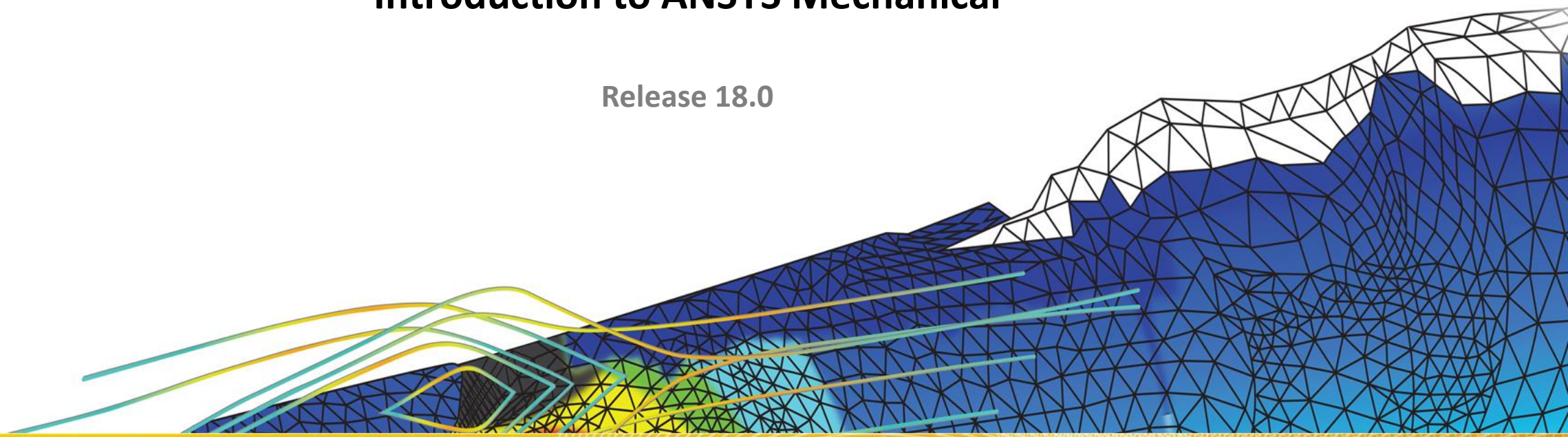
Workshop 08.2: Submodeling

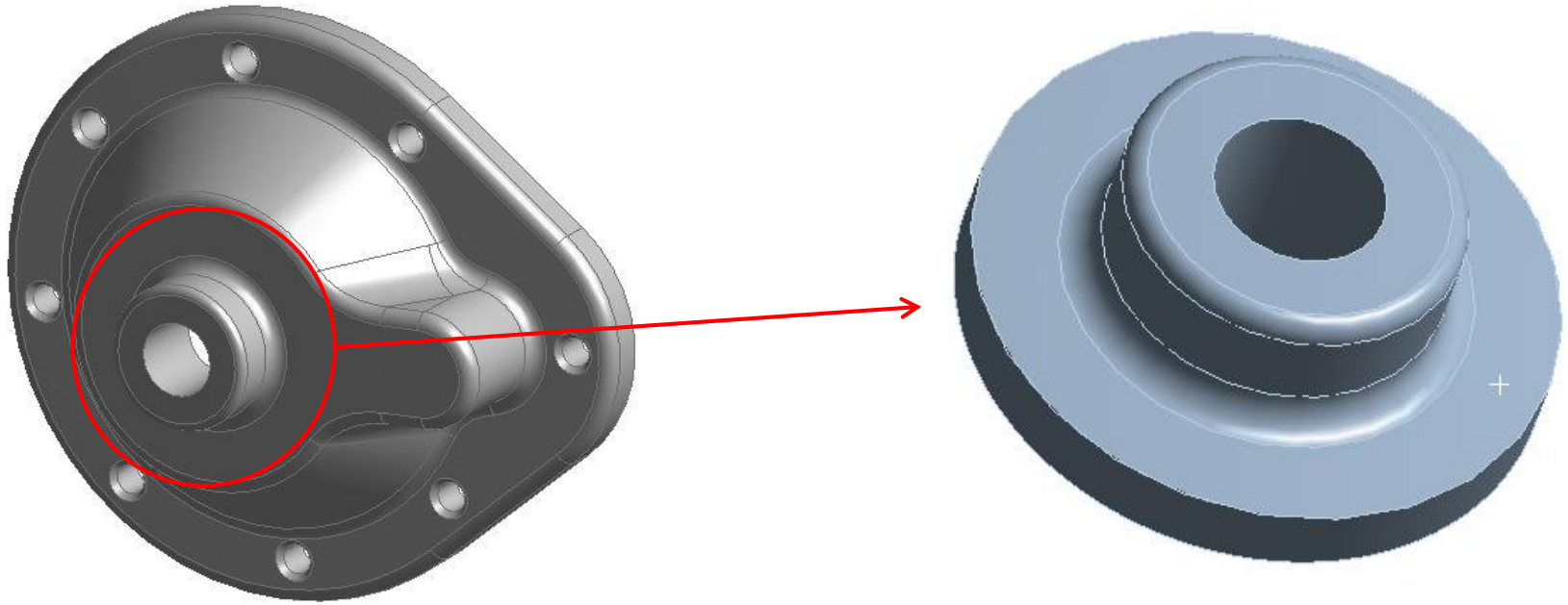
Introduction to ANSYS Mechanical

Release 18.0



Goals

Our goal is to perform a submodel analysis of the pump housing shown below by first setting up and solving a full model (coarse model) and then setting up and solving a submodel (fine model) representing a small region of the full model.

