

VPG/PrePost Tutorial

Fatigue Analysis using eta/VPG

Release Date: March, 2009

Engineering Technology Associates, Inc. assumes no liability or responsibility to any person or company for direct or indirect damages resulting from the use of any information contained herein.

Engineering Technology Associates, Inc., ETA, the ETA logo, and eta/VPG are the registered trademarks of Engineering Technology Associates, Inc. All other trademarks or names are the property of their respective owners.

Copyright 1998-2009 Engineering Technology Associates, Inc. All rights reserved.

ETA- Engineering Technology Associates, Inc.
1133 E. Maple, Suite 200
Troy, MI 48083

Phone: (248) 729-3010
Fax: (248) 729-3020
Support email: support@eta.com

VPG/PrePost Tutorial

This tutorial was created to familiarize eta/VPG users with the functions and techniques associated with the use of the fatigue analysis tools and the associated features within the VPG user environment. This tutorial will demonstrate the steps required to properly prepare a model results for fatigue analysis using VPG.

Background Information: Graphic User Interface

The graphic user interface consists of 6 areas within a single window: Main Menu, Drawing Window, View Options, Dialog Window and Display Options and the Top Menu Area.

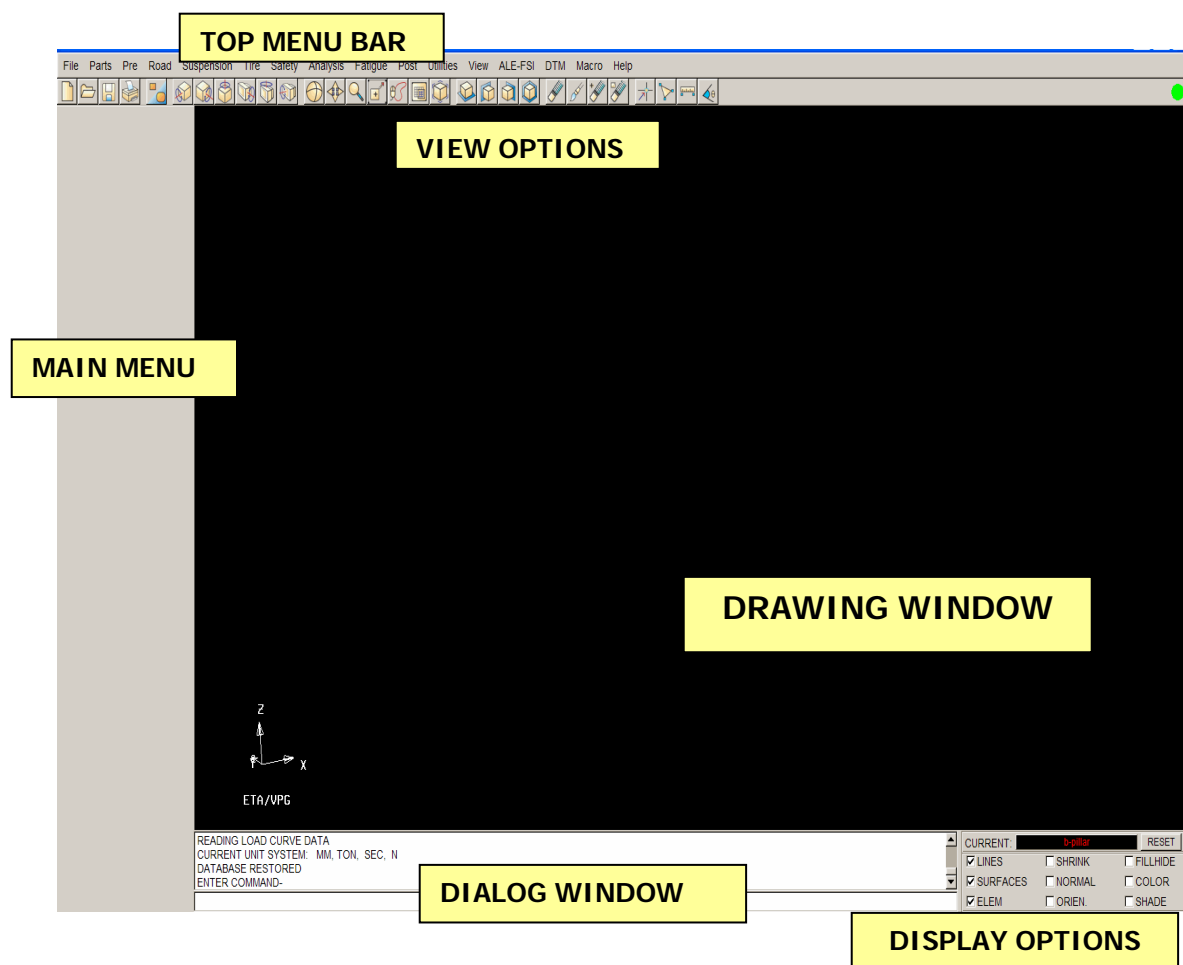


Figure 1: VPG User Interface

1. **MAIN MENU:** The Main menu options are displayed in this area after selections from the TOP MENU BAR
2. **DRAWING WINDOW**
Display graphic entities (CAD- points, lines, surfaces; FEA- nodes, elements).

3. VIEW OPTIONS
Part control, model positioning, program setup and utilities.
4. DIALOG AREA
Interface command prompt and scrollable command history.
5. DISPLAY OPTIONS
Toggle ON/OFF graphic entities to be displayed and controls display mode.
6. TOP MENU BAR
This menu lists all of the main menus as well as Utility menus and Help Menus

Function Keys

For quick access and to improve productivity, function keys are assigned for the most frequently used menus as follow:

| F1 | F2 | F3 | F4 | F5 | F6 |
|------------|-----------------|-------------|--------------|---------------|--------------|
| Clear Menu | Element Options | Import File | Line Options | Model Checker | Node Options |

| F7 | F8 | F9 |
|-----------------|---------------|-------------|
| Surface Options | Pre-Processor | Online Help |

NOTE: Please note that some images shown in this tutorial may make use of optional display features including background colors, shading, and element outlines. These may be controlled through the UTILITY Menu, and the SETUP options found there.

