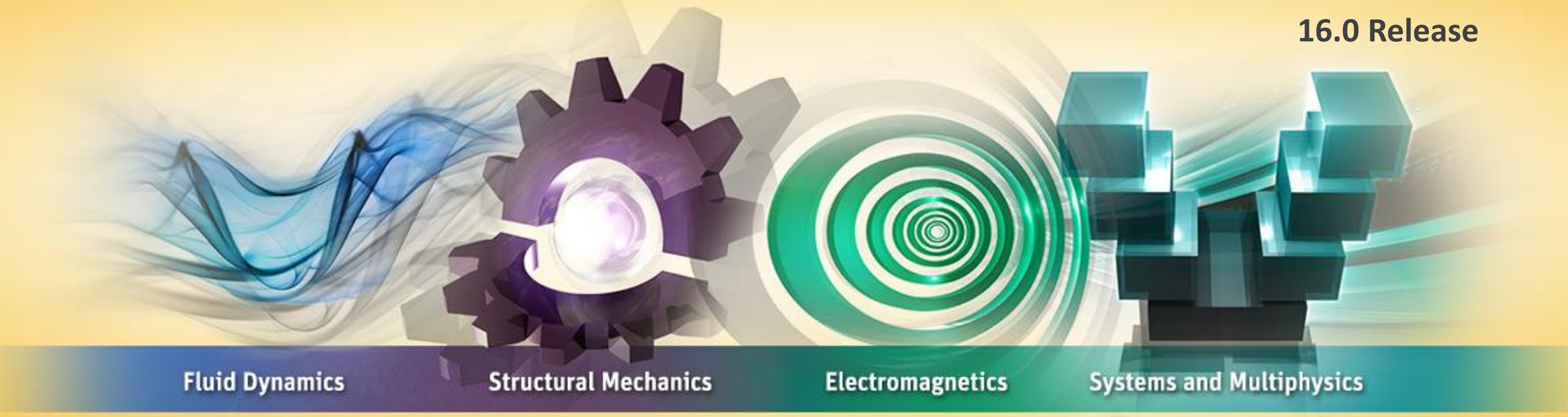
Workshop 2.1
ANSYS Mechanical Basics

16.0 Release



Introduction to ANSYS Mechanical

Realize Your Product Promise®

Notes on Workshop 2.1

Please Note: The step by step instructions for this workshop do not begin until slide #6.

The first workshop is extensively documented. As this course progresses, students will become more familiar with basic Workbench Mechanical functionality (menu locations etc.), thus subsequent workshops will contain less details.

Throughout these workshops menu paths are documented as: “First pick > Second pick > etc.”.

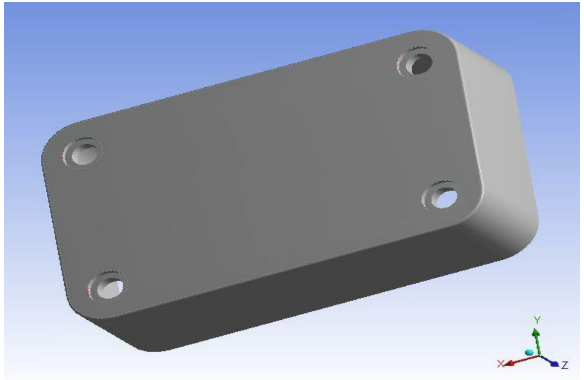
Workshops begin with a goals section followed by an assumptions section.

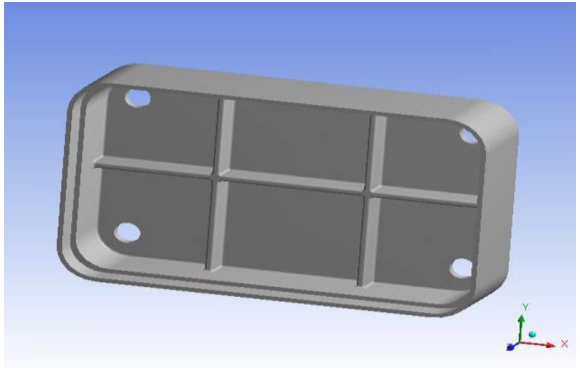
Goals

Using the Stress Wizard, set up and solve a structural model for stress, deflection and safety factor.

Problem statement:

- **The model consists of a STEP file representing a control box cover (see figure). The cover is intended to be used in an external pressure application (1.0 MPa).**
- **The cover is to be made from aluminum alloy.**
- **Our goal is to verify that the part will function in its intended environment.**

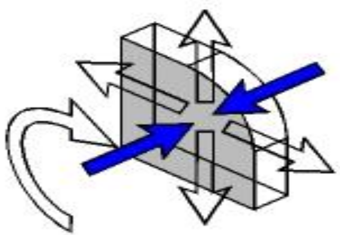


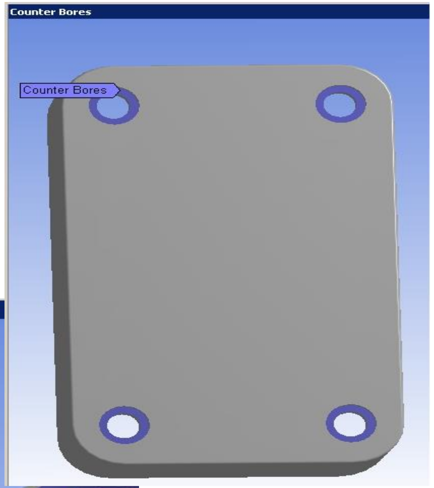


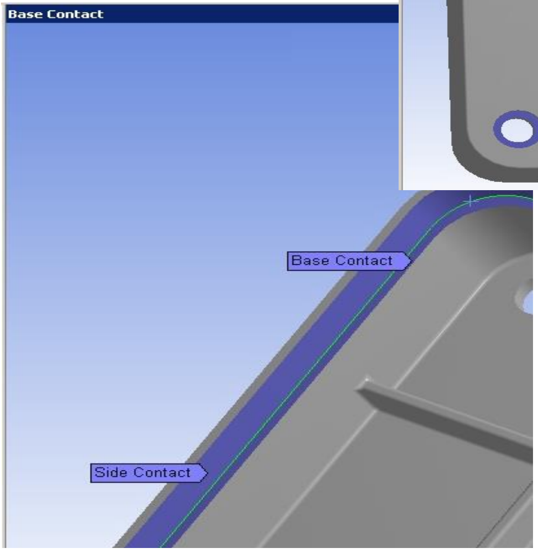
Assumptions

We will represent the constrains on the counter bores, bottom contact area and inner sides using frictionless supports.