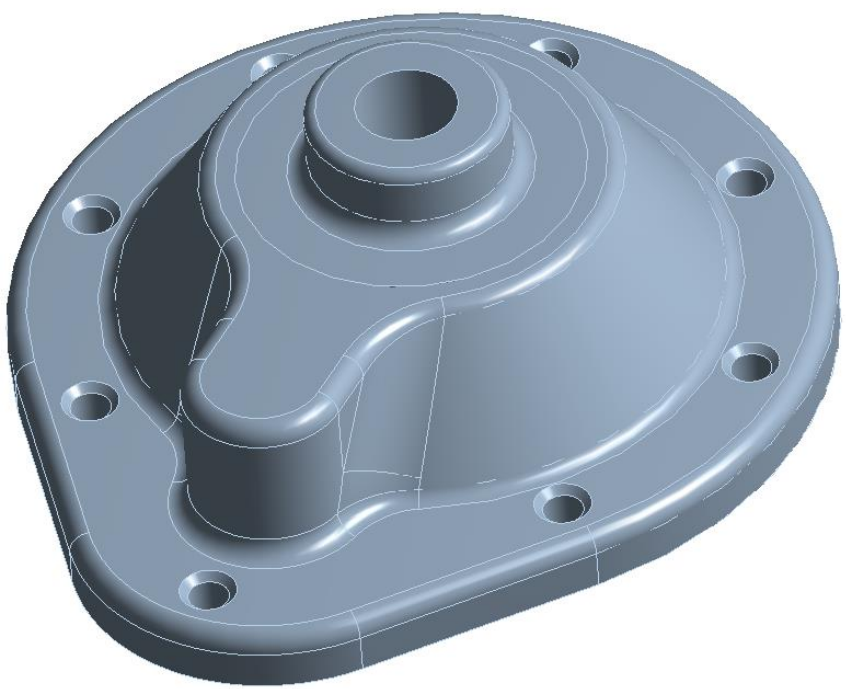


Full Model

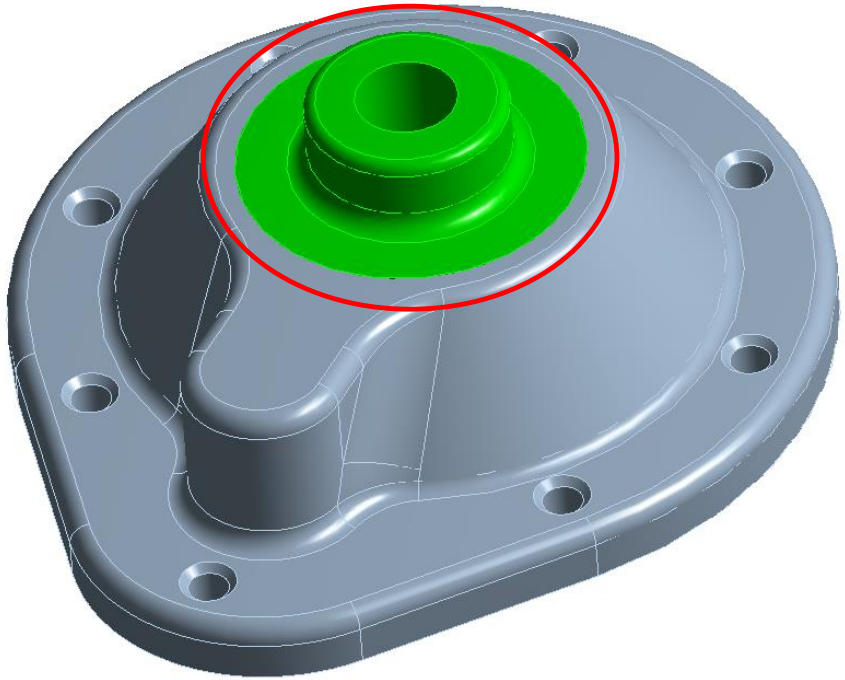
Submodel

Approach

Submodeling requires the use of two geometry models: one to represent the full geometry and one to represent a smaller portion of the full model. For this exercise, we used ANSYS DesignModeler to slice a smaller volume from the full model.



Full Model

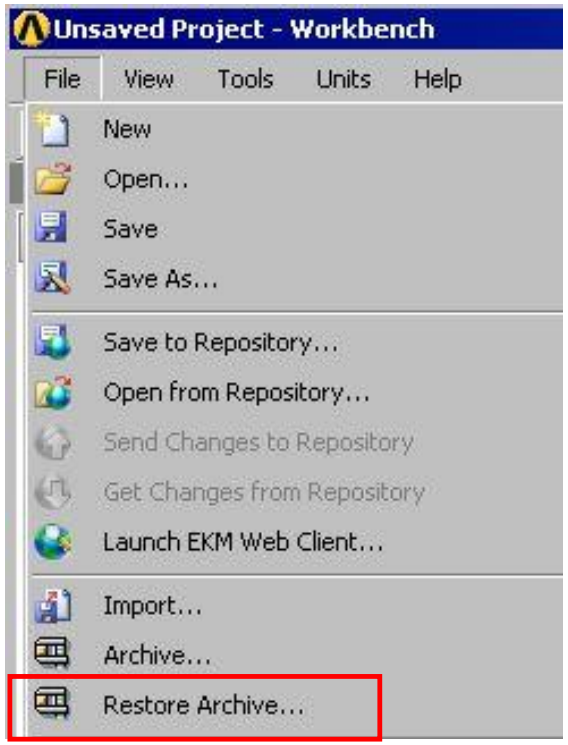


Submodel

Project Schematic

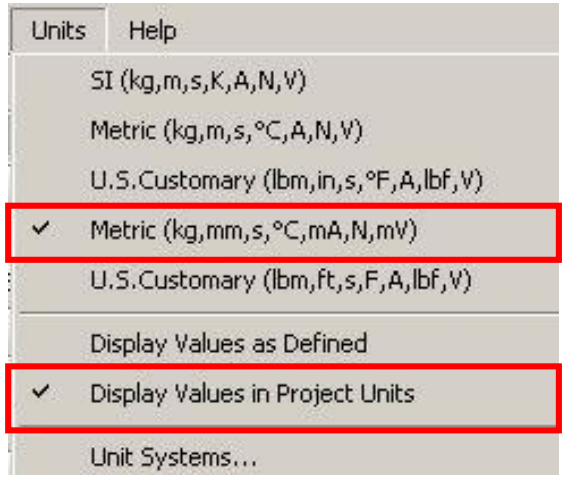
Begin a new Workbench session and, from the Workbench window, choose “Restore Archive…,” browse to file “Submodeling_WS_APPXB.wbpz,” and Open.

