FEMdesigner™ Quick Start Guide

Welcome to the FEMdesigner community! To get started, there are a few simple steps to start performing sophisticated analyses on your 3D CAD models.

Start the FEMdesigner Software
> Depending on the options you selected during the installation, there are two ways to launch the software:
> - Program group under the "Start" button
>   - Go to the FEMdesigner program group and click the "FEMdesigner" launch icon
> - Desktop shortcut
>   - Double-click the FEMdesigner shortcut icon on your Windows desktop

If you see the registration screen…

it means your copy of FEMdesigner is not yet registered. If you have purchased FEMdesigner, this dialog box tells you how many days you have left to obtain your Registration Certificate and enter it into the software before the software stops working. If you have not purchased, it tells you how many days are left in your free trial period.
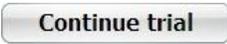
After clicking **Continue trial**, or entering your License Code and clicking **Register** and receiving your registration certificate, you'll come to the main program screen:
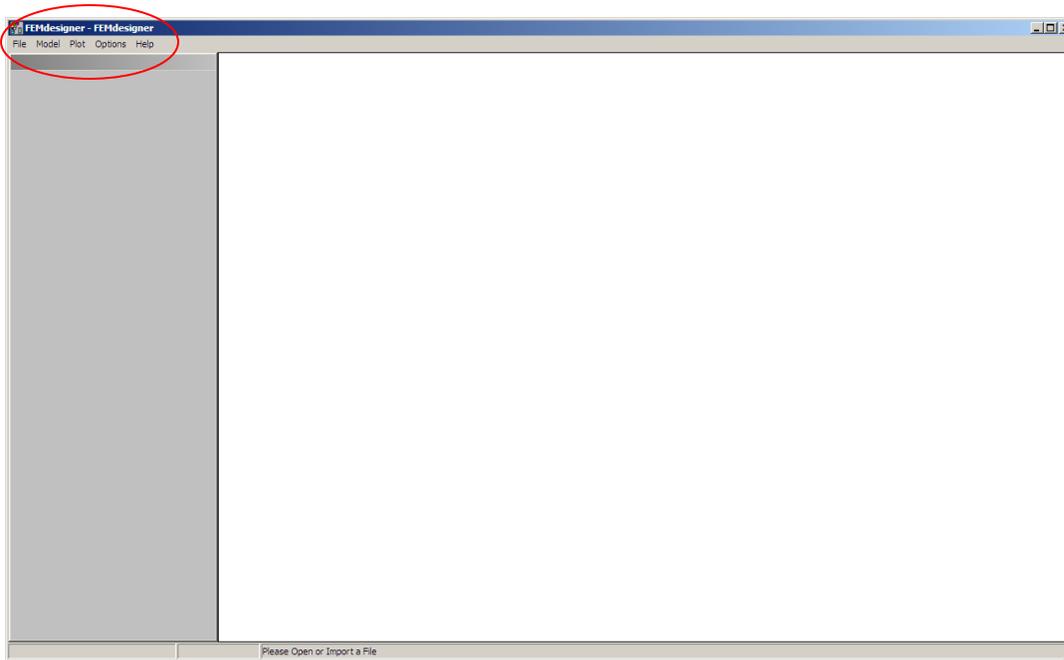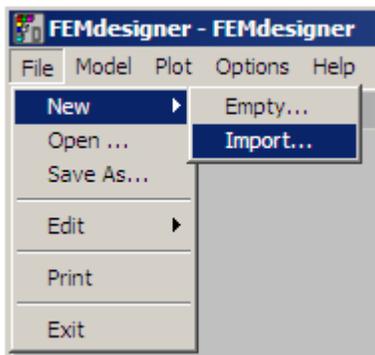
In general, the workflow for creating analyses in FEMdesigner is this:

1. **Import a Model**
2. **Assign Material**
3. **Establish Boundary Conditions**
4. **Apply Loads**
5. **Build Mesh**
6. **Solve Model**
7. **Visualize Results**

1. Import a Model

At the top left of this screen (circled in red above), click "File" then "New" then "Import…"
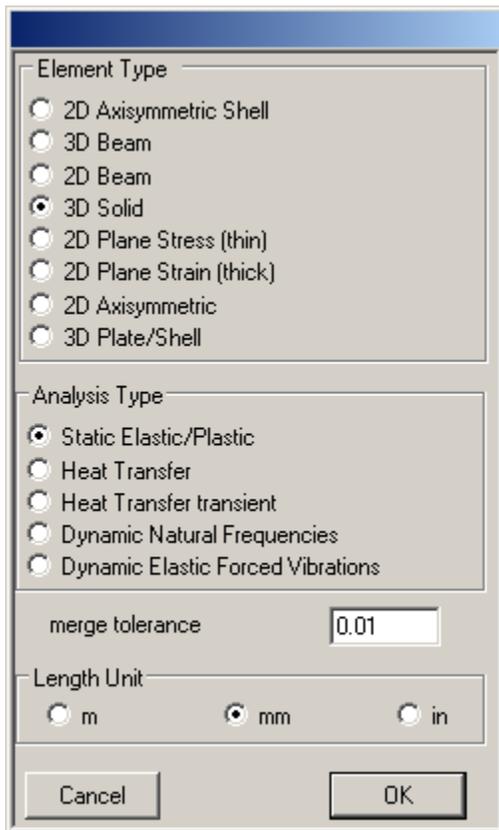

Navigate to one of the sample files provided with FEMdesigner, or an IGES or STEP file of your own to perform an analysis on. In this example we are using the file 'link.igs' in the FEMdesigner 'samples' directory. If you have already performed a FEMdesigner analysis on the file you select, you will be asked to overwrite the existing .FEM (FEMdesigner loadcase) file.

