fe-safe 2017

fe-safe/RUBBER TUTORIALS

Support

For support, contact your local SIMILUIA support office.

Legal Notices

The Endurica critical plane analysis algorithm is protected under US Patent No. 6,634,236 B1.

Endurica LLC

1219 West Main Cross St., Suite 201

Findlay, Ohio 45840

USA

www.endurica.com

Trademarks

Legal Notices

Third-Party Copyright Notices

Certain portions of *fe-safe* contain elements subject to copyright owned by the entities listed below.

© Battelle
© Endurica LLC
© Amec Foster Wheeler Nuclear UK Limited

fe-safe Licensed Programs may include open source software components. Source code for these components is available if required by the license.

The open source software components are grouped under the applicable licensing terms. Where required, links to common license terms are included below.

IP Asset Name	IP Asset Version	Copyright Notice
Under BSD 2-Clause		
UnZip (from Info-ZIP)	2.4	Copyright (c) 1990-2009 Info-ZIP. All rights reserved.
Under BSD 3-Clause		
Qt Solutions	2.6	Copyright (c) 2014 Digia Plc and/or its subsidiary(-ies) All rights reserved.

Tutorials

1 Tutorial 1: Tension on a Plate with Hole

1.1 Introduction

This tutorial outlines how to perform a standard rubber fatigue analysis using *fe-safe*/Rubber™. This tutorial is based on the Abaqus ODB format:

```
<DataDir>\rubber\tensionwithhole*.odb
```

1.1.1 Preparation

The tutorial uses an Abaqus *.odb model. However, the same techniques can be applied to all FE formats for which nominal strains and stresses can be reported.

This tutorial assumes that the user has experience using *fe-safe*, thus detailed information on how to set up an *fe-safe* analysis is not included in this tutorial. Please see the *fe-safe* User Manual including *fe-safe* Tutorials for details, for instance:

- Tutorial 106: *Using fe-safe with Abaqus .odb files*

Start *fe-safe*/Rubber™ as described in the *fe-safe* User Manual. The Configure *fe-safe* Project Directory window will be displayed:

Select an existing project, or create a new one from the welcome page

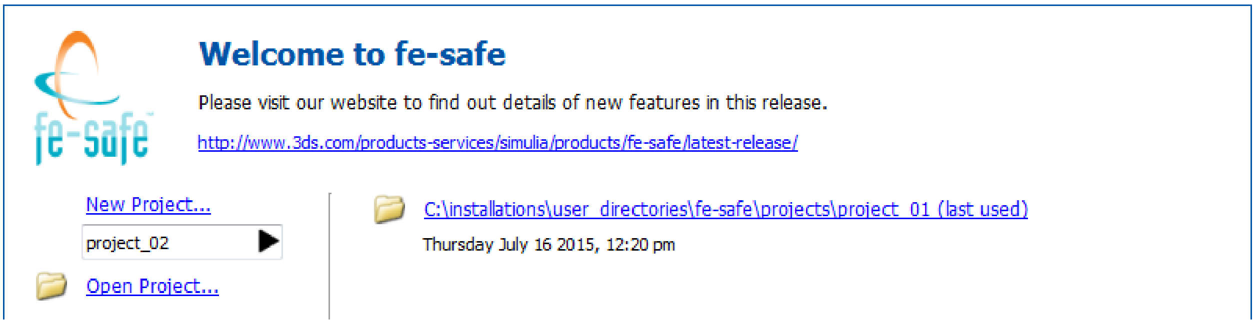


Figure 1-1

General FEA and analysis options

From the Menu bar select **Tools >> Clear Data and Settings...**, The Clear Data and Settings dialogue will appear as shown in

Figure 1-2 below, by selecting all of the Project settings and the Re-set file dialogues setting (all but the Re-set user settings checkbox). Click **OK**.

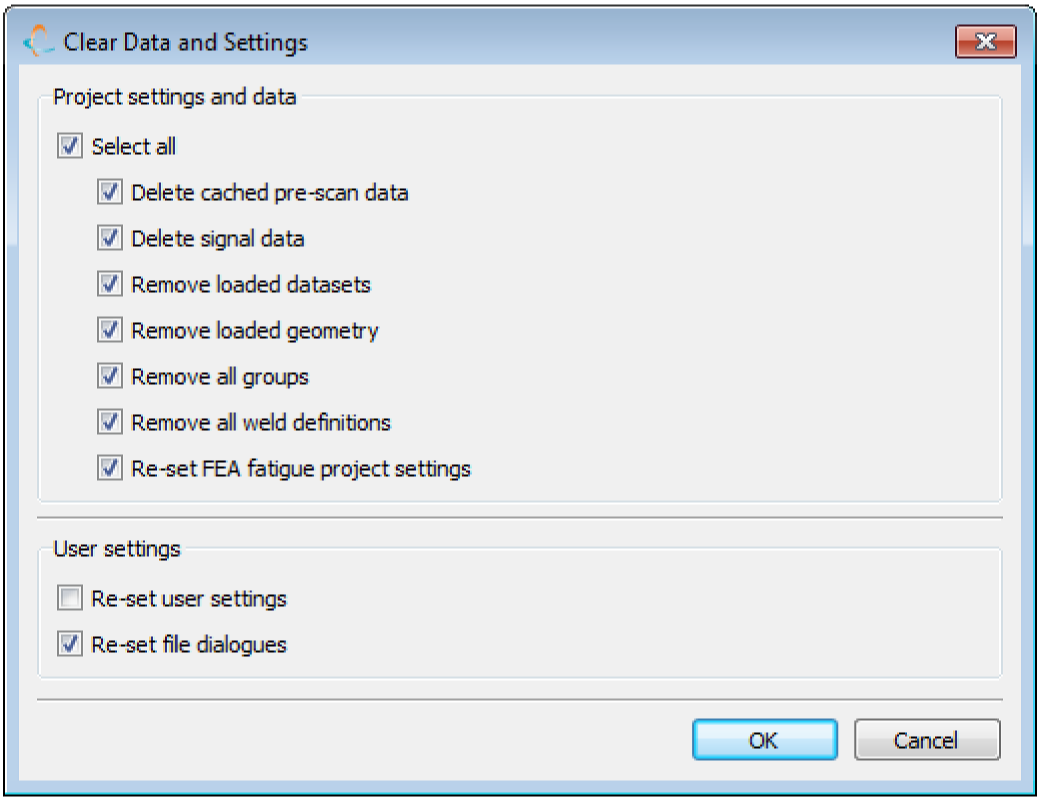


Figure 1-2 Clear Data and Settings

Configure *fe-safe*/Rubber™

This exercise must be completed with *fe-safe* using *fe-safe*/Rubber™ with the Endurica1.dll plug-in library. *fe-safe* will only be installed with the *fe-safe*/Rubber™ plug-in as well as the EnduricaMaterials database, if the *fe-safe*/Rubber option is selected in the installer configurations page. Please see the Installation and Licensing Guide for details.

Open the EnduricaMaterials_writable database

If the **`EnduricaMaterials`** database isn't shown in the Material Databases window, please select **File >> Materials >> Open Materials Database** to browse to the User Directory **`<UserDir>`** and open the *fe-safe*/Rubber database (**`EnduricaMaterials_writable.dbase`**).

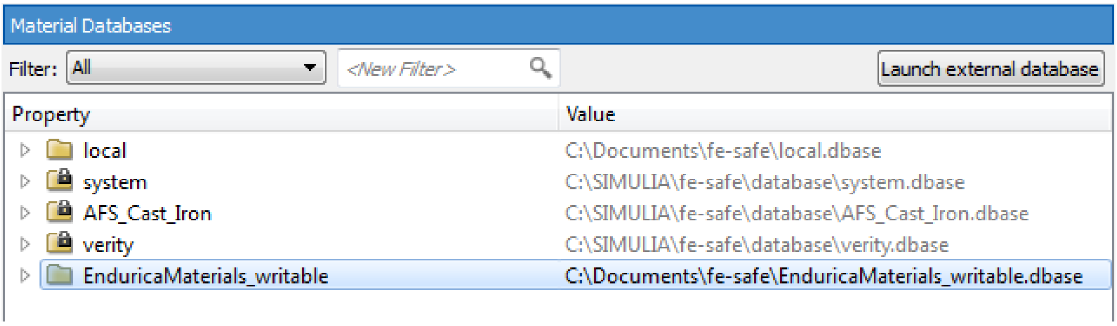


Figure 1-3 *fe-safe*/Rubber™ materials database

Load the *fe-safe*/Rubber™ plug-in

From the Menu bar select **Tools >> Load Plugin**..., and browse to the Installation directory **`<InstallDir>`** to find the Plugins subdirectory (**`/plugins`**). Select the *fe-safe*/Rubber™ Plug-in (**`Endurica1.dll`**) and click Open.

A Plug-in loaded dialogue will display. Click **OK.**

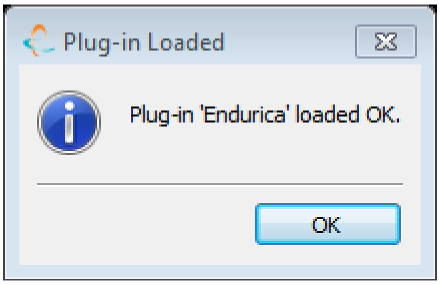


Figure 1-4

1.2 Opening the sample FE model

The model for this tutorial is a thin rubber plate with a hole in the middle, as shown in Figure 1-5.

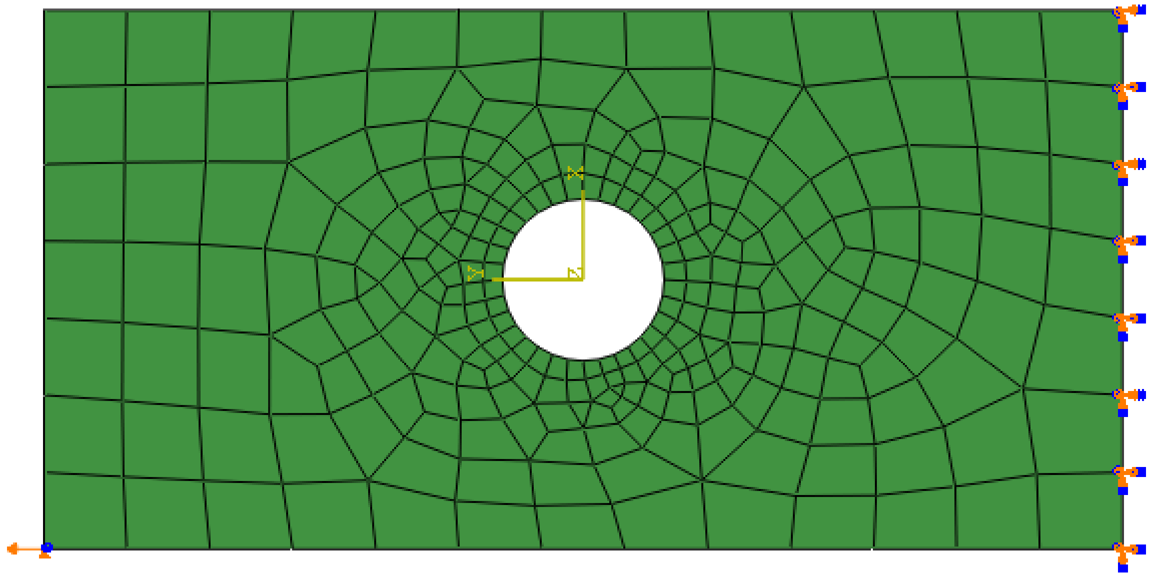


Figure 1-5 Example Finite Element Model

Encastre constraint is applied at the right, and a Y-direction displacement is applied for 1 second using 5 increments, at the node shown at the left. Displacement amplitude varies from 1 to 100 over 1 second (equally spaced time points) using the signal shown in Figure 1-6.

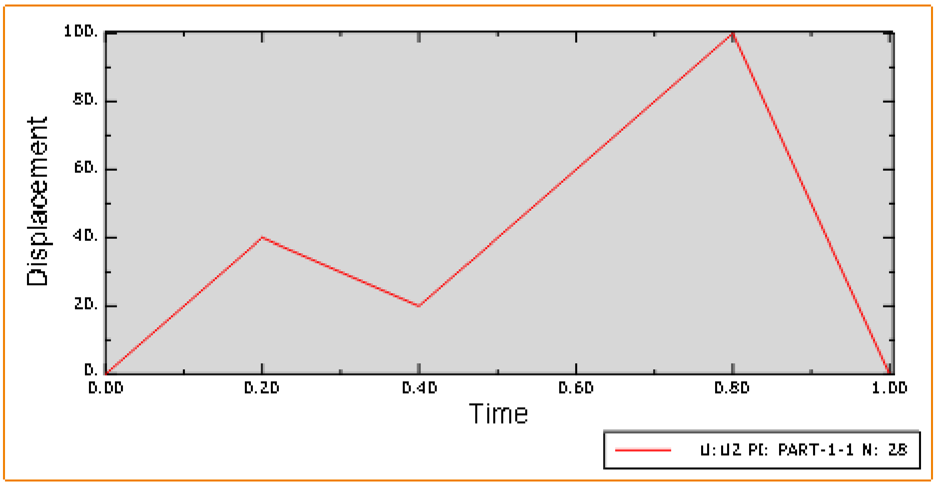


Figure 1-6 Amplitude Displacement in the Y-direction

To open the model, select the menu item **File >> FEA Solutions >> Open Finite Element Model...** and browse to the sample file **`tensionwithhole*.odb`** from the directory **`<DataDir>\rubber`**.

A prompt to pre-scan the file will be displayed. Click **Yes**.

The *Select Datasets to Read* dialogue will be displayed. Check the *Quick select* items for **Stresses**, **Strains (NE)** as shown and click **Apply to Dataset List** to apply the selections. Ensure that all of the checkboxes for the stress and strain symbols under each Increment are selected and select **OK** to load the datasets, as shown in Figure 1-7:

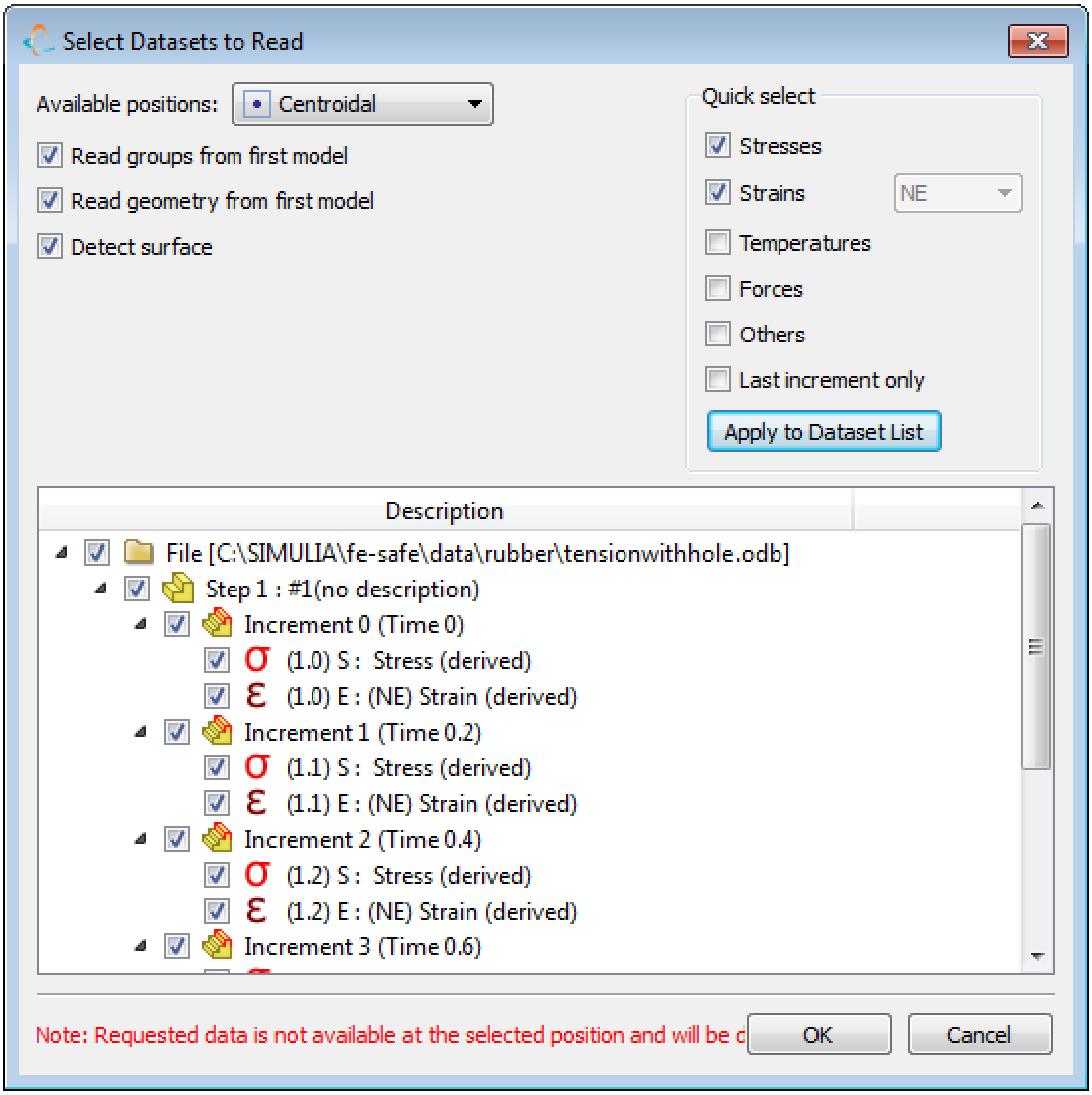


Figure 1-7 Select Datasets to Read

Note: Loading the model without pre-scanning may not load the required datasets.