When prompted, save the project using the default name and the same location as the archive file.



From the “Units” menu, verify that:

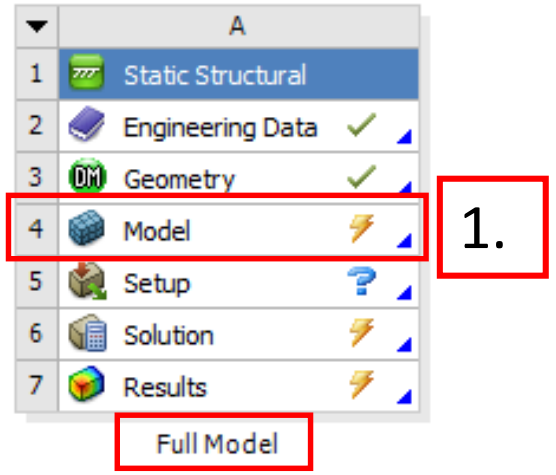
- **Project units are set to “Metric (kg,mm,s,°C,mA,N,mV).”**
- **“Display Values in Project Units” is checked.**



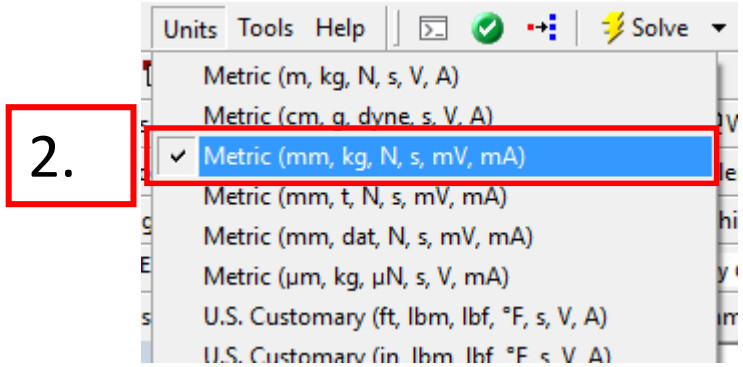
Project Schematic

Note that the existing Static Structural system has been renamed to “Full Model.”

1. **From the Static Structural system, double-click (or RMB > Edit) the “Model” cell.**



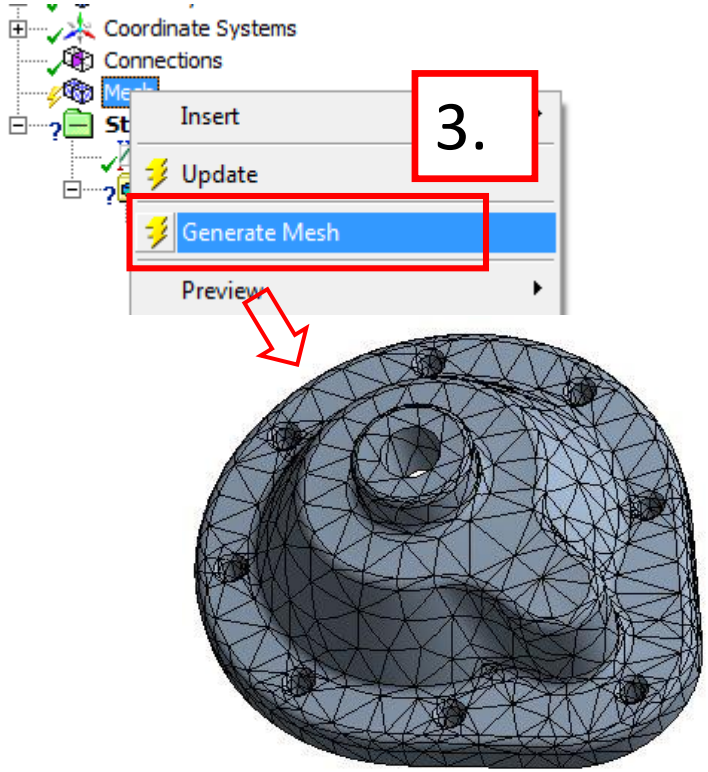
2. **When Mechanical opens, verify that the units are set to “Metric (mm, kg, N, s, mV, mA).”**