C:\Program Files\FEMdesigner\Samples\link.fem

? overwrite old file?

| Yes | No |

Click **Yes**.
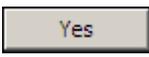
Now you will have some file import options to heal any mismatched points and edges. The following dialog box will appear:

Element Type

- ○ 2D Axisymmetric Shell
- ○ 3D Beam
- ○ 2D Beam
- ● 3D Solid
- ○ 2D Plane Stress (thin)
- ○ 2D Plane Strain (thick)
- ○ 2D Axisymmetric
- ○ 3D Plate/Shell

Analysis Type

- ● Static Elastic/Plastic
- ○ Heat Transfer
- ○ Heat Transfer transient
- ○ Dynamic Natural Frequencies
- ○ Dynamic Elastic Forced Vibrations

merge tolerance   `0.01`

Length Unit

- ○ m   ● mm   ○ in

| Cancel | OK |

Select the length units that were used for the model design, enter an appropriate merge tolerance, and click OK.

In this case, the part was designed in millimeters, and a reasonable merge tolerance is 0.01 mm, as shown here.

The merge tolerance should be slightly smaller than the smallest feature on the model, which will allow healing of mismatched points but won't consume small features.
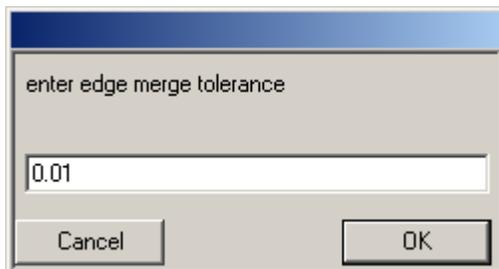
The next screen is for healing mismatched edges, and the same rule applies. Enter an appropriate merge tolerance and click 'OK'.

enter edge merge tolerance

`0.01`

| Cancel | OK |

The selected file will be imported and any mismatched points and/or edges will be healed into a 'watertight' solid model.

In this case, we imported the file 'link.igs' from the Samples directory provided with FEMdesigner.
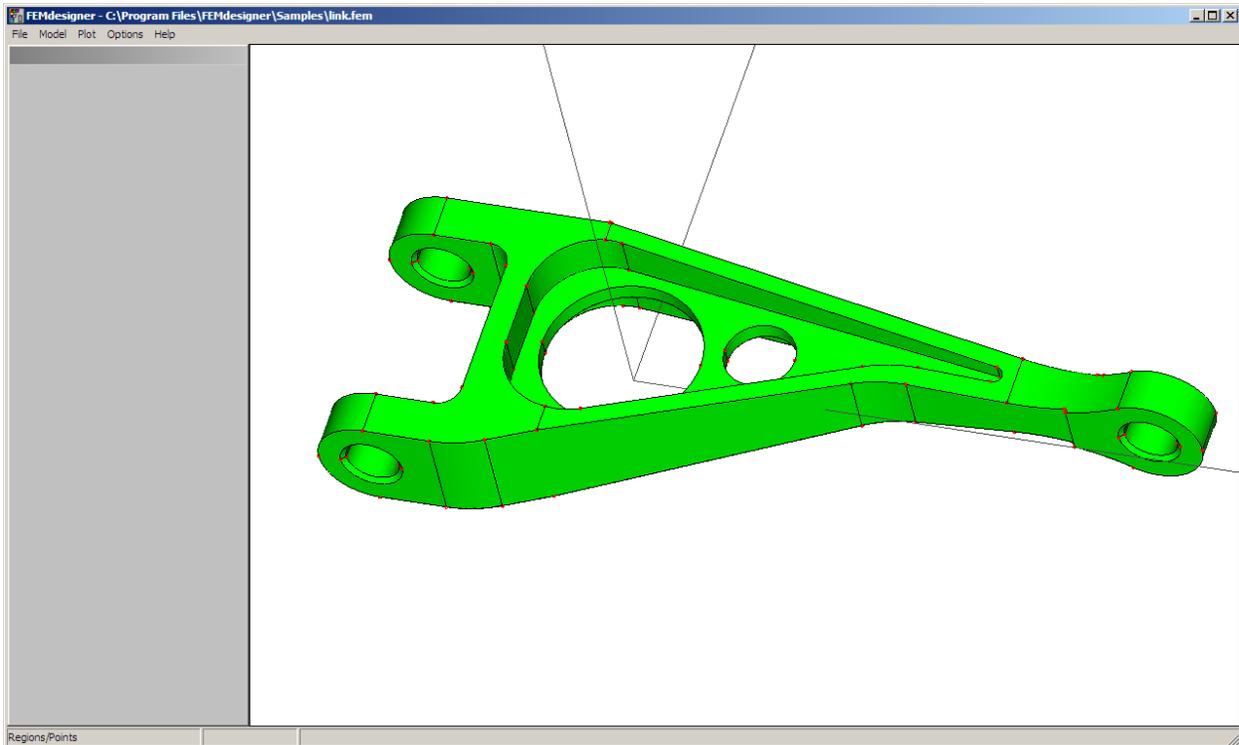
View manipulation: To obtain a better view of the part, dynamically rotate on screen by holding the right mouse button down and moving to rotate the view to the desired perspective. Zooming is done by simply drawing a window around the whole part, or the area of the part that you want to examine more closely. To quickly zoom out to see the entire model on screen, simply single-click anywhere in the model window.