- **Frictionless supports place a normal constraint on an entire surface. Translational displacement is allowed in all directions except into and out of the supported plane. Since we would expect frictional forces to act in these areas, this is a conservative approach.**

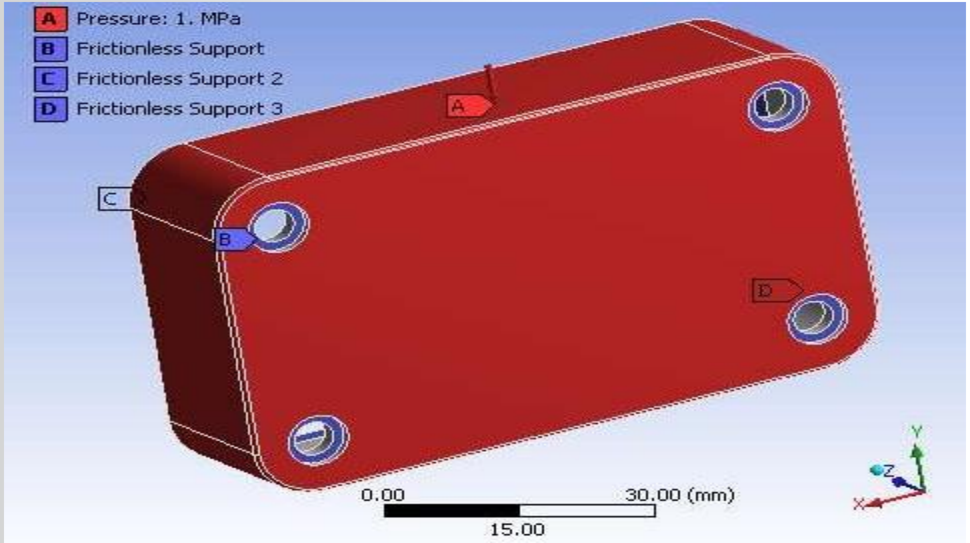






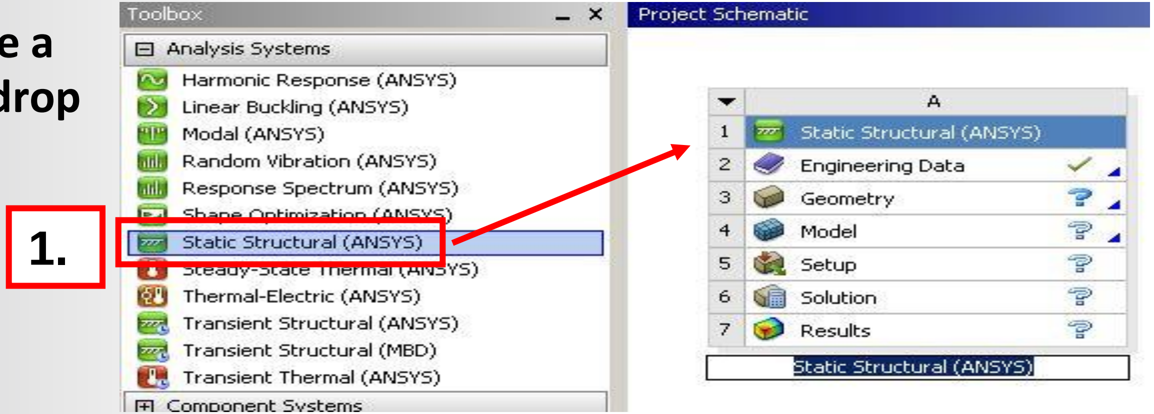
Environment

Loads: the load consists of a 1 MPa pressure applied to the 17 exterior surfaces of the cover.