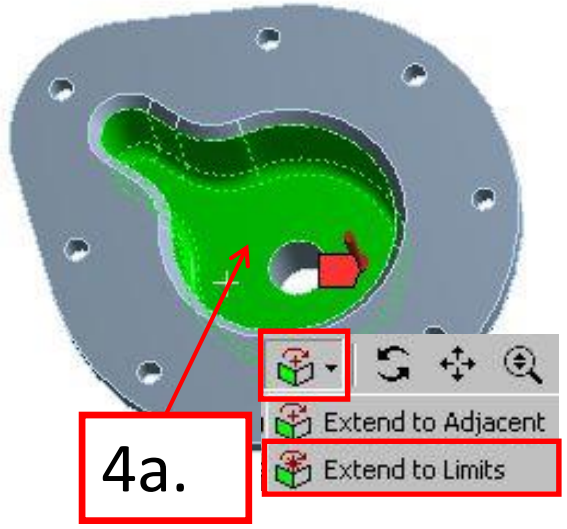


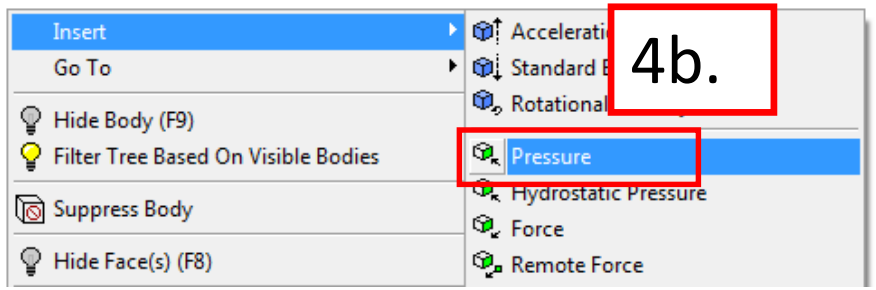
Preprocessing—Full Model

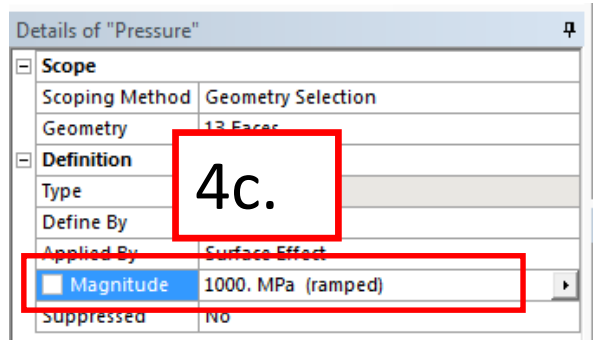
3. Select the Mesh branch, then RMB > Generate Mesh.

4. Apply a pressure load:
   a. Select the Static Structural environment branch, then select one of the interior surfaces of the housing and “Extend to Limits” to select all 13 interior faces.
   b. RMB > Insert > Pressure.
   c. In the Pressure Details view, set Magnitude to “1000 MPa.”





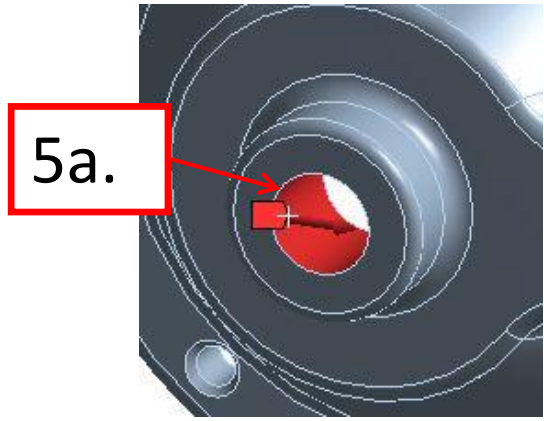


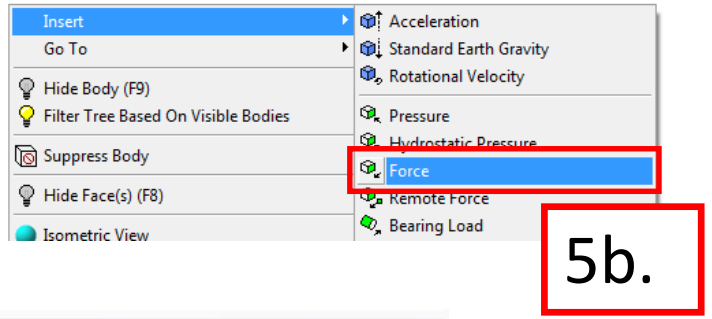


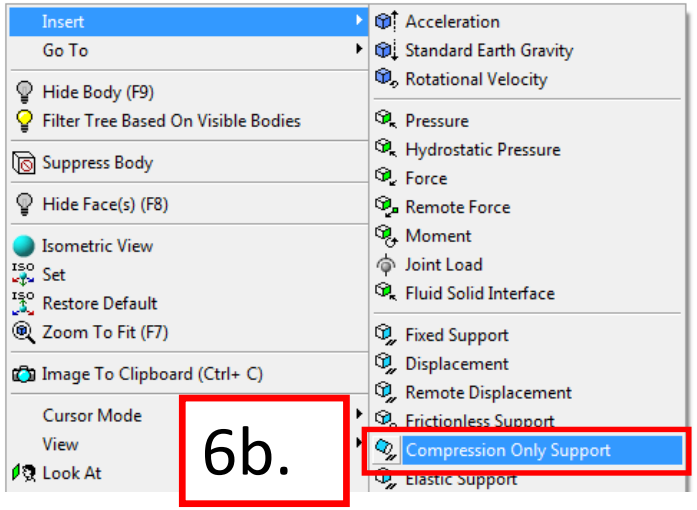
Preprocessing—Full Model

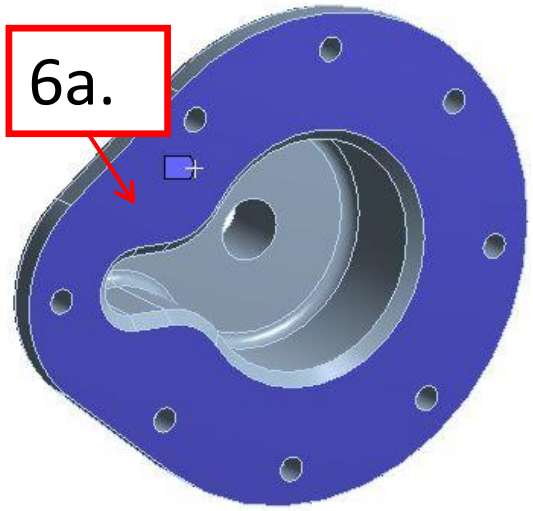
5. **Add a force to the housing:**
   a. **Select the cylindrical face of the center hole in the housing.**
   b. **RMB > Insert > Force.**
   c. **Set Define By to Components and set X Component to 200 N.**