As *fe-safe* loads the model, information about the file and the data it contains is written to the file:

`<ModelDir>``\reader.log`.

This information is also displayed in the **Message Log** window.

When the model has finished loading, the **Loaded FEA Models Properties** dialogue box appears, as shown in Figure 1-8.

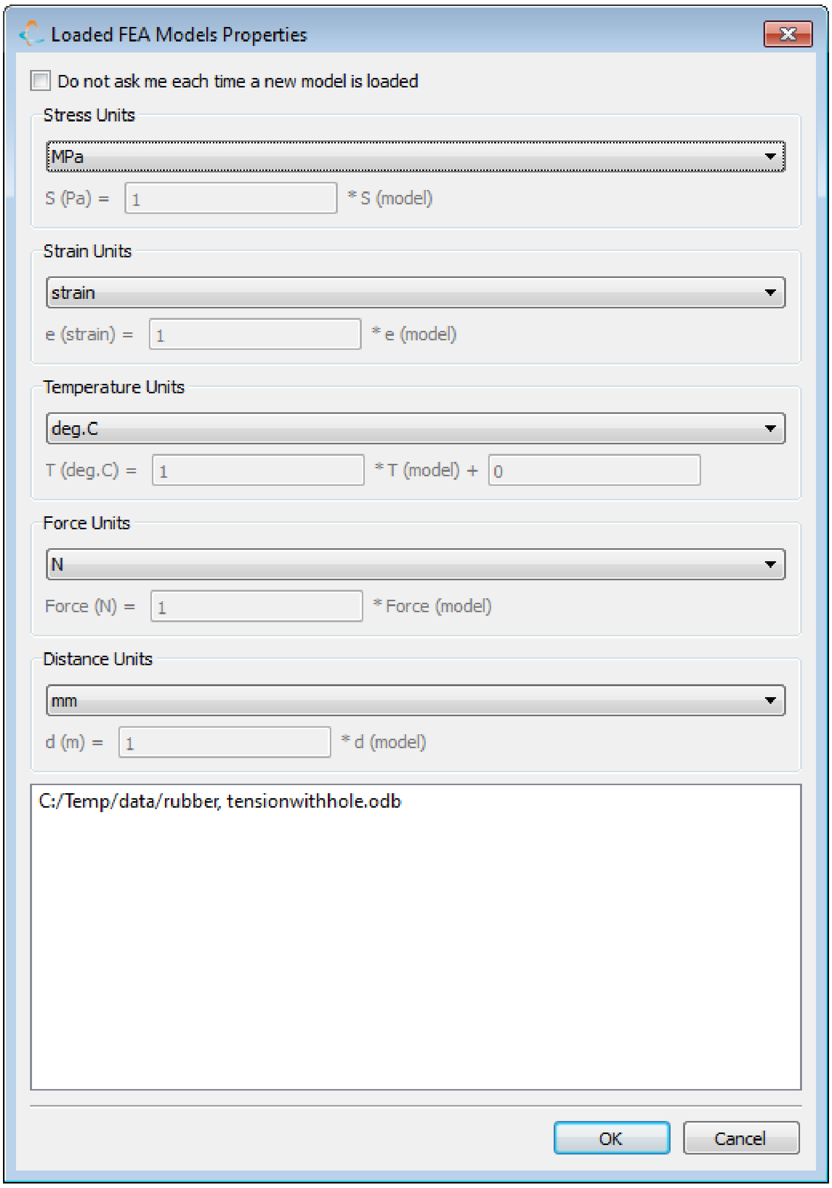


Figure 1-8 Loaded FEA Models Properties

If the dialogue box does not appear automatically, then it can be displayed by right-clicking on the SI? icon in the **Current FE Models** window and selecting Properties.

Ensure that the stress, strain and temperature units are MPa, strain and deg.C, respectively, as shown in Figure 1-8, then click **OK**.

A dialogue will show prompting to edit element groups loaded from the model, as shown in Figure 1-9, click **No**.

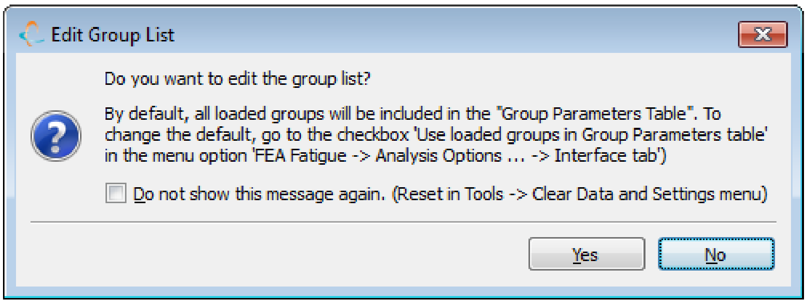


Figure 1-9 Edit Group List dialogue

You will see the following message regarding zeroes in datasets, this is normal

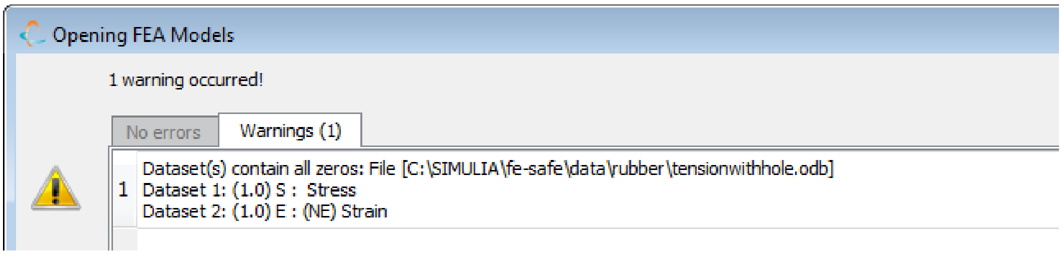


Figure 1-10

A summary of the open model appears in the **Current FE Models** window, showing the loaded datasets and element group information – see Figure 1-11.

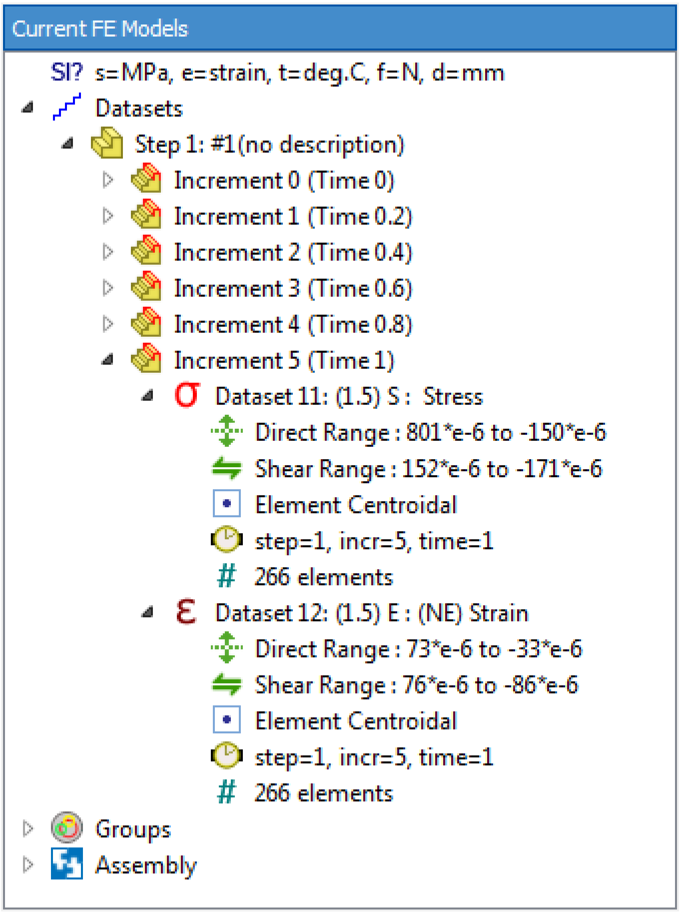


Figure 1-11 Current FE Models

Note: if the window does not appear as shown above, then expand the tree view to show more details.

The model contains six stress datasets, six strain (NE) datasets. *fe-safe* also extracts element group information from the ODB file.

When a new model is loaded, the output filename automatically defaults to:

<ResultsDir>\{source_file_name}Results.{source_file_extension}

which in this example is:

<ResultsDir>\tensionholewithResults.odb

If the correct number of datasets is not shown as above, use your right-mouse-button in the Current FE models window to select **Reload All Models** as shown in Figure 1-12.

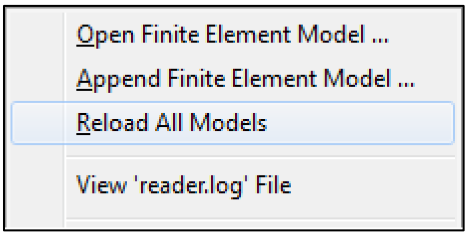


Figure 1-12 Use right-mouse-button to Reload All Models if needed

Click **Yes** as needed until the *Select Datasets to Read* dialogue is displayed as shown above.

Note: If the units of Stress, Strain and Temperature, do not appear as shown they can be changed by double-clicking on the SI? icon in the **Current FE Models** window and modifying **Properties.**

.

1.3 Exercise 1: *fe-safe*/Rubber™ analysis with dataset sequence (no time dependence)

This tutorial should be read in conjunction with the *fe-safe* User Guide and the *fe-safe*/Rubber™ User Guide.

Objective:

To perform a rubber fatigue analysis based on a sequence of surface based centroidal stress and strain (NE) solutions from FEA.

Each increment represents the hyperelastic stress and strain solutions from FEA. Each stress and strain dataset corresponds to a time increment (each increment is 0.2 seconds).

Analysis process:

For each elemental centroidal position:

- The nominal strain (NE) and stress (S) are read from the FE model database into *fe-safe*.
- The loading history is configured according to the sequence of stress and strain datasets and the length in seconds specified in Loading Settings
- The 6 components of the nominal strain tensor are calculated from the 3 in-plane nominal strain components, and from the plane stress condition (e.g. the out of plane stress is exactly 0).
- A series of material planes is generated based on the plug-in setting for damage sphere variables phi and theta. Subsequent calculations will be repeated on each material plane, in order to identify the critical plane.
- The local loading history is computed for each plane, giving the Cracking Energy Density as a function of time.
- A rainflow counting algorithm is then used to identify each individual cycle (e.g. peak and valley) contained within the entire local loading history.
- A numerical integration of the crack growth rate law is made to determine the number of repeats (the life) required to grow the initial flaw to its specified size at nucleation (see the *fe-safe*/Rubber™ Theory manual). As a part of the computation, the crack growth rate contributions of individual cycles are summed to obtain a total rate of crack growth per repeat of the entire loading history. The initial and final flaw sizes, and all crack growth properties were specified as a part of the material definition.
- Once the life has been computed for every material plane, then the minimum life is selected from among the results and reported as the life of the individual item.
- Output File containing Log Life and *fe-safe* Results Log file containing analysis configurations are automatically generated

Method:

Step 1: Define the loading:

The loading consists of a single fatigue loading block, cycling between the initial increment (Increment 0, stress dataset 1 and strain dataset 2 at time t=0.0 s) through the fifth increment (Increment 5, stress dataset 11 and strain dataset 12 at time t=1.0 s).

To define the loading:

- In the Current FE Models window, select any stress dataset:

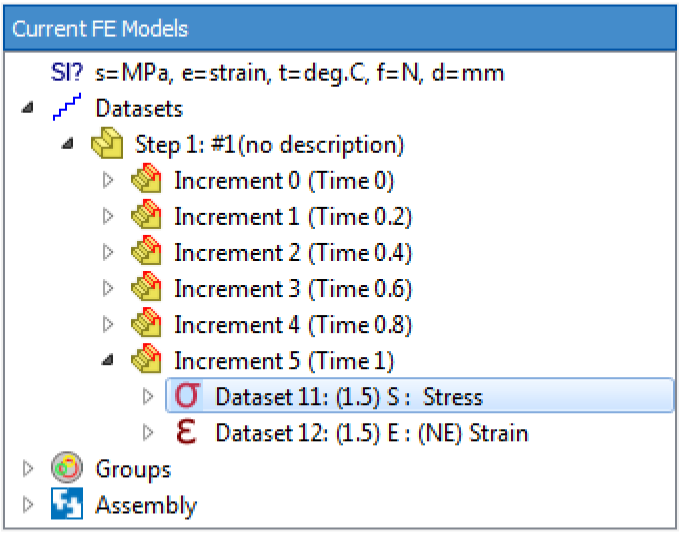


Figure 1-13 Select a Stress Dataset

- Select the **Loading Settings** tab from the *Fatigue from FEA* dialogue to switch to the loading tree;
- Use your right-mouse-button to select **Clear all loadings** and click **Yes** as shown:

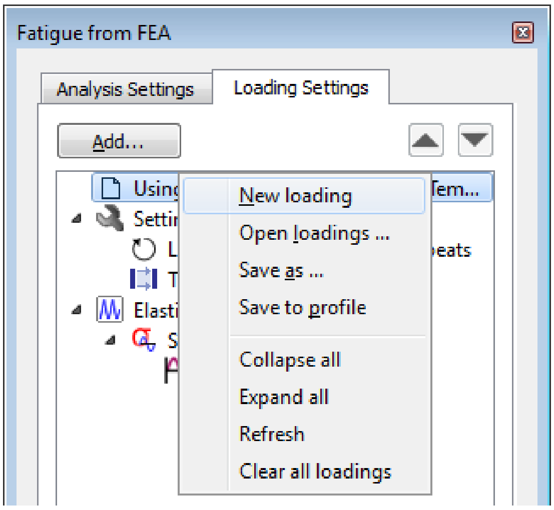


Figure 1-14 Clear all loadings

- Use the Add button in the Loading Settings window to select **Add... >> Dataset:**

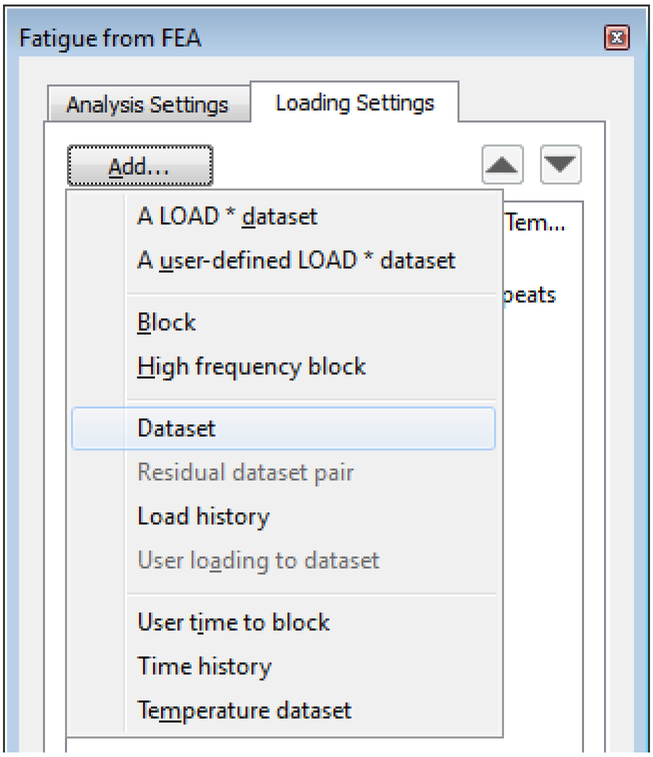


Figure 1-15 Add a Dataset

- Click **OK** when prompted
- Double-click on the Stress Dataset and using your keyboard, type the sequence '**1-11(2)**' without the ' symbols to represent every second dataset from 1 to 11 (a sequence of stress datasets):

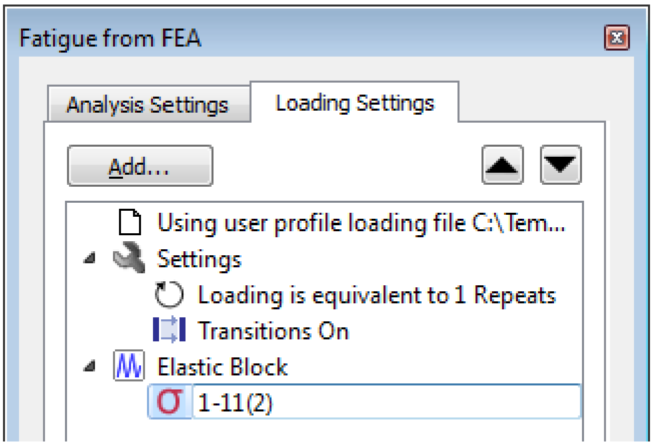


Figure 1-16 A sequence of stress datasets

This means 1 through 11 every second dataset enumerator, e.g. 1, 3, 5, 7, 9, then 11.

