of the Biaxiality Ratio** - The standard deviation of $a_e(n)$ during the loading sequence.
- **Angle Range** - To calculate this, the values of ϕ_p are collected into three degree bins, then looking for the largest continuous block of unoccupied bins and subtracting this angle from 180 degrees.
- **Most Popular Angle** - This refers to the most populated bin.
- **Biaxiality Gate (In stress units)** - A percent (%) of the material UTS, below which, no multiaxial assessment is conducted.
- **Maximum Abs Max Principal Stress Range** - The maximum range of σ_1.

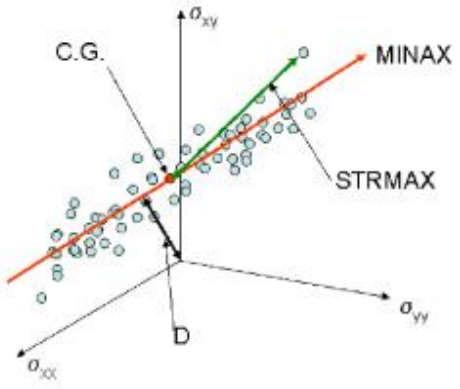
.

Standard Multiaxial Assessment

This method provides a more robust measure of the biaxiality and non-proportionality of the local loading (stress state). Visualize the stress history as a cloud of data in which every point in the stress history is represented by a point on a 3-D scatter plot of σ_{xx} vs σ_{yy} vs σ_{xy} (see image below). By analyzing the shape and orientation of this data cloud, we can obtain numerical measures of the biaxiality and non-proportionality of the loading.

The calculation procedure is as follows:

1. Determine the position of the center of gravity (C.G.) of the data cloud.
2. Determine the principal moments of inertia of the data cloud about the center of gravity $I_1 \geq I_2 \geq I_3$.
3. Find the vector corresponding to the principal axis with the minimum moment of inertia I_3 (MINAX).
4. Find the point with the largest distance from the center of gravity of the cloud (STRMAX).
5. Find the offset D of MINAX from the origin.



Essentially, a 3-D linear regression analysis is performed on the data cloud. If the cloud of data correlates closely with a straight line through the origin, the loading is proportional. The larger the aspect ratio of the cloud and/or its offset from the origin, the more non-proportional the loading. The orientation of the vector MINAX tells us the average biaxiality.

6. We define the cloud aspect ratio.

$$ASPECT = \frac{I_3}{I_1}$$

7. We define the cloud offset factor.

$$OFFSET = \frac{D}{2\ x\ STRMAX}$$

8. We define the **non-proportionality factor**.

$$NONPROP = \sqrt{ASPECT + OFFSET^2}$$

9. The three stress components (σ_{xx}, σ_{yy}, σ_{xy}) of the vector MINAX define a 2-D stress tensor σ_{ij} from which are obtained the two principal stresses σ_1 and σ_2 where $|\sigma_1| \geq |\sigma_2|$. The **mean biaxiality** is then calculated from:

$$a_e = \frac{\sigma_2}{\sigma_1}$$

10. At the same time, the orientation of the plane corresponding to MINAX is determined, i.e., the orientation of σ_1 from the previous step. This gives the **Dominant Stress Direction**, which will correspond to the most likely critical plane orientation.

Statistical Output provided in CSV File for Stresses above the Biaxiality Gate

The following quantities are added to the CSV file when vFTGPARM>MULTIAX>STAND is added to the control file:

- **Non-proportionality Factor** – The square root of the Aspect plus Offset squared.
- **Mean Biaxiality** - The mean value of $a_e(n)$ during the loading sequence.
- **Dominate Stress Direction** – The orientation of σ_1 corelating to the mean biaxiality output.

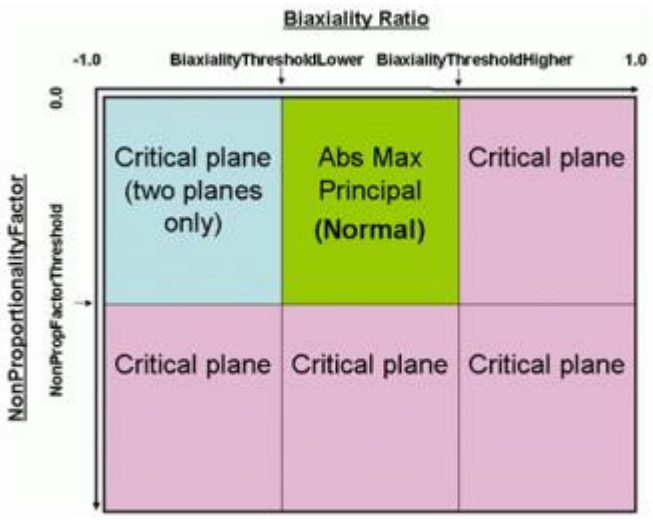
Automatic Multiaxial Assessment – not yet supported

Auto mode uses a two-pass approach, which may override the stress combination method setting (vFTGPARM>COMB). When Auto is selected, four (six for e-N analysis) additional properties must be set as shown in the table below.

1. In the first pass, the combination method is set to Abs Maximum Principal (the Normal method) and a **Standard Assessment** is carried out.
2. A second pass using a different stress combination method if the value of STRMAX exceeds the percentage of the material strength (UTS) defined by Zero Damage Stress Percent AND IF the non-proportionality and/or biaxiality values lie outside the ranges defined in the table.

Property	Default Settings
Zero Damage Stress Percent	10
Lower Non-Proportional Factor Threshold	0.25
Upper Non-Proportional Factor Threshold (e-N only)	0.5
Lower Biaxiality Threshold	-0.6
Mid Biaxiality Threshold (e-N only)	0.25
Upper Biaxiality Threshold	0.6

The Auto mode is summarized in the figure below.



The other possible option is the Two Plane option. This is similar to the Critical Plane method except that the calculations are made on two planes only - in the Dominant Stress Direction and at 90 degrees to the Dominant Stress Direction.

The following quantities are added to the CSV file when vFTGPARM>MULTIAX>AUTO is added to the control file:

- **Non-proportionality Factor** – The square root of the Aspect plus Offset squared.
- **Mean Biaxiality** - The mean value of $a_e(n)$ during the loading sequence.
- **Dominate Stress Direction** – The orientation of σ_1 corelating to the mean biaxiality output.

- **Calculation Method** – Will be abs Max Principal (Normal) method, Critical Plane method or Two Plane method.

Considerations between using S-N versus e-N

For a standard e-N fatigue analysis, the basic principles of the **Simple** and **Standard** methods are the same as for S-N fatigue analysis. The only difference in implementation is that the full strain tensor histories must first be converted to stresses (using an elastic assumption) before the assessment can take place.

For the Automatic mode, note the differences in the stress combination methods used based on the calculated biaxiality ratio and proportionality factor in tables below

Auto Stress Combination for S-N Analysis based on Biaxiality Ratio (a_e) and Proportionality (PF)

S-N Analysis		**Proportionality Factor (PF)**	
		0.0 ≤ PF ≤ 0.25	0.25 ≤ PF
Biaxiality Ratio	-1.0 ≤ a_e < -0.6	Critical plane in 2 directions – dominant and at 90 degrees to dominant	Critical plane 10 degree intervals
	-0.6 ≤ a_e < 0.6	Original calculation OK (Absolute Maximum Principal)	
	0.6 ≤ a_e ≤ 1.0	Critical plane 10 degree intervals	

Auto Stress Combination for e-N Analysis based on Biaxiality Ratio (a_e) and Proportionality (PF)

e-N Analysis		**Proportionality Factor (PF)**		
		0.0 ≤ PF ≤ 0.25	0.25 ≤ PF ≤ 0.5	0.5 ≤ PF
Biaxiality Ratio	-1.0 ≤ a_e < -0.6	Critical plane in 2 directions – dominant and at 90 degrees to dominant		Critical plane 10 degree intervals - Issue warning
	-0.6 ≤ a_e < 0.25	Original calculation OK (Absolute Maximum Principal)	Critical plane in dominant direction	
	0.25 ≤ a_e < 0.6	Signed Tresca	TBCPS* dominant direction only	TBCPS 10 degree intervals - Issue warning
	0.6 ≤ a_e ≤ 1.0	TBCPS 10 degree intervals		

*TBCPS = Type B Critical Plane Shear

Note on Duty Cycle Processing

When a duty cycle is being processed, the biaxiality parameters are calculated for the individual unique events that make up the duty cycle, as well as for the duty cycle as a whole. The statistics for the duty cycle as a whole take into account the number of repeats of each event.

15 | INTRODUCTION TO THE NEF DECK CREATION INTERFACE FOR TIME DOMAIN ANALYSIS

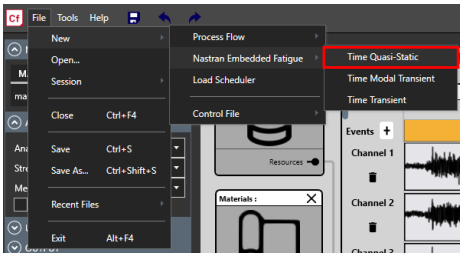
NOTE: The images shown in this section reflect CAEfatigue v2021 and may change slightly with future releases.

Overview and Introduction to the NEF Deck Creator

The MSC MSC Nastran Embedded Fatigue (NEF) deck creator tool is a capability that can be used to create a time domain MSC MSC Nastran Embedded Fatigue (NEF) execution deck or even run MSC MSC Nastran while in the Cf process flow interface.

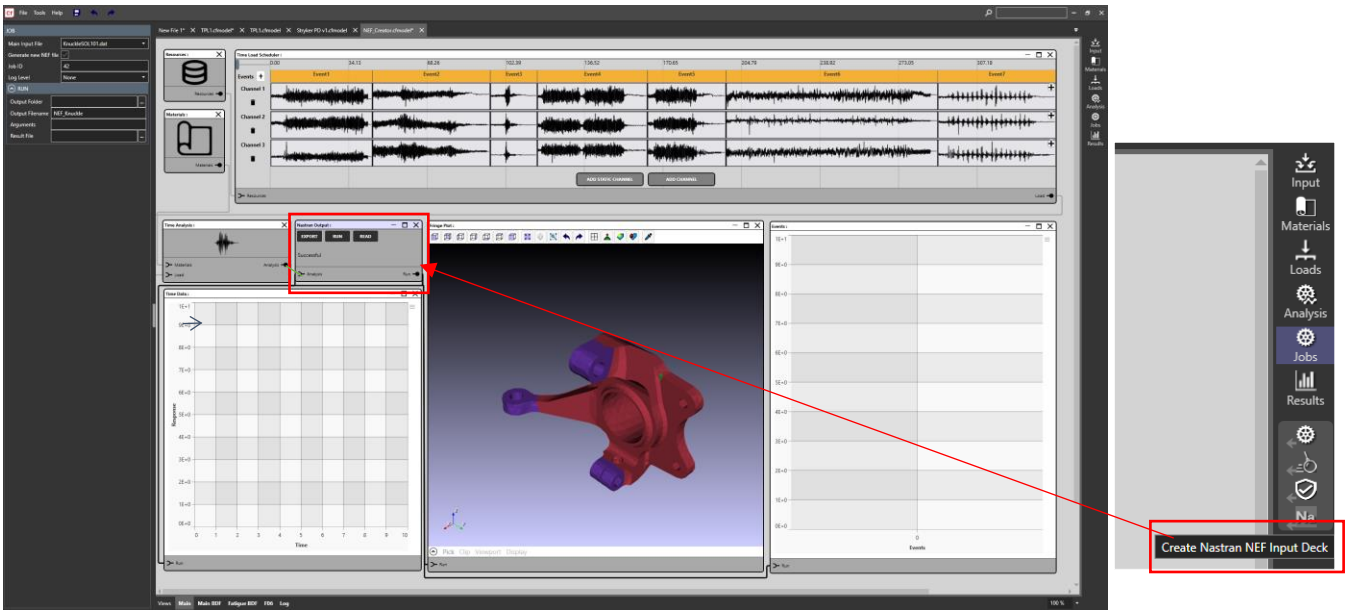
For example, below is the NEF Deck Creator feature used to create a NEF deck to run a time domain linear static superposition analysis. The process to use the NEF Deck Creator is to

a) Open a new MSC MSC Nastran Embedded Fatigue process flow template for a linear quasi-static superposition analysis.



b) Once the template is open, setup the fatigue analysis as desired. For this example, we will set up the example to match TPL352. Note: for reference, the NEF Deck Creator box is located in the JOB menu on the right.

