Getting Started with FEMdesigner AD
Cantilever Beam Tutorial

This tutorial is intended to teach the typical workflow required to create an analysis with FEMdesigner AD, the integrated FEA software for Alibre Design.

IMPORTANT: Before we continue, it is important to emphasize that FEA software results are never a substitute for real-world testing. FEA is a design optimization tool, not a substitute for testing. While FEA can give a good idea of how a part or assembly might behave in its operating environment, all designs should be fabricated and thoroughly tested in appropriate test environments with appropriate factors of safety, regardless of FEA results.

It is easy to get started learning FEMdesigner AD, with a menu structure that is logically organized to represent the process of creating an analysis. CAD-integrated FEA consists of a series of straightforward steps that are followed to obtain results, and in general these steps are:

1. <u>Create Geometry</u> – whether created directly in the CAD system or imported, we need geometry (usually 3D) to perform our analysis, which can be performed on individual parts or on whole assemblies.
2. <u>Create Boundary Conditions and Mesh</u> – "boundary conditions" is a catch-all term that refers to the materials, loads, restraints and other conditions that will govern the behavior of the geometry. In other words, these are the actual conditions that the geometry will experience. The "mesh" is a collection of individual finite elements connected together at "nodes" that is built to very closely mimic the 3D shape of the CAD geometry, representing not only the surface of the geometry, but filling the interior volume as well.
3. <u>Solve</u> – the boundary conditions (BCs) are applied to the mesh, a system of equations is generated from the loaded and constrained mesh (sometimes called an "input deck"), and the equations are solved through numerical analysis to a given error tolerance.
4. <u>Plot Results</u> – Once the equations are solved, we can visualize the results in numerous different ways, depending on the type of analysis, e.g. von Mises stress, principle stresses, geometry deformation (displacement), temperatures, plastic deformation zones, natural frequencies, etc.

In this tutorial, we will perform a linear elastic analysis on a simple cantilever beam. We will:

- Create the beam in Alibre Design
- Assign aluminum 6061-T6 material to the beam
- Create finite element mesh
  - Automesh using defaults
  - View the mesh
- Apply boundary conditions
  - Distributed load
  - Fixed beam end
- Solve for resultant elastic stresses
- Visualize the results
- Refine mesh, solve and visualize

The geometry units assumed in this tutorial are inches, but these can be any unit you are comfortable using. If you use different units, it is best to change the dimensions of the geometry and the values of the boundary conditions to reasonable values so that the FEA results are also reasonable.