2. Assign Material

Select 'Model' then 'Materials list
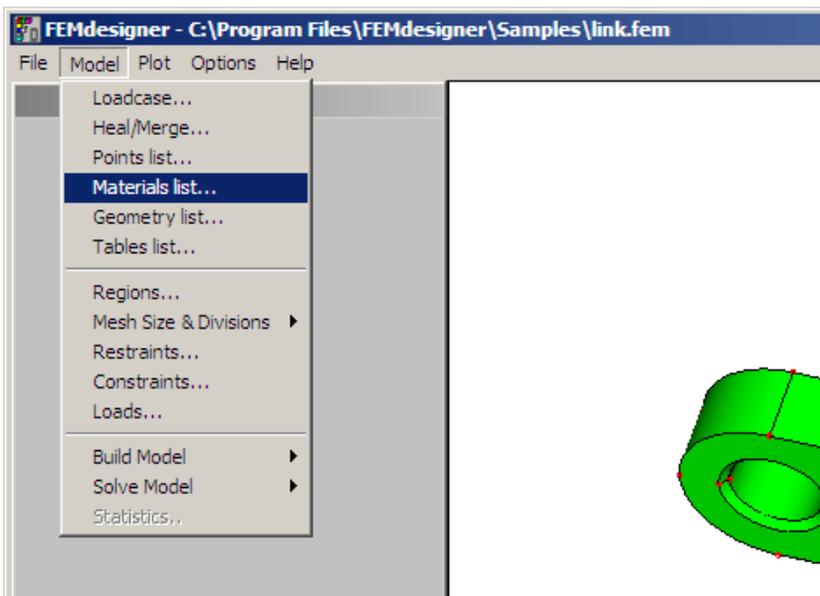
The following dialog box is displayed:
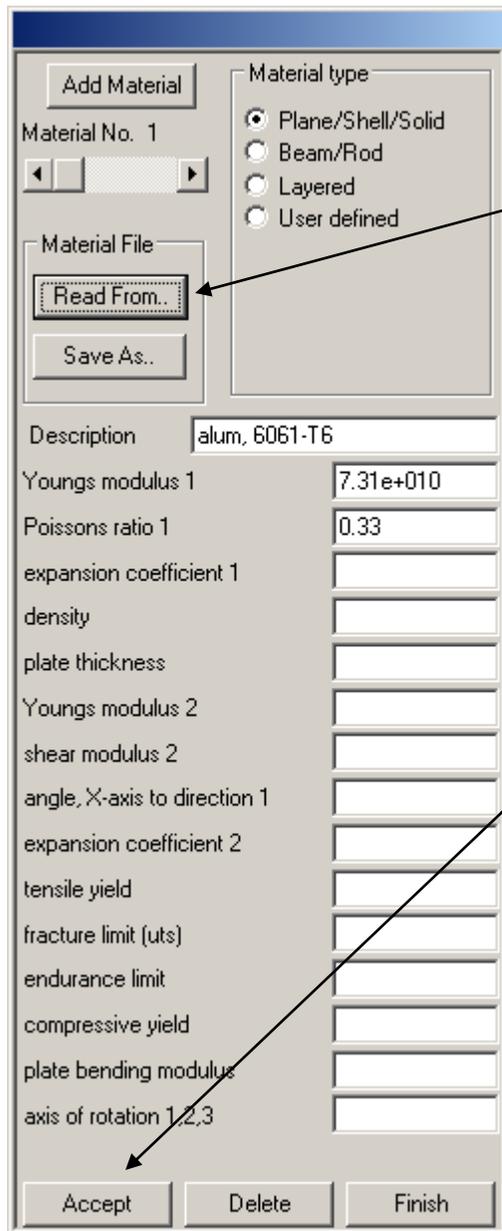
Select the 'Read From..' button and navigate to the 'materials/mechanical' directory supplied with FEMdesigner.

Select any material in the list. In this example we'll use 6061 T6 Aluminum, represented by the text file 'alum_6061-T6.txt'

Note that the only required material properties for an accurate elastic mechanical analysis are the Young's modulus (the slope of the elastic portion of the stress vs. strain curve), and the Poisson's ratio (ratio of transverse strain to axial strain).

Click the 'Accept' button and the material is assigned to this part for this analysis. Then click 'Finish'.

3. Establish Boundary Conditions

The term 'boundary conditions' is a catchall to refer to the conditions under which the part will be stressed in the real world. This includes any points, edges or faces that will be fixed, or if not fixed, any limits on motion or displacement such as sliding joints, surface contact, etc. In this example, we will apply minimal boundary conditions to simply illustrate their use. These might not reflect real-world conditions for this part.
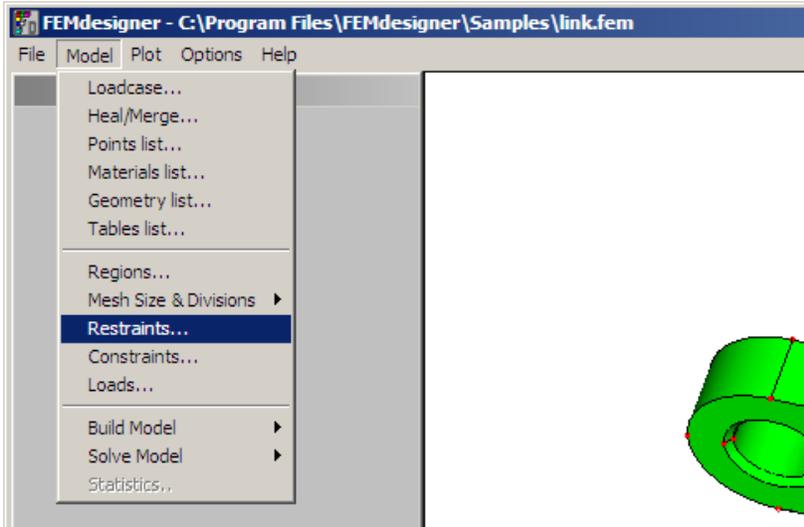
From the 'Model' menu, select 'Restraints.'