Introduction to Fatigue in eta/VPG

Structural durability analysis is an important part of finite element analyses since, the majority of the structural failures are caused by fatigue damage. It is even more important in recent decades since the automotive industry searches ways to reduce the sheet metal thickness, in order to achieve optimal structure designs and reduce the cost of materials.

For vehicle durability it has been historical practice for the maximum loads at the wheel centers to be estimated from derived formulas, which were then applied to the vehicle body for estimating the structural linear static responses. The stresses induced by the derived loads at the wheel centers were then used to evaluate the integrity of the structures for any further design improvement.

In recent years, with the introduction of VPG simulations, the linear static analysis approach was improved by implementing transient analysis. The stress history provided by the VPG approach did improve the fatigue damage evaluation, however, the element real-time stress history problem was not adequately addressed.

Current common practice of fatigue analysis is first to generate or obtain the road load data, then perform the stress analysis, and finally evaluate the structural fatigue life[1].

The road load data generated, when the prototype is not yet available, are often based on the assumptions that the vehicle structure and the suspension system consist of various rigid bodies. After obtaining the road load data, the linear elastic stress analysis is performed. These stresses are assumed to be the worst fatigue damage events and they are fully reversed under that loading condition. These assumptions limit the accuracy of the previous durability analysis methods.

The Virtual Proving Ground provides a means to simulate system behavior, using dynamic nonlinear analyses. This allows engineers to have a prediction of stress over a time period, which corresponds to a test or proving ground event. This then is the source of the dynamic stress which may be used in the calculation of fatigue life, much in the way that strain gage data may be used to evaluate a test event.

Fatigue Analysis In eta/VPG

Fatigue analysis in this example is based on the assumption that the most damage areas are experiencing uniaxial stress. Even though that the forces applied to a structure are very complex, stresses in the most critical areas are mainly uniaxial stress.