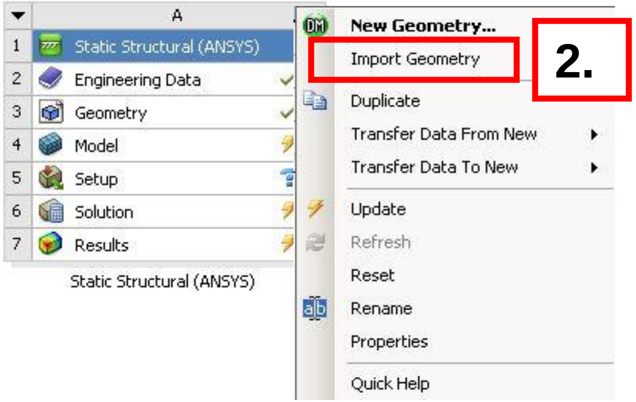


Project Schematic

1. From the Toolbox choose create a Static Structural system (drag/drop or RMB).



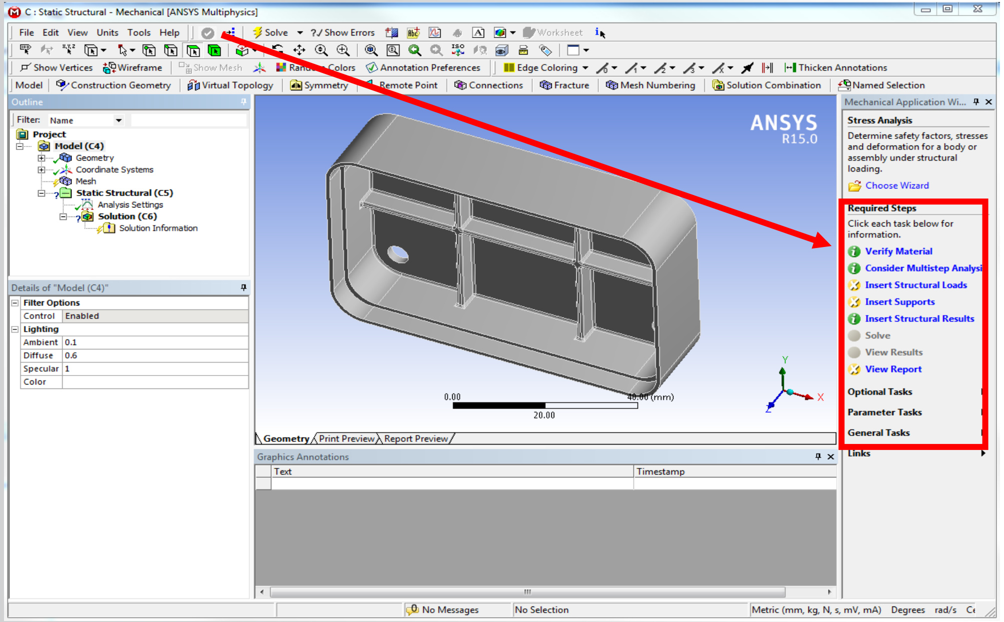
2. RMB in the Geometry cell and choose Import Geometry. Browse to the file “Cap_fillets.stp” and click Open.



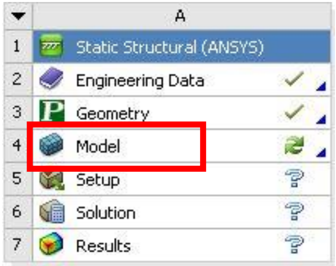
Preprocessing

3. Double click the “Model” cell to open the Mechanical application.

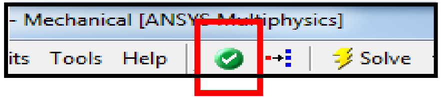
When the Mechanical application opens the model will display in the graphics window and the Mechanical Application Wizard displays on the right.



3.



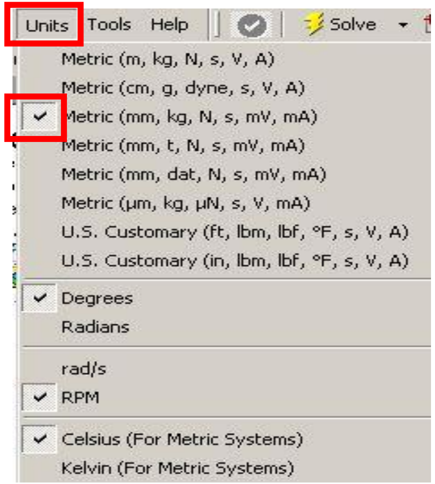
Workshop 2-1



When Mechanical starts if the Wizard is not displayed, use the icon to open it.

. . . Preprocessing

4. **Set/check the units system:**
   - **From the main menu go to “Units > Metric (mm, kg, N, s, mV, mA).**

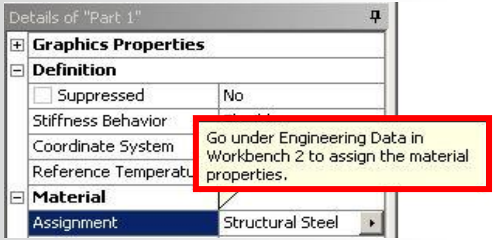


. . . Preprocessing

5. **Select a suitable material for the part:**

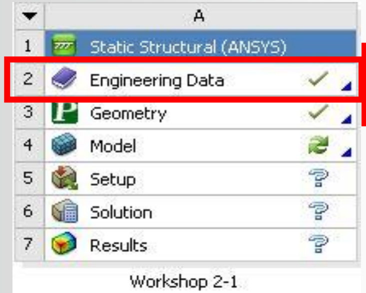
   a. From the Mechanical Wizard choose "Verify Material"

   b. Notice the callout box indicates Engineering Data is accessible from the WB2 interface (Project Schematic).



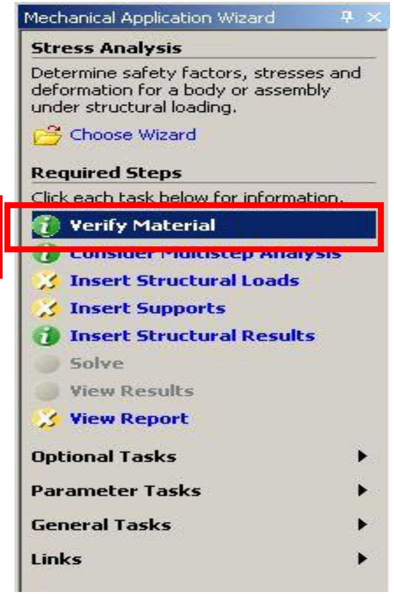
b.

   c. Return to the Project schematic window and double click "Engineering Data" to access the material properties.



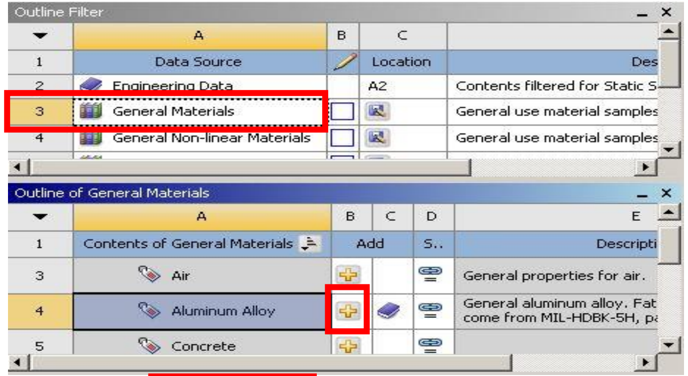
c.

a.