Figure 17 Location of the NEF Deck Creator feature in the JOBS menu.



Review of NEF Deck Creation Menus

When the MSC Nastran Output box is selected, additional menu items appear in the bottom Views bar and a left side menu appears to allow input of various parameters needed for the NEF deck creation.

Main: Provides a view of the Process Flow workspace (same as show above).

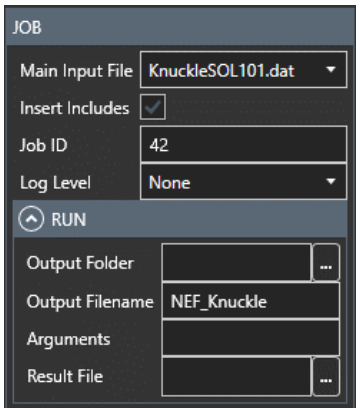
Main BDF: Provides a view of the NEF deck created using MSC Nastran Output box.

Fatigue BDF: Provides a view of the Cf INC files created by the MSC Nastran Output box.

FO6: Provides a view of the F06 file created from the MSC Nastran run, if RUN is selected in the MSC Nastran Output box.

Log: Provides a view of the LOG file created from the MSC Nastran run, if RUN is selected in the MSC Nastran Output box.

The left side menu requires the User to enter parameters to execute the export of the NEF file or to run the MSC Nastran NEF job.



JOB

Main Input File: Select the MSC Nastran input file to modify with NEF entries. This is a SOL101 BDF file when doing a Linear Static Analysis. When doing a Time Modal Transient analysis, CF requires a SOL103 or SOL112 BDF file. When doing a Time Transient analysis, CF requires a SOL112 BDF file.

Insert Includes: Allow the automatic inclusion of the CF generated INC files into the BDF deck.

Job ID: Sets the job ID to use in the MSC Nastran analysis.

Log Level: Sets the LOG level for the MSC Nastran analysis. See the MSC Nastran User Guide for further information.

RUN

Output Folder: Destination where the output files from the CF NEF deck creator will be written. If not defined, the folder will be the CF **filename_cfjob** directory.

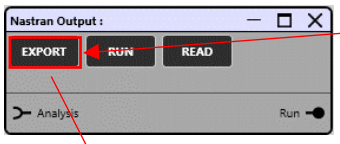
Output Jobname: Prefix name to use for the output files from the CF NEF deck creator. If not defined, the CF will use the CF filename.

Arguments: MSC Nastran arguments to include with the MSC Nastran analysis.

Result File: After selecting RUN from the MSC Nastran Output box, the User will be able to select an CF output CSV file to read into CF and few results for the NEF run.

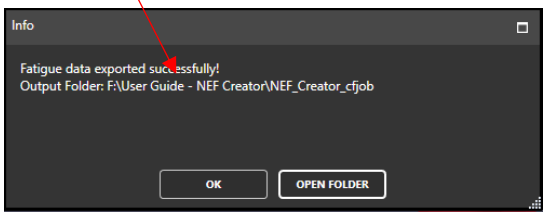
Exporting or Running a NEF Analysis

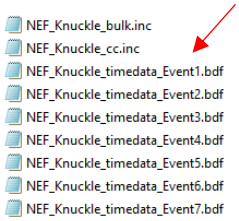
The NEF Deck Creation capability is design to support the easy creation and export of a NEF deck and also allows a MSC Nastran analysis to be run, using the INC files within the original BDF deck. To export the NEF INC files only, the Users simply needs to select EXPORT and the INC files will be created along with the TABRND files for each event. If “Insert Includes” is selected, a modified BDF file will also be created and called <*filename*>_NEF. The modified BDF file will contain the INC files and can be run directly by Nastran.



EXPORT: This button will create a number of files. The ***_bulk.inc** and ***_cc.inc** files are the INCLUDE files that are added to the exist BDF file automatically if "Insert Includes" is selected. The other file(s) is a TABRND file for each event in the loading.

For the example referenced earlier, there are 7 loading events so 7 TABLED1 files are created as part of the expert process.





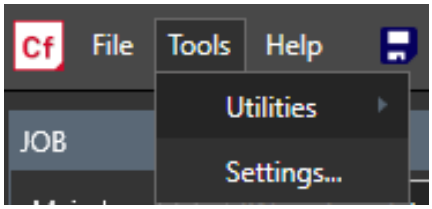
Note:

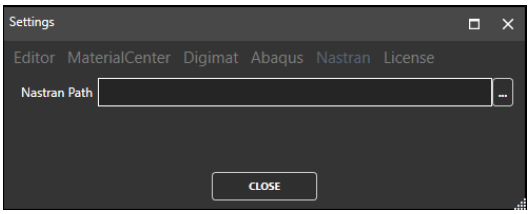
*_bulk.inc includes entries for the Bulk Data section of the MSC Nastran BDF.

*_cc.inc includes entries for the Case Control section of the MSC Nastran

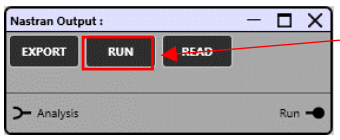
The User also has an option to run the MSC Nastran job without leaving CAEfatigue.

This action requires to User to have access to the MSC Nastran software (v2021.2 or higher) and have valid MSC Nastran license. The User must first tell Cf where to find the MSC Nastran executable file by setting the location via the **TOOLS> UTILITIES> MSC Nastran** path selection box.



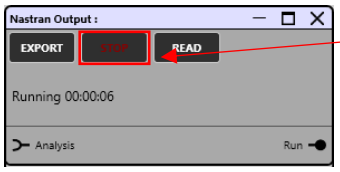


Once the path is set, the User can select **RUN** within the Cf analysis, which will execute the MSC Nastran analysis. The advantage of doing this analysis in NEF is that the OP2 file does not need to be created by MSC Nastran or read into CAEfatigue. This can be a significant time saver when dealing with very large jobs.



RUN: This button will perform the Export of the INC and TABRND file and then run the BDF file. In order to run a NEF analysis, the BDF file must be modified to include the fatigue INC and TABRND files by selecting "Insert Includes" from the Job box or the User can do this manually.

Note: The **RUN** button will create a copy of the original BDF file and append **_NEF** to the filename. The copy will automatically contain the INC files.



STOP: If desired, the User can stop the MSC Nastran job by selecting the STOP button during the NEF run.

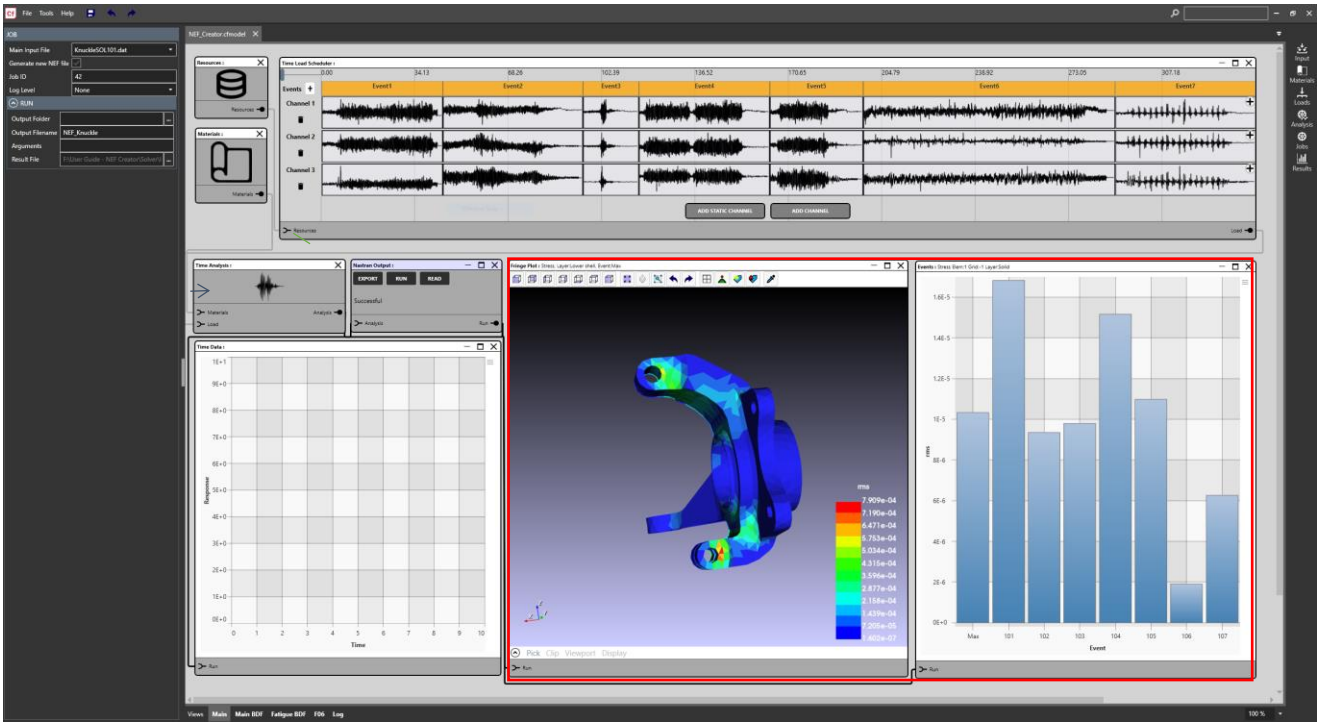
Once the job is complete, the User can see the results by selecting the READ button and select the appropriate CSV file containing the Cf results.

READ: The results of the NEF analysis can be viewed within CF, if desired. To do this, the User must select the READ button which will bring up a selection box.

The NEF output CSV file will be generated and stored where the solver files are located. The name of the file will be **<filename_nef_SN_<job id>.csv**.

Note: The NEF CSV file contains the information to view results in the **Fringe plotter** and **Event plotter** only.

Figure 18 Viewing output from NEF analysis.



16 | APPENDIX 1: CSV FILE FORMATS

Main CSV File Format

A Frequency Domain analysis Cf analysis will result (by default) in the creation of a **filename.csv** file with the following output data:

A	B	C	D	E	F	G	H	I	J
Element	Grid	Layer	Elset ID	MID	Event	M0	M1	M2	M4
K	L	M	N	O	P	Q	R	S	T
rms	rms_strain	E[0]	E[p]	Irr_Factor	Mean Stress	Mean + MAXPEAK * rms	Mean - MAXPEAK * rms	Mean + MAXPEAK *rms strain	Mean – MAXPEAK *rms strain
U	V	W	X	Y	Z	AA	AB		
Damage (for Duty Cycle)	Log Damage (for Duty Cycle)	Life (in Duration Unit)	Log of Life (in Duration Unit)	MOS / FOS *	Plasticity Index	Frequency at Max Response	Critical Angle	**	

* Margin of Safety (MOS) is replaced by Factor of Safety (FOS) when FOS is requested.

** EXTRA COLUMNS of data will be in the CSV file if vFTGPARM>MULTIAX is set to any value other than NONE. See User Guide “Introduction to Multiaxial Assessment” for further information on the additional columns added to the CSV file depending on the multiaxial assessment method chosen.