Von Mises stress at each time step is calculated from the stress tensor, which is available from the analysis. The elements stress history is then separated into individual cycles by means of rainflow cycle counting technique and the stress amplitude and the corresponding peak and valley stresses are then used to calculate element fatigue life.

The algorithm VPG uses in fatigue life prediction is the Smith-Watson-Topper equation, which considers the effect of mean stress. Since the assumption was made about the uniaxial stress, the fatigue life calculated would be in the conservative side.

The strain due to cyclic stress is determined from an incremental plasticity theory. The general form of total strain is

$$d\varepsilon = d\varepsilon^e + d\varepsilon^p$$

where $d\varepsilon$ is the total strain increment, $d\varepsilon^e$ and $d\varepsilon^p$ are the portions of elastic and plastic strain increments, respectively. The plastic strain increment can be decomposed into two yield domain and then we rewrite the total strain formula:

$$d\varepsilon = d\varepsilon^e + d\varepsilon^p = d\varepsilon^e + d\varepsilon^{y_1} + d\varepsilon^{y_2}$$

Based on the plastic flow equations, the plastic strain increments in the two domains can be written as follows:

$$d\varepsilon^{y_1} = \lambda^{y_1} G^{y_1}$$

$$d\varepsilon^{y_2} = \lambda^{y_2} G^{y_2}$$

where l_{y1} and l_{y2} are the magnitude scalars, G_{y1} and G_{y2} are the gradient vectors of the yield surfaces 1 and 2. The l_i can be derived based on assumption that the strain increment is normal to a smooth yield surface and the scalar product of stress increment and strain increment must be zero. The total incremental stress and strain relationship is :

$$d\sigma = (K^e + K^p)d\varepsilon$$

where K^e is the elastic stiffness matrix and K^p is the plastic reduction matrix. Knowing the strain, we can then calculate the fatigue life using the Smith-Watson-Topper (SWT) equation.

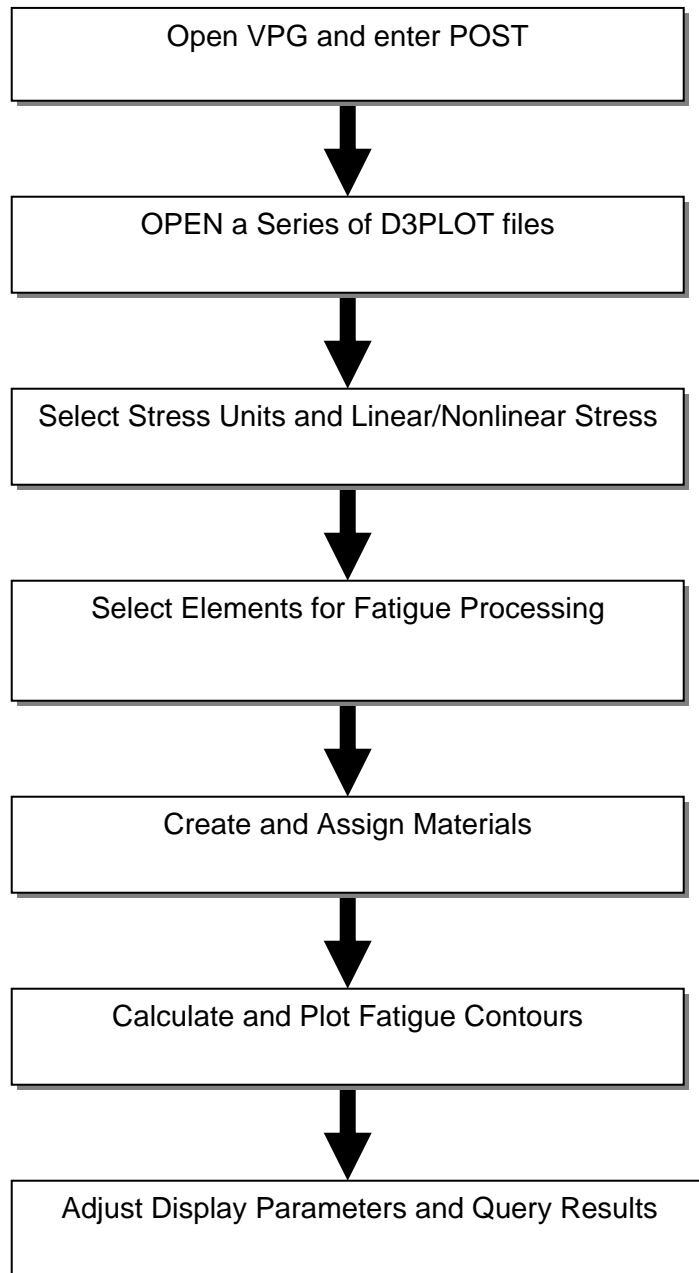
The SWT formula used for the mean stress effect, which is an influential factor of estimating fatigue life, is