- Use the enter key to apply the changes made with the Keyboard

- In the current FE Models window, select any strain dataset:

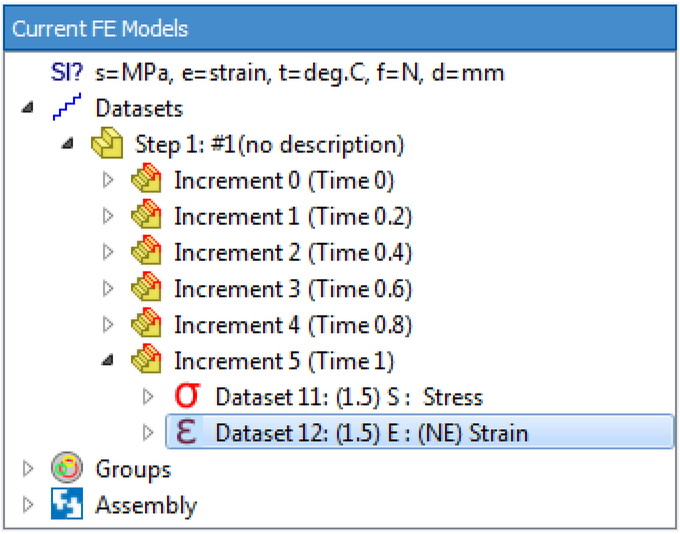


Figure 1-17 Select a Strain Dataset

- In the Loading Settings panel, right-click on the stress dataset and select **Add dataset**:

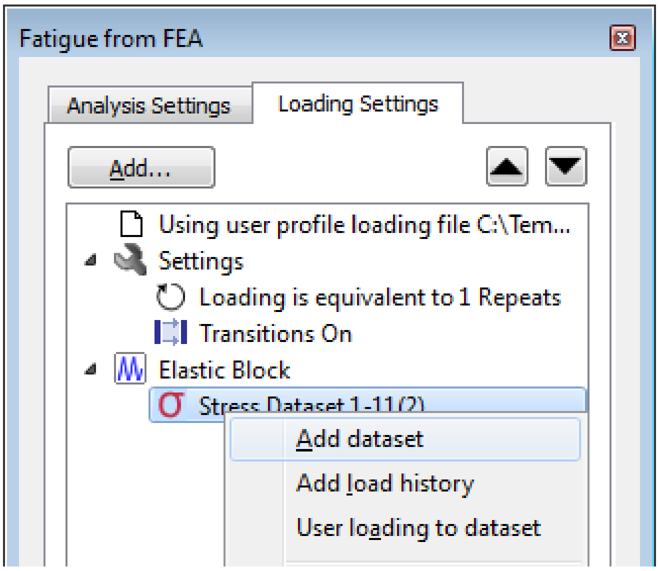


Figure 1-18 Add a Strain Dataset

- The strain dataset will appear as a branch in tree-view under the sequence of stress datasets.
- Double-click on the Strain Dataset and using your keyboard, type the sequence '**2-12(2)**' without the ' symbols to represent every second dataset from 2 to 12 (e.g. a sequence of strain datasets):

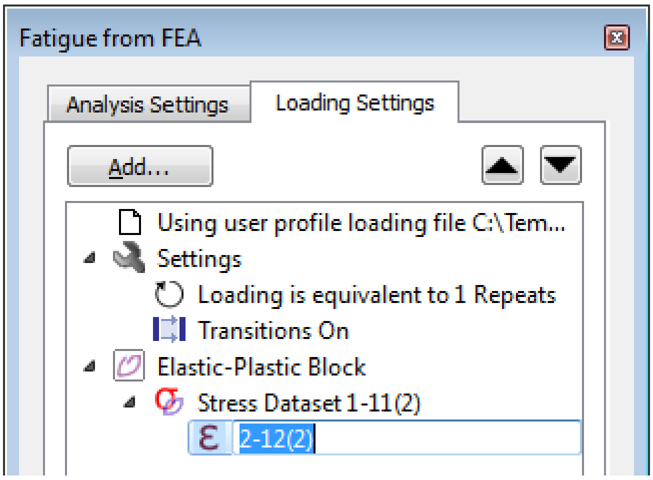


Figure 1-19 A sequence of Strain Datasets

- Use the enter key to apply the changes made with the Keyboard

Note: After adding a strain dataset to the **Elastic Block**, the block title will change to **Elastic-Plastic Block**, signifying that stress and strain are used to define the stress-history and strain-history for fatigue loading. This is only a naming convention, stresses and strains actually follow the stress-strain law the user implemented in FE.

- The defined loading appears in the Loading Settings panel, as shown below:

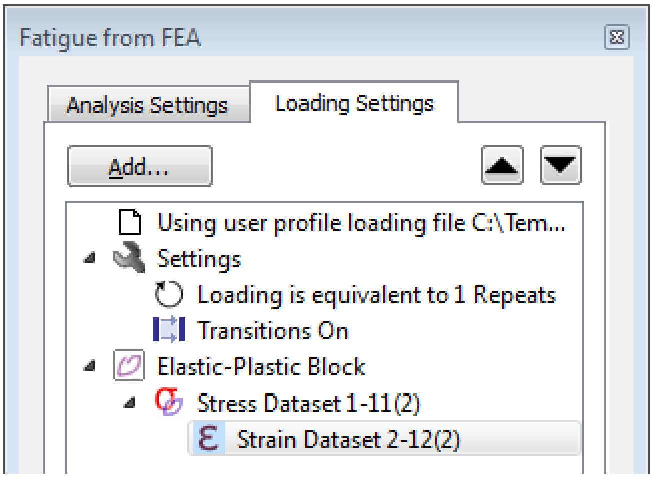


Figure 1-20 Finished sequence of stress and strain dataset pairs

Note: if the window does not appear as shown above, then expand the tree view to show more details. Time-dependent effects are accounted for in the example, so time definition in the loading is not required.

- Select the **Analysis Settings** tab from the Fatigue from FEA dialogue to switch to the Analysis Settings panel:

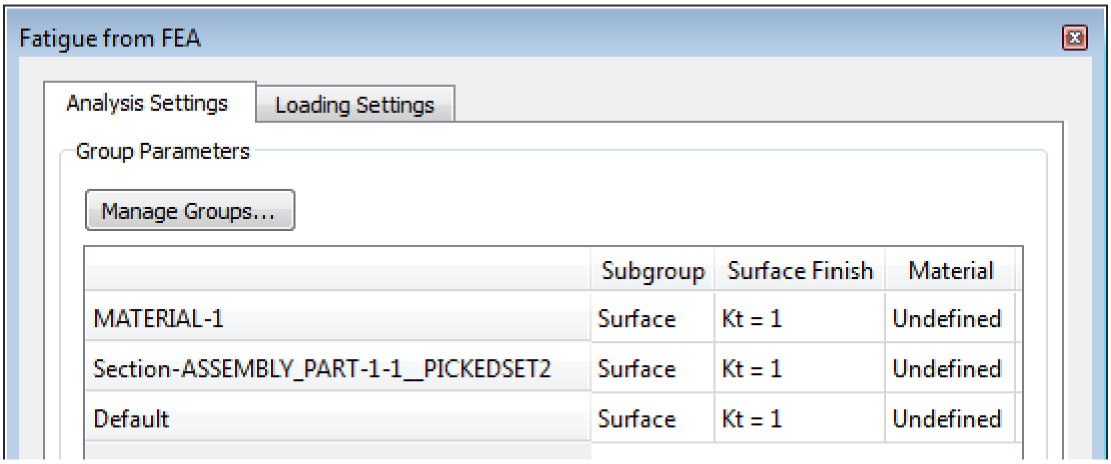


Figure 1-21 Return to Analysis Settings

Step 1: Define the subgroup option:

For all *fe-safe*/Rubber analysis the Whole group setting on the Subgroup Selection should be used at all times:

- double-click the **Subgroup** column header to open the **Subgroup Selection** dialogue for all groups.

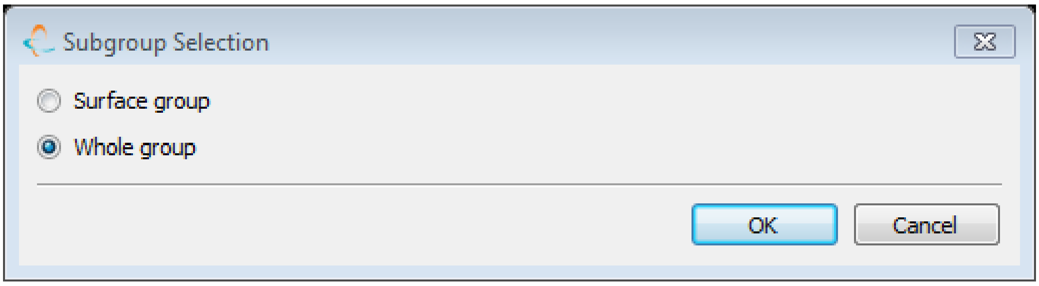


Figure 1-22 Subgroup definition

- select **Whole group**;
- click **OK**.

Step 2: Define the Surface Finish:

For all *fe-safe*/Rubber analysis the user-defined Kt value of 1 should be configured. This will ensure the effects of surface finish aren't applied to *fe-safe*/Rubber analyses.

- double-click the **Surface** column header to open the **Surface Finish Definition** dialogue for all groups:

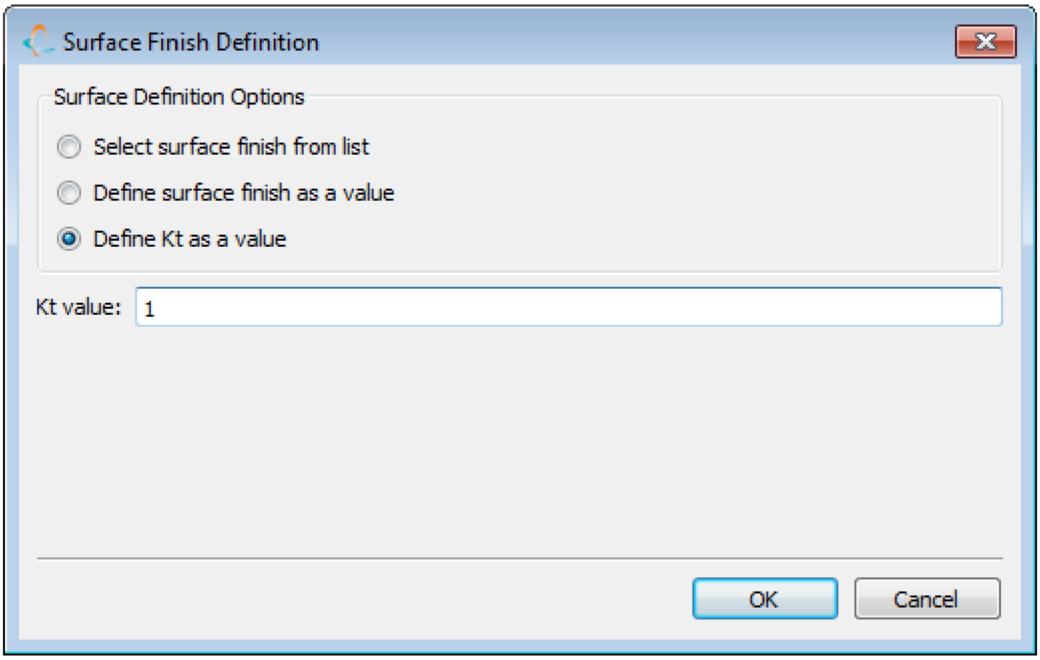


Figure 1-23 Surface Finish definition

- select **Define Kt as a value**;
- Enter **User-defined Kt value** of 1;
- click **OK**.

Step 3: Select the Material:

Since this Finite Element model has implemented a Hyperelastic law using the Reduced Polynomial model for stress-strain response, an equivalent material property must be used in *fe-safe*/Rubber as well. For this exercise, a copy of the material dataset NR_GUM will be created, and then modified to give it a hyperelastic law defined using the Reduced Polynomial model of order 2.

First create the new material:

- in the **Material Databases** window, use the left-mouse-button to select (highlight) the material **NR_GUM** from the list of available materials in the **`EnduricaMaterials_writable.dbase`** material database;
- select **Material >> Copy Material** to create a new material called **CopyOfNR_GUM**, which will appear at the bottom of the **`EnduricaMaterials_writable.dbase`** database;
- double-click the newly created material to rename it – change the name to **TUT1**.

Change the Hyperelastic model type:

- expand the **TUT1** material to show its properties;
- select the Hyperelastic model type selector: **hyperelastic : Type**;
- double-click on the value field to edit it;
- modify the value to: **REDUCEDPOLY**;

Now configure material parameters for the second order Reduced Polynomial model:

- select the C10 field: **hyperelastic : RP C10 (MPa)**;
- double-click on the value field to edit it
- modify the value to: **1.0**;
- Repeat to configure fields for C20 to **0.1**, D1 to **0.00067**, and D2 to **0.0**;
- All of the material parameters related to higher order Reduced Polynomial models are to be configured with value of **Unused**.

To define **TUT1** for the whole component:

- highlight the material **TUT1** in the **`EnduricaMaterials_writable.dbase`** database;
- double-click the **Material** header in the **Groups Parameter** box in the **Fatigue from FEA** dialogue - a **Change Material?** confirmation dialogue box appears;

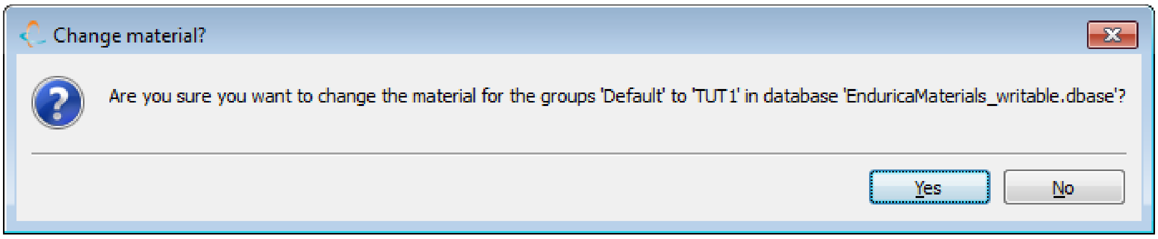


Figure 1-24 Change material dialogue

- click **YES**;
- the material name should appear for all groups in the **Material** column.

Step 4: Select the plug-in Algorithm:

This exercise uses a plug-in algorithm. To define the plug-in algorithm for the whole model (i.e. all element groups):

- double-click the **Algorithm** column header to open the in **Group Algorithm Selection** dialogue box for all groups;
- select the **Analyse using a plug-in algorithm** option;
- select the **Plug-in algorithm** drop-down box, and select **Surface** from the drop-down menu as shown in Figure 1-25;

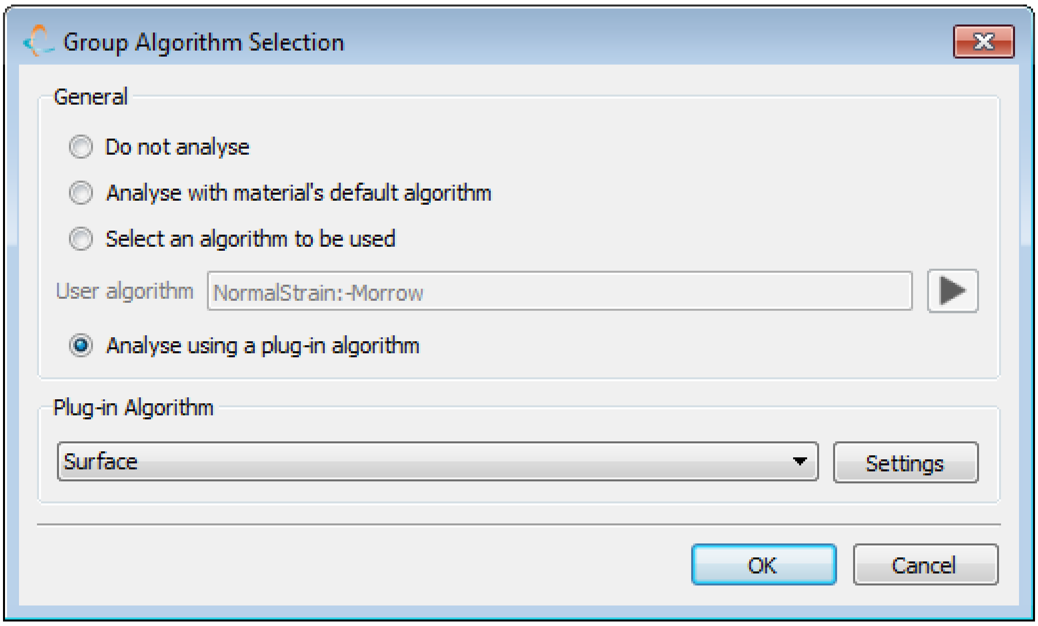


Figure 1-25 Group Algorithm Selection

Note: If ‘Analyse using a plug-in algorithm’ is not an option, go back to Load the *fe-safe*/RubberTM plug-in above to follow the steps to Load the plug-in.

Step 5: Define the output file:

When the FE model was loaded, the Output File was automatically defaulted to a standard file name in the Project Directory.