Some comments:

- The order and listing of output will change with various selected parameters (i.e. selection of SIMSINE, inclusion of Event name, etc.). However, the above listing is typical for stress output.
- **RMS Strain** (column L) is the elastic plastic strain calculated using the elastic FEA stress results and the Neuber Rule for conversion. This column will only be populated if the cyclic strength coefficient (K') and strain hardening exponent (n') are provided in the vFTGMAT or VFTGSTAT entries.
- For a **Time Domain** analysis, columns Q, R, S, T are replaced by MAX Stress, MIN Stress, MAX Strain and MIN Strain.
- The **Margin of Safety - MOS** (column Y) is a percentage value that can be positive or negative. If positive, the User can increase the loading until the positive percentage becomes zero. At this point, the model will fail within the required duration specified by the User (vFTGSEQ: MOS_DUR). If the percentage is negative, the model will fail sooner than the required duration.
- The **Factor of Safety – FOS** will output instead of MOS if specified on the vFTGPARM>FOS entry. FOS is a multiplier to the stress points in the output time history that can be applied before the required LIFE is reached. i.e. a FOS of 2 means the equivalent stress results can be doubled before reaching the required life.
- The **Plasticity Index** (column Z) is defined as the real (elastic plastic) max strain (***Mean + MAXPEAK*rms_strain***) divided by the FEA (elastic) strain (***Mean + MAXPEAK * rms_stress*** / ***E***). There is no Plasticity Index for a Time Domain analysis.
- The **Critical Plane Angle** (column AB) is populated if vFTGPARM>COMB = critical plan angle when doing a Time Domain analysis. Also, the MEAN + / - columns are replaced with MAX / MIN data when doing a Time Domain analysis.
- **Damage** values are presented as Damage for a Duty Cycle i.e., damage / sec * event duration (converted to seconds from duration units), whereas **Life** is presented as life in duration units. LIFE is the number of duration units required to FAIL due to the input loading applied in the Cf file.
- A summary of the fatigue damage results (for top 10 elements) is presented in the LOG file.

Example output below:

Element	Grid	Layer	Elset Id	Material ID	Event	m0	m1	m2	m4	rms	rms_strain	E(p)	E(p)	Ir_Factor	Mean Stress	Mean*rms	Mean-S*rms	Mean+S*rms_strain	Mean-S*rms_strain	Damage(Duty Cycle)	Log_Damage (Duty Cycle)	Life(seconds)	Log of Life(seconds)	MOS	Plasticity Index	Freq at peak response
5971	0	none	600	60	100	4.40E+02	4.08E+03	8.01E+04	6.74E+07	2.10E+01	0.00E+00	1.35E+01	2.90E+01	4.65E-01	0.00E+00	1.05E+02	-1.05E+02	0.00E+00	0.00E+00	1.24E-11	-1.09E+01	8.07E+10	1.09E+01	7.67E+01	0.00E+00	4.88E-01
5971	2658	none	600	60	100	8.06E+02	7.23E+03	1.37E+05	1.14E+08	2.84E+01	0.00E+00	1.30E+01	2.89E+01	4.51E-01	0.00E+00	1.42E+02	-1.42E+02	0.00E+00	0.00E+00	1.43E-09	-8.85E+00	7.01E+08	8.85E+00	7.30E+01	0.00E+00	8.85E+00
5971	2622	none	600	60	100	3.75E+02	4.12E+03	8.60E+04	7.65E+07	1.94E+01	0.00E+00	1.51E+01	2.98E+01	5.08E-01	0.00E+00	9.68E+01	-9.68E+01	0.00E+00	0.00E+00	4.01E-12	-1.14E+01	2.49E+11	1.14E+01	7.66E+01	0.00E+00	8.85E+00
5971	2671	none	600	60	100	3.63E+02	3.24E+03	6.06E+04	4.87E+07	1.91E+01	0.00E+00	1.29E+01	2.84E+01	4.55E-01	0.00E+00	9.53E+01	-9.53E+01	0.00E+00	0.00E+00	2.54E-12	-1.16E+01	3.94E+11	1.16E+01	7.67E+01	0.00E+00	8.85E+00
5971	2616	none	600	60	100	6.27E+03	5.57E+04	1.03E+06	8.15E+08	7.92E+01	0.00E+00	1.28E+01	2.81E+01	4.56E-01	0.00E+00	3.96E+02	-3.96E+02	0.00E+00	0.00E+00	8.31E-03	-2.08E+00	1.20E+02	2.08E+00	2.65E+01	0.00E+00	8.85E+00
13897	0	lower	600	60	100	1.57E+03	1.36E+04	2.55E+05	2.30E+08	3.97E+01	0.00E+00	1.27E+01	3.00E+01	4.24E-01	0.00E+00	1.98E+02	-1.98E+02	0.00E+00	0.00E+00	2.55E-07	-6.59E+00	3.93E+06	6.59E+00	6.23E+01	0.00E+00	8.85E+00
13897	0	upper	600	60	100	1.57E+03	1.36E+04	2.55E+05	2.30E+08	3.97E+01	0.00E+00	1.27E+01	3.00E+01	4.24E-01	0.00E+00	1.98E+02	-1.98E+02	0.00E+00	0.00E+00	2.55E-07	-6.59E+00	3.93E+06	6.59E+00	6.23E+01	0.00E+00	8.85E+00
13897	2858	lower	600	60	100	1.72E+03	1.48E+04	2.73E+05	2.43E+08	4.15E+01	0.00E+00	1.26E+01	2.98E+01	4.22E-01	0.00E+00	2.07E+02	-2.07E+02	0.00E+00	0.00E+00	5.09E-07	-6.29E+00	1.96E+06	6.29E+00	6.06E+01	0.00E+00	8.85E+00
13897	2858	upper	600	60	100	1.72E+03	1.48E+04	2.73E+05	2.43E+08	4.15E+01	0.00E+00	1.26E+01	2.98E+01	4.22E-01	0.00E+00	2.07E+02	-2.07E+02	0.00E+00	0.00E+00	5.09E-07	-6.29E+00	1.96E+06	6.29E+00	6.06E+01	0.00E+00	8.85E+00
13897	2859	lower	600	60	100	2.29E+02	1.87E+03	3.15E+04	2.62E+07	1.51E+01	0.00E+00	1.17E+01	2.88E+01	4.07E-01	0.00E+00	7.57E+01	-7.57E+01	0.00E+00	0.00E+00	3.69E-14	-1.34E+01	2.71E+13	1.34E+01	7.67E+01	0.00E+00	8.85E+00
13897	2859	upper	600	60	100	2.29E+02	1.87E+03	3.15E+04	2.62E+07	1.51E+01	0.00E+00	1.17E+01	2.88E+01	4.07E-01	0.00E+00	7.57E+01	-7.57E+01	0.00E+00	0.00E+00	3.69E-14	-1.34E+01	2.71E+13	1.34E+01	7.67E+01	0.00E+00	8.85E+00
13897	2756	lower	600	60	100	4.09E+03	3.59E+04	6.91E+05	6.36E+08	6.39E+01	0.00E+00	1.30E+01	3.03E+01	4.29E-01	0.00E+00	3.20E+02	-3.20E+02	0.00E+00	0.00E+00	4.28E-04	-3.37E+00	2.34E+03	3.37E+00	3.93E+01	0.00E+00	8.85E+00
13897	2756	upper	600	60	100	4.09E+03	3.59E+04	6.91E+05	6.36E+08	6.39E+01	0.00E+00	1.30E+01	3.03E+01	4.29E-01	0.00E+00	3.20E+02	-3.20E+02	0.00E+00	0.00E+00	4.28E-04	-3.37E+00	2.34E+03	3.37E+00	3.93E+01	0.00E+00	8.85E+00

PSD.CSV File Format

A Cf analysis will result (by default) in the creation of a **filenamePSD.csv** file with the following output data:

A	B	C	D	E	F	G	H	I	J
Frequency	Element	***Direction***	Grid	Layer	Event	Input PSD	Transfer Function	Response PSD	M0
K	L	M	N	O	P	Q	R	S	T
M1	M2	M4	Rel Node	Damage					

Some comments:

- **Frequency** is the X axis frequency point that will be used by the PSD Plotter to display the Input PSD, Transfer Function and Response PSD.
- **Element** / **Grid** / **Layer** / **Event** are used to specify the exact location of the output.
- ***Direction*** *is currently not used.*
- **Input PSD** / **Transfer Function** / **Response PSD** are the output Y axis values that will be used by the PSD Plotter.

Example output below:

Freq	Elem	Direction	Grid	Layer	Event	Input PSD	Transfer function	Response PSD	M0	M1	M2	M4
9.77E-02	13897		0	lower	100	2.54E+07	1.94E-05	4.94E+02	2.41E+01	1.57E+00	1.15E-01	7.32E-04
9.77E-02	13897		0	lower	101	5.09E+07	1.94E-05	9.89E+02	4.83E+01	3.14E+00	2.30E-01	1.46E-03
9.77E-02	13897		0	upper	100	2.54E+07	1.94E-05	4.94E+02	2.41E+01	1.57E+00	1.15E-01	7.32E-04
9.77E-02	13897		0	upper	101	5.09E+07	1.94E-05	9.89E+02	4.83E+01	3.14E+00	2.30E-01	1.46E-03
9.77E-02	13897		2858	lower	100	2.54E+07	2.08E-05	5.30E+02	2.59E+01	1.69E+00	1.23E-01	7.85E-04
9.77E-02	13897		2858	lower	101	5.09E+07	2.08E-05	1.06E+03	5.18E+01	3.37E+00	2.47E-01	1.57E-03
9.77E-02	13897		2858	upper	100	2.54E+07	2.08E-05	5.30E+02	2.59E+01	1.69E+00	1.23E-01	7.85E-04
9.77E-02	13897		2858	upper	101	5.09E+07	2.08E-05	1.06E+03	5.18E+01	3.37E+00	2.47E-01	1.57E-03
9.77E-02	13897		2859	lower	100	2.54E+07	2.41E-06	6.13E+01	2.99E+00	1.95E-01	1.43E-02	9.07E-05
9.77E-02	13897		2859	lower	101	5.09E+07	2.41E-06	1.23E+02	5.99E+00	3.90E-01	2.85E-02	1.81E-04
9.77E-02	13897		2859	upper	100	2.54E+07	2.41E-06	6.13E+01	2.99E+00	1.95E-01	1.43E-02	9.07E-05
9.77E-02	13897		2859	upper	101	5.09E+07	2.41E-06	1.23E+02	5.99E+00	3.90E-01	2.85E-02	1.81E-04
9.77E-02	13897		2756	lower	100	2.54E+07	5.19E-05	1.32E+03	6.44E+01	4.19E+00	3.07E-01	1.95E-03
9.77E-02	13897		2756	lower	101	5.09E+07	5.19E-05	2.64E+03	1.29E+02	8.39E+00	6.14E-01	3.91E-03
9.77E-02	13897		2756	upper	100	2.54E+07	5.19E-05	1.32E+03	6.44E+01	4.19E+00	3.07E-01	1.95E-03
9.77E-02	13897		2756	upper	101	5.09E+07	5.19E-05	2.64E+03	1.29E+02	8.39E+00	6.14E-01	3.91E-03

_DVA.CSV File Format

When requesting **vFTGDEF: ABSRESP**, the follow outputs are generated in the **filename_dva.csv** output file:

A	B	C	D	E	F	G	H	I	J
Element	Grid	Layer	Elset ID	Event	M0	M1	M2	M4	rms
K	L	M	N	O	P	Q	R	S	T
E[0]	E[p]	Irr_Factor	Mean Stress	Mean + MAXPEAK * rms	Mean - MAXPEAK * rms	X	Y	Z	Rx
U	V	W	X	Y	Z	AA			
Ry	Rz	Frequency at Peak Response	Minimum Clearance	MAX Collision Probability Gaussian	MAX Collision Probability Rayleigh	MAX Collision Probability Davenport			

Some comments:

- X, Y and Z are the RMS values of the X-motion, Y-motion and Z-motion.
- Rx, Ry and Rz are the RMS values of the rotation about X, rotation about Y and rotation about Z.
- IF RELRESP is also specified, the **Minimum Clearance** (column X) is the smallest distance a node from the ABSRESP set will get to any node in the RELRESP set. The calculation considers the relative motion between the nodes including any phasing in the motion between the nodes.
- The **MAX Collision Probabilities** represent the probability of collision based on the Gaussian, Rayleigh or Davenport probabilities for the actual distance between the nodes in the ABSRESP and RELRESP groups.