. . . Preprocessing

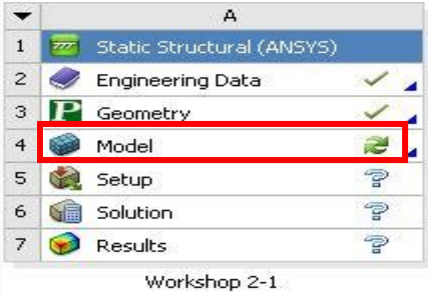
6. **Activate the Engineering Data Sources toggle and highlight “General Materials” then click the ‘+’ next to “Aluminum Alloy”.**

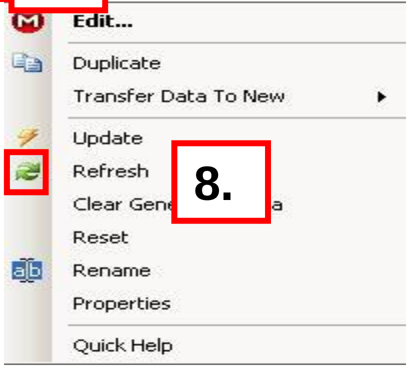


7. **Return to the Project.**

- **Notice the Model cell indicates a refresh is necessary.**

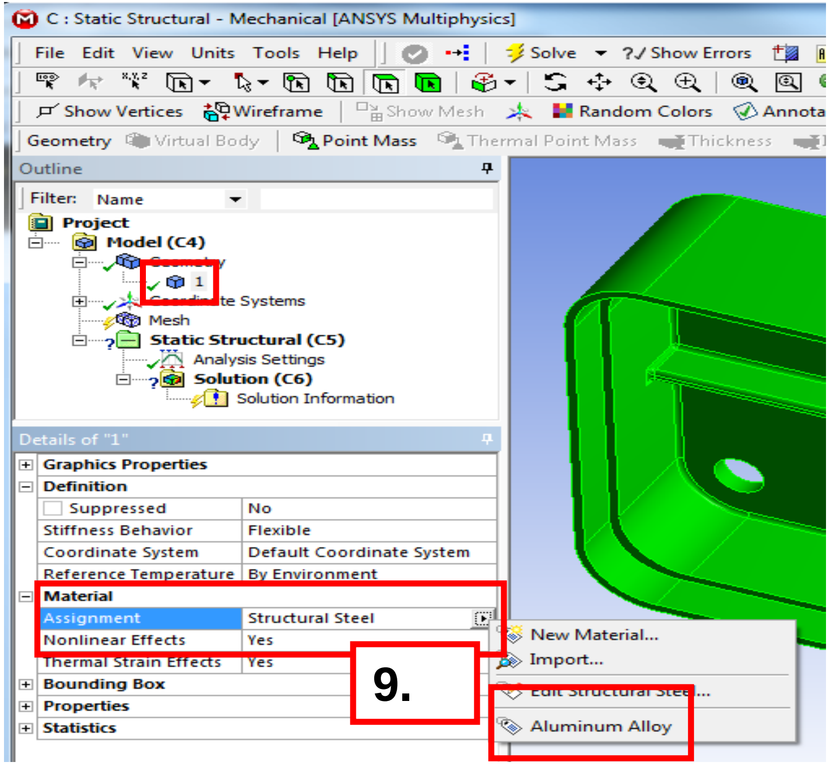
8. **Refresh the Model cell (RMB), then return to the Mechanical window.**





. . . Preprocessing

9. **Highlight “Part 1” and click the “Material > Assignment” field to change the material property to aluminum alloy.**



. . . Preprocessing

10. **Insert Loads:**

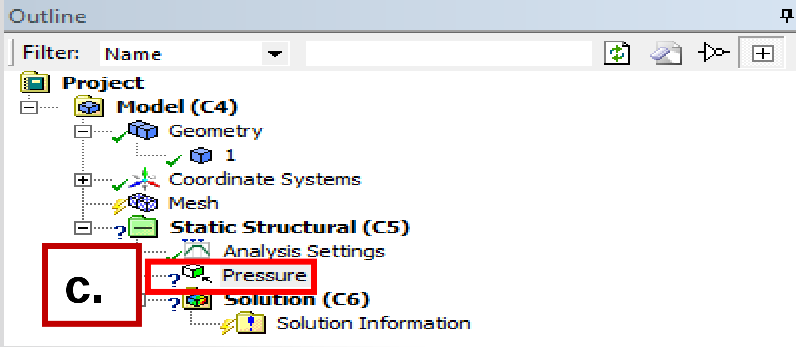
   a. Select “Insert Structural Loads” from the Wizard

   b. Follow the call out box to insert a “Pressure” load

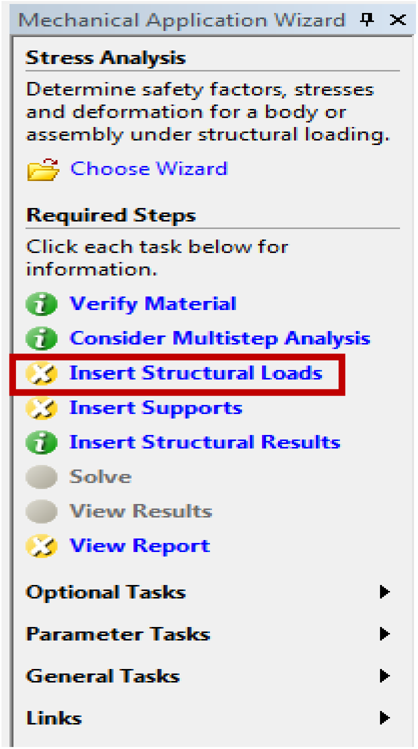
   c. The tree will now include a Pressure load in the “Static Structural” environment branch

b.

c.



a.