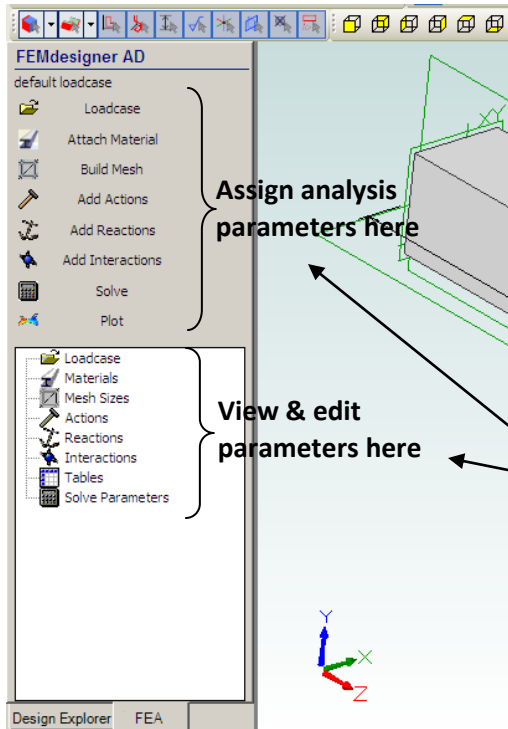© **Copyright FEMdesigner Ltd.**

STEP 1 – Create Geometry

In Alibre Design, with the XY plane as the sketch plane, draw a 1x1 square and extrude 10 units:
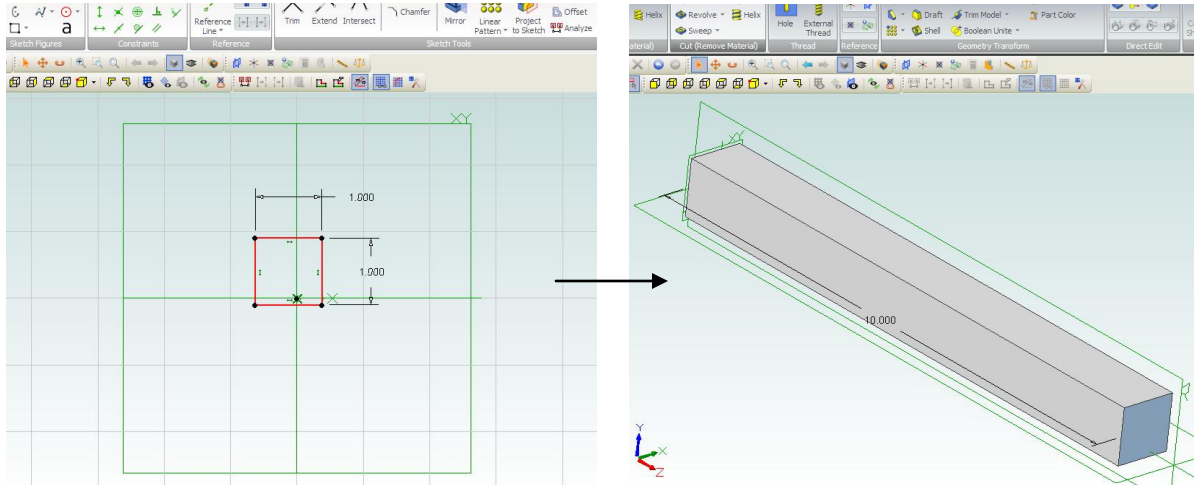
Now we can start the analysis…
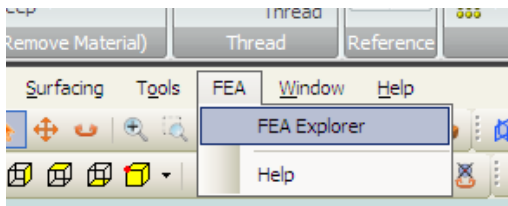
To activate the FEMdesigner Explorer in Alibre Design 2012, press the <ALT> key to display the Alibre drop-down menu bar, then select FEA->FEA Explorer.


This will create a new tab in the Design Explorer to guide and help you manage your FEMdesigner analysis.

Notice the menu choices are organized from top to bottom in the approximate order of the steps needed to perform an analysis.

The upper and lower sections are similar, but not the same: the upper section is for geometry selection and data input, and the lower section is a treeview of the current analysis parameters for viewing and editing. You can also view current parameters by selecting any menu choice in the upper section and the parameters for that menu will be displayed directly on the Alibre Design geometry.

## STEP 2 – Create Boundary Conditions and Mesh
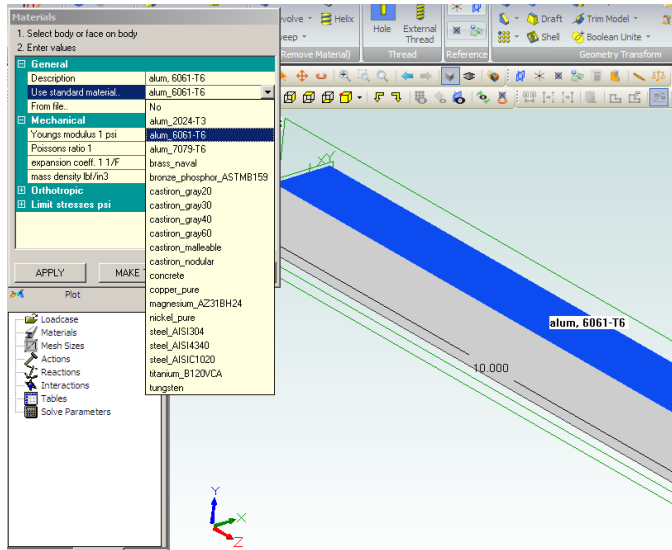
Assign Material

- Select "Attach Material" from the Explorer menu and the Materials dialog box will appear