This will present the Restraints dialog box, shown below left.

Select the 'X' 'Y' and 'Z' checkboxes under 'Fixities'. This will fix, or create a "ground" or unmovable face. More than one face can be grounded.

Next, we need to select the faces to fix.

IMPORTANT - Selecting items on your model always happens the same way:

1. Hold down the 'Shift' key
2. Draw a window around the points, edges, or faces that you wish to select

Note: If you don't press the 'Shift' key first, you will simply create a zoom window. Also, items to be selected must be entirely within the selection window. If this requires selecting unwanted items, we can remove these after selection.
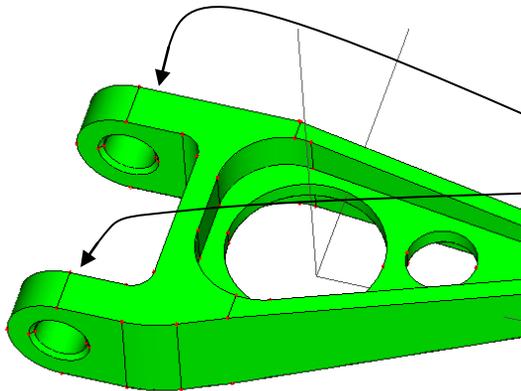

From our current view, let's say we want to fix the faces of the two axially aligned holes that are facing away from us.
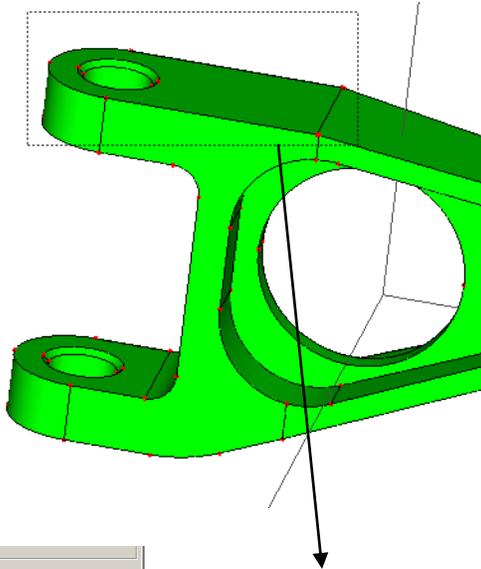
Right-click rotate the view until we can see those faces, as shown here.

Select the upper face with Shift-window as shown. It is best to use the smallest window that will select the items you are interested in.

Any face that was entirely within your selection window will be highlighted, as seen at left. A new selection dialog box will appear in the lower left, indicating that you have selected some items. Now we have to tell the software which items we want to keep or delete.
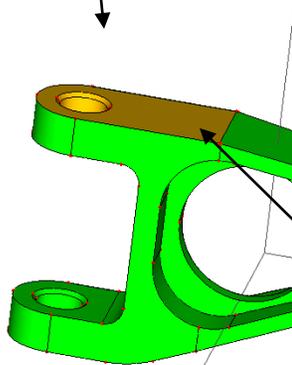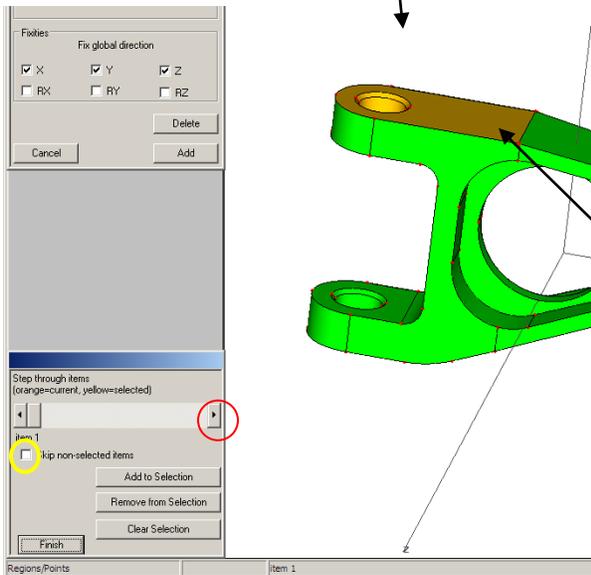
Check the 'Skip non-selected items' checkbox (circled in yellow), and click the right-hand scrolling arrow once (circled in red). Clicking this arrow will advance through all the faces that have been selected, one at a time, changing their highlight color to a darker orange color.