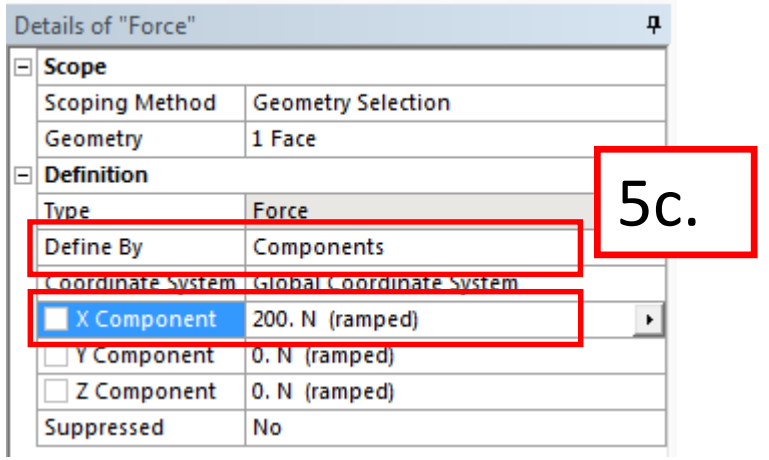
6. **Add a Compression Only support:**
   a. **Select the planar mounting surface on the housing.**
   b. **RMB > Insert > Compression Only Support.**





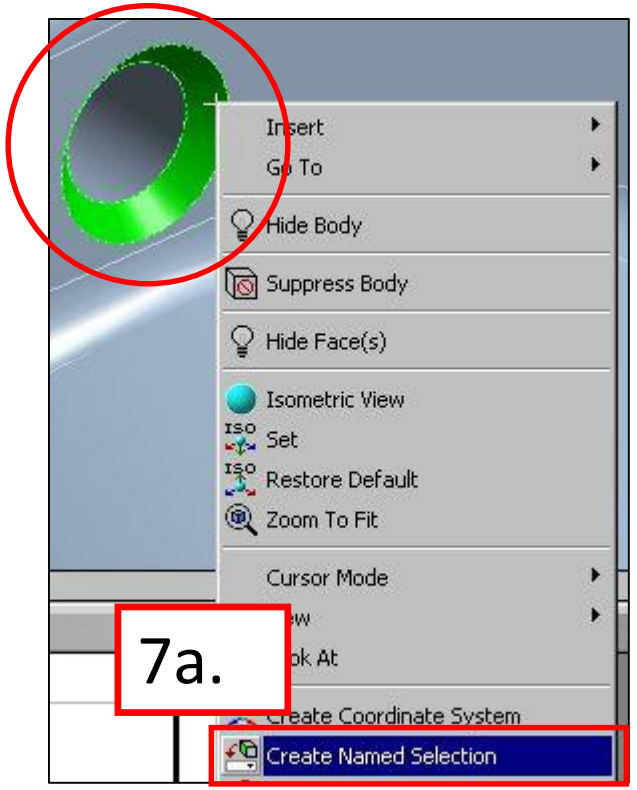


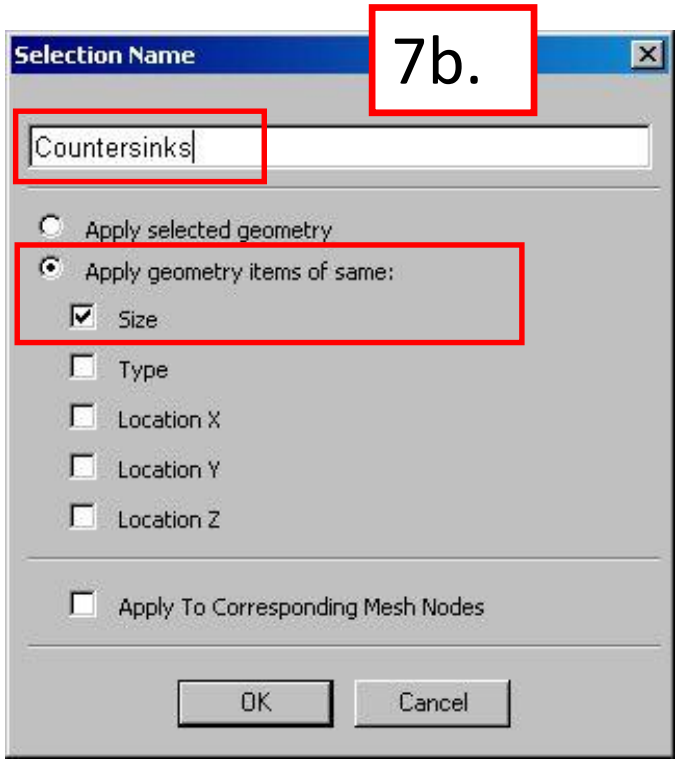




Preprocessing—Full Model

7. **Create a named selection containing the countersink faces:**
   a. **Select one countersink face, then RMB > Create Named Selection.**
   b. **In the Selection Name dialog, enter the name “Countersinks,” select “Apply geometry items of same:,” check “Size,” and click OK. The resulting named selection should contain 8 faces as shown.**