- Select "Use standard material" and from the drop down list of materials, select "alum_6061-T6" which represents the 6061 alloy of aluminum with a T6 temper treatment

- Select any face on your beam and click "APPLY" in the dialog – the face will be highlighted and tagged with a material label denoting the material assigned
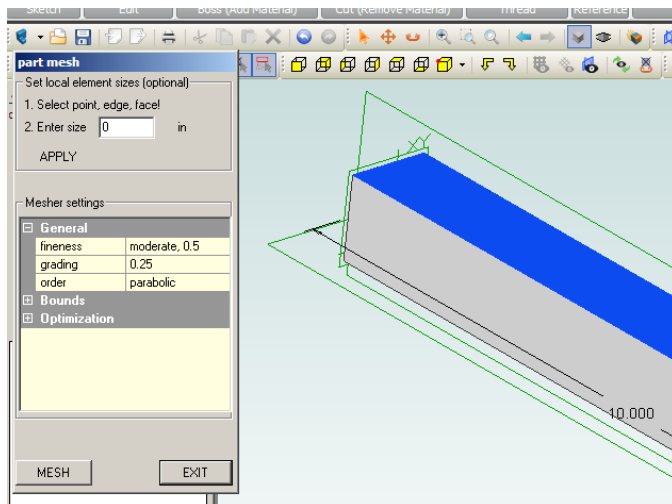
Create Mesh

- Select "Build Mesh" from the Explorer menu and the Part Mesh dialog box will appear

- For a first pass analysis, we will use the default values for the mesh. We can refine these values for later analyses if necessary

- Select the "MESH" button and a mesh will automatically be generated using the default mesh parameters
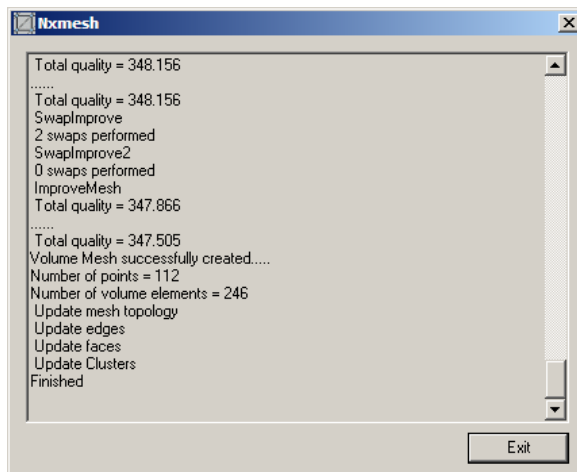
- A dialog box will pop up with information about the meshing process and its progress

- When the mesher is finished, simply click Exit, or you can review the mesher log displayed in the dialog before you exit
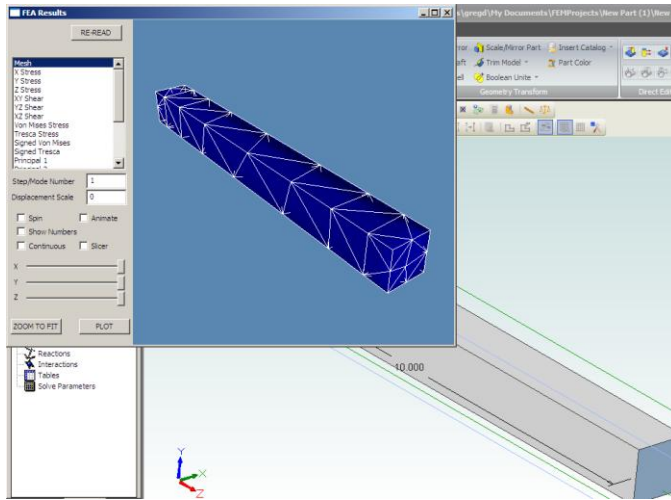
View Mesh

- Select "Plot" from the menu and a new graphics window with the beam mesh is displayed