. . . Preprocessing

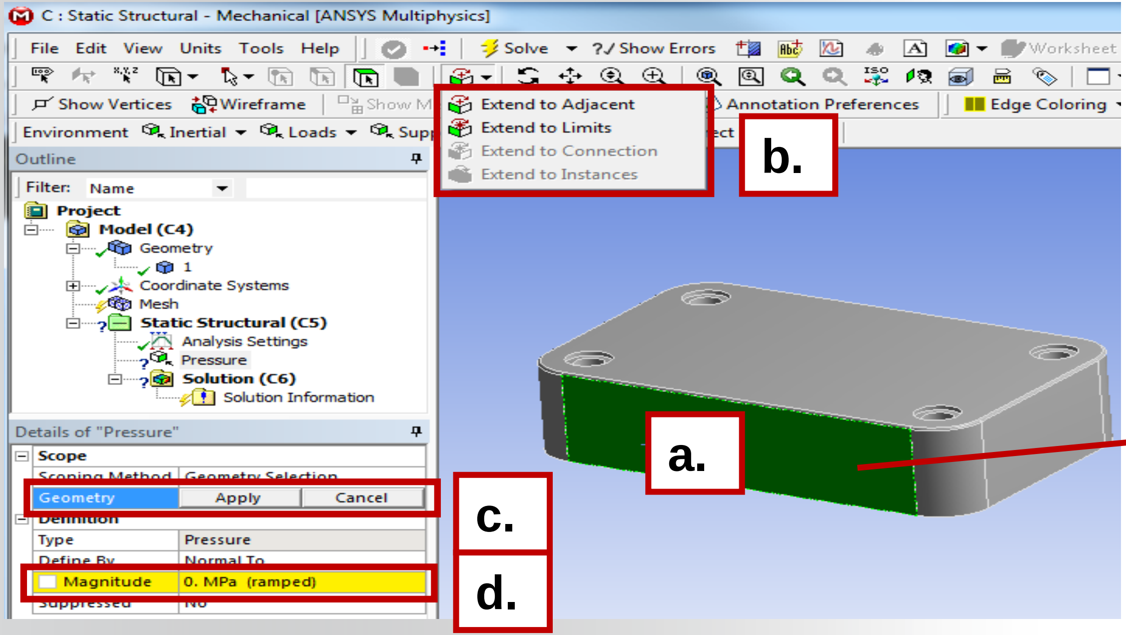
11. Apply the load to geometry:

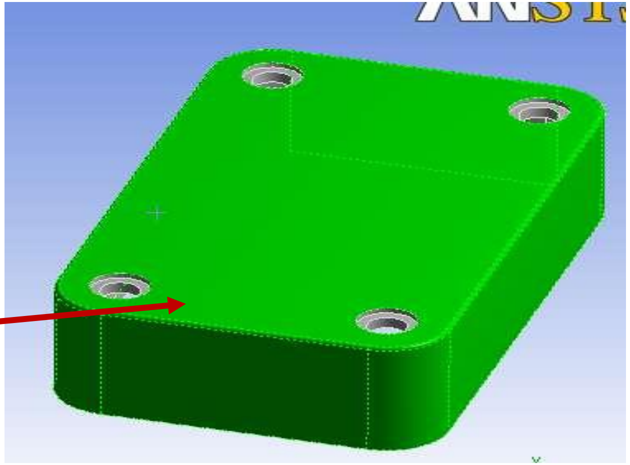
a) Highlight one of the outer faces of the part.

b) Use the “Extend to Limits” icon to select the remaining 16 faces (total 17 faces selected).

c) Click “Apply” to accept the faces.

d) Enter a “Magnitude” of 1MPa.





  February 27, 2015

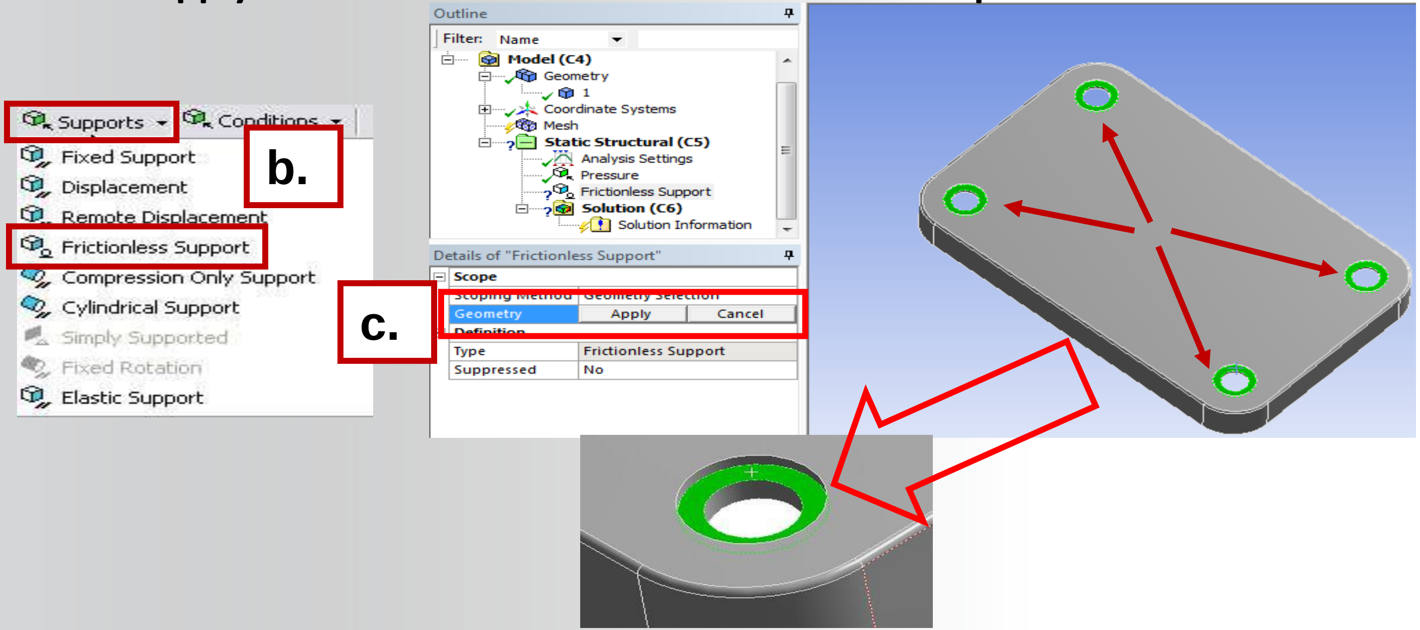
. . . Preprocessing

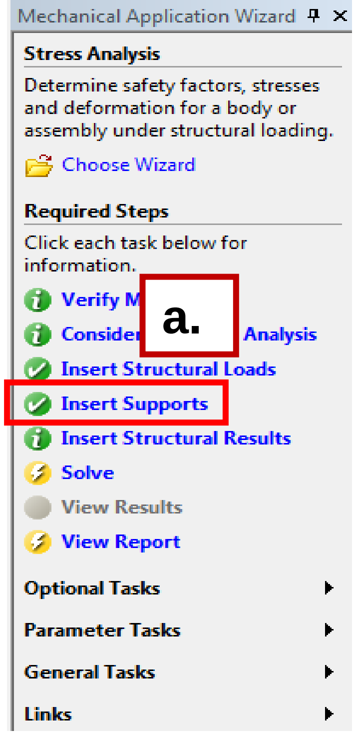
12. Apply supports to constrain the part:

  a. Select “Insert Supports” from the Wizard.

  b. Follow the callout box to insert a “Frictionless Support”.