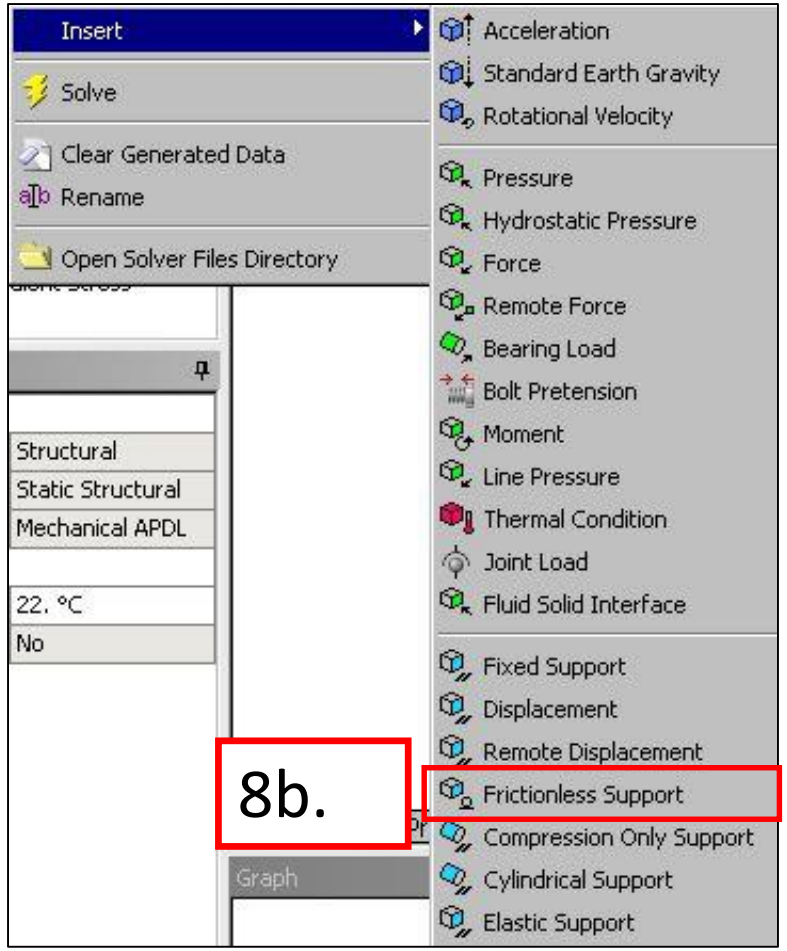




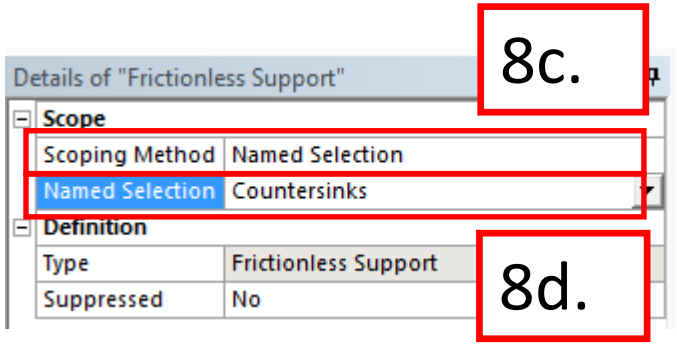
Preprocessing—Full Model

8. **Add frictionless supports to the countersink faces:**
   a. **Select the Static Structural environment branch.**
   b. **RMB > Insert > Frictionless Support.**
   c. **In the Details view, set Scoping Method to “Named Selections.”**
   d. **Set Named Selection to “Countersinks.”**

8a.

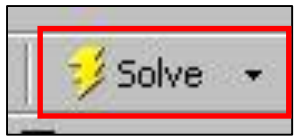


8b.



8c.
8d.

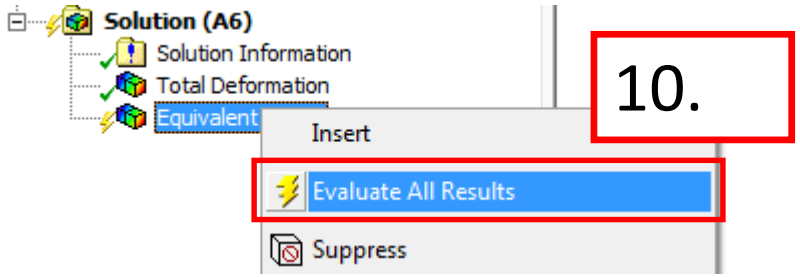
9. **Solve.**

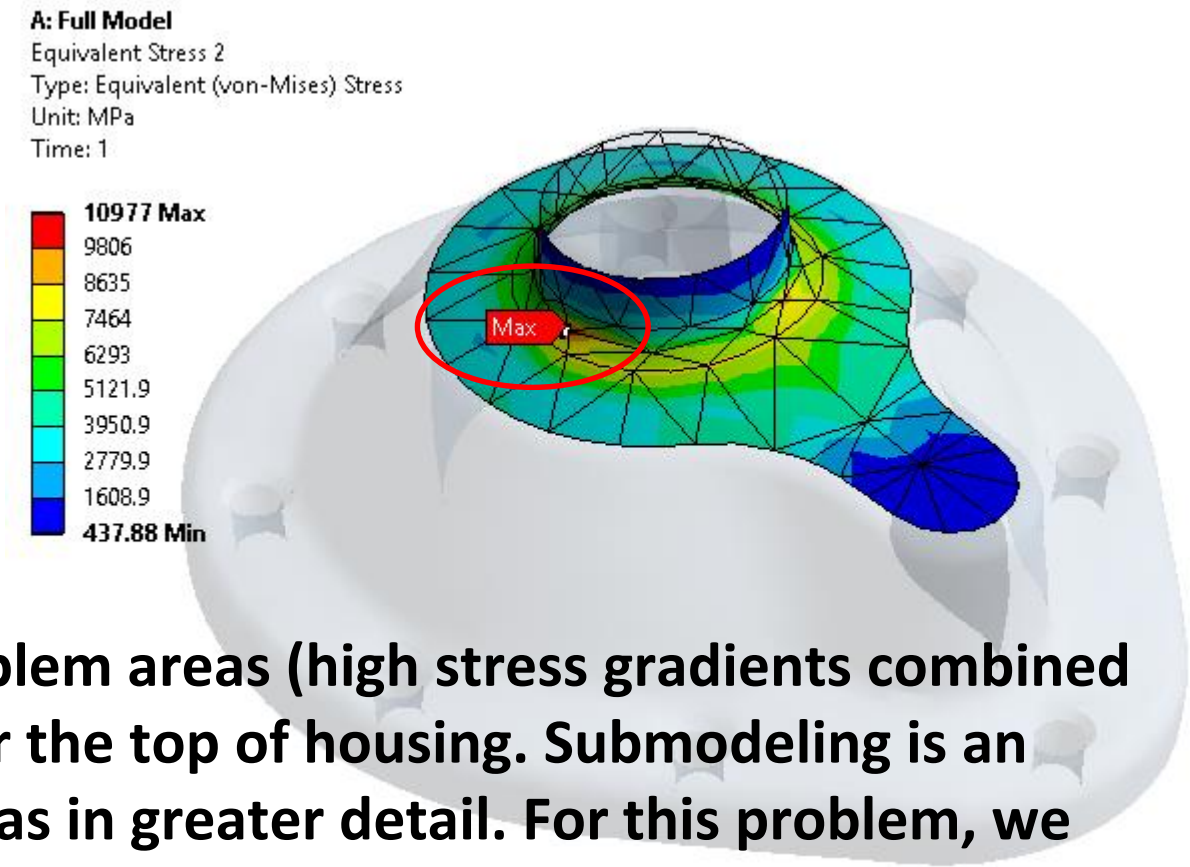


9.