- The mouse operations for view rotation and panning are the same as in Alibre Design, so you will be immediately comfortable with view manipulation
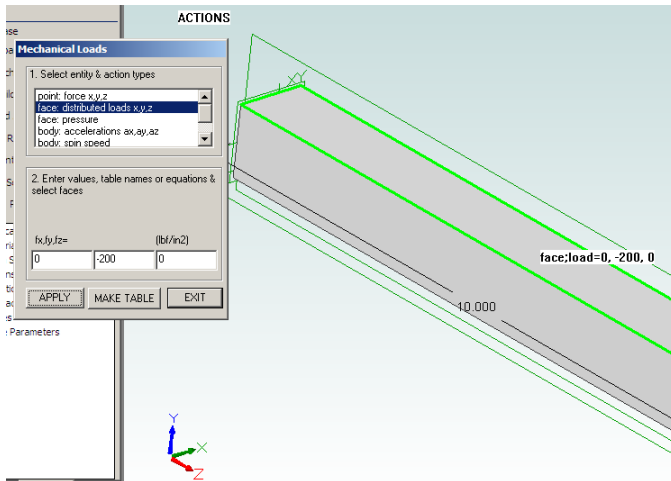

Assign Load

- Select "Add Actions" from the menu, then "face: distributed loads x, y, z"

- In the lower section of the dialog box, enter 0, -200, and 0 in the data entry fields as shown. These represent the x, y, and z values of the applied force vector

- Select the top face of the beam (having the positive y-axis normal to its surface) and click "APPLY" to create a 200-lb load evenly distributed across this face
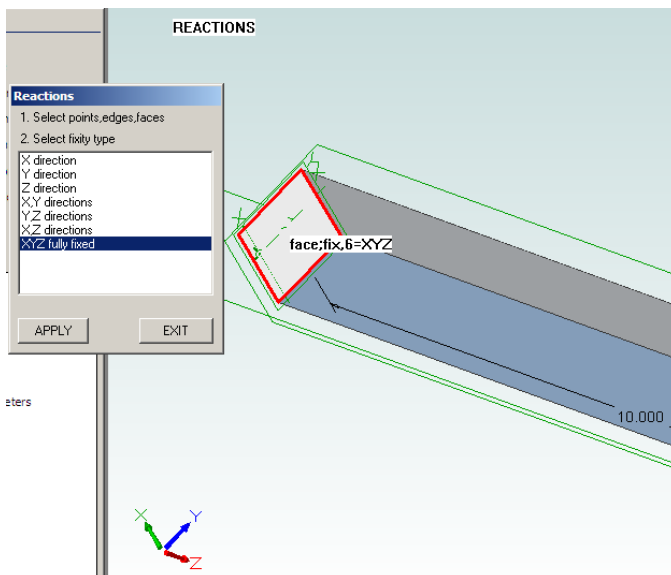
Fix Beam End

- Select "Add Reactions" from the menu, then "XYZ fully fixed"

- Rotate the view so that we can select the end of the beam that is facing away from us and select that face that is on the original sketch plane (XY plane)

- Click "APPLY" to fix that end of the beam and create a cantilever boundary condition

STEP 3 – Solve
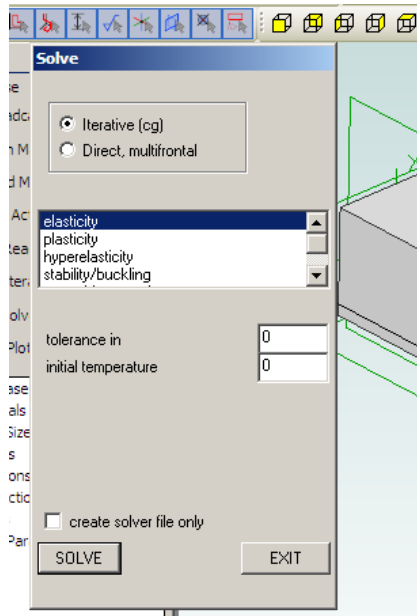
Run Solver

- Select "Solve" from the menu, then "Iterative (cg)" and "elasticity"

- Click the "SOLVE" button

- At this point FEMdesigner AD will derive the system of equations from the mesh and boundary conditions and calculate the solution

- An additional dialog box will pop up to report progress toward the solution – when this dialog box disappears, the solution is done