Example output below:

Element	Grid	Layer	Elset Id	Event	m0	m1	m2	m4	rms	E[0]	E[p]	Irr_Factor	Mean	Mean+ s0.01* rms	Mean- s0.01* rms	X	Y	Z	RX	RY	RZ	Freq at peak respon-se	Minimum Clearance
0	32	D	7	102	1.64E+01	9.35E+01	6.26E+02	4.08E+04	4.04E+00	6.19E+00	8.07E+00	7.66E-01	0.00E+00	2.01E+01	-2.01E+01	3.20E-02	2.93E-01	4.03E+00	9.28E-03	7.41E-04	4.54E-05	7.00E+00	
0	32	D	7	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN(	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.00E+00	
0	32	D	7	SUM (or max event 102)					4.04E+00							3.20E-02	2.93E-01	4.03E+00	9.28E-03	7.41E-04	4.54E-05	7.00E+00	
0	21577	D	8	102	3.99E+01	2.69E+02	2.02E+03	1.45E+05	6.32E+00	7.11E+00	8.47E+00	8.40E-01	0.00E+00	3.16E+01	-3.16E+01	2.86E-01	5.38E-01	6.29E+00	2.08E-03	1.01E-03	5.34E-04	7.00E+00	
0	21577	D	8	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN(	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.00E+00	
0	21577	D	8	SUM (or max event 102)					6.32E+00							2.86E-01	5.38E-01	6.29E+00	2.08E-03	1.01E-03	5.34E-04	7.00E+00	
0	21578	D	8	102	3.96E+01	2.66E+02	2.00E+03	1.42E+05	6.29E+00	7.10E+00	8.44E+00	8.41E-01	0.00E+00	3.15E+01	-3.15E+01	2.88E-01	5.30E-01	6.26E+00	2.23E-03	1.35E-03	4.65E-04	7.00E+00	
0	21578	D	8	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN(	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.00E+00	
0	21578	D	8	SUM (or max event 102)					6.29E+00							2.88E-01	5.30E-01	6.26E+00	2.23E-03	1.35E-03	4.65E-04	7.00E+00	
0	21579	D	8	102	4.07E+01	2.73E+02	2.04E+03	1.40E+05	6.38E+00	7.08E+00	8.28E+00	8.55E-01	0.00E+00	3.19E+01	-3.19E+01	9.65E-02	1.59E-01	6.38E+00	2.22E-03	1.21E-03	5.88E-04	7.00E+00	
0	21579	D	8	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN(	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.00E+00	
0	21579	D	8	SUM (or max event 102)					6.38E+00							9.65E-02	1.59E-01	6.38E+00	2.22E-03	1.21E-03	5.88E-04	7.00E+00	
0	21580	D	8	102	4.07E+01	2.73E+02	2.04E+03	1.40E+05	6.38E+00	7.08E+00	8.28E+00	8.54E-01	0.00E+00	3.19E+01	-3.19E+01	1.15E-01	1.83E-01	6.37E+00	2.46E-03	1.23E-03	2.29E-04	7.00E+00	
0	21580	D	8	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN(	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.00E+00	

_D_REL.CSV File Format

When requesting **vFTGDEF: RELRESP**, the follow outputs are generated in the **filename_d_rel.csv** output file:

A	B	C	D	E	F	G	H	I	J
Grid	minus Grid	Layer	Elset ID	Distance	Event	M0	M1	M2	M4
K	L	M	N	O	P	Q	R	S	T
rms undirected	rms	MAXPEAK * rms	E[0]	E[p]	Irr_Factor	Clearance	Gaussian Collision Probability	Rayleigh Collision Probability	Davenport Collision Probability
U	V	W	X	Y	Z				
X	Y	Z	Rx	Ry	Rz				

Some comments:

- "**Grid**" nodes are nodes from the ABSRESP set and "**minusGrid**" nodes are nodes from the RELRESP set. With the grids and minusGrids, Cf will calculate the relative motion of the nodes taking into account any phasing in motion between the nodes. Note: only nodes that fall within a certain relative distance (determined by RMS or probability model) will be used as grid and minusGrid points.
- **RMS undirected** means the RMS of the relative displacement vector between Grid and minusGrid.
- **RMS** means the RMS of the relative displacement vector projected in the direction between minusGrid and Grid.
- **Clearance** means the minimum distance between a reference node (minusGrid) and all other nodes (grids) in the ABSRESP reference set.
- **Collision Probability** means the probability that a reference node will contact any grid point. Cf offers three different probability models; Gaussian, Rayleigh and Davenport (sometimes referred to as Sarafin).
- **X**, **Y** and **Z** are the RMS displacement values of the X-motion, Y-motion and Z-motion.
- **Rx**, **Ry** and **Rz** are the RMS displacement values of the rotation about X, rotation about Y and rotation about Z.

Example output below:

Grid	minusGrid	Layer	Elset Id	Distance	Event	m0	m1	m2	m4	rms undirected	rms	>0.01*rms	E[0]	E[p]	Irr_Factor	Clearance	Gaussian Collision P	Rayleigh Collision P	Davenport Collision P	X	Y	Z	RX	RY	RZ
63678	21582	D_rel	7	4.34E+01	102	4.04E-01	3.75E+00	3.89E+01	5.75E+03	3.32E+00	6.35E-01	3.22E+00	9.82E+00	1.22E+01	8.07E-01	4.02E+01	0.00E+00	0.00E+00	0.00E+00	6.34E-01	4.55E-01	3.23E+00	5.30E-03	2.74E-03	1.39E-03
63678	21582	D_rel	7	4.34E+01	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
63678	21582	D_rel	7	4.34E+01	Max event	102				3.32E+00	6.35E-01	-1.#IND00e+000				-1.#IND00	0.00E+00	0.00E+00	0.00E+00	6.34E-01	4.55E-01	3.23E+00	5.30E-03	2.74E-03	1.39E-03
63654	21587	D_rel	7	3.34E+01	102	1.64E+00	1.47E+01	1.43E+02	1.79E+04	3.21E+00	1.28E+00	6.48E+00	9.34E+00	1.12E+01	8.34E-01	2.70E+01	0.00E+00	0.00E+00	0.00E+00	6.62E-01	4.67E-01	3.11E+00	6.12E-03	2.62E-03	1.52E-03
63654	21587	D_rel	7	3.34E+01	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
63654	21587	D_rel	7	3.34E+01	Max event	102				3.21E+00	1.28E+00	-1.#IND00e+000				-1.#IND00	0.00E+00	0.00E+00	0.00E+00	6.62E-01	4.67E-01	3.11E+00	6.12E-03	2.62E-03	1.52E-03
63672	21587	D_rel	7	3.32E+01	102	2.04E+00	1.82E+01	1.78E+02	2.22E+04	3.24E+00	1.43E+00	7.23E+00	9.34E+00	1.12E+01	8.35E-01	2.60E+01	0.00E+00	0.00E+00	0.00E+00	6.59E-01	4.67E-01	3.13E+00	6.12E-03	2.62E-03	1.52E-03
63672	21587	D_rel	7	3.32E+01	103	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	0.00E+00	0.00E+00	0.00E+00	-1.#IND00	1.#QNAN0	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00	0.00E+00
63672	21587	D_rel	7	3.32E+01	Max event	102				3.24E+00	1.43E+00	-1.#IND00e+000				-1.#IND00	0.00E+00	0.00E+00	0.00E+00	6.59E-01	4.67E-01	3.13E+00	6.12E-03	2.62E-03	1.52E-03

Note: If equivalent values for "NAME" are specified in the vFTGEVNT entry, then an additional column is written which includes the Users specified event names. If equivalent value for "UNITS" is specified in the vFTGSEQ section, then two additional columns are written which includes the Life and Log of Life in the new units.

_SPOT.CSV File Format

When requesting **vFTGDEF: SPOTW**, the follow outputs are generated in the **filename_spot.csv** output file. The file is virtually identical to the Main CSV file except for the inclusion of FACE and ANGLE.

A	B	C	D	E	F	G	H	I	J
Element	Face	Grid	Angle	Elset ID	Material ID	Event	M0	M1	M2
K	L	M	N	O	P	Q	R	S	T
M4	rms	rms_strain	E[0]	E[p]	Irr_Factor	Mean Stress	Mean + MAXPEAK * rms	Mean - MAXPEAK * rms	Mean + MAXPEAK *rms strain
U	V	W	X	Y	Z	AA	AB		
Mean – MAXPEAK *rms strain	Damage (Duty Cycle)	Log Damage (Duty Cycle)	Life (/ Duration Unit)	Log of Life (/ Duration Unit)	MOS	Plasticity Index	Frequency at Max Response		

Some comments:

- **Face** is the location within the spot weld where the highest stress results are generated for each Event. Outputs can be Top Face, Bottom Face or Nugget.
- **Angle** is the direction (angle), on the face, where the highest stress results are generated for each Event.

_SEAM.CSV File Format

When requesting **vFTGDEF: SEAMW**, the follow outputs are generated in the **filename_seam.csv** output file. The file is virtually identical to the Main CSV file except for the addition of FLEXIBILITY.

A	B	C	D	E	F	G	H	I	J
Element	Grid	Layer	Elset ID	Material ID	Event	M0	M1	M2	M4
K	L	M	N	O	P	Q	R	S	T
rms	rms_strain	E[0]	E[p]	Irr_Factor	Mean Stress	Mean + MAXPEAK * rms	Mean - MAXPEAK * rms	Mean + MAXPEAK *rms strain	Mean – MAXPEAK *rms strain
U	V	W	X	Y	Z	AA	AB		
Damage (Duty Cycle)	Log Damage (Duty Cycle)	Life (/ Duration Unit)	Log of Life (/ Duration Unit)	MOS	Plasticity Index	Frequency at Max Response	Flexibility		

Some comments:

- **Flexibility** is an indicator of what SN curve is being used in the seam weld analysis. If Flexibility=0, then the “stiff” SNBR0 curve is dominating the analysis. Any value between 0 and 1 means an interpolated SN value is being used between SNBR0 and SNBR1. If the value is 1, then the “flexible” SNBR1 curve is dominating the analysis.

_FORCE.CSV File Format

When requesting **vFTGDEF: ABSRELP with ABSVARi=F** (force) or **vFTGDEF: USERWHS**, the follow outputs are generated in the **filename_force.csv** output file.

A	B	C	D	E	F	G	H	I	J
Element	Grid	*GeomF*	Fangle	*Fvariance*	Frequency at Max Response	Layer	Elset ID	Event	M0
K	L	M	N	O	P	Q	R	S	T
M1	M2	M4	RMS	E[0]	E[p]	Irr_Factor	Mean Stress	Mean + MAXPEAK * rms	Mean - MAXPEAK * rms
U	V	W	X	Y	Z	AA	AB	AC	AD
X	Y	Z	Rx	Ry	Rz	Static X	Static Y	Static Z	Static RX
AE	AF	AG							
Static RY	Static RZ	Damage							

Some comments:

- ***GeomF*** *and* ***Fvariance*** *are currently not being used.*
- **Fangle** is only generated for an USERWHS analysis and is the direction (angle), on the face, where the highest stress results are generated for each Event.
- **STATIC** values are obtained from the solver file and are simply reported in the CSV file. No calculations are done using the STATIC values.