Postprocessing—Full Model

10. When the solution completes, select the Solution branch, insert an Equivalent Stress object, and evaluate.





As the plot shows, there are potential problem areas (high stress gradients combined with a coarse mesh) around the blend near the top of housing. Submodeling is an efficient approach to investigate these areas in greater detail. For this problem, we have used ANSYS DesignModeler to slice out the volumetric region to be used for the submodel.

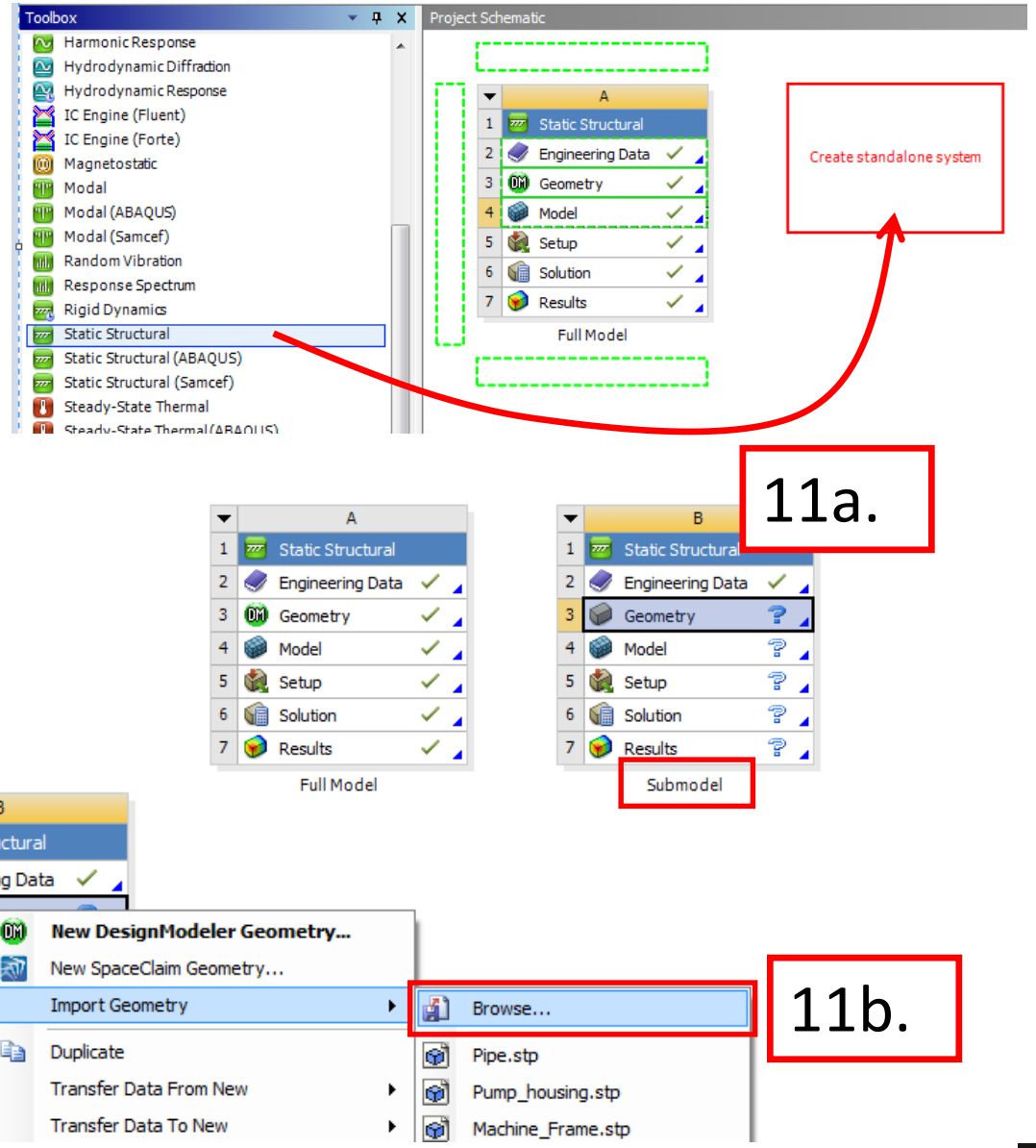
Project Schematic

Return to the Project Schematic.

11. Set up the Submodel:

a. Drag and drop a new standalone Static Structural system onto the project and rename it to "Submodel."

b. Select the Geometry cell in the new system, then RMB > Import Geometry > Browse… > "Submodelv150.stp" > Open.