- in the **Fatigue from FEA** dialogue select the browse button to the right of the **Output File** field.
- Change the output filename to: **`tensionwithholeResults_ex01.odb`**

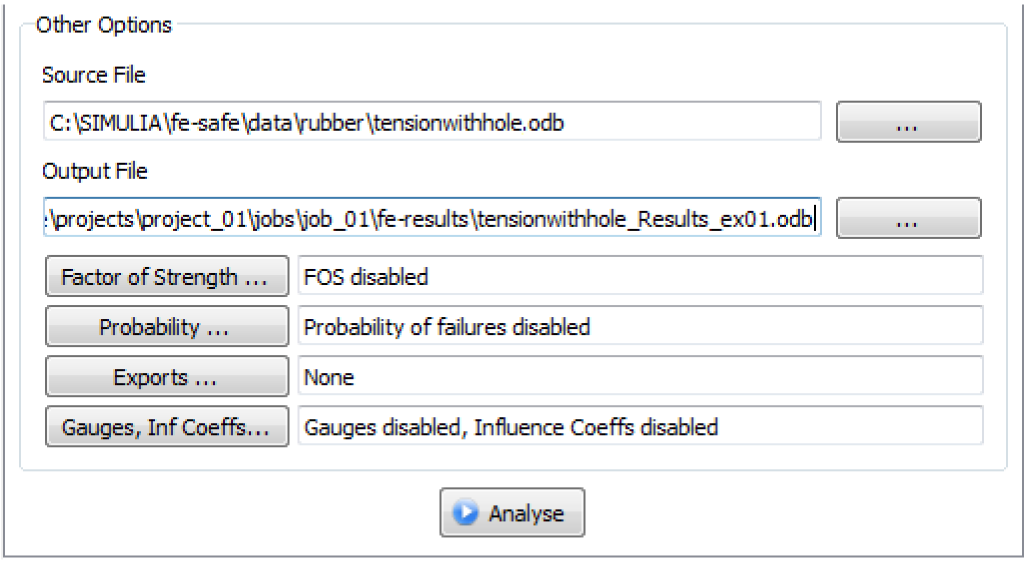
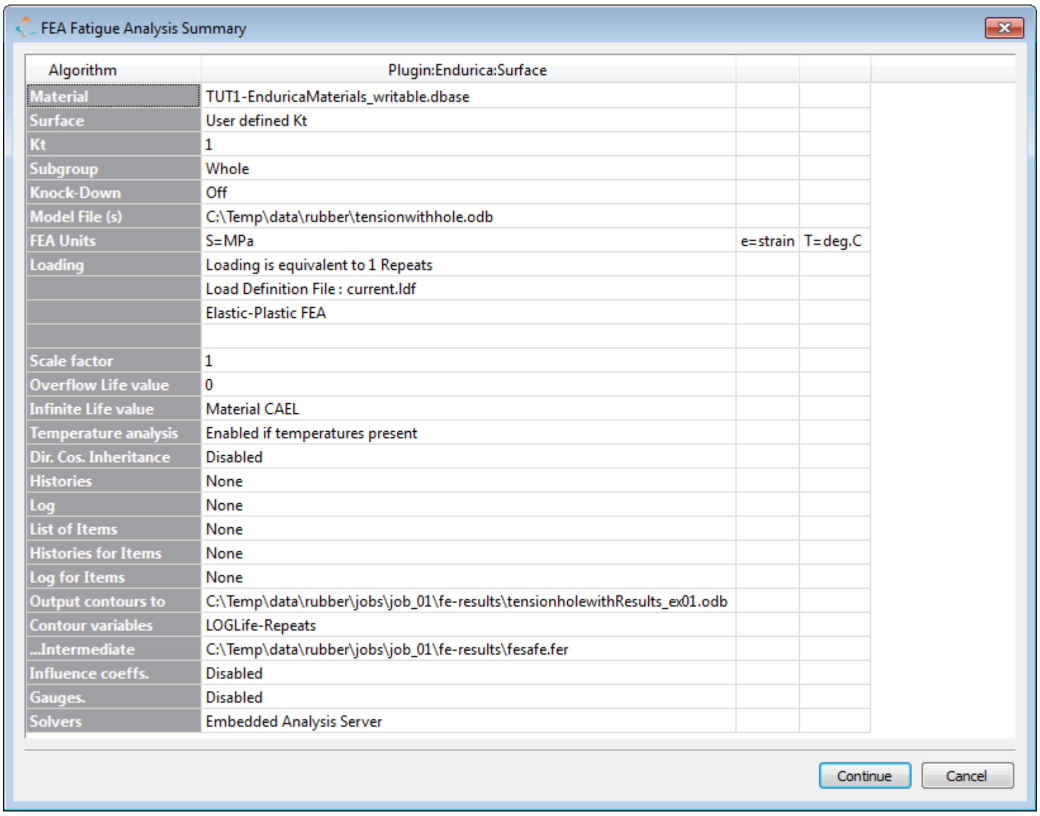


Figure 1-26 Define the Output File

Step 6: Run the analysis:

fe-safe is now configured to run the analysis.

Press the **Analyse** button. A summary of analysis parameters is displayed;



Algorithm	Plugin:Endurica:Surface		
Material	TUT1-EnduricaMaterials_writable.dbase		
Surface	User defined Kt		
Kt	1		
Subgroup	Whole		
Knock-Down	Off		
Model File (s)	C:\Temp\data\rubber\tensionwithhole.odb		
FEA Units	S=MPa	e=strain	T=deg.C
Loading	Loading is equivalent to 1 Repeats		
	Load Definition File : current.ldf		
	Elastic-Plastic FEA		
Scale factor	1		
Overflow Life value	0		
Infinite Life value	Material CAEL		
Temperature analysis	Enabled if temperatures present		
Dir. Cos. Inheritance	Disabled		
Histories	None		
Log	None		
List of Items	None		
Histories for Items	None		
Log for Items	None		
Output contours to	C:\Temp\data\rubber\jobs\job_01\fe-results\tensionholewithResults_ex01.odb		
Contour variables	LOGLife-Repeats		
...Intermediate	C:\Temp\data\rubber\jobs\job_01\fe-results\fesafe.fer		
Influence coeffs.	Disabled		
Gauges.	Disabled		
Solvers	Embedded Analysis Server		

Figure 1-27 FEA Fatigue Analysis Summary

Check that the analysis is configured as shown in Figure 1-27, and then click **Continue**.

As the analysis is being performed, the following information is written to the analysis log file.

```
 %      Time        Life-Repeats
100    0:00:02      4829@[0]61.1         266 of 266
```

The analysis log file has the same file name as the output file, except that the extension is **`.log`**, for instance:

`<ResultsDir>\tensionwithholeResults_ex01.log`

This information is also displayed in the **Message Log** window and includes

```
Summary
=======
          Worst Life-Repeats                     : 4828.717
  at Element [0]61.1
Analysis time                                    : 0:00:11.294
```

Step 7: Reviewing the results

The analysis log shows that the worst-case life for the whole model is:

4828 repeats of the loading at the centroid of element 61.

Note: In fe-safe, the calculated fatigue life always refers to the number of repeats of the complete defined fatigue loading cycle. An optional conversion factor can be used to convert the fatigue life in repeats to fatigue life with respect to some other quantity, for example hours or miles (see the fe-safe User Guide for details).

Step 8: Viewing the fatigue life contour:

A copy of the original `.odb` file was created, onto which a new step containing the fatigue results was appended.

In the last step of the file **`<ResultsDir>/tensionwithholeResults_ex01.odb`**, the results for the exported variable should look similar to *:*

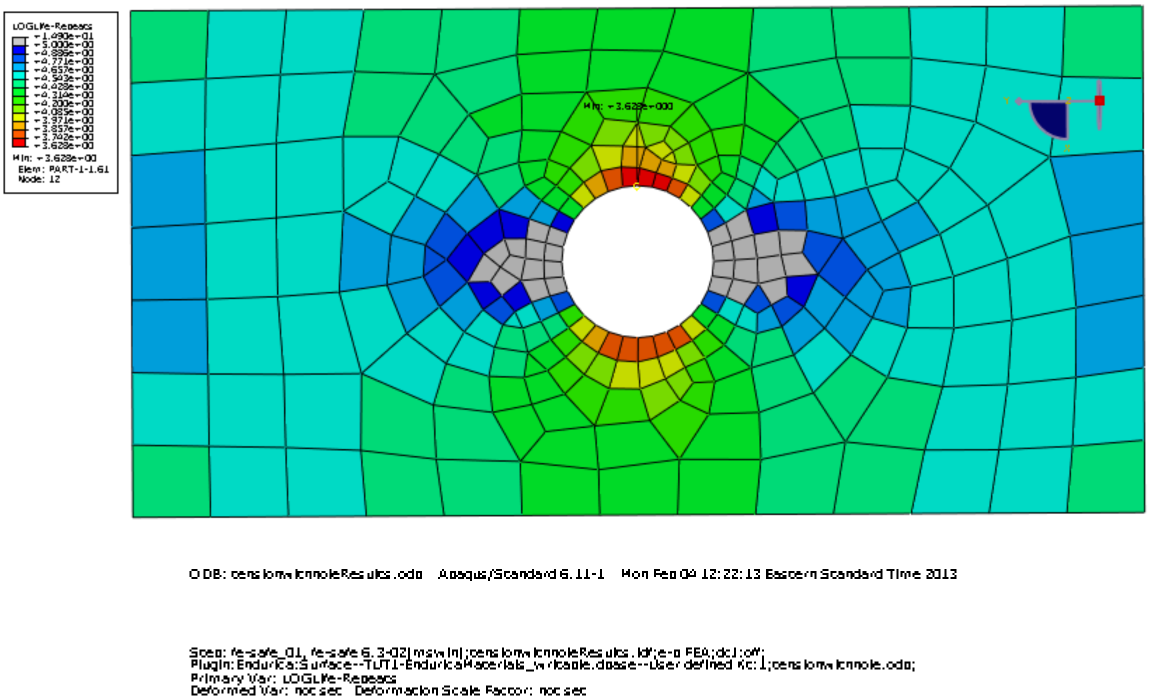


Figure 1-28 Fatigue Life Contour in Log(Repeats) for Example 1

Note: Contours may appear differently depending on post-processor, contour legend limits and averaging scheme.

1.4 Exercise 2: using Loading Definition and Loading Equivalence for Rubber

Objective:

This exercise is a continuation of Exercise 1. Continue using *fe-safe* without changing fatigue configurations.

To define a complex loading scenario – in this case 5 seconds worth of loading based on only 1 second of FEA analysis. The exercise will illustrate use of the loading definition file (*.LDF). The *fe-safe* User Guide includes a full description of the LDF file, including syntax.

Preparation:

Below is listed a tabular representation of Figure 1-6, describing the Y-direction displacement applied in the FE solver as it relates to the time increments in the solution (from the pre-processor):

```
Time (s)       Displacement (mm)
0              0
0.2            40
0.4            20
0.6            60
0.8            100
1              0
```

The desired 5-second displacement that's required is as follows:

```
Time (s)       Displacement (mm)
0              0
0.2            40
0.4            20
0.6            60
0.8            100
1              0
1.2            20
1.4            40
1.6            20
1.8            40
2              20
2.2            40
2.4            20
2.6            40
2.8            0
3              60
3.2            20
3.4            0
3.6            100
3.8            20
4              100
4.2            60
4.4            40
4.6            100
4.8            60
5              0
```

So it will be required to represent the displacements desired in seconds 2 through 5 by using solved displacement stress and strain pairs from the 1 second FEA solution, at those desired time instances.

fe-safe/Rubber™ analysis requires time instants to be equally spaced. In this case we've used 0.2 second time incrementation for the entire loading definition.

Method:

Step 1: Define the loading:

- select **File >> Loadings >> Save Current FEA Loadings As...**;
- save the file as **`ex01.ldf`** and click Save;
- open the file in a text editor to display the contents

```
# .ldf file created by fe-safe compliant product [mswin]

INIT
transitions=Yes
END

# Block number 1
BLOCK
ds=1, de=2
ds=3, de=4
ds=5, de=6
ds=7, de=8
ds=9, de=10
ds=11, de=12
END
```

Note: comment lines starting in # mark may vary):

- Edit the text file to add stress(ds) and strain (de) pairs for each additionally required time instant, as follows

```
# .ldf file created by fe-safe compliant product [mswin]

INIT
transitions=Yes
END

# Block number 1
BLOCK
ds=1, de=2
ds=3, de=4
ds=5, de=6
ds=7, de=8
ds=9, de=10
ds=11, de=12
ds=5, de=6
ds=3, de=4
ds=5, de=6
ds=3, de=4
ds=5, de=6
ds=3, de=4
ds=5, de=6
ds=3, de=4
ds=11, de=12
ds=7, de=8
ds=5, de=6
ds=11, de=12
ds=9, de=10
ds=5, de=6
ds=9, de=10
ds=7, de=8
ds=3, de=4
ds=9, de=10
ds=7, de=8
ds=11, de=12
END
```

- Use the text editor to save the modified file as **`ex02.ldf`**;
- Use *fe-safe* to open the modified loading definition, select **File >> Loadings >> Open FEA Loadings File...**;
- The Loading Settings window will be displayed

- Use the right-mouse-button to click the Elastic-Plastic Block and select **Length in Seconds**:

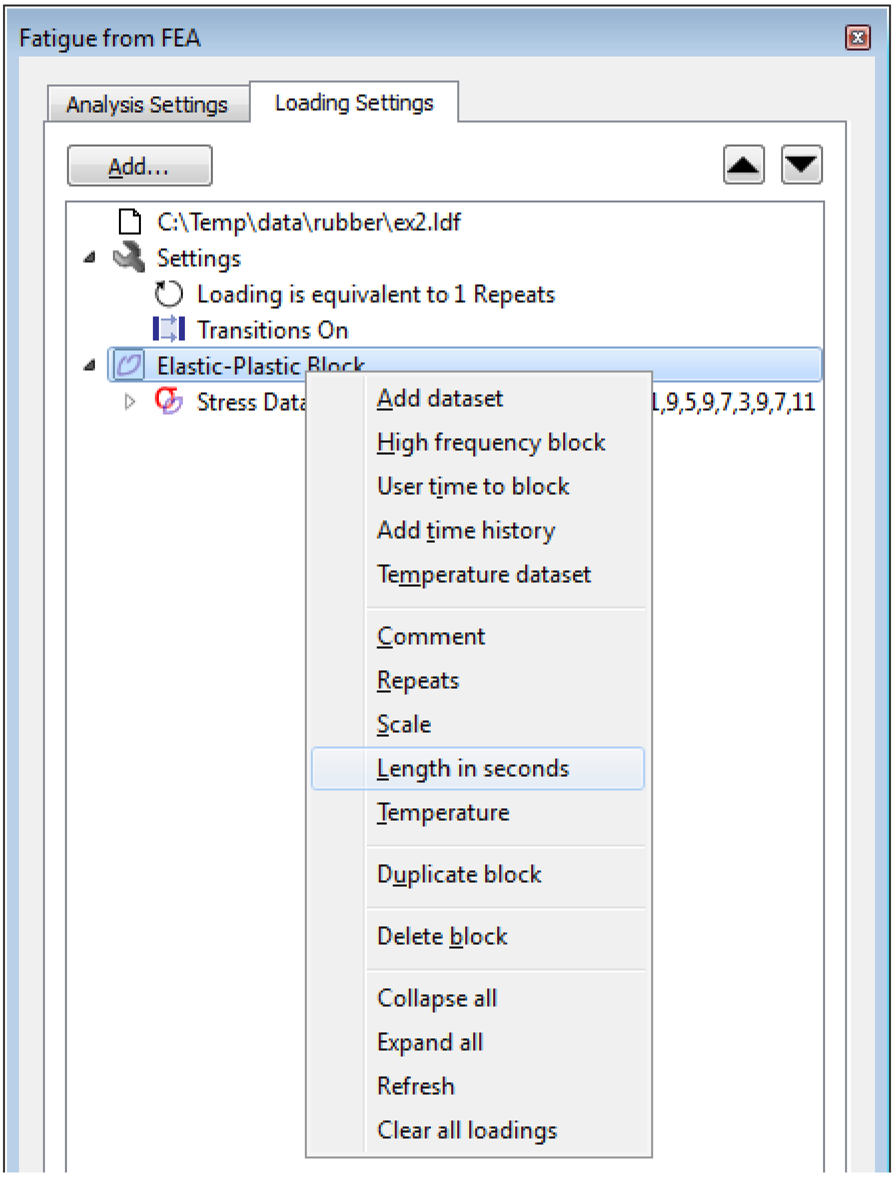


Figure 1-29 Add Length in Seconds to Elastic-Plastic Block

- Enter the length of the loading definition: **5.0**;
- Double-click on **Loading is equivalent to 1 Repeats** in the **Settings** tree;
- The *Conversion factors for Lives from Repeats to User Units* dialogue will be displayed;

- Click on the button for **LDF Time** to calculate the number of Hours defined in the Loading Definition:

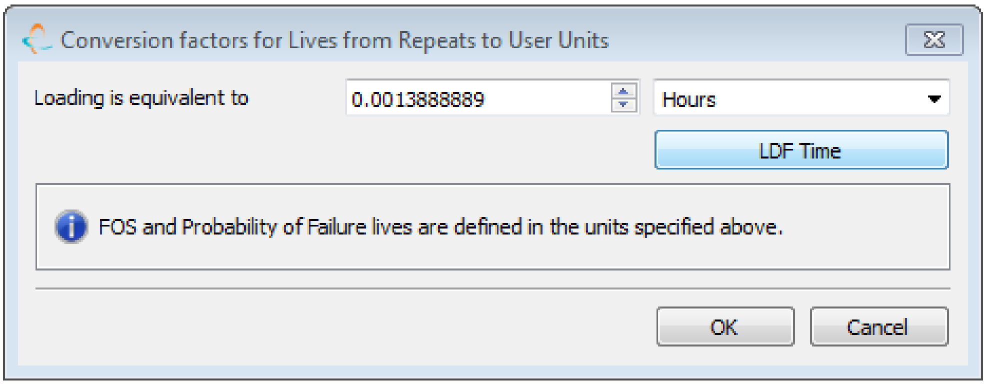


Figure 1-30 Conversion from Repeats to hours

- Click **OK**;
- The loading definition will appear as shown in Figure 1-31:

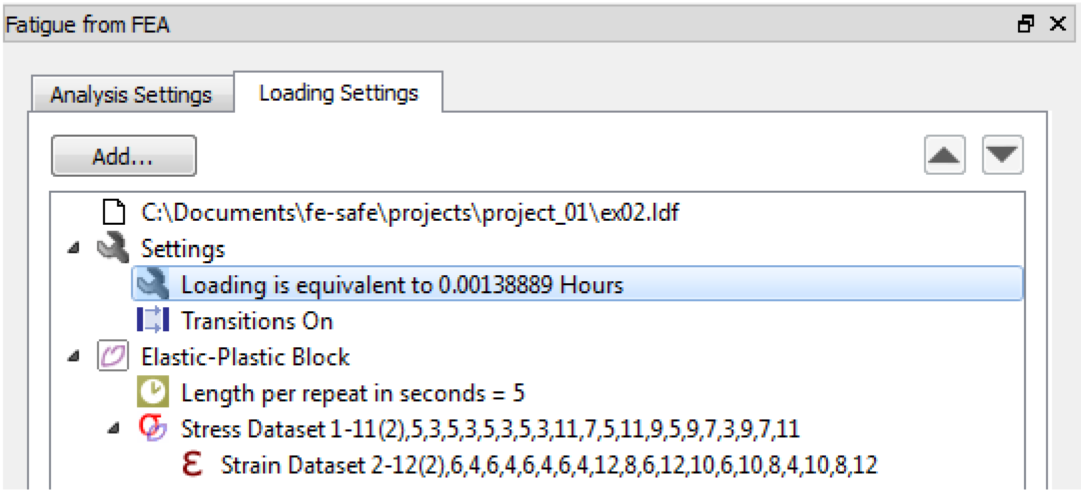


Figure 1-31 Five second loading definition with Loading in Hours

Note: if the window does not appear as shown above, then expand the tree view to show more details.

- Select the **Analysis Settings** tab from the Fatigue from FEA dialogue to switch to the Analysis Settings panel
- A dialogue will be displayed prompting to save the loading definition to ex2.ldf which has been modified:
-

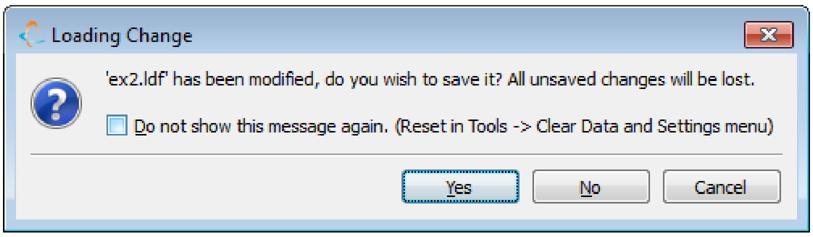


Figure 1-32 Save Loading Definition after change

- Click **Yes**;

Step 2: Define the Output File:

When the FE model was loaded, the Output File was automatically defaulted to a standard file name in the Project Directory.

- in the **Fatigue from FEA** dialogue select the browse button to the right of the **Output File** field.
- Change the output filename to: **`tensionholewithResults_ex02.odb`**

Step 3: Run the analysis:

- *fe-safe* is now configured to run the analysis.
- Press the **Analyse** button. A summary of analysis parameters is displayed;
- Check that the analysis is configured as described above and then click **Continue**.
- As the analysis is being performed, the following information is written to the analysis log file.

```
%      Time          Life-Hours
100   0:00:10        2.6@[0]61.1                266 of 266
```

The analysis log file has the same file name as the output file, except that the extension is **`.log`**, for instance:

<ResultsDir>\tensionholewithResults_ex02.log

This information is also displayed in the **Message Log** window and includes

```
Summary
=======
          Worst Life-Hours                       : 2.604
  at Element [0]61.1
Analysis time                                    : 0:00:57.825
```

Step 4: Reviewing the results

The analysis log shows that the worst-case life for the whole model is:

2.604 Hours of the loading at the centroid of element 61.

***Note**: An optional conversion factor was used in this example to convert the fatigue life in repeats to fatigue life with respect to hours (see the fe-safe User Guide for details).*

Step 5: Viewing the fatigue life contour:

A copy of the original `.odb` file was created, onto which a new step containing the fatigue results was appended.

In the last step of the file **`<ResultsDir>/tensionwithholeResults_ex01.odb`**, the results for the exported variable should look similar to *:*

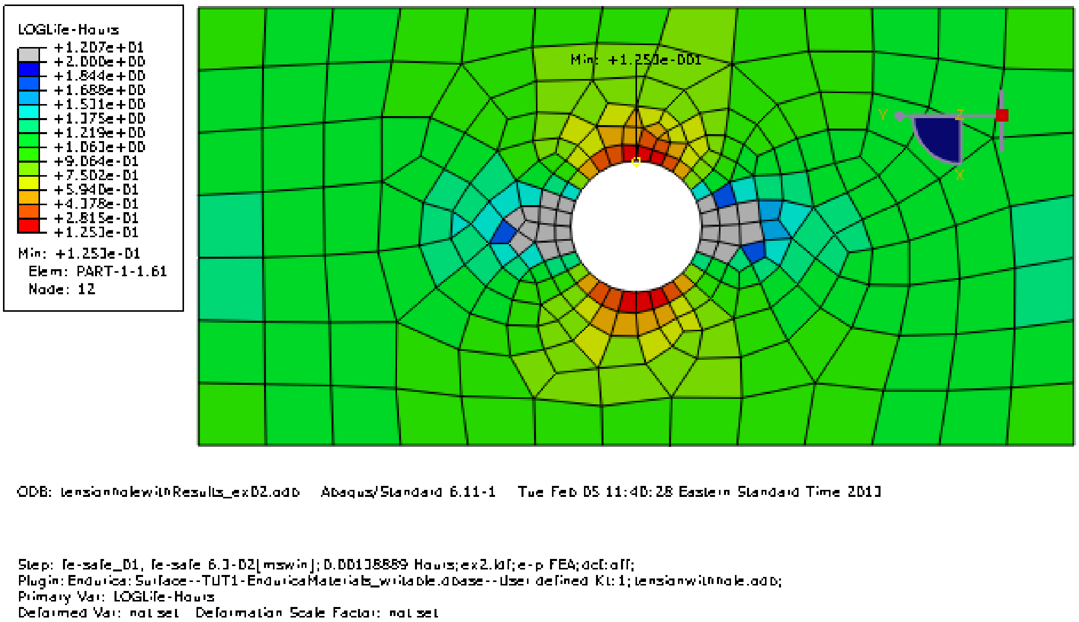


Figure 1-33 Fatigue Life Contour in Log(Hours) for Example 2

Note: Contours may appear differently depending on post-processor, contour legend limits and averaging scheme.

1.5 Exercise 3: Request Exports and Outputs for an element of interest

Objective:

This exercise is a continuation of Exercise 2. Continue using *fe-safe* without changing fatigue configurations.

This exercise will define a single element for a secondary FEA Fatigue analysis, and configure plug-in Settings to request exports and outputs for an element of interest.

Preparation:

The element at which the lowest life in Hours was reported during Exercise 2 was element number 61. This will be used for a secondary post-processing analysis of only one element. Such an analysis may be desirable in order to:

1. Speed up delivery of exports and outputs at a known position, without calculating lives in the remainder of the model
2. Limit log output from *fe-safe*/Rubber to ASCII text file formats
3. Limit size of Output Files generated during the secondary analysis

Method:

Step 1: Define the Output File:

When the FE model was loaded, the Output File was automatically defaulted to a standard file name in the Project Directory.

- in the **Fatigue from FEA** dialogue select the browse button to the right of the **Output File** field.
- Change the output filename to: **`tensionholewithResults_ex03.csv`**

Note: Since only one element will be analysed a text format such as *.csv or *.fer is used to reduce output file size. See the Appendices of the *fe-safe* User Manual for details of Output File fatigue results formats.

Step 2: Define the List of Items:

- in the **Fatigue from FEA** dialogue select the **Exports...** button ;
- The *Exports and Outputs* dialogue will appear, select the **List of Items** tab;
- In the field on the List of Items tab enter the instance and element number: [0]61
- Select the checkbox labelled **Only analyse listed items**:

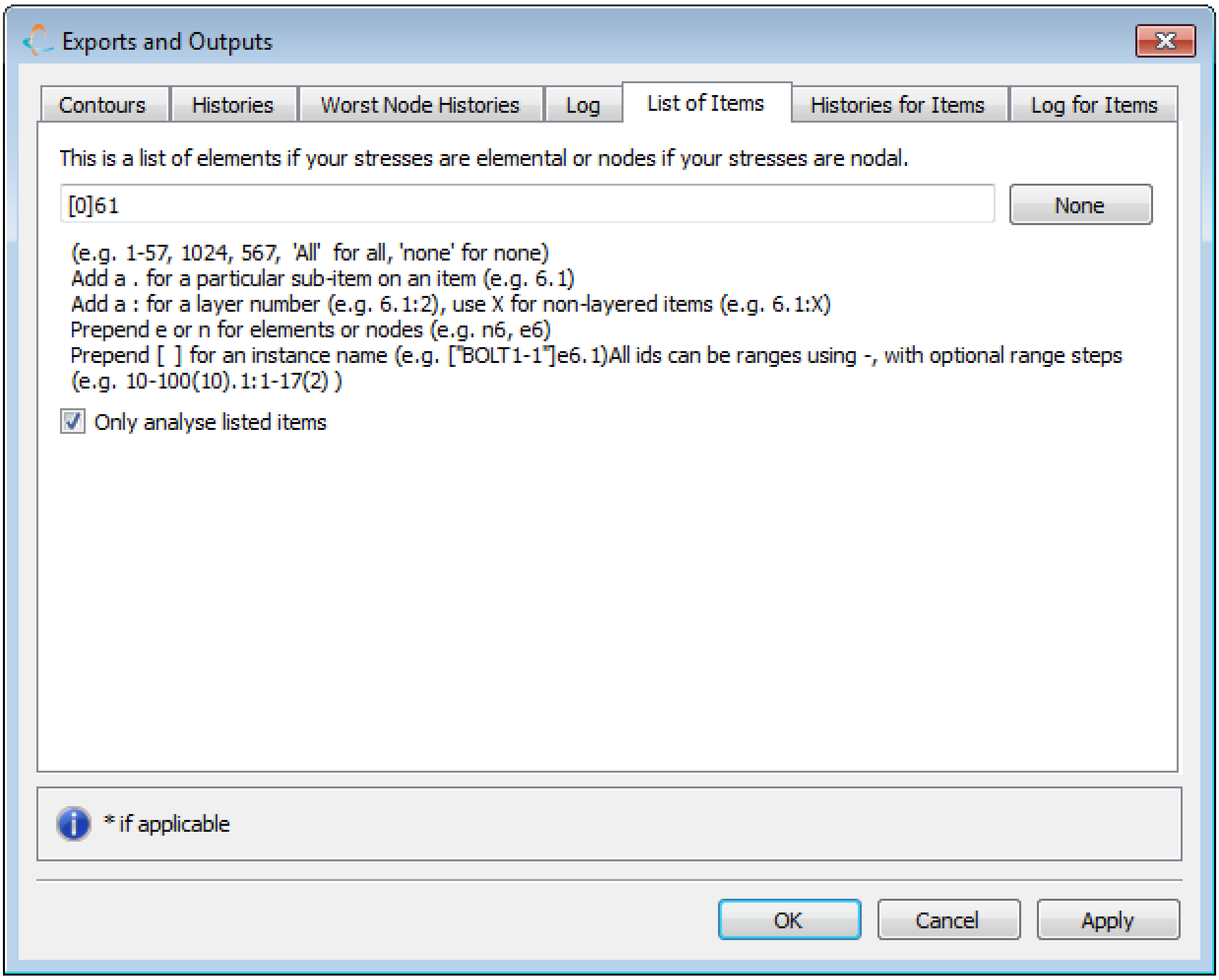


Figure 1-34 Configure the List of Items to only analyse the listed items

- Click **OK**;

- The **Fatigue from FEA** dialogue will appear as shown:

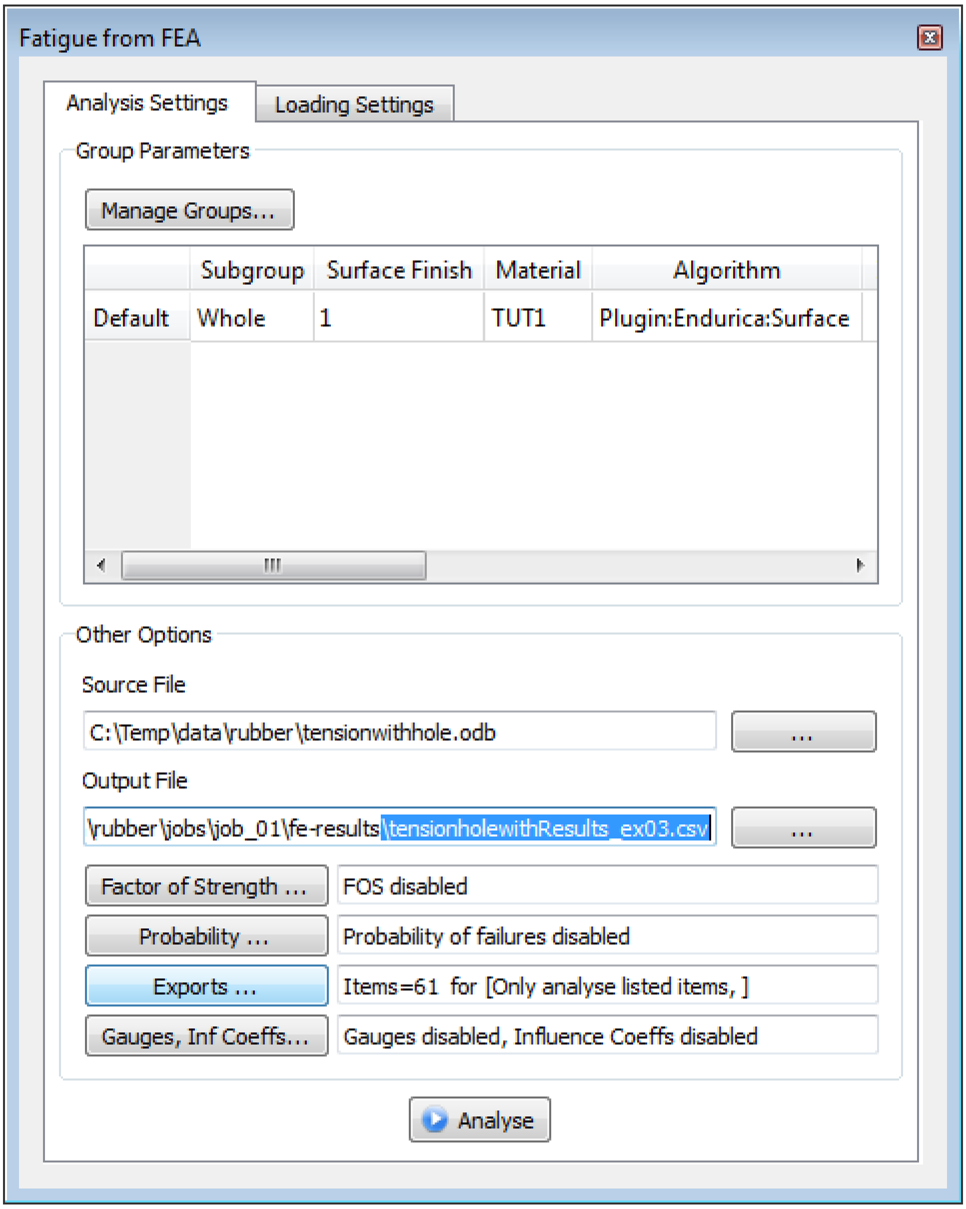


Figure 1-35 Output File and Exports after Exercise 3 Definition

Step 3: Configure the plug-in Settings:

- In the **Fatigue from FEA** dialogue, identify the **Group Parameters** table
- double-click the **Algorithm** column header to open the in **Group Algorithm Selection** dialogue box for all groups;
- The **Group Algorithm Selection** dialogue will appear as shown:

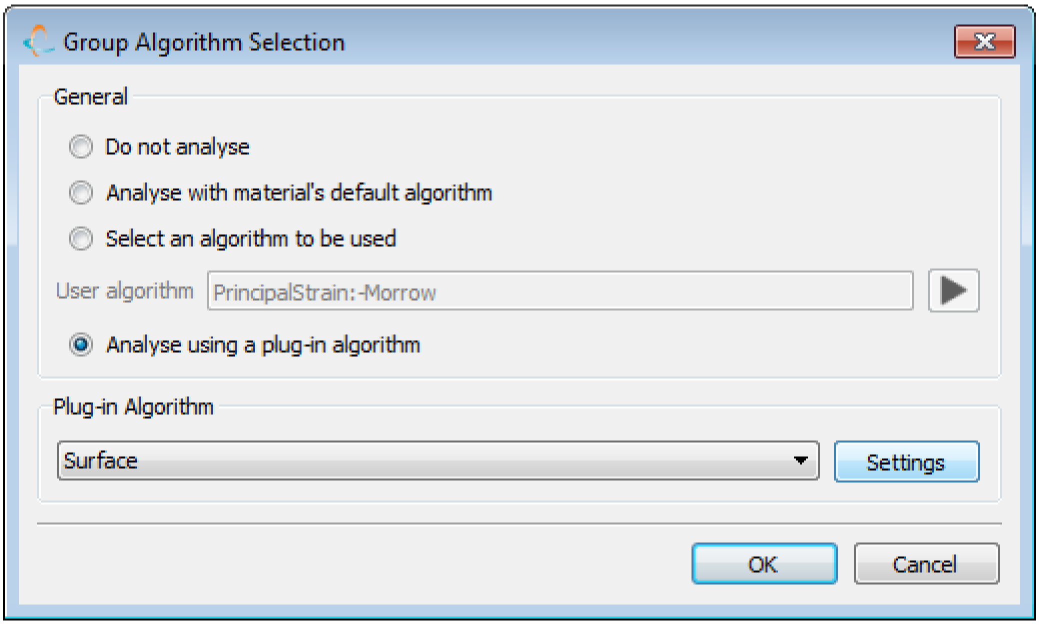


Figure 1-36

- select the **Settings** button, next to the plug-in algorithm **(Surface)**;

- The **Edit Plug-in Settings** dialogue will appear as shown:

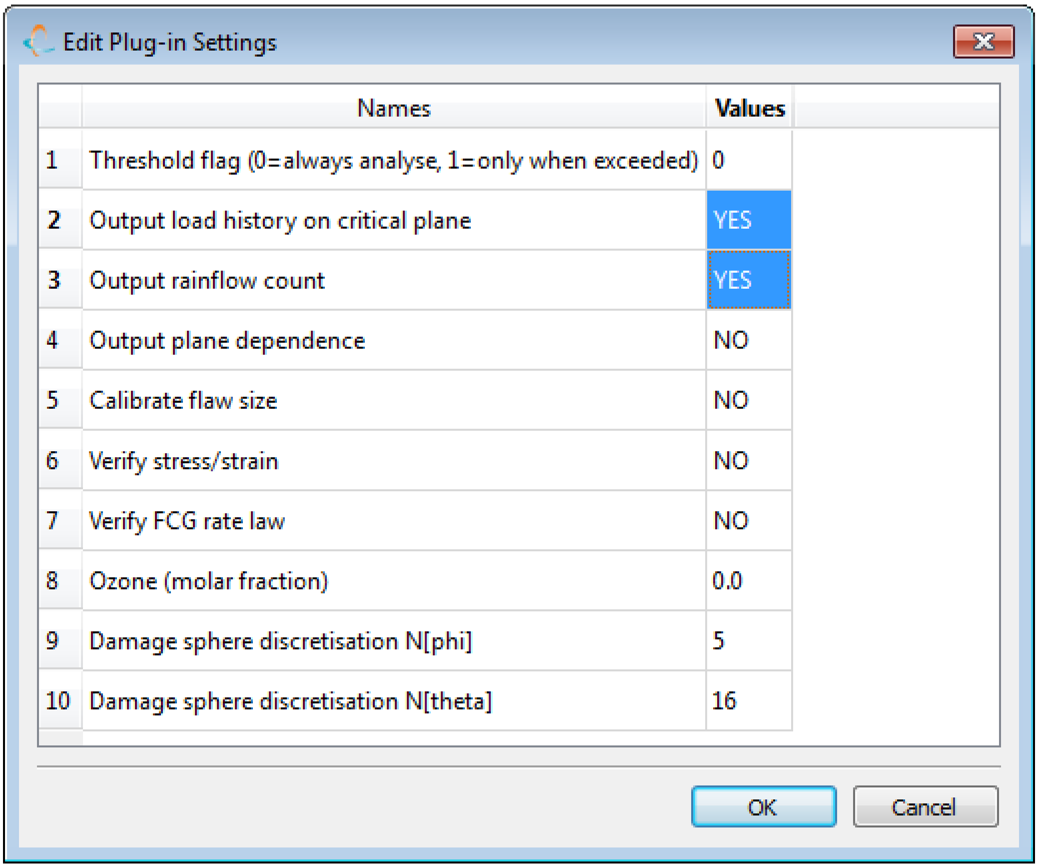


Figure 1-37 Configure plug-in Settings to request load history and rainflow count

- Edit the value of Setting 2 *Output load history on critical plane*: **YES**
- Edit the value of Setting 3 *Output rainflow count*: **YES**
- Click **OK** to dismiss the **Edit Plug-in Settings** dialogue;
- Click **OK** to dismiss the **Group Algorithm Selection** dialogue;

Step 4: Run the analysis:

fe-safe is now configured to run the analysis.

Press the **Analyse** button. A summary of analysis parameters is displayed;

Check that the analysis is configured as described above, and then click **Continue**.

As the analysis is being performed, the following information is written to the analysis log file.

```
%       Time          Life-Hours
100     0:00:00       2.6@[0]61.1                  266 of 266
```

The analysis log file has the same file name as the output file, except that the extension is **`.log`**, for instance:

`<ResultsDir>\tensionholewithResults_ex03.log`

This information is also displayed in the **Message Log** window and includes

```
Summary
=======
          Worst Life-Hours                      : 2.604
  at Element [0]61.1
Analysis time                                   : 0:00:00.260
```

Step 5: Reviewing the results

The analysis log shows that the worst-case life for the whole model is:

1.335 Hours of the loading at the centroid of element 61.

***Note**: The List of Items was used to Only analyse listed items in this Exercise. This results in a very quick secondary fatigue analysis as show above, but the fatigue life results in Hours for the element have not changed*

Step 6: Viewing the Exports and Outputs from the fe-safe/Rubber plug-in:

- Use the operating system to browse to the Results Directory (e.g. **`C:\Temp\data\rubber\jobs\job_01\fe-results`**);
- Find the log file generated for load history on the critical plane (e.g. **`tensionholewithResults_ex03CEDHISTORY.log`**);
- Find the log file generated for Rainflow Count (e.g. **`tensionholewithResults_ex03RAINFLOW.log`**);
- Inspect the log files in a text editor and plot histories an rainflow counts if desired

2 Tutorial 2: Time-dependent Analysis of a single element with Exports

2.1 Introduction

This tutorial outlines how to perform a time-dependent rubber fatigue analysis using *fe-safe*/Rubber™. This tutorial is based on the ASCII FE tensor format:

`<DataDir>\rubber\TUT2.txt`

2.1.1 Preparation

The tutorial uses an ASCII FE Tensor format. However, the same techniques can be applied to all FE formats for which nominal strains and stresses can be reported. Results may differ while using other FE formats since the ASCII file contains stress and strain solutions for only one element. Please see the Appendices of the *fe-safe* User Manual for details on the Proprietary FE file format.

This tutorial assumes that the user has experience using *fe-safe*, thus detailed information on how to set up an *fe-safe* analysis is not included in this tutorial. Please see Tutorial 1 for details.

- Start *fe-safe*/Rubber™;
- The Configure *fe-safe* Project Directory window will be displayed;
- Select an appropriate Project Directory **`<ProjectDir>`** and Click **OK**.

General FEA and analysis options

- From the Menu bar select **Tools >> Clear Data and Settings...**;
- The Clear Data and Settings dialogue will appear
- Select all of the Project settings and the Re-set file dialogues setting (all but the Re-set user settings checkbox).
- Click **OK**.

Open the EnduricaMaterials (writable copy) database

If the **`EnduricaMaterials`** database is not shown in the Material Databases window, please see section 1.1.1 above.

Load the *fe-safe*/Rubber™ plug-in

If the *fe-safe*/Rubber plug-in is not loaded, please see section 1.1.1 above.

2.2 Opening the sample FE model

The model for this tutorial is a one-element ASCII file. Displaying the contour is not useful in such cases.

Loading is applied for 1 second using 5 increments.

- Select the menu item **File >> FEA Solutions >> Open Finite Element Model...** and browse to the sample file **TUT2.txt** from the directory **<DataDir>\rubber**.
- A prompt to pre-scan the file will be displayed. Click **Yes**.
- The *Select Datasets to Read* dialogue will be displayed
- Check the *Quick select* items for **Stresses**, **Strains** as shown
- click **Apply to Dataset List** to apply the selections
- select **OK** to load the datasets

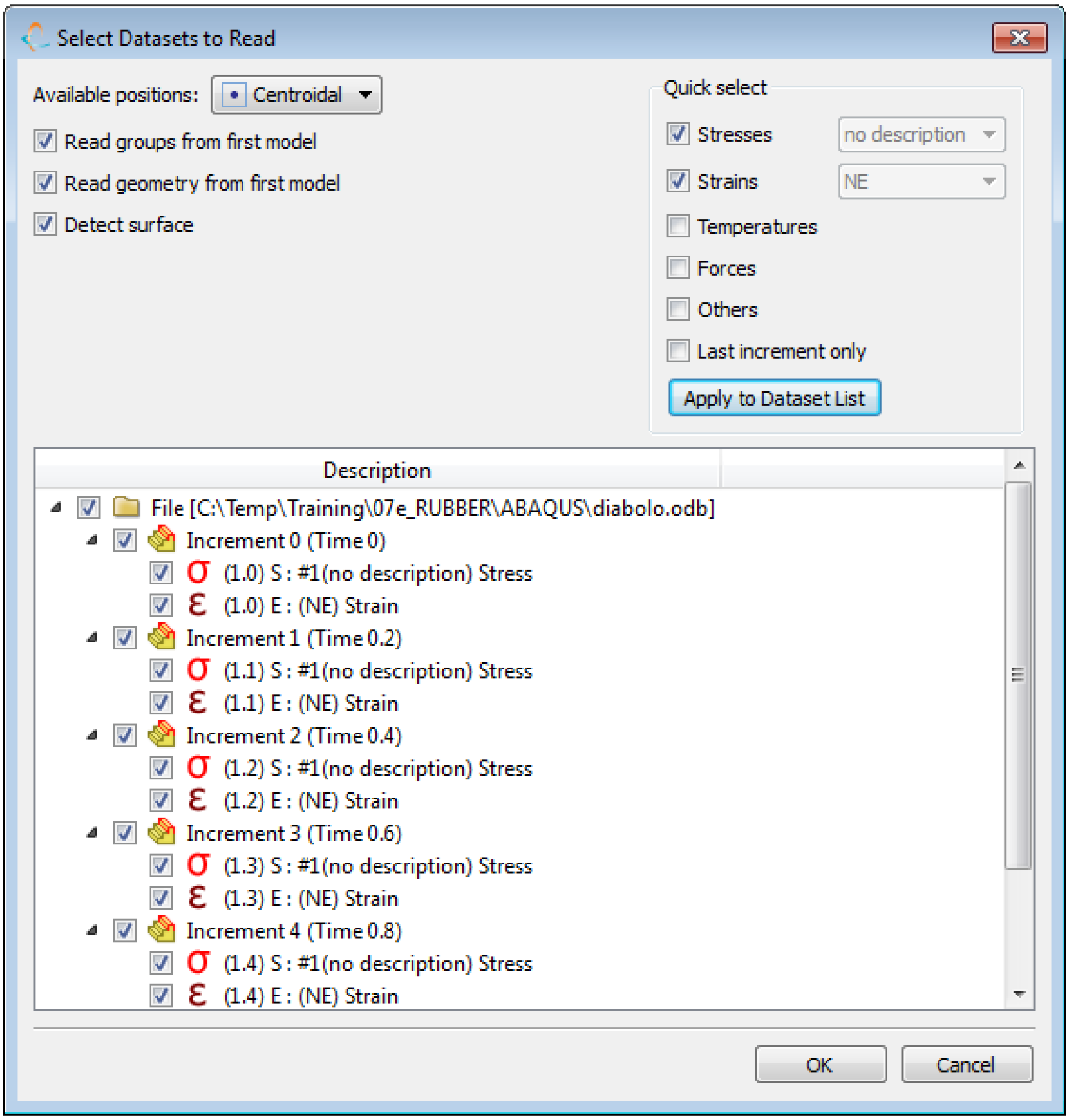


Figure 2-1 Select all stress and strain datasets

- When the model has finished loading, the **Loaded FEA Models Properties** dialogue box
- Ensure that the stress, strain and temperature units are psi, strain and deg.C, respectively, as shown:

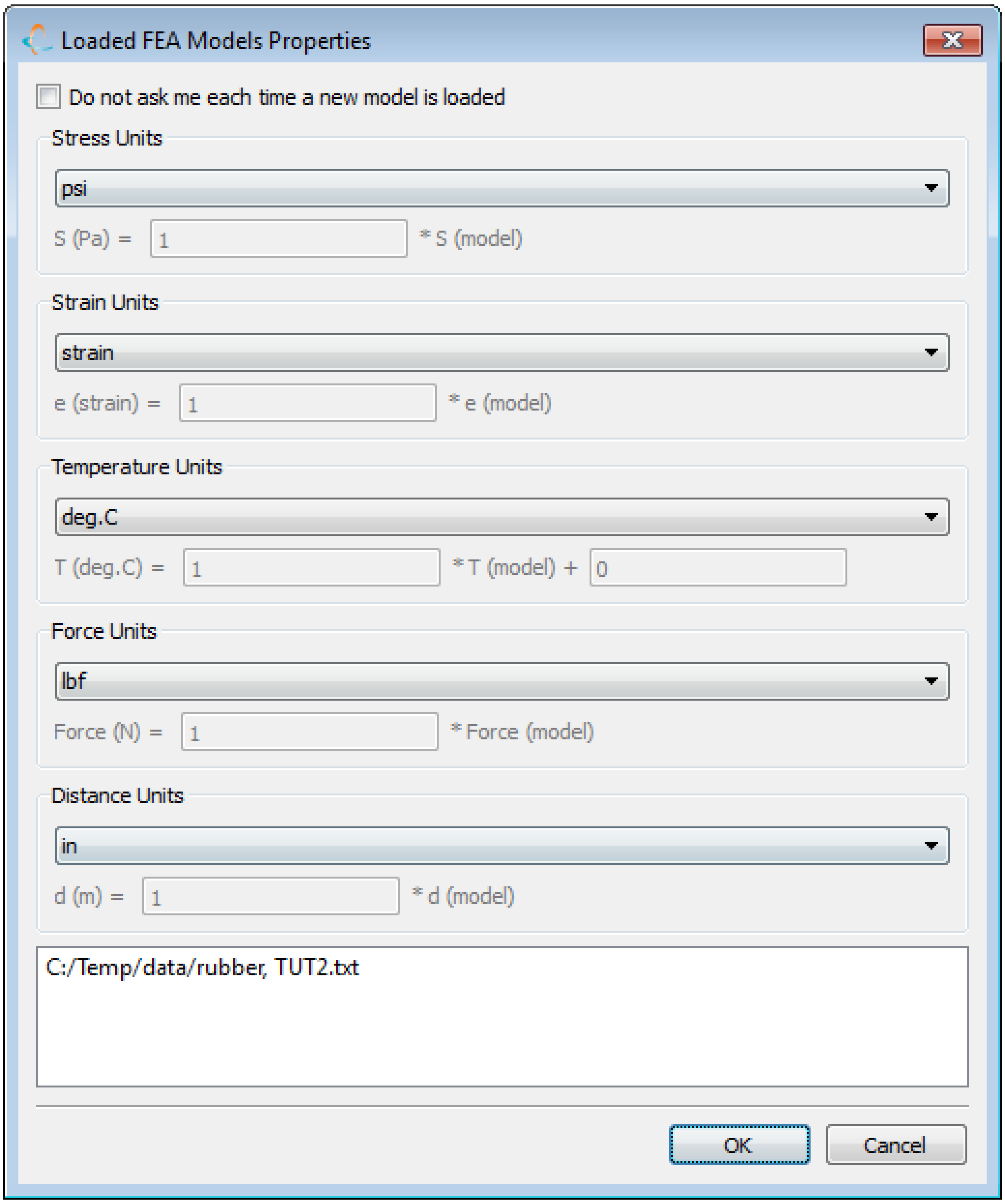


Figure 2-2 Configure Loaded FE Models Properties

- click **OK**.
- A dialogue will prompt to edit element groups loaded from the model, click **No**.