Example output below:

Element	Grid	GeomF	Fangle	Fvariance	Freq at Peak Response	Layer	Elset Id	Event	m0	m1	m2	m4	rms	E[0]	E[p]	Irr_Factor	Mean	Mean+5*rms	Mean-5*rms	X	Y	Z	RX	RY	RZ	Static X	Static Y	Static Z	Static RX	Static RY	Static RZ	damage
1	0	0	20	0	94.6027	none	0	101	0.073057	5.16615	444.677	3.85E+06	0.27029	78.0175	93.0253	0.83867	0	0	0	0.072031	0.204785	0.06227	0.472353	0.810175	0.144624							5.52E-05
1	0	0	160	0	94.6027	none	0	102	0.001416	0.075912	7.95064	169117	0.03763	74.9328	145.846	0.513782	0	0	0	0.019587	0.031054	0.036816	0.128759	0.118965	0.031147							1.55E-08
1	0	0	0	0	94.6027	none	0	103	3.50272	320.293	30429.8	2.91E+08	1.87155	93.2066	97.8096	0.952939	0	0	0	0.694519	1.22783	0.536795	3.91122	6.54898	0.968769							0.091513
1	0	0	0	0	94.6027	none	0	SUM (or max event 103)	3.50272	320.293	30429.8	2.91E+08	1.87155	93.2066	97.8096	0.952939	0	0	0	0.694519	1.22783	0.536795	3.91122	6.54898	0.968769	0	0	0	0	0	0	0.091568
2	0	0	180	0	94.6027	none	0	101	0.003553	0.301979	28.7235	319027	0.059605	89.9162	105.389	0.853184	0	0	0	0.018198	0.050767	0.057148	0.107969	0.088881	0.055476							2.89E-07
2	0	0	340	0	94.6027	none	0	102	0.000119	0.010451	1.50289	42316.9	0.01092	112.266	167.8	0.669046	0	0	0	0.009495	0.007545	0.013628	0.024553	0.03156	0.013178							1.09E-11
2	0	0	180	0	94.6027	none	0	103	0.279518	24.8098	2522.4	3.29E+07	0.528694	94.9953	114.143	0.832249	0	0	0	0.363061	0.327087	0.449774	0.900809	0.893543	0.465361							0.000979
2	0	0	180	0	94.6027	none	0	SUM (or max event 103)	0.279518	24.8098	2522.4	3.29E+07	0.528694	94.9953	114.143	0.832249	0	0	0	0.363061	0.327087	0.449774	0.900809	0.893543	0.465361	0	0	0	0	0	0	0.000979
7	0	0	0	0	94.6027	none	0	101	0.097303	7.70494	693.493	6.29E+06	0.311934	84.4225	95.1993	0.886797	0	0	0	0.186149	0.149648	0.055234	0.487155	0.642453	0.250556							9.39E-05
7	0	0	200	0	94.6027	none	0	102	0.000559	0.07461	12.0852	357182	0.023642	147.04	171.917	0.855298	0	0	0	0.041748	0.021568	0.023674	0.162777	0.091899	0.040505							1.73E-10
7	0	0	0	0	94.6027	none	0	103	5.49328	502.164	47672.1	4.54E+08	2.34377	93.1572	97.6229	0.954255	0	0	0	1.34394	1.11086	0.458452	3.76014	4.96725	1.8663							0.204155
7	0	0	0	0	94.6027	none	0	SUM (or max event 103)	5.49328	502.164	47672.1	4.54E+08	2.34377	93.1572	97.6229	0.954255	0	0	0	1.34394	1.11086	0.458452	3.76014	4.96725	1.8663	0	0	0	0	0	0	0.204249

Note: The above example is from a USERWHS analysis, so **Fangle** has values. This example had no static data in the solver file.

_CRITOUT.CSV File Format

When requesting **vFTGDEF> CRITOUT**, the follow outputs are generated in the **filename_force.csv** output file.

A	B	C	D	E	F	G	H	I	J
Element	Grid	Layer	Event	Theta Angle	PHI Angle	Vector X	Vector Y	Vector Z	Damage
K	L	M	N	O	P	Q	R	S	T
RMS Stress									
U	V	W	X	Y	Z	AA	AB	AC	AD

Some comments:

- ***Theta Angle*** is the normal angle from the critical plane.
- ***PHI Angle*** is the direction within the critical plane.
- ***Vector X, Y, Z*** form the components of the normal vector magnitude from the critical plane.

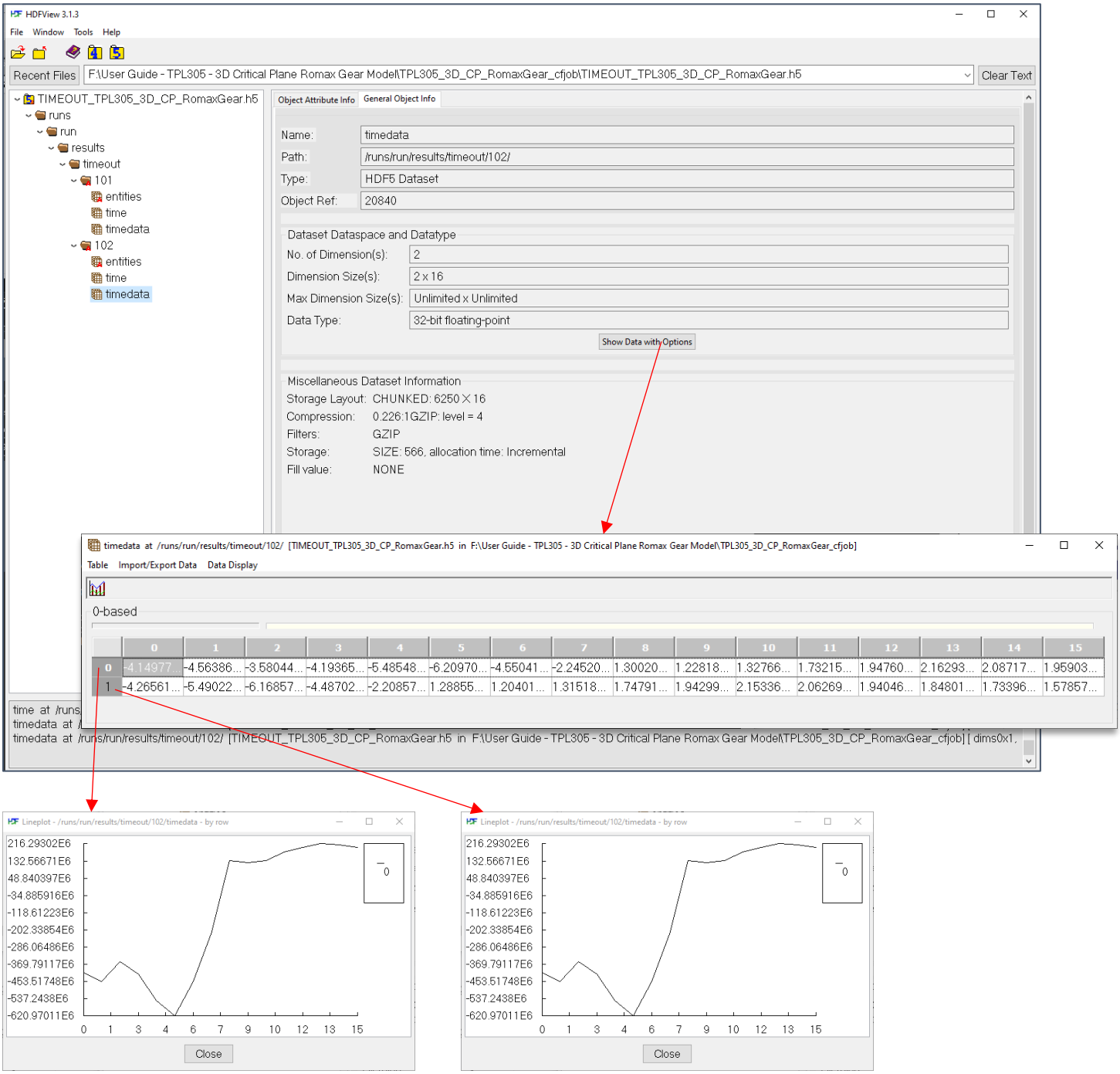
Example output below:

Element	Grid	Layer	Event	Theta	Phi	Vector X	Vector Y	Vector Z	Damage	RMS Stress
9630	9964	Solid	101	0	90	1	0	-8.74E-08	2.00E-30	867139
9630	9964	Solid	101	4	90	0.997564	0.069757	-8.74E-08	2.00E-30	1.20E+06
9630	9964	Solid	101	8	90	0.990268	0.139173	-8.74E-08	3.00E-30	1.57E+06
9630	9964	Solid	101	12	90	0.978148	0.207912	-8.74E-08	3.00E-30	1.98E+06
9630	9964	Solid	101	16	90	0.961262	0.275637	-8.74E-08	1.00E-30	2.41E+06
9630	9964	Solid	101	20	90	0.939693	0.34202	-8.74E-08	1.00E-30	2.85E+06

TIMEOUT_<filename>.h5 File Format

When requesting **vFTGDEF> TIMEOUT**, the software produces an HDF5 (h5) format file of the output time history data. To view an h5 file, the User must use an HDF5 viewer, something like HDFView™.

Below is an example for TIMEOUT results from TPL305 Romax Gear example, which has 2 events with a request for TIMEOUT data from 2 elements. See **TIMEOUT_TPL305_3D_CP_RomaxGear.h5.**



17 | APPENDIX 2: EXAMPLE MSC Nastran SOL112 INPUT FILES WHEN USING FORCE INPUTS

```
SOL 112
CEND
SET 1 = 1035259,1050094,1057009,1068132,8158502
RESVEC (INRLOD, APPLOD, ADJLOD, RVDOF, DAMPLOD, NODYNRSP) = YES
STRESS (SORT1,PRINT,CENTER) = 1
LOADSET = 1
METHOD = 1
SDAMPING=1
$!- First event (12 input time histories) defined by SUBCASE 1
SUBCASE 1
   SUBTITLE=01-ea10
   TSTEP = 1
   DLOAD = 1

Repeat for all 10 events

BEGIN BULK
PARAM     WTMASS     1.0
TABDMP1 1       CRIT
        .1      .03     1000.   .03      ENDT
include model
GRID         101          539.268 2877.30 1804.5
Include 10 sets of time histories in TABLED1 format (ref 101, 102, etc. on TLOAD1
entry)
EIGRL          1            80.0               0                     MASS

LSEQ           1      101      101
LSEQ           1      102      102
LSEQ           1      103      103
Repeat for remaining 12 loads


$`-- Left Front X, Y and Z
FORCE        101      101      101      1.0      1.0      0.0      0.0
FORCE        102      101      101      1.0      0.0      1.0      0.0
FORCE        103      101      101      1.0      0.0      0.0      1.0
Repeat for left rear, right front and right rear


$=............................. Event 1: ea10 ..................................
TSTEP          1     3800      .01       1
Repeat time steps for remaining 10 events
DLOAD          1      1.0      1.0     1101      1.0     1102      1.0     1103
             1.0     1201      1.0     1202      1.0     1203      1.0     1301
             1.0     1302      1.0     1303      1.0     1401      1.0     1402
             1.0     1403
TLOAD1      1101      101                       101
TLOAD1      1102      102                       102
TLOAD1      1103      103                       103
Repeat for remaining 12 loads
Then repeat for all 10 events
CORD2R   101             188.510 3579.02 1906.14 188.510 3579.02 5965.48
         188.509-480.325 1906.14
ENDDATA
```