$$\sigma_{\max} \varepsilon_a = \frac{\sigma_f'^2}{E} (2N_f)^{2b} + \varepsilon_f' \sigma_f' (2N_f)^{b+c}$$

where σ_{\max} : maximum stress,
 ε_a : amplitude of strain,
 b : fatigue strength exponent,
 c : fatigue ductility exponent,
 ε_f' : fatigue ductility coefficient,
 σ_f' : fatigue ductility coefficient,
 $2N_f$: reversals to failure (2 reversals = 1 cycle)

In the above equation, the σ_{\max} is used to correct the mean stress effect. True stress and strain are required in this equation and Glinka's energy density method is used to calculate them, if the stress and strain are obtained from linear elastic analysis.

Procedural Flowchart



Fatigue Analysis Example

The example that we will consider here is a suspension system, which is undergoing an enforced displacement and the wheel center, which is intended to simulate the displacement inputs of a pothole event.

The model is constructed using solid, shell, beam and spring elements. The structural components, the upper control arm, the lower control arm and the knuckle are modeled using solid and shell elements. Bushings, springs and a shock absorber are included in the model and are used to connect the various components and accurately transmit the forces between the components.

Materials have been defined for the various components. In our case, we have defined the upper control arm and knuckle as RIGID materials, hence they will have no stresses as a result of the simulation. The lower control arm is modeled as a steel material, and has the flexibility of the component included in the dynamic analysis. This component will be the focus of our fatigue analysis.

Together, these components form a system which allows for the component interaction and transmits forces amongst the various components.

Suspension System Fatigue Analysis: Step-by-Step Tutorial

1. Opening a New Database

1. Open any VPG database. For fatigue calculations, VPG does not require that the model in the database be processed for fatigue analysis.
 - a. Start VPG by selecting the VPG Icon from the Desktop, or by entering the command that executes VPG (UNIX/ LINUX).
 - b. Open any database. For our example we will create a new file called "*suspfat.vpg*".
 - c. VPG will prompt "CREATE NEW FILE", select **YES** from the menu.
 - d. A database format must be specified for the new VPG database. For convenience our example we will select **LS-DYNA** from the menu.
 - e. As with any other LS-DYNA model database, the user must select a UNIT SYSTEM. While this is unimportant for our case here, we will select **MM, TON, SEC, N**



Figure 1: eta/VPG Top Menu Bar

2. Entering the Fatigue Post Processing Tools

Select **POST** from the VPG Main Menu, which is located across the top of the Window.

3. Loading Stress Results

VPG now needs to import the model results (i.e. D3PLOT, ELOUT or PUNCH files). Navigate to the results directory on your computer, and select the result file name. For the case of D3PLOT files, the user specifies the first D3PLOT, and VPG will import all associated D3PLOT files in that directory. This operation is shown in Figure 2.