- You will see the following message regarding zeroes in datasets, this is normal

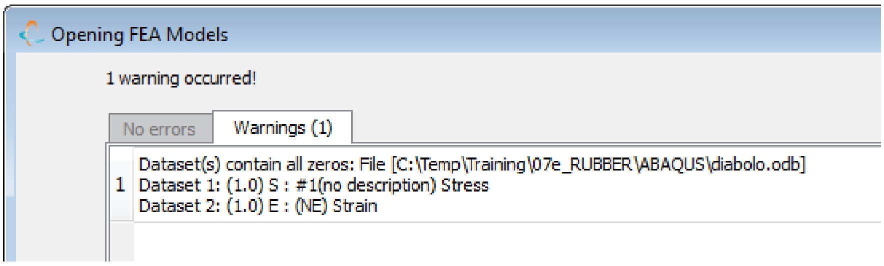


Figure 2-3

- A summary of the open model appears in the **Current FE Models** window, showing the loaded datasets as shown:

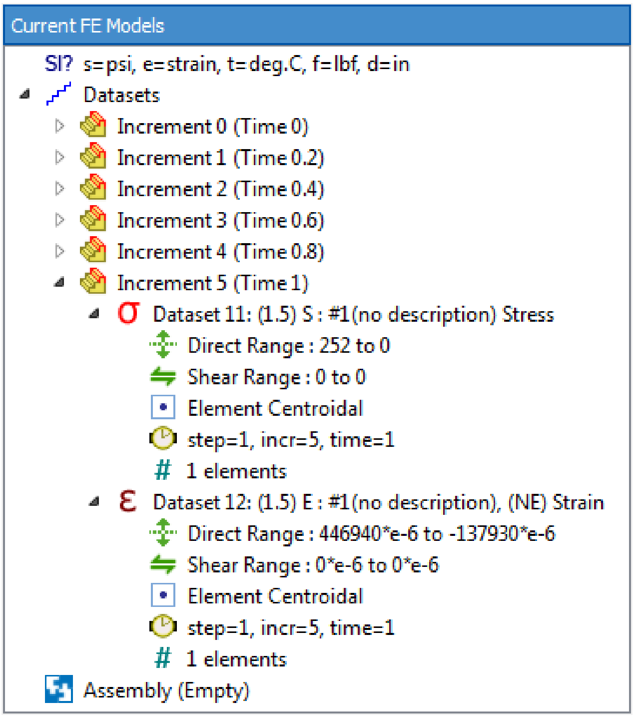


Figure 2-4 Current FE Models are displayed

Note: if the window does not appear as shown above, then expand the tree view to show more details.

- The model contains six stress datasets, six strain (NE) datasets. *fe-safe* also extracts element group information from the ASCII FE file.
- If the correct number of datasets is not shown as above, use your right-mouse-button in the Current FE models window to select **Reload All Models**
- Click **Yes** as needed until the *Select Datasets to Read* dialogue is displayed as shown above.
- If the units of Stress, Strain and Temperature, do not appear as shown they can be changed by double-clicking on the SI? icon in the **Current FE Models** window and modifying **Properties**

2.3 Exercise 1: *fe-safe*/Rubber™ analysis with time-dependent effects

This tutorial should be read in conjunction with the *fe-safe* User Guide and the *fe-safe*/Rubber™ User Guide.

Objective:

To perform a rubber fatigue analysis based on a sequence of surface based centroidal stress and strain (NE) solutions from FEA, with time dependent effects included.

Each increment represents the hyperelastic stress and strain solutions from FEA. Each stress and strain dataset corresponds to a time increment (each increment is 0.2 seconds).

Analysis process:

For each elemental centroidal position:

- The nominal strain (NE) and stress (S) are read from the FE model database into *fe-safe*.
- The loading history is configured according to the sequence of stress and strain datasets and the length in seconds specified in Loading Settings
- The 6 components of the nominal strain tensor are calculated from the 3 in-plane nominal strain components, and from the plane stress condition (e.g. the out of plane stress is exactly 0).
- A series of material planes is generated based on the plug-in setting for damage sphere variables phi and theta. Subsequent calculations will be repeated on each material plane, in order to identify the critical plane.
- The local loading history is computed for each plane, giving the Cracking Energy Density as a function of time.
- A rainflow counting algorithm is then used to identify each individual cycle (e.g. peak and valley) contained within the entire local loading history.
- A numerical integration of the crack growth rate law is made to determine the number of repeats (the life) required to grow the initial flaw to its specified size at nucleation (see the *fe-safe*/Rubber™ Theory manual). As a part of the computation, the crack growth rate contributions of individual cycles are summed to obtain a total rate of crack growth per repeat of the entire loading history. The initial and final flaw sizes, and all crack growth properties were specified as a part of the material definition.
- Once the life has been computed for every material plane, then the minimum life is selected from among the results and reported as the life of the individual item.
- Output File containing Log Life and *fe-safe* Results Log file containing analysis configurations are automatically generated
- Any requested auxiliary outputs are written to their respective log files.

Method:

Step 1: Define the loading:

As in Tutorial 1, Exercise 1, generate a single fatigue loading block, containing a sequence of stress datasets. The load will represent a history between the initial increment (Increment 0, stress dataset 1 and strain dataset 2 at time t=0.0 s) through the fifth increment (Increment 5, stress dataset 11 and strain dataset 12 at time t=1.0 s).

- In the Current FE Models window, select any stress dataset;
- Select the **Loading Settings** tab from the *Fatigue from FEA* dialogue to switch to the loading tree;
- Use your right-mouse-button to select **Clear all loadings** and click **Yes**;
- Use the Add button in the Loading Settings window to select **Add... >> Dataset;**
- Click **OK** when prompted
- Double-click on the Stress Dataset and using your keyboard, type the sequence **1-11(2)**
- In the current FE Models window, select any strain dataset;
- In the Loading Settings panel, right-click on the stress dataset and select **Add dataset**;
- The strain dataset will appear as a branch in tree-view under the sequence of stress datasets.
- Double-click on the Strain Dataset and using your keyboard, type the sequence **2-12(2)**
- Use the right-mouse-button to click the Elastic-Plastic Block and select **Length in Seconds**:
- Enter the length of the loading definition: **1.0**
- The Loading Settings window will appear as shown:

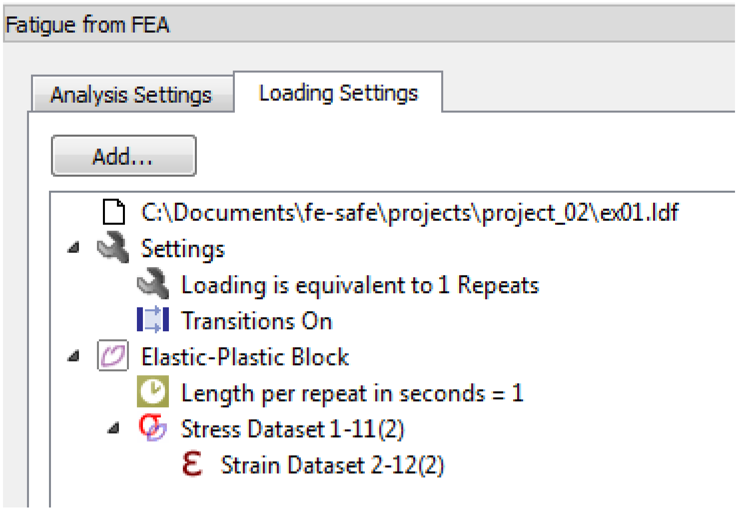


Figure 2-5 Time-dependent dataset sequence loading

Note: if the window does not appear as shown above, then expand the tree view to show more details.

- Select the **Analysis Settings** tab from the Fatigue from FEA dialogue to switch to the Analysis Settings panel;

Step 2: Define the subgroup option:

- double-click the **Subgroup** column header to open the **Subgroup Selection** dialogue for all groups.
- select **Whole group**;
- click **OK**.

- *Step 3: Define the Surface Finish:*

For all *fe-safe*/Rubber analysis the user-defined Kt value of 1 should be configured. This will ensure the effects of surface finish aren't applied to *fe-safe*/Rubber analyses.

- double-click the **Surface** column header to open the **Surface Finish Definition** dialogue for all groups
- select **Define Kt as a value**;
- Enter **User-defined Kt value** of 1;
- click **OK**.

Step 4: Select the Material:

For this exercise, a copy of the material dataset SBR_GUM will be created, and then modified to give it a lower threshold parameter (**`fcgr:T0`**).

First create the new material:

- in the **Material Databases** window, use the left-mouse-button to select (highlight) the material **SBR_GUM** from the list of available materials in the **`EnduricaMaterials_writable.dbase`** material database;
- select **Material >> Copy Material** to create a new material called **CopyOfSBR_GUM**, which will appear at the bottom of the **`EnduricaMaterials_writable.dbase`** database;
- double-click the newly created material to rename it – change the name to **TUT2**.

Configure the material parameters for the **`LAKELINDLEY`** fatigue crack growth rate model:

- expand the **TUT2** material to show its properties;
- select the T0 field: **fcgr : T0 (kJ/m^2)**;
- double-click on the value field to edit it
- modify the value to: **0.04**;
- All of the remaining material parameters should remain as-is

To define **TUT2** for the whole component:

- highlight the material **TUT2** in the **`EnduricaMaterials_writable.dbase`** database;
- double-click the **Material** header in the **Groups Parameter** box in the **Fatigue from FEA** dialogue - a **Change Material?** confirmation dialogue box appears;
- click **YES**;
- the material name should appear for all groups in the **Material** column.

Step 5: Select the plug-in Algorithm and configure plug-in Settings:

This exercise uses a plug-in algorithm. To define the plug-in algorithm for the whole model (i.e. all element groups):

- double-click the **Algorithm** column header to open the in **Group Algorithm Selection** dialogue box for all groups;
- select the **Analyse using a plug-in algorithm** option;

Note: If 'Analyse using a plug-in algorithm' is not an option, go back to Load the *fe-safe*/RubberTM plug-in above to follow the steps to Load the plug-in.

- select the **Plug-in algorithm** drop-down box, and select **Surface** from the drop-down menu;
- select the **Settings** button, next to the plug-in algorithm;

- The **Edit Plug-in Settings** dialogue will appear as shown:

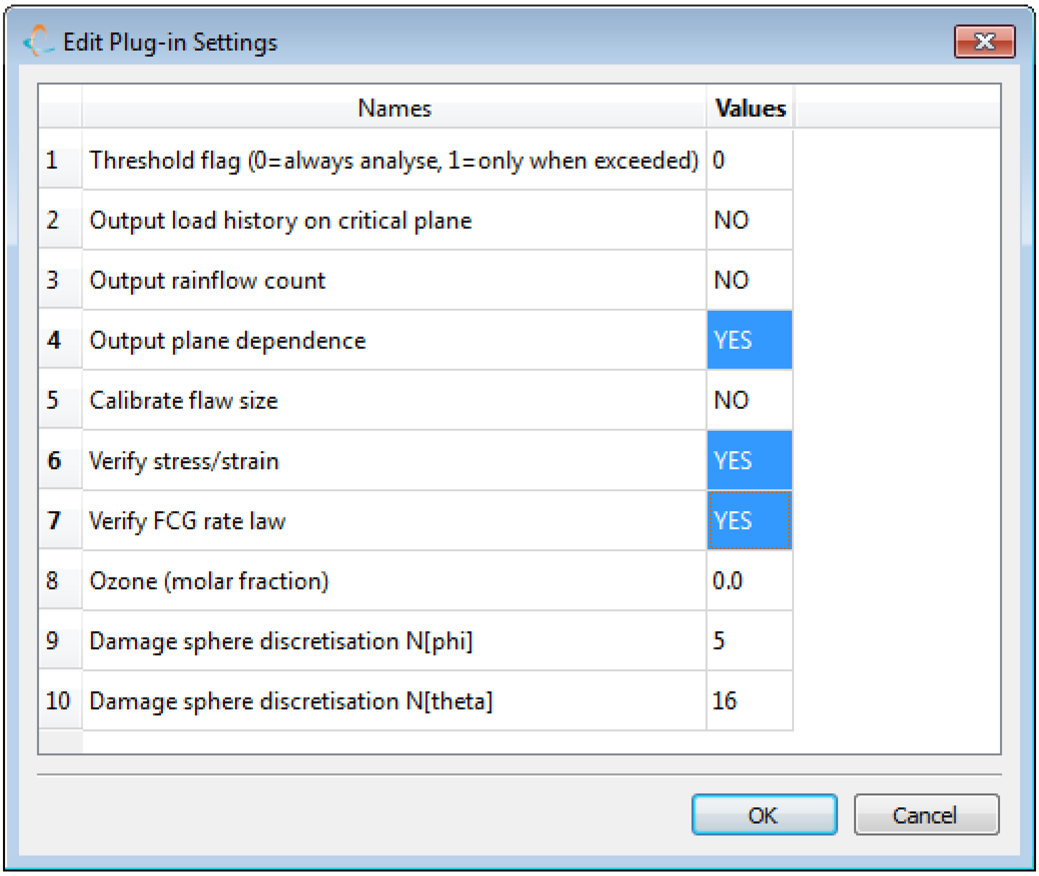


Figure 2-6 Edit plug-in settings

- Edit the value of Setting 4 *Output plane dependence*: **YES**
- Edit the value of Setting 6 *Verify stress/strain*: **YES**
- Edit the value of Setting 7 *Verify FCG rate law*: **YES**
- Click **OK** to dismiss the **Edit Plug-in Settings** dialogue;
- Click **OK** to dismiss the **Group Algorithm Selection** dialogue;

Step 6: Define the output file:

When the FE model was loaded, the Output File was automatically defaulted to a standard file name in the Project Directory.

- in the **Fatigue from FEA** dialogue select the browse button to the right of the **Output File** field.
- Change the output filename to: **`TUT2Results_ex01.txt`**

Note: Since only one element will be analysed a text format such as *.txt or *.fer is used to reduce output file size. See the Appendices of the *fe-safe* User Manual for details of Output File fatigue results formats.

Step 7: Define the Export options:

- in the **Fatigue from FEA** dialogue select the **Exports...** button ;
- The *Exports and Outputs* dialogue will appear, select the **List of Items** tab;
- In the field on the List of Items tab enter the element number: **2827**
- select the **Histories for Items** tab;
- **check** the box next to *stress tensors*; in the Stress/Strain section
- select the **Log for Items** tab;
- **check** the box next to D*ataset stresses*; in the Stress/Strain section
- Click **OK**;
- The **Fatigue from FEA** dialogue will appear as shown:

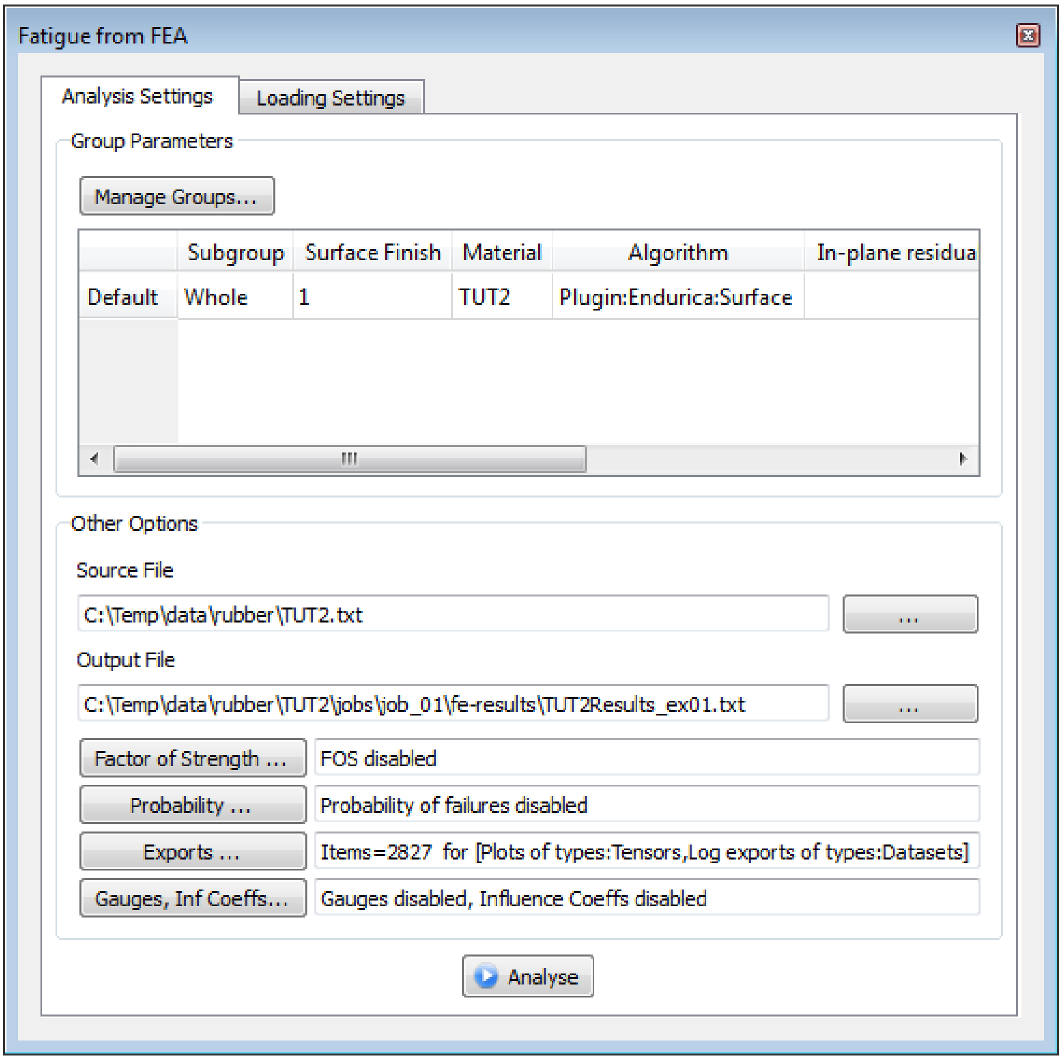


Figure 2-7 Output File and Exports after Exercise 1 Definition

Note: If the dialogue does not appear exactly as above, it can be re-sized so that the fields can be shown.