18 | APPENDIX 3: EXAMPLE MSC Nastran SOL111 INPUT FILES WHEN USING FORCE INPUTS

```
SOL 111
CEND
SET 11 = all elements for analysis
RESVEC (INRLOD, APPLOD, ADJLOD, RVDOF, DAMPLOD, NODYNRSP) = YES
STRESS(PLOT,center) = 11
LOADSET = 1
METHOD = 1
SDAMPING=1
FREQ = 100
$!- Left Front
SUBCASE 1
   SUBTITLE = Left Front X
   DLOAD = 101
SUBCASE 2
   SUBTITLE = Left Front Y
   DLOAD = 102
SUBCASE 3
   SUBTITLE = Left Front Z
   DLOAD = 103

Repeat for Left Rear X,Y,Z, Right Front X,Y,Z and Right Rear X,Y,Z

BEGIN BULK
PARAM     WTMASS     1.0
TABDMP1 1       CRIT
        .1      .03     1000.   .03      ENDT
include model
GRID          101          539.268 2877.30 1804.5
EIGRL           1             80.0               0                    MASS
FREQ1         100     0.0     0.2       76
FREQ1         100    15.5     0.5       10
FREQ          100    4.82    7.76    9.03   12.16   13.35   16.72   16.79

LSEQ            1      101      101
LSEQ            1      102      102
LSEQ            1      103      103

Repeat for left rear, right front and right rear

$`-- Left Front X, Y and Z
FORCE         101      101      101      1.0      1.0      0.0      0.0
FORCE         102      101      101      1.0      0.0      1.0      0.0
FORCE         103      101      101      1.0      0.0      0.0      1.0
Repeat for left rear, right front and right rear

$!-- left front
DLOAD         101      1.0      1.0     1101
DLOAD         102      1.0      1.0     1102
DLOAD         103      1.0      1.0     1103

Repeat for left rear, right front and right rear
$!-- left front
RLOAD1       1101      101                       600
RLOAD1       1102      102                       600
RLOAD1       1103      103                       600
TABLED1       600
              0.0      1.0    100.0      1.0     ENDT
Repeat for left rear, right front and right rear
CORD2R    101              188.510 3579.02 1906.14 188.510 3579.02 5965.48
          188.509-480.325 1906.14
ENDDAT
```

19 | APPENDIX 4: MMPDS MATERIAL DATA TABLE

The following data is extracted from DOT/FAA/AR-MMPDS-01 (Metallic Materials Properties Development and Standardization Handbook).

Chapter	Page	type	direction	name	Kt	A1	A2	A3	A4	t	min_R	max_R
2	179	Stainless Steel bar	long and long tran	15-5PH(H1025)	3.0	19.690	-9.14	0.595	18.160		-1.00	0.50
2	180	Stainless Steel plate	long and long tran	15-5PH(H1025)	1.0	110.100	-47.22	1.000	0.000		0.10	0.10
3	182	Al. alloy plate	long	2219-T851	2.0	7.920	-2.69	0.640	16.000		-1.00	0.75
3	183	Al. alloy plate	long	2219-T851	3.2	8.460	-2.83	0.760	3.930		-1.00	0.75
3	184	Al. alloy plate	long tran	2219-T851	4.0	10.850	-4.34	0.686	0.000		-1.00	0.75
3	185	Al. alloy plate	long	2219-T851	5.0	8.760	-3.05	0.722	0.000		-1.00	0.75
3	289	Al. alloy var wrought	long	6061-T6	1.0	20.680	-9.84	0.630	0.000		-1.00	0.50
3	344	Al. alloy plate	long tran t/2 loc	7050-T7451	1.0	12.484	-4.878	0.420	7.500	8.000	-1.00	0.02
3	344	Al. alloy plate	long tran t/2 loc	7050-T7451	1.0	12.484	-4.878	0.420	9.231	6.5	-1.00	0.02
3	344	Al. alloy plate	long tran t/2 loc	7050-T7451	1.0	12.484	-4.878	0.420	14.118	4.25	-1.00	0.02
3	347	Al. alloy plate	long tran t/4 loc	7050-T7451	1.0	10.700	-3.81	0.590	10.000		-1.00	0.50
3	348	Al. alloy plate	long and long tran t/4 loc	7050-T7451	3.0	10.000	-3.96	0.640	0.000		-1.00	0.50
3	349	Al. alloy extruded	long	7050-T7451X	1.0	8.230	-2.82	0.300	10.000		-0.40	0.20
3	350	Al. alloy hand forging	long	7050-T7452	1.0	7.060	-1.89	0.600	30.000		-1.00	0.50
3	351	Al. alloy hand forging	long tran and short tran	7050-T7452	1.0	7.580	-2.14	0.570	21.000		-1.00	0.50
3	352	Al. alloy hand forging	long, long tran and short tran	7050-T7452	3.0	8.210	-2.96	0.680	5.000		-1.00	0.50
3	353	Al. alloy die forging	long	7050-T74	1.0	16.800	-6.97	1.000	0.000		0.00	0.00
3	354	Al. alloy die forging	long	7050-T74	3.0	10.500	-4.14	0.629	0.000		-1.00	0.33
3	361	Al. alloy extruded	long and long tran	7050-T7651X	1.0	11.800	-4.38	0.610	12.000		-1.00	0.50
3	362	Al. alloy extruded	long and long tran	7050-T7651X	3.0	10.380	-4.26	0.563	0.000		-1.00	0.50
3	406	Al. alloy var	long	7075-T6	1.0	18.220	-7.77	0.620	10.150		-1.00	0.50
3	407	Al. alloy bar	long	7075-T6	1.6	8.260	-2.62	0.525	15.300		-1.00	0.74
3	408	Al. alloy bar	long	7075-T6	3.4	9.190	-3.646	0.386	5.360		-1.00	0.46
3	409	Al. alloy sheet	long	7075-T6	1.0	14.860	-5.8	0.490	5.800		-1.00	0.40
3	410	Al. alloy sheet	long	7075-T6	1.5	9.540	-3.52	0.490	18.700		-1.00	0.75
3	411	Al. alloy sheet	long	7075-T6	2.0	7.500	-2.46	0.540	18.600		-1.00	0.75
3	412	Al. alloy sheet	long	7075-T6	4.0	10.200	-4.63	0.510	5.300		-1.00	0.75
3	413	Al. alloy sheet	long	7075-T6	5.0	7.510	-2.92	0.580	6.700		-1.00	0.75
3	436	Al. alloy extrusion	long	7150-T77511	1.0	21.890	-9.32	1.000	0.000		0.10	0.10
3	437	Al. alloy extrusion	long tran	7150-T77511	1.0	17.980	-7.57	1.000	0.000		0.10	0.10
3	438	Al. alloy extrusion	long and long tran	7150-T77511	3.0	5.710	-1.31	1.000	16.920		0.10	0.10
3	447	Al. alloy extrusion	long	7175-T73511	5.0	7.630	-2.78	0.560	7.300		-1.00	0.10
3	468	Al. alloy sheet	long and long tran	7475-T61	1.0	22.700	-10.1	1.000	0.000		0.00	0.00
3	468	Al. alloy sheet	long and long tran	7475-T761	1.0	22.700	-10.1	1.000	0.000		0.00	0.00
3	469	Al. alloy sheet	long and long tran	7475-T61	3.0	13.400	-6.29	1.000	0.000		0.00	0.00
3	469	Al. alloy sheet	long and long tran	7475-T761	3.0	13.400	-6.29	1.000	0.000		0.00	0.00
4	24	Mg. alloy extr bar	long	ZK60A-T5	1.0	7.560	-2.73	0.400	23.700		-1.00	0.60
4	25	Mg. alloy extr bar	long	ZK60A-T5	2.4	5.510	-1.36	0.420	13.200		-1.00	0.90
4	26	Mg. alloy extr bar	long	ZK60A-T5	3.4	9.270	-4.13	0.460	5.630		-1.00	0.90
5	115	Ti. alloy annealed sheet		Ti-4.5Al-3V-2Fe-2Mo	1.0	7.720	-2.59	0.130	114.680		0.05	0.50
5	116	Ti. alloy annealed sheet		Ti-4.5Al-3V-2Fe-2Mo	2.8	7.220	-1.96	0.650	44.050		0.05	0.50
6	67	Inc. sheet at room temp	long tran	inconel 718	1.0	8.630	-2.07	0.580	58.480		-1.00	0.50
6	68	Inc. sheet at room temp	long tran	inconel 718	3.0	8.170	-2.23	0.680	30.580		-1.00	0.50
6	69	Inconel sheet at 1000F	long tran	inconel 718	1.0	23.510	-10.57	0.620	50.000		-1.00	0.60
6	70	Inconel sheet at 1000F	long tran	inconel 718	3.0	11.020	-3.93	0.910	20.000		-1.00	0.82
6	71	Inconel sheet at 1400F	long tran	inconel 718	3.0	10.290	-4.02	0.620	20.000		-1.00	0.60

20 | APPENDIX 5: TYPICAL STRESS-LIFE (SN) MATERIAL DATA TABLE

ID	UTS	E	SRI1	B1	NC1	B2	NFC
Steel Weld BS5400 ClassB	500	2.07E5	6955	-0.25	1E7	-0.1667	1E30
Steel Weld BS5400 ClassC	500	2.07E5	1.0222E4	-0.2857	1E7	-0.1818	1E30
Steel Weld BS5400 ClassD	500	2.07E5	1.5861E4	-0.3333	1E7	-0.2	1E30
Steel Weld BS5400 ClassE	500	2.07E5	1.4873E4	-0.3333	1E7	-0.2	1E30
Steel Weld BS5400 ClassF	500	2.07E5	1.2005E4	-0.3333	1E7	-0.2	1E30
Steel Weld BS5400 ClassF2	500	2.07E5	1.0714E4	-0.3333	1E7	-0.2	1E30
Steel Weld BS5400 ClassG	500	2.07E5	8291	-0.3333	1E7	-0.2	1E30
Steel Weld BS5400 ClassW	500	2.07E5	7179	-0.3333	1E7	-0.2	1E30
Generic Al_Alloy_UTS200	200	7.3E4	473.6	-0.07606	5E8	0	1E30
Generic Al_Alloy_UTS400	400	7.3E4	947	-0.07606	5E8	0	1E30
Generic Al_Alloy_UTS600	600	7.3E4	1420.6	-0.07606	5E8	0	1E30
Steel generic Material For UTS of 200	200	2.1E5	907.6	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 400	400	2.1E5	1815.2	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 600	600	2.1E5	2722	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 800	800	2.1E5	3630	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 1000	1000	2.1E5	4538	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 1200	1200	2.1E5	5446	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 1400	1400	2.1E5	6352	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 1600	1600	2.1E5	7260	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 1800	1800	2.1E5	8168	-0.1339	1E6	0	1E30
Steel generic Material For UTS of 2000	2000	2.1E5	9076	-0.1339	1E6	0	1E30
Ti_Alloy generic Material For UTS of 600	600	1.1E5	2502	-0.1387	1E6	0	1E30
Ti_Alloy generic Material For UTS of 800	800	1.1E5	3336	-0.1387	1E6	0	1E30
Ti_Alloy generic Material For UTS of 1000	1000	1.1E5	4170	-0.1387	1E6	0	1E30
Ti_Alloy generic Material For UTS of 1200	1200	1.1E5	5003	-0.1387	1E6	0	1E30
Ti_Alloy generic Material For UTS of 1400	1400	1.1E5	5837	-0.1387	1E6	0	1E30

SN Curve Defaults for the **CODE** entry in the vFTGMAT: STATIC flag or when Spotweld or Seamweld material data is not entered.