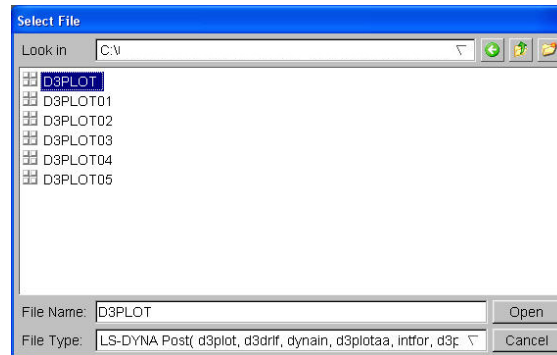


Figure 2: File Open Dialogue Window

a. From the **FILE** menu, select the **OPEN** command. Use the File Open Window. Select the **D3PLOT** files associated with the simulation you would like to process.

b. Your model geometry will now appear. This data was also contained in the D3PLOT file (see Figure 3)

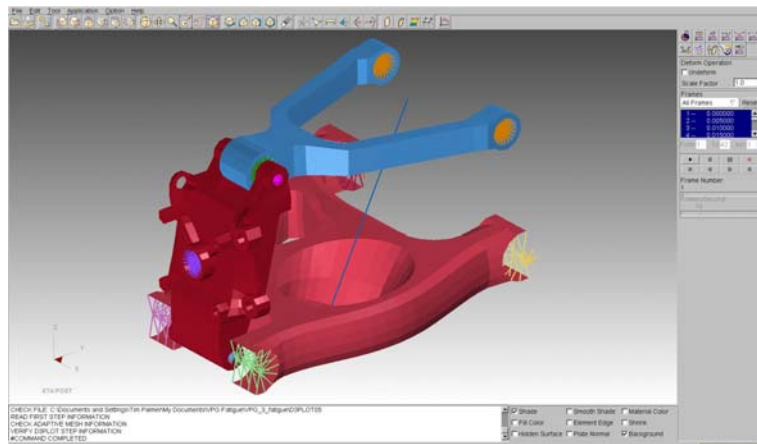


Figure 3: Model Display

To enter the Fatigue post processing menus, the user may select the Fatigue Icon on the upper right menu, or select the **APPLICATIONS** and the **FATIGUE** option. The methods for accessing the Fatigue Menus are shown in Figure 4.

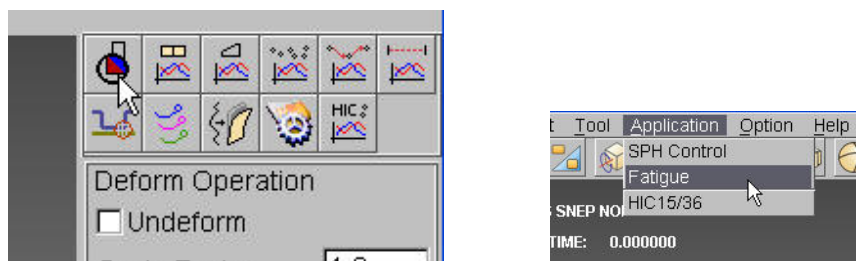


Figure 4: Options for Accessing Fatigue Menus

4. Stress Units, Type

After selecting the Fatigue Application icon, a **FATIGUE OPERATION** window will open (Figure 5). This window allows the user to specify the units of stress which will be used on the calculation, the type of input file which will be used, the type of stresses (linear or nonlinear), and the material type which will be used in the fatigue analysis.

a. The user should select **MPA** units for this example. The original model was set up a Newtons for the force units and millimeters as the distance units therefore our stresses would be N/mm², or MPA.

b. The stress results we will use in this example are from a **D3PLOT** file. This is the output file from an LS-DYNA analysis. Other options are an ELOUT file, which can provide more detailed stress information about our model. Typically the sampling rate or output frequency of a D3PLOT file is much lower than that of the ELOUT file. However, the ELOUT file is typically used for smaller models or portions of a large model.

c. The stresses we obtained in our simulation are from a nonlinear analysis therefore we will select the **TRUE STRESS** option.