## STEP 4 – Plot Results
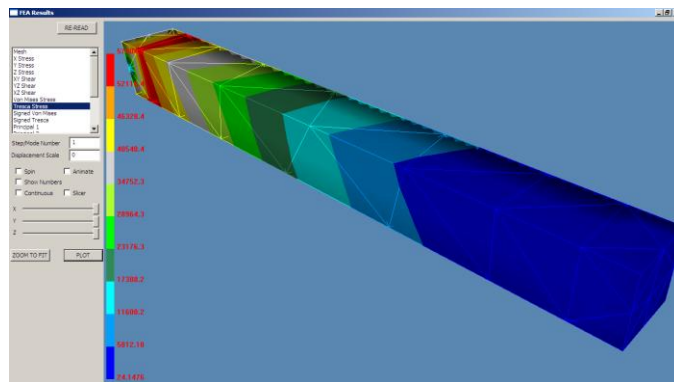
Stress Plot

- Select "Plot" from the menu, and the results graphics window appears again, displaying the mesh

- Make this new window full screen if you like, then select "Von Mises Stress" and then the "PLOT" button

- Rotate the view as shown
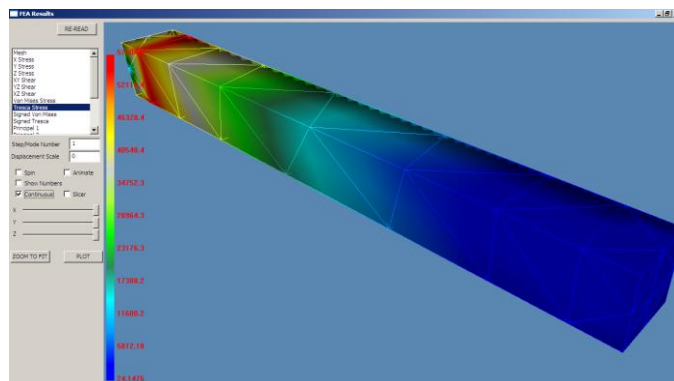
- Instead of sharp lines between the color bands representing different stress values, we can smooth out these colors with the "Continuous" check box
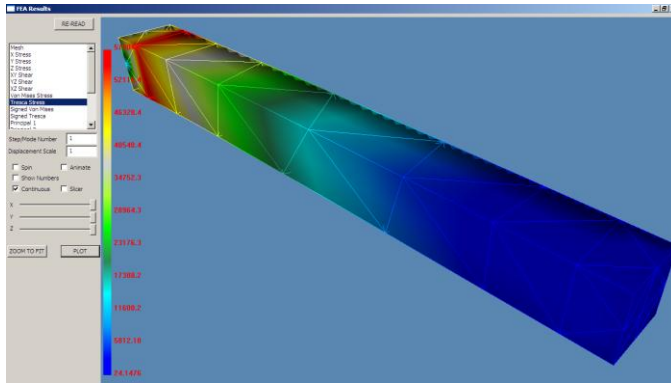
- To visualize how much the beam will bend (displacement) under this load, while viewing the stress plot at the same time, enter "1" into the "Displacement Scale" field

- This is the actual bending of the beam, to scale, calculated in the solution
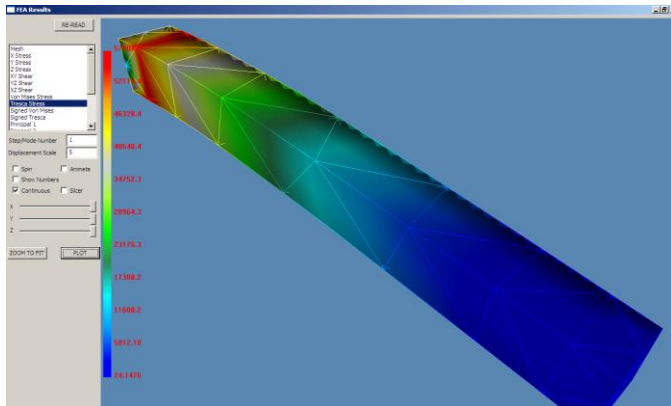
- Many times the displacement is too small to visualize when plotted to scale, so we can magnify the displacement for easier visualization

- Enter "5" in the "Displacement Scale" box and click "PLOT"