Property	Generic Nugget Spot Weld		Generic Top/Bot Sheet Spot Weld		Generic Flexible Seam Weld (r=1)		Generic Stiff Seam Weld (r=0)	
CODE	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199	steel <100	alum 100-199
UTS (MPa)	4000	4000	4000	4000	1.8E4	4000	1.8E4	4000
YS (MPa)	355	150	355	150	Blank	Blank	Blank	Blank
E (MPa)	2.1E5	7E4	2.1E5	7E4	2.1E5	7E4	2.1E5	7E4
SRI1	2100	2462	2900	2462	3.6E4	3140	1.8E4	1275
b1	-0.1667	-0.2	-0.1667	-0.2	-0.3333	-0.1734	-0.3333	-0.1625
Nc1	1E6	1E7	1E6	1E7	1E7	1E7	1E7	1E7
b2	-0.09091	-0.1111	-0.09091	-0.1111	-0.3333	-0.1734	-0.3333	-0.1625
SE	0.334	0.33	0.33	0.33	0.4	0.4	0.4	0.4
RR	0.0	0.0	0.0	0.0	-1.0	-1.0	-1.0	-1.0
Nfc	1E30	1E30	1E30	1E30	1E30	1E30	1E30	1E30
MSS	0.1	0.2	0.1	0.2	Blank	Blank	Blank	Blank
M1	Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank
M2	-0.1	-0.2	-0.1	-0.2	-0.25	-0.25	-0.25	-0.25
M3	-0.1	-0.2	-0.1	-0.2	-0.1	Blank	-0.1	Blank
M4	Blank	Blank	Blank	Blank	-0.1	Blank	-0.1	Blank
BTHRESH	Blank	Blank	Blank	Blank	Blank	Blank	Blank	Blank
RTHICK	Blank	Blank	Blank	Blank	1 (mm)	1 (mm)	1 (mm)	1 (mm)
nTHICK	Blank	Blank	Blank	Blank	0.16667	0.16667	0.16667	0.16667

21 | APPENDIX 6: TYPICAL STRAIN-LIFE (EN) MATERIAL DATA TABLE

Name	E	UTS	YS	K'	n'	Sf'	b	Ef'	c
5454_NONE_CF	69000	334	294	373.0	0.047	554.00	-0.089	0.310	-0.620
5083_87_CF	69000	385	285	417.0	0.035	650.00	-0.094	2.260	-1.010
6061-T6_NONE_CF	69000	389	369	422.0	0.030	689.00	-0.094	0.350	-0.670
5052-H32	69600	231	167	297.0	0.047	365.00	-0.066	0.300	-0.610
6061-T6_NONE_SHEET	69600	314	289	426.0	0.062	535.00	-0.082	1.340	-0.830
7175-T73_NONE_HF	71300	524	447	529.0	0.033	765.00	-0.082	6.180	-1.140
6061-T6_80_HF	72700	340	313	416.0	0.042	645.00	-0.097	0.220	-0.600
2014-T6_125_HF	72700	483	437	605.0	0.049	976.00	-0.120	0.880	-0.880
2TA11	117000	1233	1185	1822.0	0.113	2070.00	-0.106	0.946	-0.710
StructuralSteelBS4360Grade-50D	191400	480	355.00	912.0	0.163	1036.00	-0.123	0.622	-0.618
Cold_rolled_sheet	200000	303	221	1103.0	0.260	1407.00	-0.185	1.930	-0.686
AISI1012	200000	333	233	738.0	0.190	2551.00	-0.218	0.760	-0.591
HotRolledHSLASteelFeE37D	200000	388	242.00	890.0	0.210	635.00	-0.093	0.271	-0.489
unsg10200	200000	393	255	1393.0	0.290	820.00	-0.121	0.207	-0.447
B40PK	200000	394	304	779.0	0.160	1310.00	-0.137	0.768	-0.597
AISI1020	200000	416	280	1482.0	0.290	834.00	-0.114	0.169	-0.453
B55XF	200000	488	398	627.0	0.090	1082.00	-0.100	5.737	-0.818
HotRolledHSLASteelFeE420TM	200000	490	410.00	785.0	0.110	930.00	-0.092	0.775	-0.650
B60RO	200000	503	469	510.0	0.070	896.00	-0.084	1.600	-0.675
sra_60	200000	531	482	1434.0	0.210	1517.00	-0.140	1.380	-0.684
HotRolledHSLASteelFeE52D	200000	550	425.00	940.0	0.140	860.00	-0.081	0.662	-0.628
sra_70	200000	570	529	1641.0	0.210	1545.00	-0.134	0.834	-0.644
CarbonSteelRQT501	200000	590	472.00	1102.3	0.165	892.00	-0.089	0.202	-0.504
B80RK	200000	610	562	724.0	0.090	1875.00	-0.130	0.950	-0.651
B80XF	200000	645	569	869.0	0.080	1530.00	-0.110	3.139	-0.852
Nitro-sa	200000	648	641	862.0	0.120	979.00	-0.098	3.130	-0.814
DualPhaseSteelDP1	200000	659	397.00	1087.0	0.150	1325.00	-0.120	0.266	-0.575
DualPhaseSteelDP2	200000	753	400.00	1242.0	0.150	2030.00	-0.154	2.570	-0.775
CarbonSteelRQT701	200000	825	735.00	1048.6	0.105	955.10	-0.063	2.084	-0.790
EngrSteelBS970-817M40X-Condn	200000	1277	1166.00	1166.0	0.025	3956.00	-0.178	1.265	-0.811
MANTEN	203000	552	324	1103.0	0.190	917.00	-0.095	0.260	-0.470
RQC100	203000	863	827	1151.0	0.100	1158.00	-0.075	1.060	-0.750
CarbonSteelSAE1045_shaft	204000	621	380.00	1258.0	0.208	948.00	-0.092	0.260	-0.445
INC718	204000	1304	1110	1994.0	0.112	2355.00	-0.103	6.480	-0.973
CarbonSteelSAE1006_85_HR	207000	318	248.00	1028.0	0.240	629.00	-0.090	0.150	-0.400
Ni-Cr-MoSteelSAE8630_254_NORM	207000	785	709.00	961.0	0.080	1049.00	-0.110	0.200	-0.860
C-MnSteelSAE1561_234_HR	207000	836	447.00	1448.0	0.190	1278.00	-0.110	0.530	-0.540
CarbonSteelSAE1080_371_QT	207000	1298	1118.00	1820.0	0.130	1787.00	-0.090	1.000	-0.690
Cr-MoSteelSAE4142_670_QT	207000	2446	1619.00	3484.0	0.070	2727.00	-0.080	0.060	-1.200
alphafe	210000	206	190	694.0	0.208	683.00	-0.127	0.799	-0.597
Beryllium	289000	323	261	1168.0	0.207	528.00	-0.099	0.023	-0.490

22 | APPENDIX 7: CAEfatigue – Open Source Software

Open Source Software used in CAEfatigue

See below for the relevant third party software, additional terms and conditions and /or notices applicable to such third party software, and, in the case of some open source third party software, where the software may be downloaded.

VTK - 8.0

Copyright (c) 1993-2008 Ken Martin, Will Schroeder, Bill Lorensen
All rights reserved.
Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met:
Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer.
Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution.
Neither name of Ken Martin, Will Schroeder, or Bill Lorensen nor the names of any contributors may be used to endorse or promote products derived from this software without specific prior written permission.
THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS “AS IS” AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE AUTHORS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Mahapps.metro – 2.4.9

MIT License

Copyright (c) .NET Foundation and Contributors. All rights reserved.

Permission is hereby granted, free of charge, to any person obtaining a copy
of this software and associated documentation files (the "Software"), to deal
in the Software without restriction, including without limitation the rights
to use, copy, modify, merge, publish, distribute, sublicense, and/or sell
copies of the Software, and to permit persons to whom the Software is
furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all
copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR
IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY,
FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE
AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER
LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM,
OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE
SOFTWARE.

ControlzEx – 5.0.1

The MIT License (MIT)

Copyright (c) 2015-2019 Jan Karger, Bastian Schmidt

Permission is hereby granted, free of charge, to any person obtaining a copy
of this software and associated documentation files (the "Software"), to deal
in the Software without restriction, including without limitation the rights
to use, copy, modify, merge, publish, distribute, sublicense, and/or sell
copies of the Software, and to permit persons to whom the Software is
furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all

copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Newtonsoft.Json – 13.0.1

The MIT License (MIT)
Copyright (c) 2007 James Newton-King
Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:
The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.
THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

jacobslusser.ScintillaNET.signed – 3.6.3

The MIT License (MIT)

Copyright (c) 2017, Jacob Slusser, https://github.com/jacobslusser

Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions:

The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Boost – 1.76

Boost Software License - Version 1.0 - August 17th, 2003

Permission is hereby granted, free of charge, to any person or organization obtaining a copy of the software and accompanying documentation covered by this license (the "Software") to use, reproduce, display, distribute, execute, and transmit the Software, and to prepare derivative works of the Software, and to permit third-parties to whom the Software is furnished to do so, all subject to the following:

The copyright notices in the Software and this entire statement, including the above license grant, this restriction and the following disclaimer, must be included in all copies of the Software, in whole or in part, and all derivative works of the Software, unless such copies or derivative works are solely in the form of machine-executable object code generated by

```
a source language processor.

THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR
IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY,
FITNESS FOR A PARTICULAR PURPOSE, TITLE AND NON-INFRINGEMENT. IN NO EVENT
SHALL THE COPYRIGHT HOLDERS OR ANYONE DISTRIBUTING THE SOFTWARE BE LIABLE
FOR ANY DAMAGES OR OTHER LIABILITY, WHETHER IN CONTRACT, TORT OR OTHERWISE,
ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER
DEALINGS IN THE SOFTWARE.
```

gRPC – 1.42.0

```
Copyright 2014 gRPC authors.

Licensed under the Apache License, Version 2.0 (the "License");
you may not use this file except in compliance with the License.
You may obtain a copy of the License at

    http://www.apache.org/licenses/LICENSE-2.0

Unless required by applicable law or agreed to in writing, software
distributed under the License is distributed on an "AS IS" BASIS,
WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied.
See the License for the specific language governing permissions and
limitations under the License.
```

HDF5 – 1.10.7

```
Copyright Notice and License Terms for
HDF5 (Hierarchical Data Format 5) Software Library and Utilities
-----------------------------------------------------------------------------

HDF5 (Hierarchical Data Format 5) Software Library and Utilities
Copyright (c) 2006-2018, The HDF Group.

NCSA HDF5 (Hierarchical Data Format 5) Software Library and Utilities
Copyright (c) 1998-2006, The Board of Trustees of the University of Illinois.

All rights reserved.

Redistribution and use in source and binary forms, with or without
modification, are permitted for any purpose (including commercial purposes)
provided that the following conditions are met:

1. Redistributions of source code must retain the above copyright notice,
   this list of conditions, and the following disclaimer.

2. Redistributions in binary form must reproduce the above copyright notice,
   this list of conditions, and the following disclaimer in the documentation
   and/or materials provided with the distribution.

3. Neither the name of The HDF Group, the name of the University, nor the
   name of any Contributor may be used to endorse or promote products derived
   from this software without specific prior written permission from
   The HDF Group, the University, or the Contributor, respectively.

DISCLAIMER:
THIS SOFTWARE IS PROVIDED BY THE HDF GROUP AND THE CONTRIBUTORS
"AS IS" WITH NO WARRANTY OF ANY KIND, EITHER EXPRESSED OR IMPLIED. IN NO EVENT SHALL THE HDF GROUP OR THE CONTRIBUTORS
BE LIABLE FOR ANY DAMAGES SUFFERED BY THE USERS ARISING OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE
POSSIBILITY OF SUCH DAMAGE.

You are under no obligation whatsoever to provide any bug fixes, patches, or upgrades to the features, functionality
or performance of the source code ("Enhancements") to anyone; however, if you choose to make your Enhancements
available either publicly, or directly to The HDF Group, without imposing a separate written license agreement for
```

such Enhancements, then you hereby grant the following license: a non-exclusive, royalty-free perpetual license to install, use, modify, prepare derivative works, incorporate into other computer software, distribute, and sublicense such enhancements or derivative works thereof, in binary and source code form.

Limited portions of HDF5 were developed by Lawrence Berkeley National
Laboratory (LBNL). LBNL's Copyright Notice and Licensing Terms can be
found here: COPYING_LBNL_HDF5 file in this directory or at
http://support.hdfgroup.org/ftp/HDF5/releases/COPYING_LBNL_HDF5.