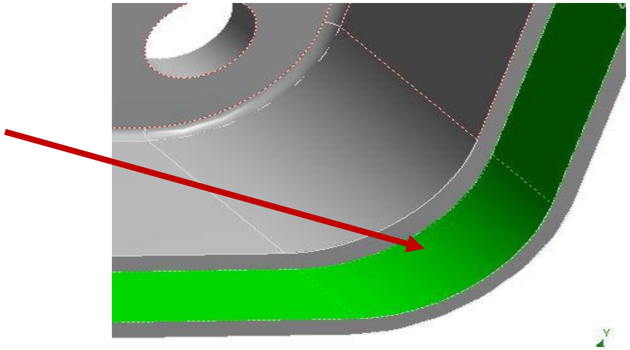
  c. “Apply” it to the 4 counter bore surfaces of the part.



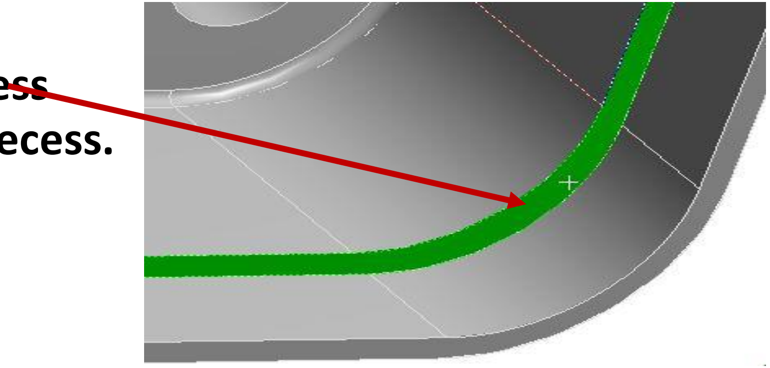


. . . Preprocessing

13. Repeat Steps 12.a. and 12.b. to insert a “Frictionless Support” on the inner surfaces of the bottom recess (use extend to limits after selecting one of the inner surfaces.



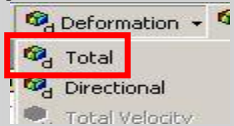
14. Repeat Steps 12.a. and 12.b. to insert a “Frictionless Support” on the lip surface at the bottom of the recess.



ANSYS . . . Preprocessing

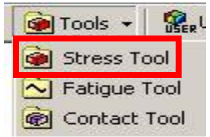
15. **From the Mechanical Wizard request:**
    a) Insert Structural Results (the call out will point to the Solution toolbar).
    b) Deformation > Total.
    c) Stress > Equivalent (von-Mises).
    d) Tools > Stress Tool.

b.

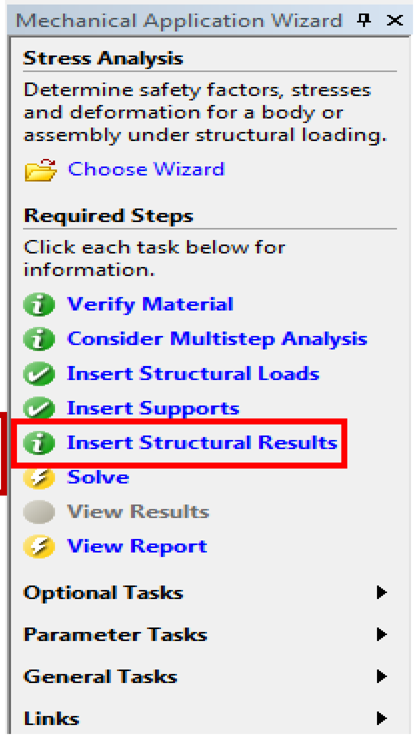


c.

d.



a.



Note the Stress Tool detail allows 4 different configurations (explained later). For this workshop we will leave the tool specified as "Max Equivalent Stress" theory.

Details of "Stress Tool"

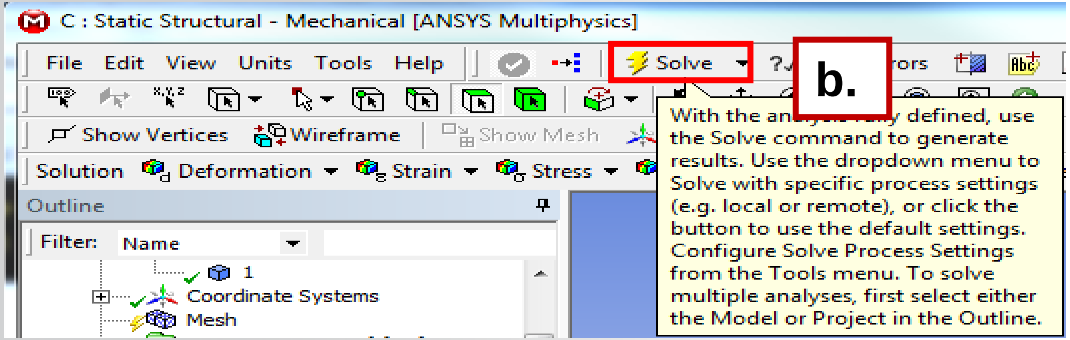
Definition	
Theory	Max Equivalent Stress
Stress Limit Type	Tensile Yield Per Material

Solution

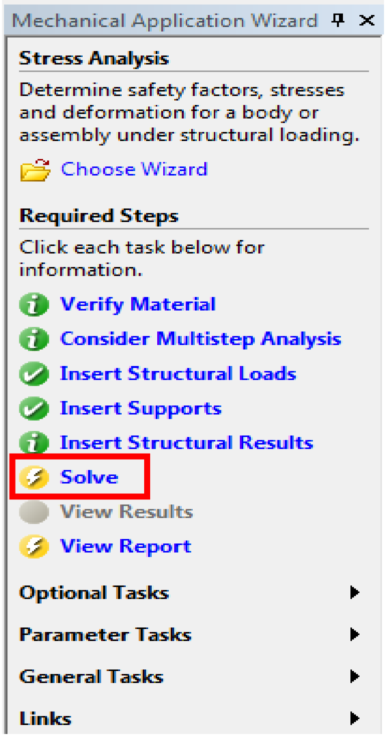
16. Solve the model:

a. Select “Solve” from the Wizard.

b. Follow the callout box and click on “Solve”.