5. Element Selection

The elements to be considered for the fatigue calculation must be selected by the user.

a. The user must now select the parts or elements that are to be included in the fatigue calculation. Select **ELEMENT PICKING** from the Menu.

b. Options by which to select elements will be displayed. We want to calculate only the fatigue damage to the lower control arm, so the user should select the elements of the lower control arm part using a window or multi-point region selection method.

To complete the selection the user selects **EXIT** and **DONE**.

If the part contains any beam elements or elements for which there are no stress results (for example rigids), VPG will display error messages, alerting the user to this situation.

6. Material Creation and Assignment

VPG needs to have specific material properties defined for the components in the model, to interpret the stress results for fatigue analysis. The data for several common materials are included in a library for VPG users. Additional materials can be added to the fatigue analysis and applied to the model.

a. Select **MATERIAL DEFINITION** from the Fatigue Operation window. A window will appear allowing the user to define and assign materials to the model. The Fatigue Material Window is shown in Figure 6.

Our component is a 1008 Steel material. Select **SAE-1008** from the CREATE drop down menu. The materials found in this section are typical automotive

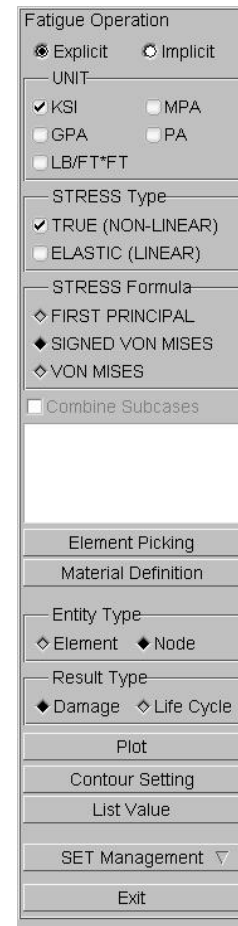


Figure 5: Fatigue Operations Menu

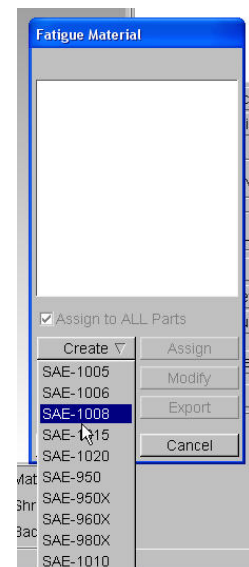


Figure 6: Material Creation/Assignment Window

materials, for which VPG provides the necessary fatigue parameters. The user may create additional material models that can be defined in subsequent steps.

b. A material parameter window will appear, displaying the material fatigue coefficients and exponents for that particular material. The user may modify these parameters at this time, if desired. To accept the parameters, select **OK**. This will return the user to the Fatigue Material window. An example of this window is shown in Figure 7.

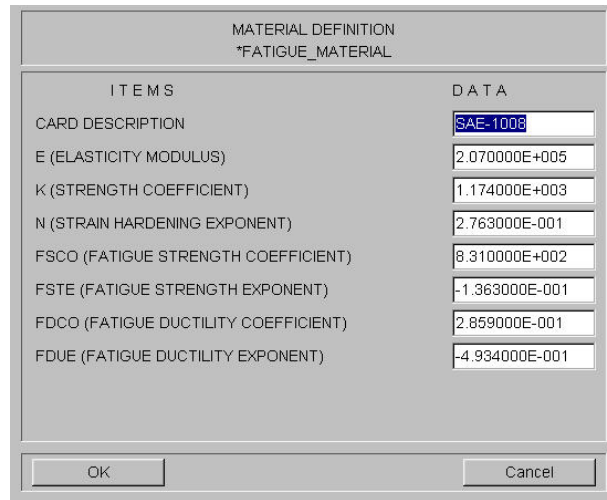


Figure 7: Material Property Definitions

In our example model only a single material is defined. VPG assumes that the entire model will use this material definition for the fatigue calculation. No ASSIGN operation is needed.