Since we want to fix only one of these faces, we need to remove all other faces that have been selected. With each click of the scroll arrow (including this first one), select "Remove from Selection" until the face we want to fix is highlighted in orange, then select "Add to Selection." Continue scrolling through the faces until all faces have been either removed or added, and there is only this single face highlighted. To test if there is only this single face, click both the left and right scrolling arrows, and if the item number doesn't change, there is only one face in the selection.
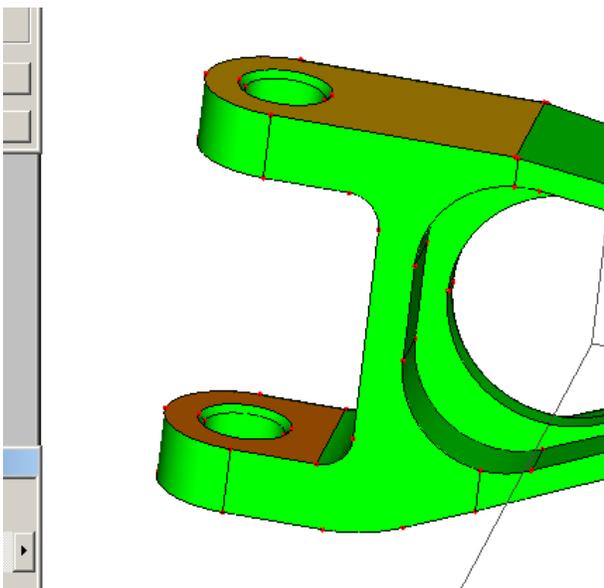
While the selection dialog remains open, repeat this "shift-window" procedure for the face on the lower hole to select the two faces we want to fix, shown at left.
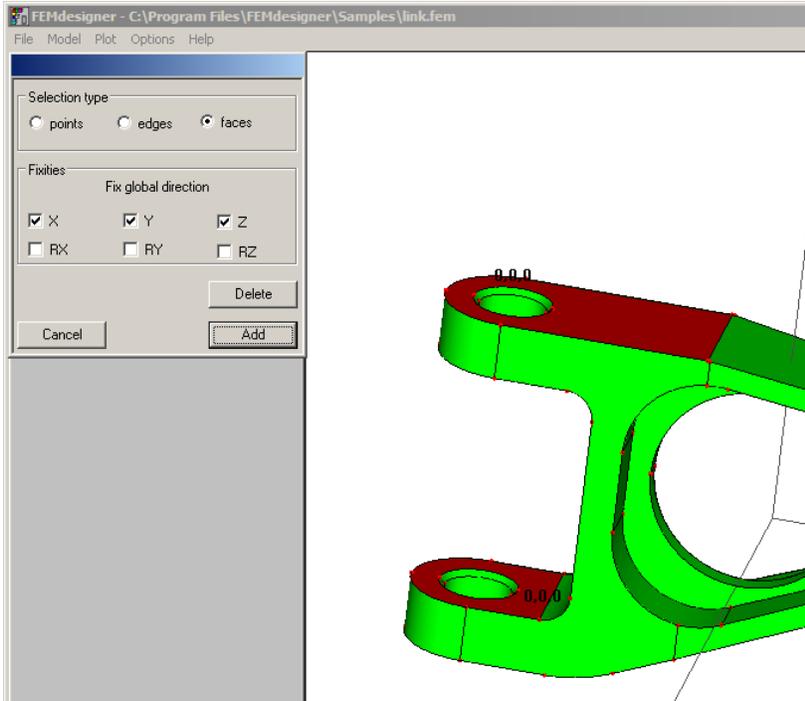
With the selection dialog box still open below, clicking "Add" in the restraints dialog box will fix the selected faces, indicated by those faces highlighting in red and displaying the "0,0,0" notation on each face.

Clicking "Cancel" dismisses the restraints dialog, but don't worry, it does NOT cancel the boundary conditions you just placed.

4. Apply Loads

Applying loads will simulate the loading conditions in the real world. Factors of safety and other loading factors that will affect the performance of the final design should be included in the loading conditions that are modeled here. Additionally, Finite Element Analysis (FEA) is not an alternative to real-world testing, but it will help to shorten the design process by enabling the designer to create higher quality designs faster, and reducing the number of parts that will be required for real-world testing.
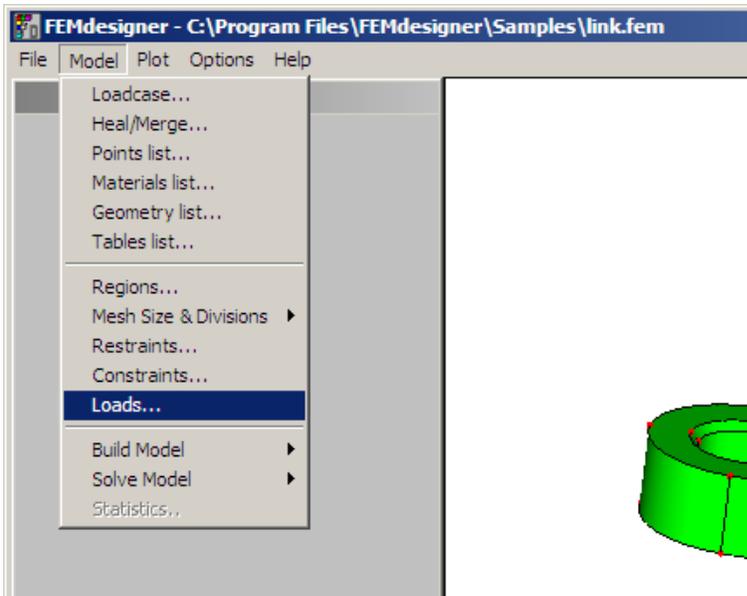
From the Model menu, select "Loads…"

This will present the Load dialog box, shown below.
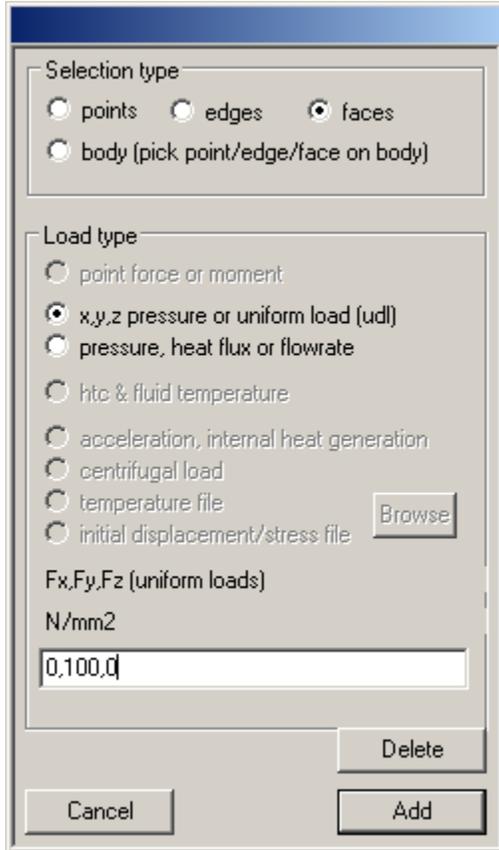
For our mechanical elastic analysis, "Selection type" should be 'faces' and "Load type" should be 'x,y,z pressure or uniform load (udl)'.