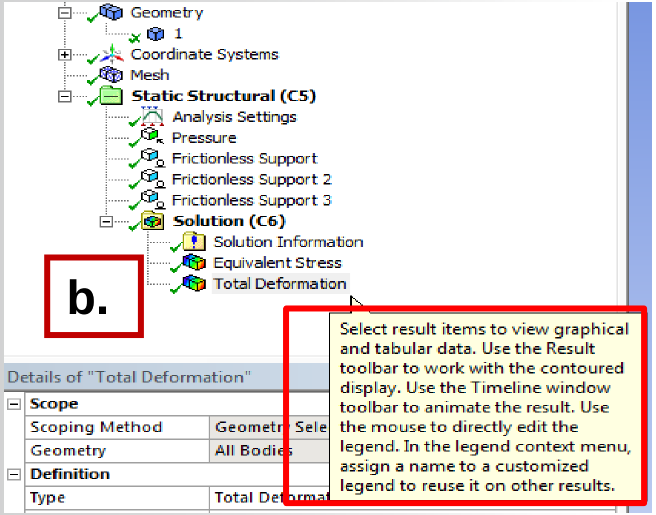
a.



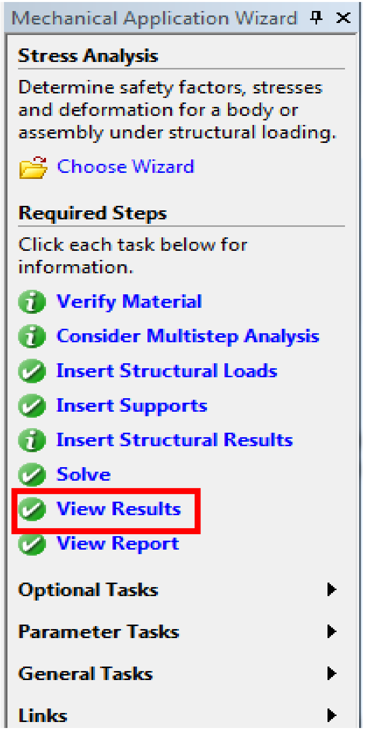
- *Note how clicking on “Solve” in the Wizard does not automatically start solving the model but instead, points out the “Solve” icon to the user. Alternatively, you could right click on any branch in the “outline” and choose “Solve”*

Results

17. **View the results:**
   a. Click “View Results” from the Wizard
   b. Follow the callout box to where the results are available under the “Solution” branch

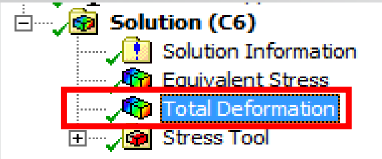


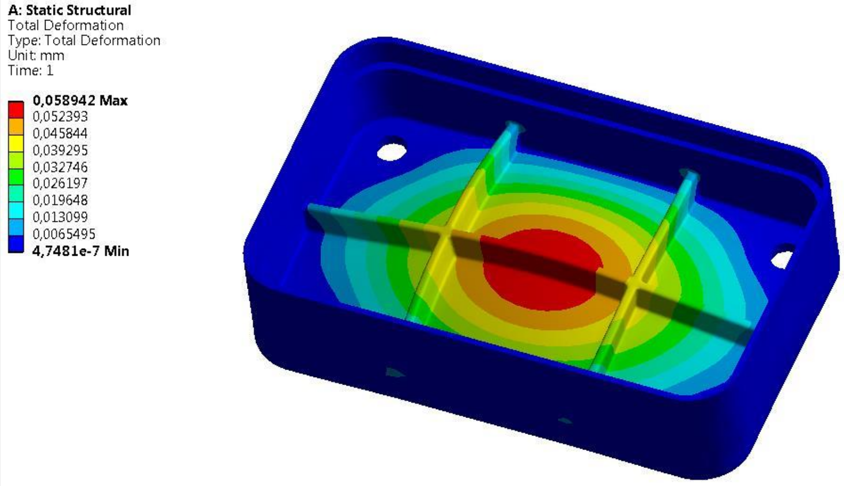
a.



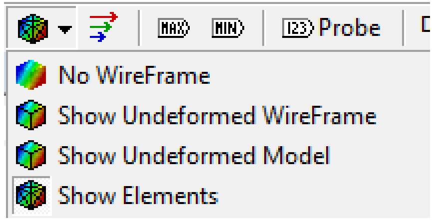
. . . Results

Plotting a model's deformation often provides a "reality check" in structural analysis. Verifying the general nature (direction and amount) of deflection can help avoid obvious mistakes in model setup. Animations are often used as well.