Note: If additional materials are defined, the user may selectively assign the materials to parts or elements of the model. If this is the case, see steps c., d., e. and f. below.

c. An additional material may be created by following steps a. and b., above.

d. To assign a material to a selected set of elements, highlight one of the newly-created materials from the list displayed in the Fatigue Material window. If you wish to assign that material to all elements of the model, select the 'Assign to ALL Parts option'.

e. Select the ASSIGN button from the options on the window. Element selection options will be displayed. By default, the selection method is by parts.

f. When the material assignment is completed the user selects **EXIT**.

Select **OK** on the Fatigue Material window to complete this operation and return to the Fatigue Operation menu.

7. Fatigue Calculation

Once VPG has the necessary information to calculate a fatigue life (stress, material type, and material properties), the fatigue damage calculation may be performed.

a. the user may select the manner in which the fatigue results are displayed. By selecting **NODE**, the results will be averaged over the nodes of the element. This results in a smoother contour. If the **ELEMENT** option is selected the fatigue results will be displayed by element. See Figure 8 to view the options available.

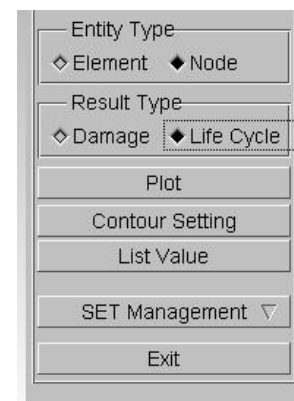


Figure 8: Fatigue Display Options

The user may also select the quantity to display, **DAMAGE** (per application of the load), or **LIFECYCLE** (cycles to failure).

Select **Node** and **Life Cycle** options

b. To start the calculation, the user then selects **PLOT** from the **FATIGUE OPERATION** window.

c. VPG now executes the fatigue calculation process in the background. The user must wait for the completion of the calculations.

If additional load cases are available for processing, the current fatigue results can be stored and combined using the SET Management tools. This feature is not discussed in this tutorial.

8. Displaying the Results

VPG will automatically display the results upon completion of the calculation. Any elements which do not have fatigue results, or were not selected for the calculation, are shown with the element in a gray color. The user may modify the display to provide a better visualization of the results.

Based on the options selected for our example, the results are displayed as Nodal values of cycles to failure. This is shown in Figure 9.