We will apply 100 MPa (Newtons/mm^2) in the positive 'Y' direction as a uniform pressure load on the face(s) we select. Enter into the text box at the bottom '0,100,0'.

Had we selected 'inches' as our import unit for the IGES file, this dialog would present us with pounds per square inch, or psi, as the pressure unit.

Now we select the face(s) to which we want to apply this load. Same as before, right-click rotate the view until we see the faces we want, then use 'Shift-window' to select, and remove unwanted faces with the selection dialog box scroll bar, making sure to add the face(s) we want with the 'Add to selection' button.

In this case, we want to load the single hole on the other end of the part, the face that is facing the opposite direction as the faces we just fixed.
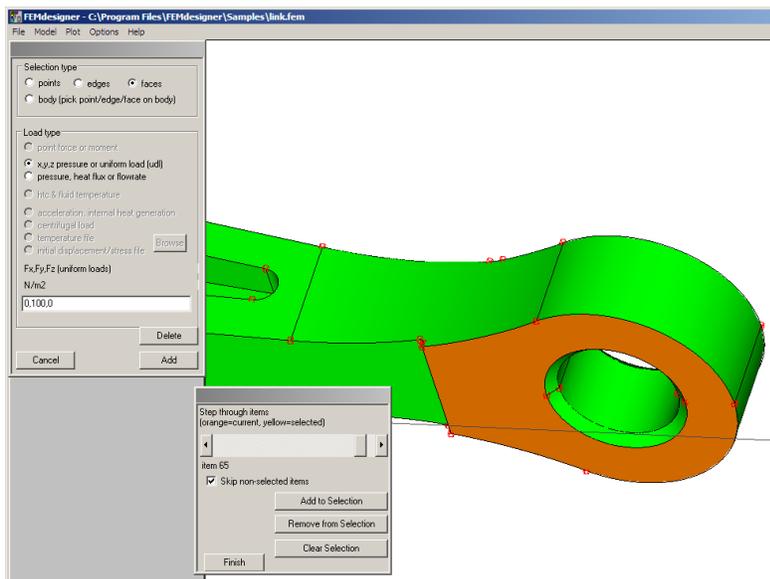

When you have the selection set to the single face shown at left, click the 'Add' button in the Load dialog to apply the load to this face. Selecting 'Cancel' dismisses the dialog box, but retains any loads you have applied.

All we have left to do is to build the mesh and run the solver.

5. Build the Mesh

This is where we actually create the finite elements that become the basis for the analysis. The mesher basically covers the surface of the part with interlocking triangles, and then fills the

volume of the part with interlocking triangular pyramids, or tetrahedrons ('tets' for short). From these elements, a system of equations is derived and solved using numerical analysis.
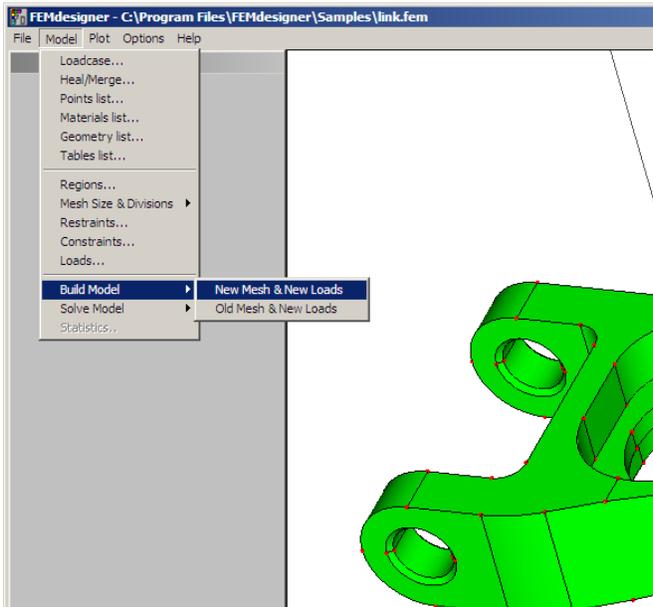
Before building the mesh, left-click anywhere in the model window to zoom out displaying the whole model on screen, then zoom in with a window to fill the screen with the model.

From the Model menu, select 'Build Model' then 'New Mesh & New Loads'.
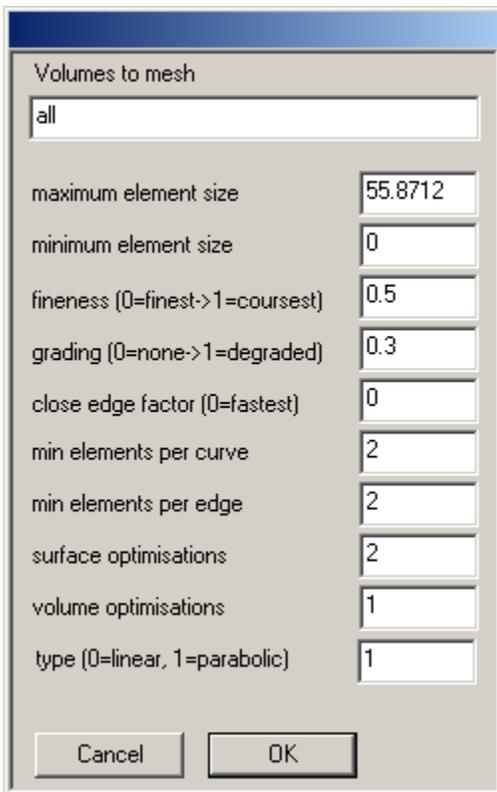

This brings up the mesh dialog box, with pre-computed defaults displayed. While we could change any of these meshing parameters if needed, there is no need to change any of the defaults for this analysis, so click 'OK'.