Step 8: Run the analysis:

fe-safe is now configured to run the analysis.

- Press the **Analyse** button. A summary of analysis parameters is displayed;
- Check that the analysis is configured as described above, and then click **Continue**.

As the analysis is being performed, the following information is written to the analysis log file.

```
   %       Time         Life-Repeats
100     0:00:00       47867@2827.1              1 of 1
```

The analysis log file has the same file name as the output file, except that the extension is **`.log`**, for instance:

`<ResultsDir>\TUT2Results_ex01.log`

This information is also displayed in the **Message Log** window and includes

```
Summary
=======
          Worst Life-Repeats                     : 47867.07
  at Element 2827.1
Analysis time                                    : 0:00:00.180
```

Step 9: Reviewing the fe-safe Results log file

- Use a text editor to open the log file and inspect the table marked **`DATASETS for Element 2827.1`**
- Note the details of the conversion from stress in units of psi in the ASCII FE file to MPa in the Safe Technology *fe-safe* Finite Element Data (*.fed) folder:

```
DATASETS for Element 2827.1

   DS#     Units           xx or T           yy            xy...
     1       psi       0.000e+000    0.000e+000    0.000e+000...
     1       MPa       0.000e+000    0.000e+000    0.000e+000...
     2    strain       0.000e+000    0.000e+000    0.000e+000...
     2        uE                0             0             0...
     3       psi       5.850e+000    4.760e+001    1.760e-007...
     3       MPa       4.034e-002    3.282e-001    1.214e-009...
     4    strain      -3.132e-002    8.538e-002    9.828e-010...
     4        uE           -31321         85380             0...
    ...
```

Note: the stress datasets shown in the log file may not be those used by *fe-safe*/Rubber. Exported Log for Items come from the *fe-safe* User Interface as opposed to the *fe-safe*/Rubber plug-in algorithm.

- Find the **`List of plottable file(s)`** in the log file and copy the fill path to the Exported history for items tensor file, e.g.:

`C:\Temp\data\rubber\TUT2\jobs\job_01\fe-results\TUT2Results_ex01.txt_Element_2827.1.txt`

- Retain the copied file path in the clipboard for later use in Step 10 below.

Step 10: Plotting the History file

- Use *fe-safe* to open the Exported history by selecting **File >> Data Files >> Open Data File...**
- Select the exported history file identified above in Step 9
- The data file will be shown in the **Loaded Data Files** window:

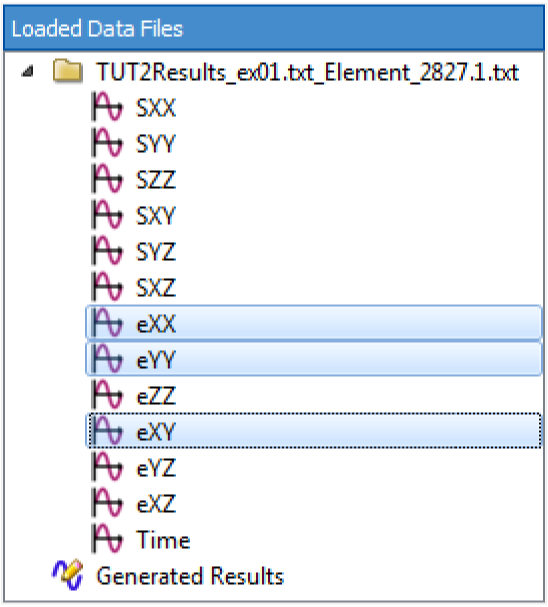


Figure 2-8 Exported history shown in Loaded Data Files

Note: the stress histories shown above in Figure 2-8 Exported history shown in Loaded Data Files may not be those used by *fe-safe*/Rubber. Exported Histories for Items come from the *fe-safe* User Interface as opposed to the *fe-safe*/Rubber plug-in algorithm.

- Use the left-mouse-button and shift key to select the three components of strain (**eXX**, **eYY**, and **eXY**)
- Use *fe-safe* to plot the three plots in one space using the menu selection **View >> Stack Plots**.

- A stacked plot will be displayed as shown:

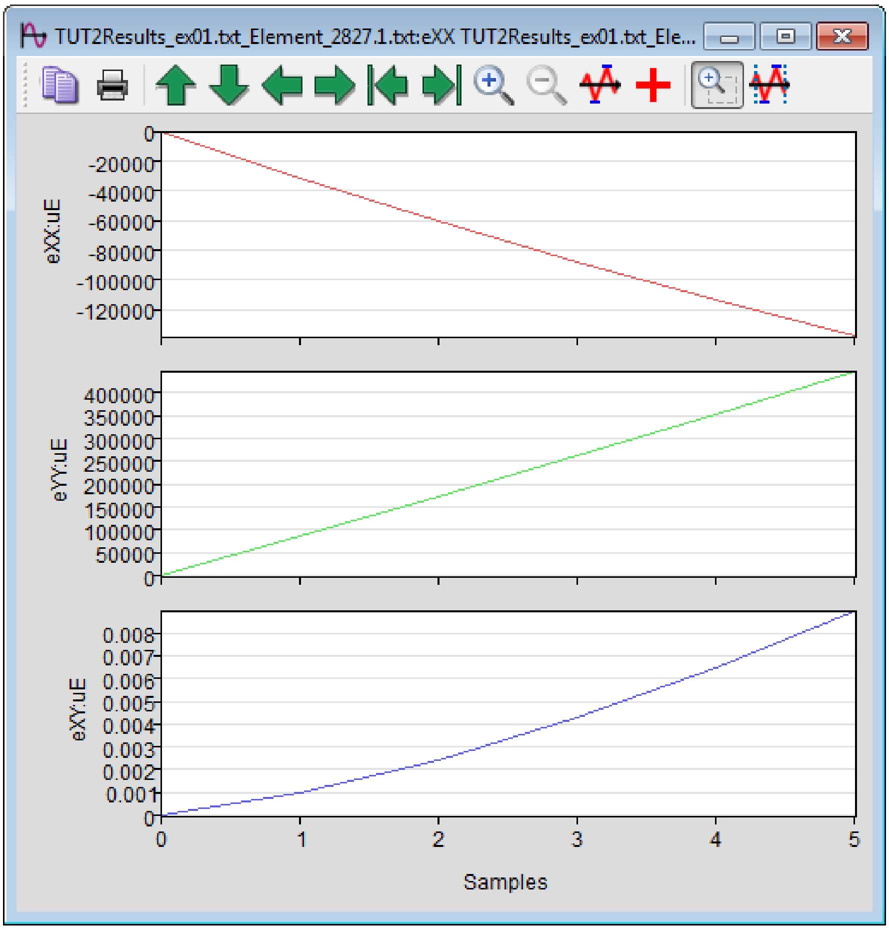


Figure 2-9 strain components on the surface displayed

Step 11: Viewing the Exports and Outputs from the fe-safe/Rubber plug-in:

- Use the operating system to browse to the Results Directory (e.g. **`C:\Temp\data\rubber\TUT2\jobs\job_01\fe-results`**);
- Find the log file generated for the output of plane dependence, also known as a damage sphere (e.g. **`TUT2Results_ex01DAMAGESPHERE.log`**);
- Find the log file generated for the output of stress/strain verification setting (e.g. **`TUT2Results_ex01VERIFYSTRESS.log`**);
- Find the log file generated for the output the fatigue crack growth rate verification setting (e.g. **`TUT2Results_ex01VERIFYFCGR.log`**);
- Inspect the log files in a text editor and plot histories an rainflow counts if desired

2.4 Exercise 2: Define temperature dependence with existing configurations

Objective:

This exercise is a continuation of Exercise 1. Continue using *fe-safe* without changing fatigue configurations.

This exercise will define an operating temperature for the entire model.

Method:

Step 1: Configure the temperature Settings

- In the **Fatigue from FEA** dialogue, switch to the **Loading Settings** tab
- Right-click on the Elastic-Plastic Block, and select Temperature from the context menu

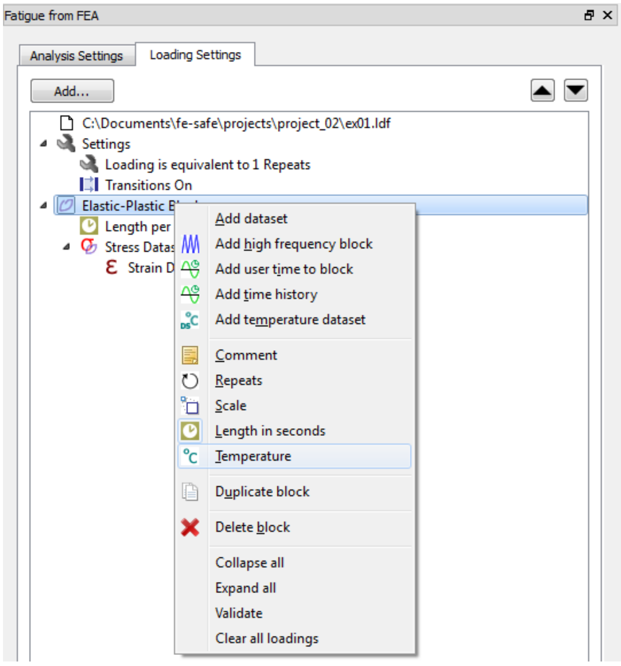


Figure 2-10

- Enter **100** in the text field

- The Loading Settings should now look as follows

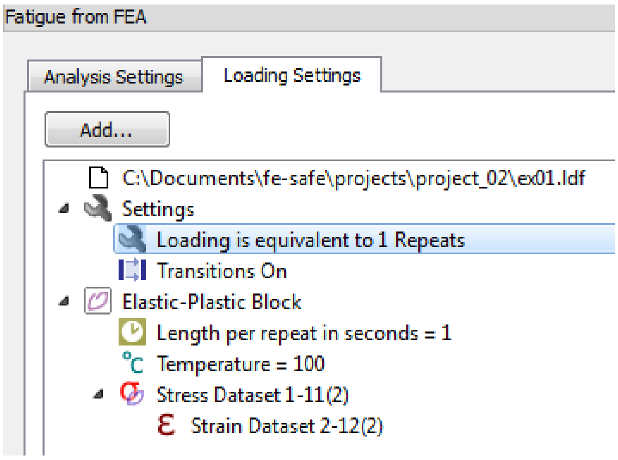


Figure 2-11

- *Step 2: Define the output file:*

When the FE model was loaded, the Output File was automatically defaulted to a standard file name in the Project Directory.

- in the **Fatigue from FEA** dialogue select the browse button to the right of the **Output File** field.
- Change the output filename to: **`TUT2Results_ex02.txt`**

Note: Since only one element will be analysed a text format such as *.txt or *.fer is used to reduce output file size. See the Appendices of the *fe-safe* User Manual for details of Output File fatigue results formats.

Step 3: Run the analysis:

fe-safe is now configured to run the analysis.

- Press the **Analyse** button. A summary of analysis parameters is displayed;
- Check that the analysis is configured as described above, and then click **Continue**.

As the analysis is being performed, the following information is written to the analysis log file.

```
   %      Time        Life-Repeats
100     0:00:00        774@2827.1            1 of 1
```

The analysis log file has the same file name as the output file, except that the extension is **`.log`**, for instance:

`<ResultsDir>\TUT2Results_ex01.log`

This information is also displayed in the **Message Log** window and includes

```
Summary
=======
          Worst Life-Repeats                    : 773.695
  at Element 2827.1
Analysis time                                   : 0:00:00.180
```

2.5 Exercise 3: Define ozone dependence with existing configurations

Objective:

This exercise is a continuation of Exercise 1. Continue using *fe-safe* without changing fatigue configurations.

This exercise will define an operating temperature for the entire model.

Method:

Step 1: Configure the plug-in Settings:

- In the **Fatigue from FEA** dialogue, identify the **Group Parameters** table
- double-click the **Algorithm** column header to open the in **Group Algorithm Selection** dialogue box for all groups;
- The **Group Algorithm Selection** dialogue will appear as shown;
- select the **Settings** button, next to the plug-in algorithm **(Surface)**;
- The **Edit Plug-in Settings** dialogue will appear as shown:

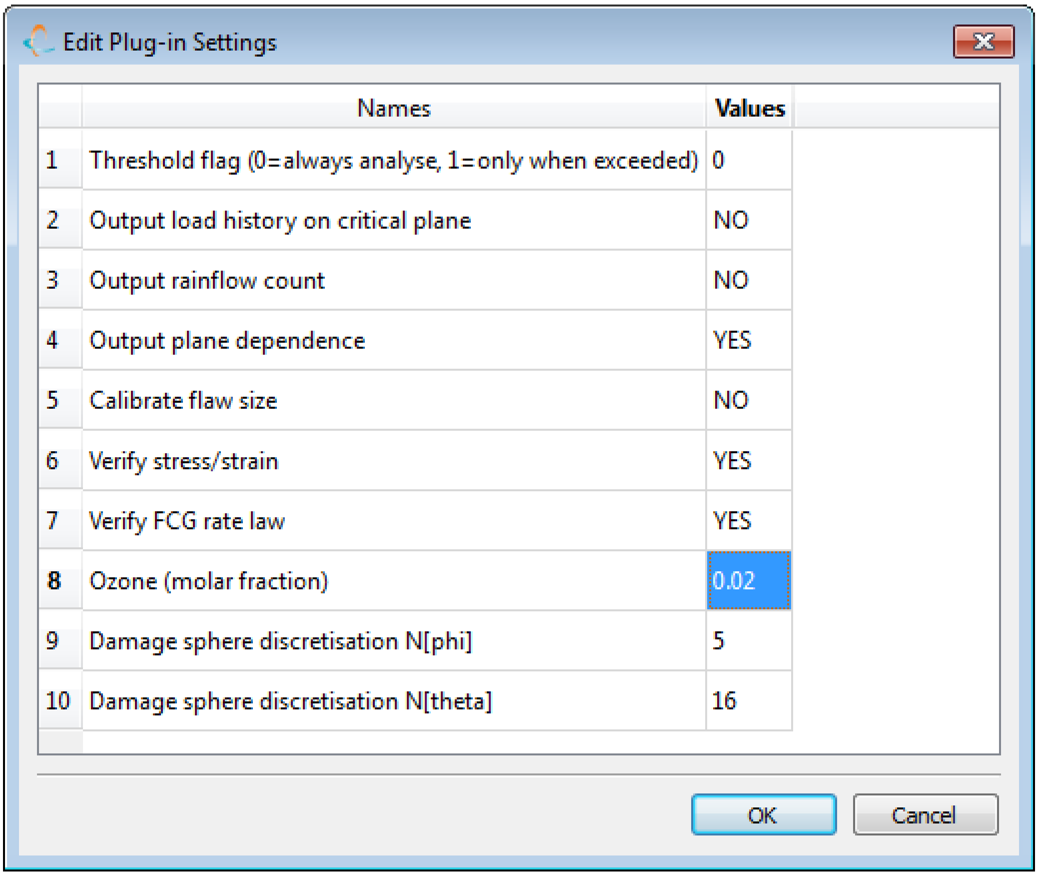


Figure 2-12 Configure plug-in Settings to specify Ozone concentration

- Edit the value of Setting 9 *Ozone (molar fraction)*: **0.02**
- Click **OK** to dismiss the **Edit Plug-in Settings** dialogue;

- Click **OK** to dismiss the **Group Algorithm Selection** dialogue;

- *Step 2: Define the output file:*

When the FE model was loaded, the Output File was automatically defaulted to a standard file name in the Project Directory.

- in the **Fatigue from FEA** dialogue select the browse button to the right of the **Output File** field.
- Change the output filename to: **TUT2Results_ex03.txt**

Note: Since only one element will be analysed a text format such as *.txt or *.fer is used to reduce output file size. See the Appendices of the *fe-safe* User Manual for details of Output File fatigue results formats.

Step 3: Run the analysis:

fe-safe is now configured to run the analysis.

- Press the **Analyse** button. A summary of analysis parameters is displayed;
- Check that the analysis is configured as described above, and then click **Continue**.

As the analysis is being performed, the following information is written to the analysis log file.

```
   %      Time        Life-Repeats
100    0:00:00         563@2827.1            1 of 1
```

The analysis log file has the same file name as the output file, except that the extension is **.log**, for instance:

<ResultsDir>\TUT2Results_ex01.log

This information is also displayed in the **Message Log** window and includes

```
Summary
=======
          Worst Life-Repeats                    : 563.308
  at Element 2827.1
Analysis time                                   : 0:00:00.180
```