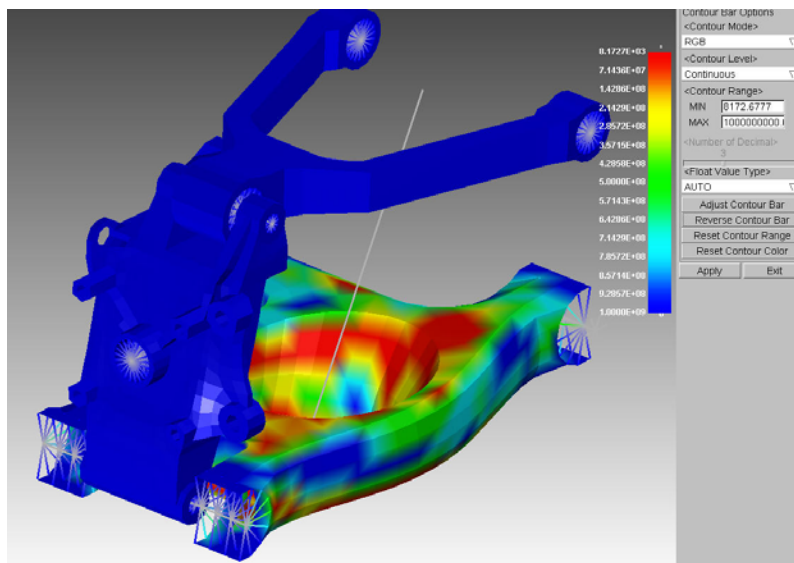


Figure 9: Fatigue Results Contour Plot

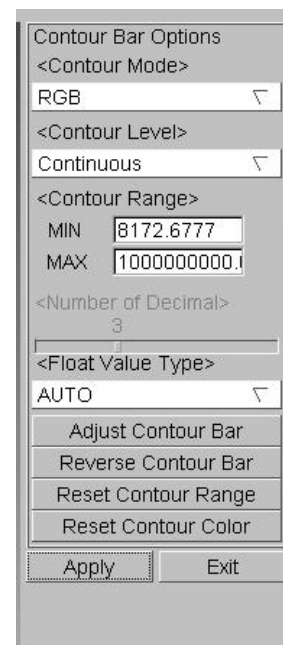


Figure 10: Contour Options

a. To modify the contour display, the user may select CONTOUR SETTING from the FATIGUE OPERATION window. The user can select the number of contours, the contour ranges, the contour colors and reverse the contours, such that red is referenced to the lowest life, and blue to the highest life areas. The Contour Setting options are shown in Figure 10.

These settings are applied to the model by selecting the APPLY button. The Contour Setting window will close and the user will return to the Fatigue Operation window when the EXIT button is selected.

9. Additional Display Options

The user can query the results by selecting the LIST VALUE button on the FATIGUE OPERATIONS window. This feature allows the user to select a set of nodes and list the values of damage or lifecycles in

a table, which will be displayed along with the fatigue contour plot. An example of this type of display is shown in Figure 11, below.

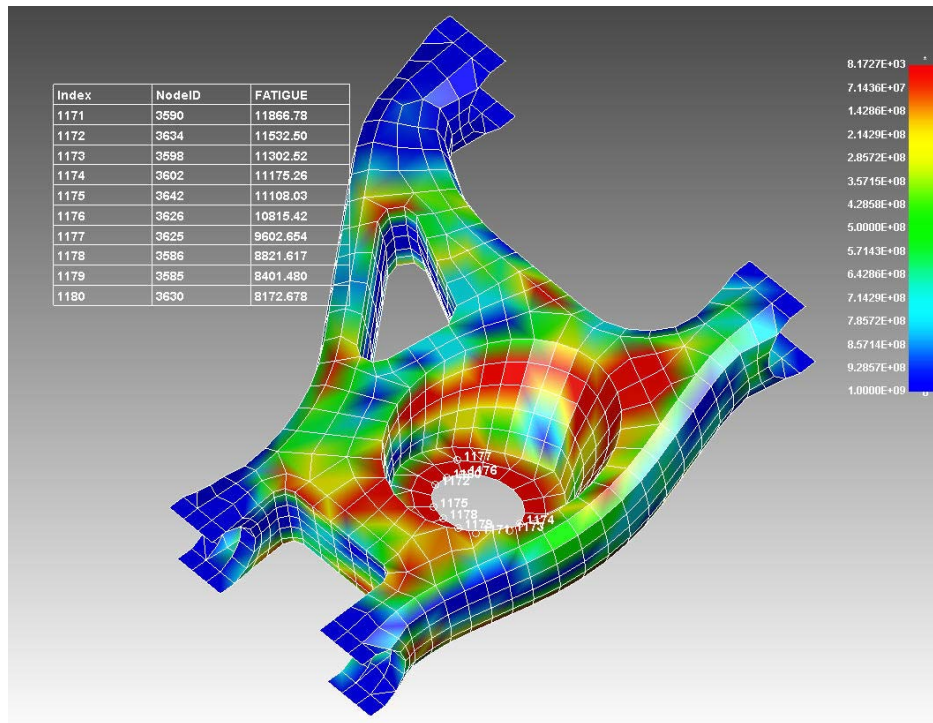


Figure 11: Contour Plot with Lifecycle Values shown in Table Format

Conclusion

The calculation of fatigue life using various forms of stress data obtained from a simulation may be performed using eta/VPG 3.4. The general process by which the fatigue life can be calculated have been presented.

Various options for the multiple event fatigue and data conversion options are discussed in the VPG 3.4 Manual. Additional information may be obtained by contacting support@eta.com.