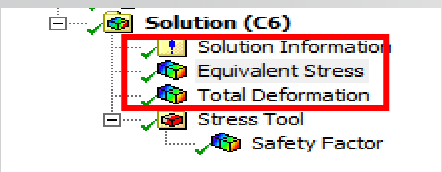


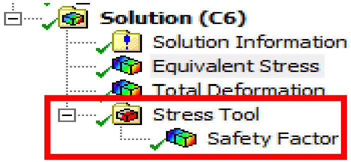
Element visibility can be toggled on and off quickly using the options

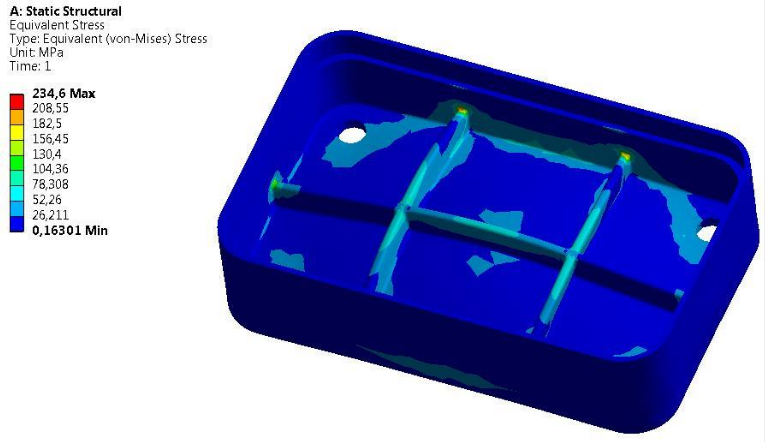


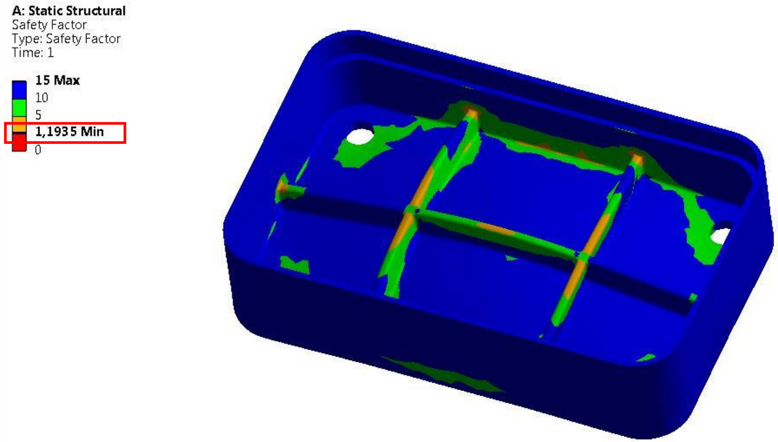
. . . Results

After reviewing stress results expand the Stress Tool and plot safety factor. Notice the failure theory selected predicts a minimum safety factor of just over 1.







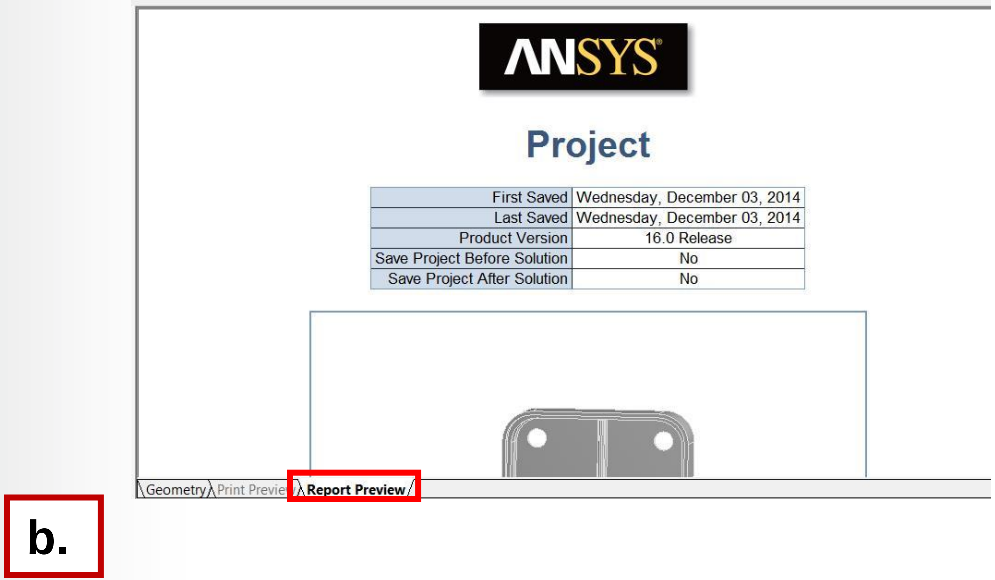


Report

18. Create an html report:

   a. First choose the graphical items you wish to include in your report and insert a figure for each one (this is your choice).

   b. Click the "Report Preview" tab to generate the report.



. . . Report

Notes on Figures:

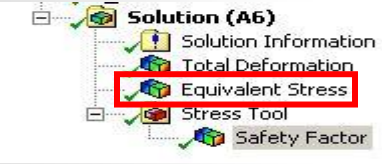
Figures are not limited to results items. Adding a plot of the environment branch, for example, will include an image of model boundary conditions in the Report.

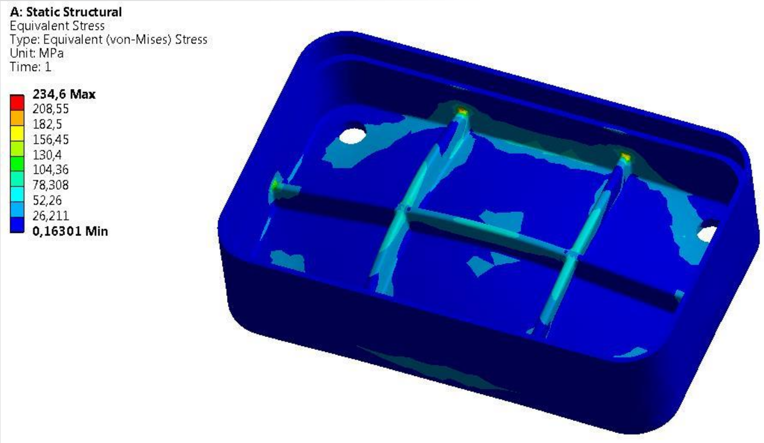
Figures are independent. You may set up individual figures and have their orientation, zoom level, etc. retained regardless of the active model orientation or other figures.

Individual branches can have multiple figures associated with them.

Go further!

If the yield stress of the material is equal to 250Mpa, do you think that we can validate the beginning of production of this part if we do not accept to enter in the plastic domain of the material?





. . . Go further!

Try to mesh a little bit finer by defining a global element size.