- Another feature that can be valuable for visualization is animation – click the "Animate" check box to watch the beam bend under loading
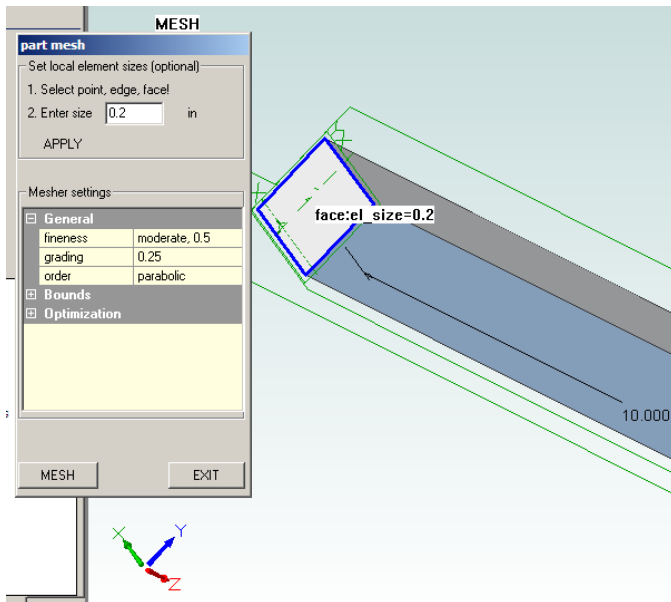
- Notice at the cantilever end where the stress is highest, the elements are relatively large, which doesn't allow us to localize the stress very precisely

- To give a more detailed picture of the stresses in that area, we can refine the mesh in order to create smaller elements

- Select "Build Mesh" again, then select the cantilever end face where the stress is highest, as shown

- We'll limit the maximum element size on that face by entering 0.2 in the "Enter size" box, then clicking "APPLY"

- Mesh and solve as before, then plot the results
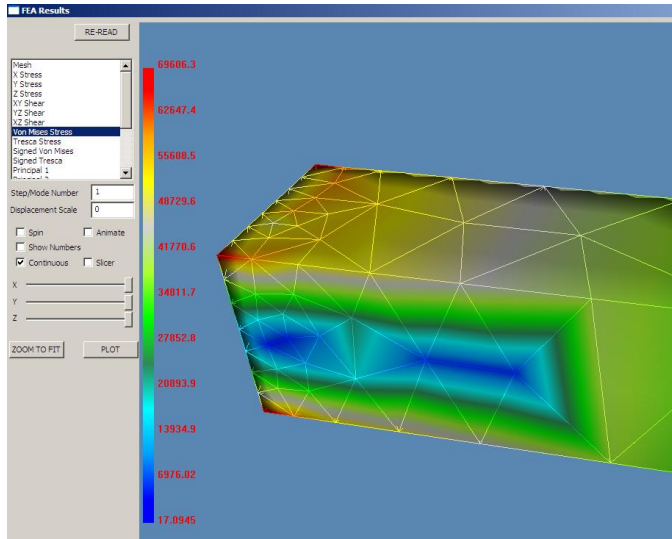
- As we inspect the new solution, we notice that the stresses at this end are exactly what we would expect from theory – there is a clear area of near-zero stress in the center of the beam (the "neutral axis"), and the stress is highest at the corners of the cantilever end, i.e. the points with the highest bending moment that are also farthest away from this neutral axis

Review

In this tutorial we covered:

Meshing
- Default auto mesh
- Mesh refinement

Boundary Conditions
- Assign material
- Fix face
- Distributed vector load on face

Solver
- Iterative solution
- Elastic model

Results
- Mesh plot
- Unsigned Von Mises stress plot
- Continuous color
- Displacement scale
- Displacement animation

Conclusion

This was an introductory tutorial designed to familiarize you with the basic FEMdesigner AD workflow necessary to set up and execute a linear static analysis. FEMdesigner AD has many more sophisticated analysis capabilities available, such as assemblies, non-linear materials, buckling, steady-state and transient heat transfer, natural frequencies and forced vibrations, but the analysis process you learned in this tutorial is essentially the same for all of these sophisticated analyses. It's simply a matter of knowing which boundary conditions to apply, and how to appropriately apply them.

Thanks for using FEMdesigner AD!