This will launch the meshing process, with progress being displayed on the status bar at the bottom of the program window.

The mesh will now be displayed on screen. It has boundary conditions and loads applied, and all that remains is to solve for stresses, displacements, etc.

6. Solve the Model
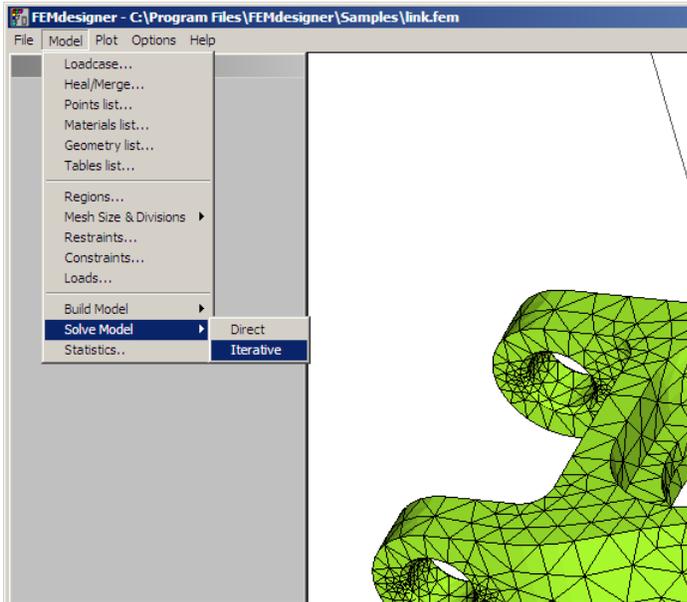
From the Model menu, select 'Solve Model' then 'Iterative' to start the solver.

Select the 'Elastic small displacements' radio button, which will bring up the 'convergence tolerance' dialog box.


This is where you can enter the allowable error in the solution. The default, which is 0.000001, is usually sufficient.

Click 'OK'.

Again, the status bar at the bottom of the window will update with messages about the progress of the solution, and when the solution is complete, a dialog box will be presented with the time taken for calculating the solution to a value within the convergence tolerance.

7. Visualize Results
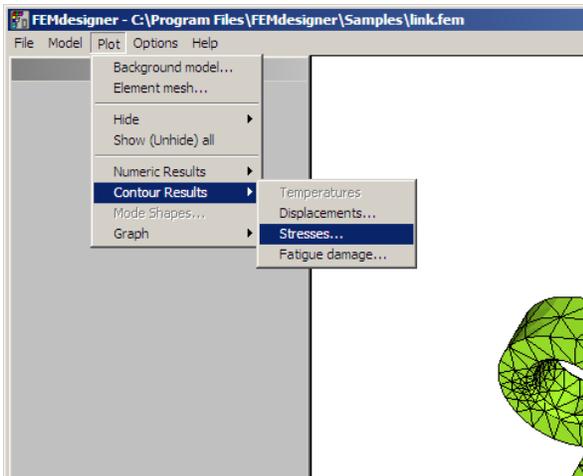
You may wonder why nothing has changed on the screen. Since there are a number different results that have been calculated, we have to choose which results to plot to the screen.

From the Plot menu, select 'Contour Results' then 'Stresses'.
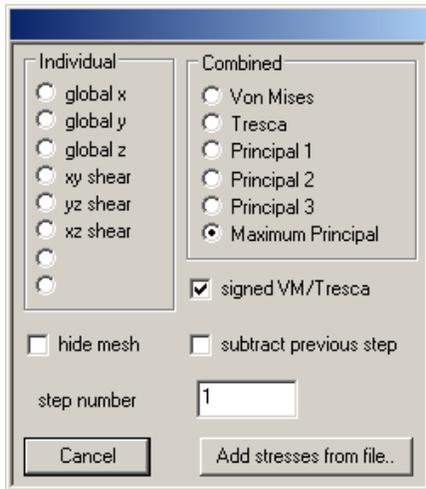
In the stress display dialog box, select 'Maximum Principal' and 'signed VM/Tresca' if not already selected by default.