Contributors: National Center for Supercomputing Applications (NCSA) at
the University of Illinois, Fortner Software, Unidata Program Center (netCDF), The Independent JPEG Group (JPEG),
Jean-loup Gailly and Mark Adler (gzip), and Digital Equipment Corporation (DEC).

Portions of HDF5 were developed with support from the Lawrence Berkeley
National Laboratory (LBNL) and the United States Department of Energy
under Prime Contract No. DE-AC02-05CH11231.

Portions of HDF5 were developed with support from the University of
California, Lawrence Livermore National Laboratory (UC LLNL).
The following statement applies to those portions of the product and must
be retained in any redistribution of source code, binaries, documentation,
and/or accompanying materials:

   This work was partially produced at the University of California,
   Lawrence Livermore National Laboratory (UC LLNL) under contract
   no. W-7405-ENG-48 (Contract 48) between the U.S. Department of Energy
   (DOE) and The Regents of the University of California (University)
   for the operation of UC LLNL.

   DISCLAIMER:
   This work was prepared as an account of work sponsored by an agency of
   the United States Government. Neither the United States Government nor
   the University of California nor any of their employees, makes any
   warranty, express or implied, or assumes any liability or responsibility
   for the accuracy, completeness, or usefulness of any information,
   apparatus, product, or process disclosed, or represents that its use
   would not infringe privately- owned rights. Reference herein to any
   specific commercial products, process, or service by trade name,
   trademark, manufacturer, or otherwise, does not necessarily constitute
   or imply its endorsement, recommendation, or favoring by the United
   States Government or the University of California. The views and
   opinions of authors expressed herein do not necessarily state or reflect
   those of the United States Government or the University of California,
   and shall not be used for advertising or product endorsement purposes.

HDF5 is available with the SZIP compression library but SZIP is not part
of HDF5 and has separate copyright and license terms. See SZIP Compression
in HDF Products (www.hdfgroup.org/doc_resource/SZIP/) for further details.

Zlib – 1.2.11

zlib.h -- interface of the 'zlib' general purpose compression library
  version 1.2.11, January 15th, 2017

```
Copyright (C) 1995-2017 Jean-loup Gailly and Mark Adler

This software is provided 'as-is', without any express or implied
warranty.  In no event will the authors be held liable for any damages
arising from the use of this software.

Permission is granted to anyone to use this software for any purpose,
including commercial applications, and to alter it and redistribute it
freely, subject to the following restrictions:

1. The origin of this software must not be misrepresented; you must not
   claim that you wrote the original software. If you use this software
   in a product, an acknowledgment in the product documentation would be
   appreciated but is not required.
2. Altered source versions must be plainly marked as such, and must not be
   misrepresented as being the original software.
3. This notice may not be removed or altered from any source distribution.

Jean-loup Gailly        Mark Adler
jloup@gzip.org          madler@alumni.caltech.edu
```

Microsoft Automatic Graph Layout,MSAGL – 1.1.11

```
Microsoft Automatic Graph Layout,MSAGL

Copyright (c) Microsoft Corporation

All rights reserved.

MIT License

Permission is hereby granted, free of charge, to any person obtaining
a copy of this software and associated documentation files (the
""Software""), to deal in the Software without restriction, including
without limitation the rights to use, copy, modify, merge, publish,
distribute, sublicense, and/or sell copies of the Software, and to
permit persons to whom the Software is furnished to do so, subject to
the following conditions:

The above copyright notice and this permission notice shall be
included in all copies or substantial portions of the Software.

THE SOFTWARE IS PROVIDED *AS IS*, WITHOUT WARRANTY OF ANY KIND,
EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF
MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND
NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE
LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION
OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION
WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.
```

Threading Building Blocks (TBB) – 2020.4

```
                                 Apache License
                           Version 2.0, January 2004
                        http://www.apache.org/licenses/

   TERMS AND CONDITIONS FOR USE, REPRODUCTION, AND DISTRIBUTION

   1. Definitions.

      "License" shall mean the terms and conditions for use, reproduction,
      and distribution as defined by Sections 1 through 9 of this document.

      "Licensor" shall mean the copyright owner or entity authorized by
      the copyright owner that is granting the License.
```

"Legal Entity" shall mean the union of the acting entity and all
other entities that control, are controlled by, or are under common
control with that entity. For the purposes of this definition,
"control" means (i) the power, direct or indirect, to cause the
direction or management of such entity, whether by contract or
otherwise, or (ii) ownership of fifty percent (50%) or more of the
outstanding shares, or (iii) beneficial ownership of such entity.

"You" (or "Your") shall mean an individual or Legal Entity
exercising permissions granted by this License.

"Source" form shall mean the preferred form for making modifications,
including but not limited to software source code, documentation
source, and configuration files.

"Object" form shall mean any form resulting from mechanical
transformation or translation of a Source form, including but
not limited to compiled object code, generated documentation,
and conversions to other media types.

"Work" shall mean the work of authorship, whether in Source or
Object form, made available under the License, as indicated by a
copyright notice that is included in or attached to the work
(an example is provided in the Appendix below).

"Derivative Works" shall mean any work, whether in Source or Object
form, that is based on (or derived from) the Work and for which the
editorial revisions, annotations, elaborations, or other modifications
represent, as a whole, an original work of authorship. For the purposes
of this License, Derivative Works shall not include works that remain
separable from, or merely link (or bind by name) to the interfaces of,
the Work and Derivative Works thereof.

"Contribution" shall mean any work of authorship, including
the original version of the Work and any modifications or additions
to that Work or Derivative Works thereof, that is intentionally
submitted to Licensor for inclusion in the Work by the copyright owner
or by an individual or Legal Entity authorized to submit on behalf of
the copyright owner. For the purposes of this definition, "submitted"
means any form of electronic, verbal, or written communication sent
to the Licensor or its representatives, including but not limited to
communication on electronic mailing lists, source code control systems,
and issue tracking systems that are managed by, or on behalf of, the
Licensor for the purpose of discussing and improving the Work, but
excluding communication that is conspicuously marked or otherwise
designated in writing by the copyright owner as "Not a Contribution."

"Contributor" shall mean Licensor and any individual or Legal Entity
on behalf of whom a Contribution has been received by Licensor and
subsequently incorporated within the Work.

2. Grant of Copyright License. Subject to the terms and conditions of
   this License, each Contributor hereby grants to You a perpetual,
   worldwide, non-exclusive, no-charge, royalty-free, irrevocable
   copyright license to reproduce, prepare Derivative Works of,
   publicly display, publicly perform, sublicense, and distribute the
   Work and such Derivative Works in Source or Object form.

3. Grant of Patent License. Subject to the terms and conditions of
   this License, each Contributor hereby grants to You a perpetual,
   worldwide, non-exclusive, no-charge, royalty-free, irrevocable
   (except as stated in this section) patent license to make, have made,
   use, offer to sell, sell, import, and otherwise transfer the Work,
   where such license applies only to those patent claims licensable
   by such Contributor that are necessarily infringed by their
   Contribution(s) alone or by combination of their Contribution(s)
   with the Work to which such Contribution(s) was submitted. If You

institute patent litigation against any entity (including a
cross-claim or counterclaim in a lawsuit) alleging that the Work
or a Contribution incorporated within the Work constitutes direct
or contributory patent infringement, then any patent licenses
granted to You under this License for that Work shall terminate
as of the date such litigation is filed.

4. Redistribution. You may reproduce and distribute copies of the
Work or Derivative Works thereof in any medium, with or without
modifications, and in Source or Object form, provided that You
meet the following conditions:

(a) You must give any other recipients of the Work or
Derivative Works a copy of this License; and

(b) You must cause any modified files to carry prominent notices
stating that You changed the files; and

(c) You must retain, in the Source form of any Derivative Works
that You distribute, all copyright, patent, trademark, and
attribution notices from the Source form of the Work,
excluding those notices that do not pertain to any part of
the Derivative Works; and

(d) If the Work includes a "NOTICE" text file as part of its
distribution, then any Derivative Works that You distribute must
include a readable copy of the attribution notices contained
within such NOTICE file, excluding those notices that do not
pertain to any part of the Derivative Works, in at least one
of the following places: within a NOTICE text file distributed
as part of the Derivative Works; within the Source form or
documentation, if provided along with the Derivative Works; or,
within a display generated by the Derivative Works, if and
wherever such third-party notices normally appear. The contents
of the NOTICE file are for informational purposes only and
do not modify the License. You may add Your own attribution
notices within Derivative Works that You distribute, alongside
or as an addendum to the NOTICE text from the Work, provided
that such additional attribution notices cannot be construed
as modifying the License.

You may add Your own copyright statement to Your modifications and
may provide additional or different license terms and conditions
for use, reproduction, or distribution of Your modifications, or
for any such Derivative Works as a whole, provided Your use,
reproduction, and distribution of the Work otherwise complies with
the conditions stated in this License.

5. Submission of Contributions. Unless You explicitly state otherwise,
any Contribution intentionally submitted for inclusion in the Work
by You to the Licensor shall be under the terms and conditions of
this License, without any additional terms or conditions.
Notwithstanding the above, nothing herein shall supersede or modify
the terms of any separate license agreement you may have executed
with Licensor regarding such Contributions.

6. Trademarks. This License does not grant permission to use the trade
names, trademarks, service marks, or product names of the Licensor,
except as required for reasonable and customary use in describing the
origin of the Work and reproducing the content of the NOTICE file.

7. Disclaimer of Warranty. Unless required by applicable law or
agreed to in writing, Licensor provides the Work (and each
Contributor provides its Contributions) on an "AS IS" BASIS,
WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or
implied, including, without limitation, any warranties or conditions
of TITLE, NON-INFRINGEMENT, MERCHANTABILITY, or FITNESS FOR A
PARTICULAR PURPOSE. You are solely responsible for determining the

```
   appropriateness of using or redistributing the Work and assume any
   risks associated with Your exercise of permissions under this License.

8. Limitation of Liability. In no event and under no legal theory,
   whether in tort (including negligence), contract, or otherwise,
   unless required by applicable law (such as deliberate and grossly
   negligent acts) or agreed to in writing, shall any Contributor be
   liable to You for damages, including any direct, indirect, special,
   incidental, or consequential damages of any character arising as a
   result of this License or out of the use or inability to use the
   Work (including but not limited to damages for loss of goodwill,
   work stoppage, computer failure or malfunction, or any and all
   other commercial damages or losses), even if such Contributor
   has been advised of the possibility of such damages.

9. Accepting Warranty or Additional Liability. While redistributing
   the Work or Derivative Works thereof, You may choose to offer,
   and charge a fee for, acceptance of support, warranty, indemnity,
   or other liability obligations and/or rights consistent with this
   License. However, in accepting such obligations, You may act only
   on Your own behalf and on Your sole responsibility, not on behalf
   of any other Contributor, and only if You agree to indemnify,
   defend, and hold each Contributor harmless for any liability
   incurred by, or claims asserted against, such Contributor by reason
   of your accepting any such warranty or additional liability.

END OF TERMS AND CONDITIONS

APPENDIX: How to apply the Apache License to your work.

   To apply the Apache License to your work, attach the following
   boilerplate notice, with the fields enclosed by brackets "[]"
   replaced with your own identifying information. (Don't include
   the brackets!)  The text should be enclosed in the appropriate
   comment syntax for the file format. We also recommend that a
   file or class name and description of purpose be included on the
   same "printed page" as the copyright notice for easier
   identification within third-party archives.

Copyright [yyyy] [name of copyright owner]

Licensed under the Apache License, Version 2.0 (the "License");
you may not use this file except in compliance with the License.
You may obtain a copy of the License at

    http://www.apache.org/licenses/LICENSE-2.0

Unless required by applicable law or agreed to in writing, software
distributed under the License is distributed on an "AS IS" BASIS,
WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied.
See the License for the specific language governing permissions and
limitations under the License.
```