The mesh will then be displayed with a colored stress plot, along with another dialog box with a color-coded legend describing the stress values, shown below.
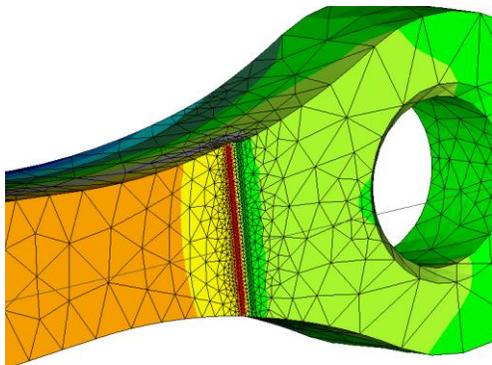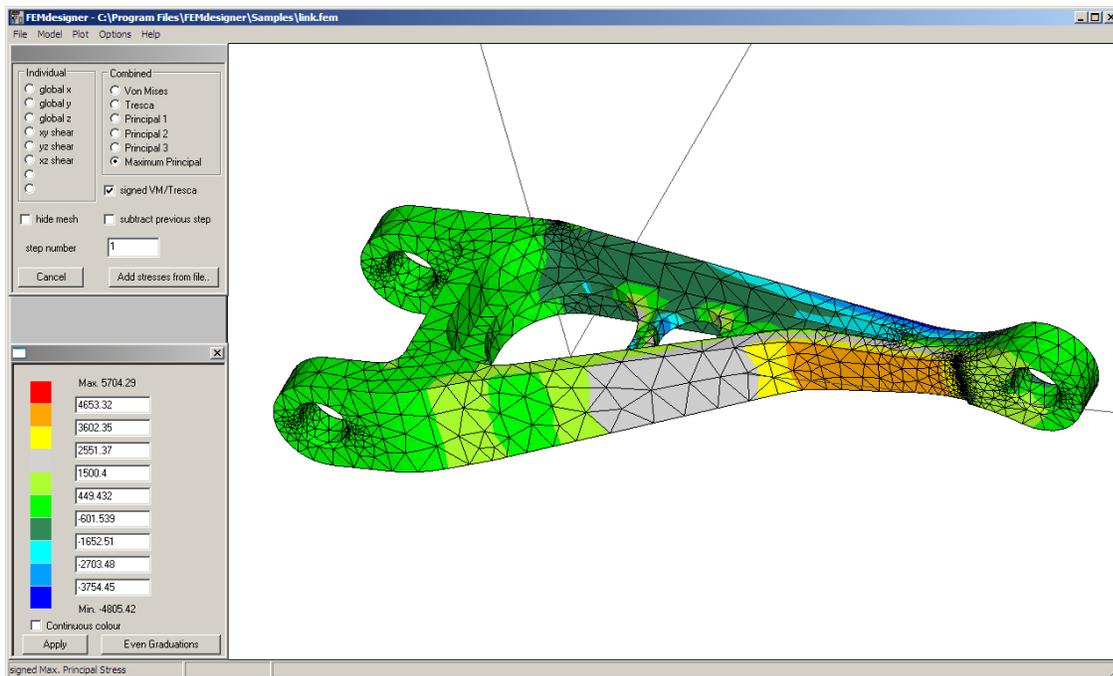
As one would expect for this analysis, the highest stresses occur at the thinnest cross-sections of the part, and maximum stress occurs on this thin cross-section at the sharp corner that is transverse to the direction of the applied load, shown at left.

Another calculated result that we can visualize is the displacement of the mesh, which corresponds to the deformed shape of the part under stress.
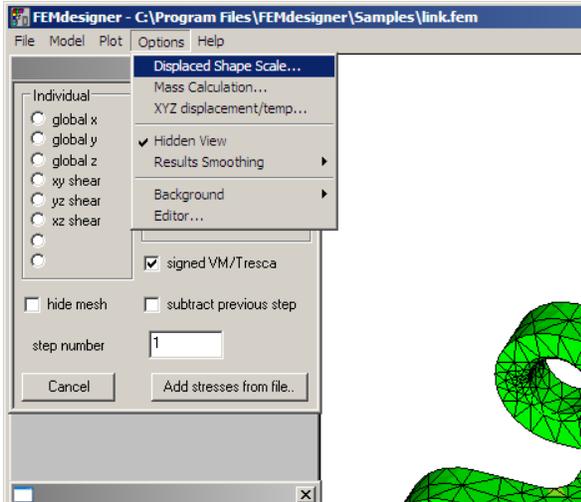
We can visualize the colored stress plot with the associated deformation at the same time.

From the Options menu, select 'Displaced Shape Scale…'

In the displacement scale dialog box that appears, set the Displacement Scale to '1' and the Animation Cycles to '3' and click 'OK'.
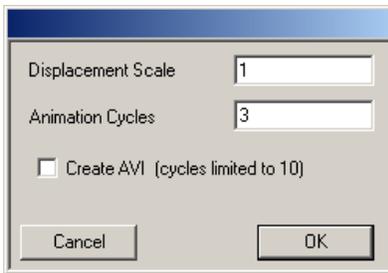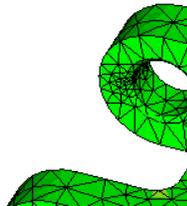
This will animate the deformation of the part from its at-rest position to its fully-deformed shape due to the load applied for three animation cycles, and then display the fully deformed shape, all while the displaying the color-coded stress plot.
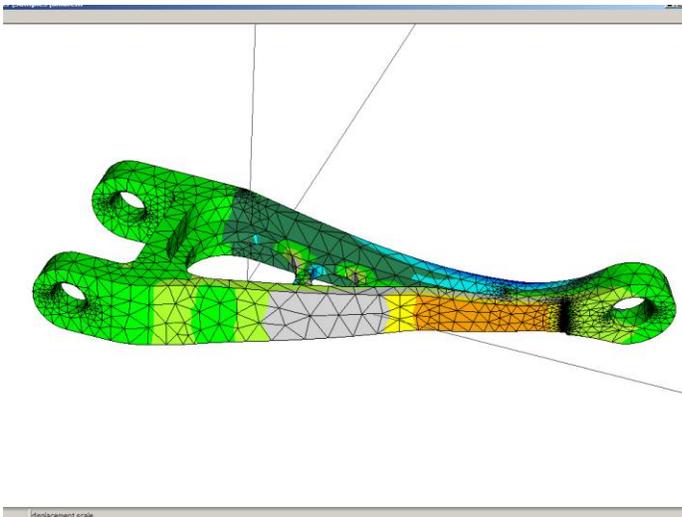
With the Displacement Scale set to '1', this represents the actual deformation that the part will experience under this load, which tells us that we may want to use a different material or a different design, or change the operating environment for this part.

For smaller displacements, we may have to increase the Displacement Scale in order to visualize the deformed shape.

The purpose of this Quick Start Guide has been to help you get started and create a quick analysis in order for you to get an idea of the general workflow for FEMdesigner. Obviously, there are many other options for analysis in FEMdesigner, and we encourage you to explore them.

Thanks for choosing FEMdesigner!