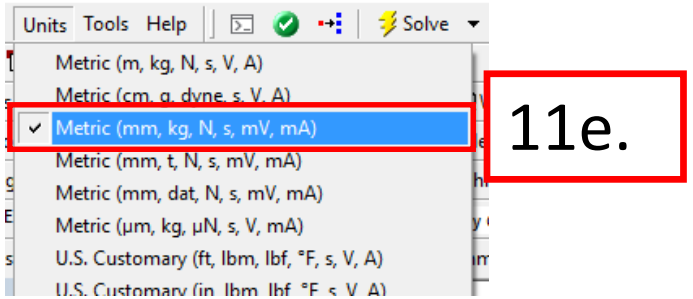
Project Schematic

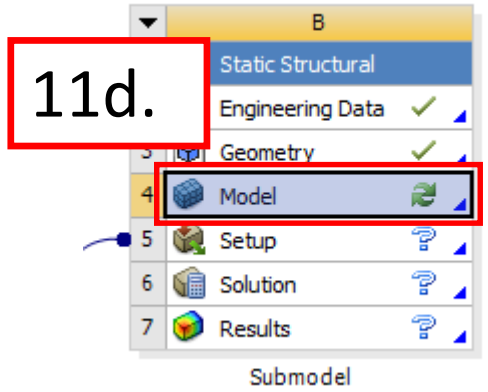
11. Continue …

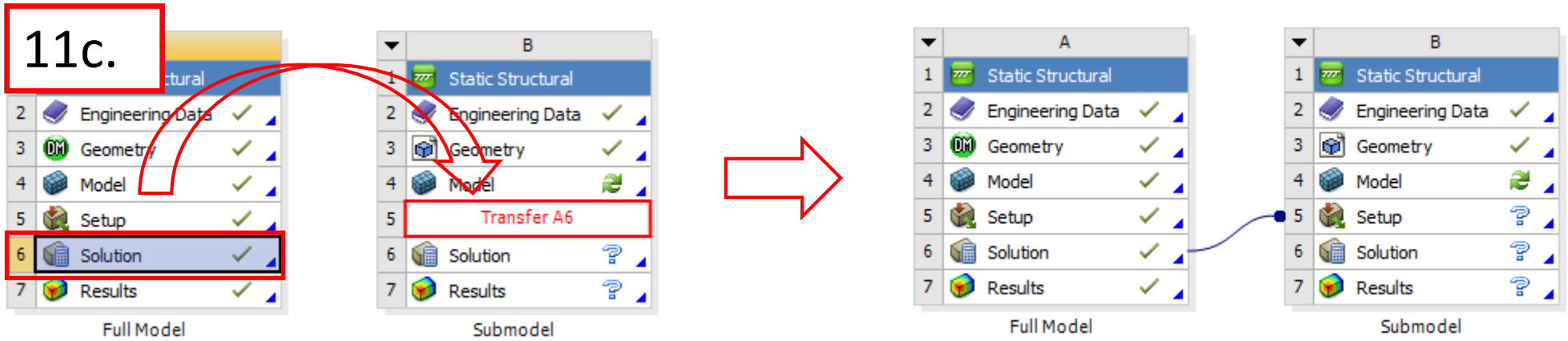
c. **Drag and drop the Solution cell from the full model onto the Setup cell of the submodel.**

d. **Double-click the Model cell in system Submodel to open a second instance of Mechanical.**

e. **When the new Mechanical window opens, verify that units are set to "Metric (mm, kg, N, s, mV, mA)."**







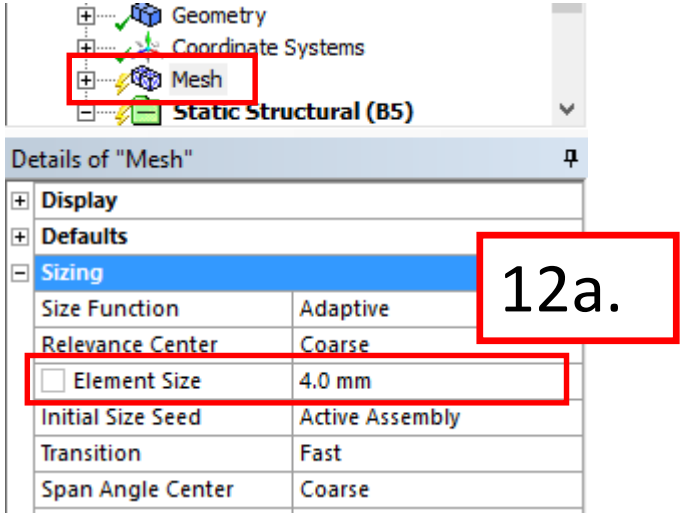
Preprocessing—Submodel

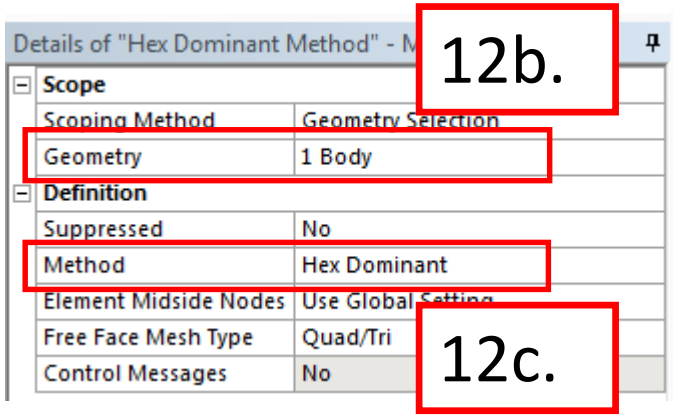
12. Define some mesh controls:

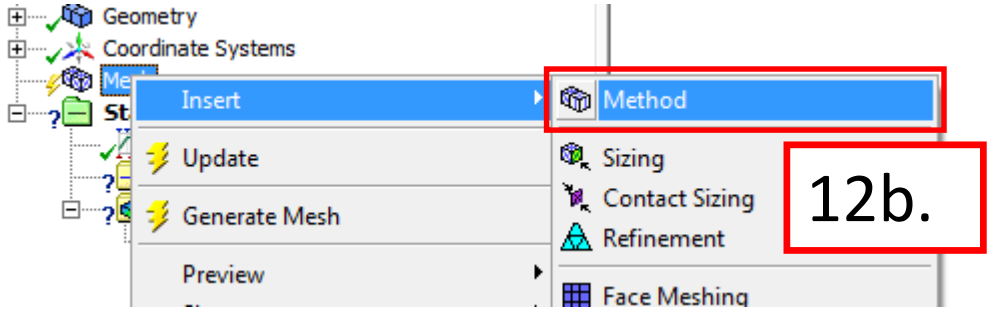
a. Select the Mesh branch and, in the Details view, expand Sizing and set Element Size to “4 mm.”

b. Select the Mesh branch, then RMB > Insert > Method and scope it to the body.

c. Change the method to “Hex Dominant.”





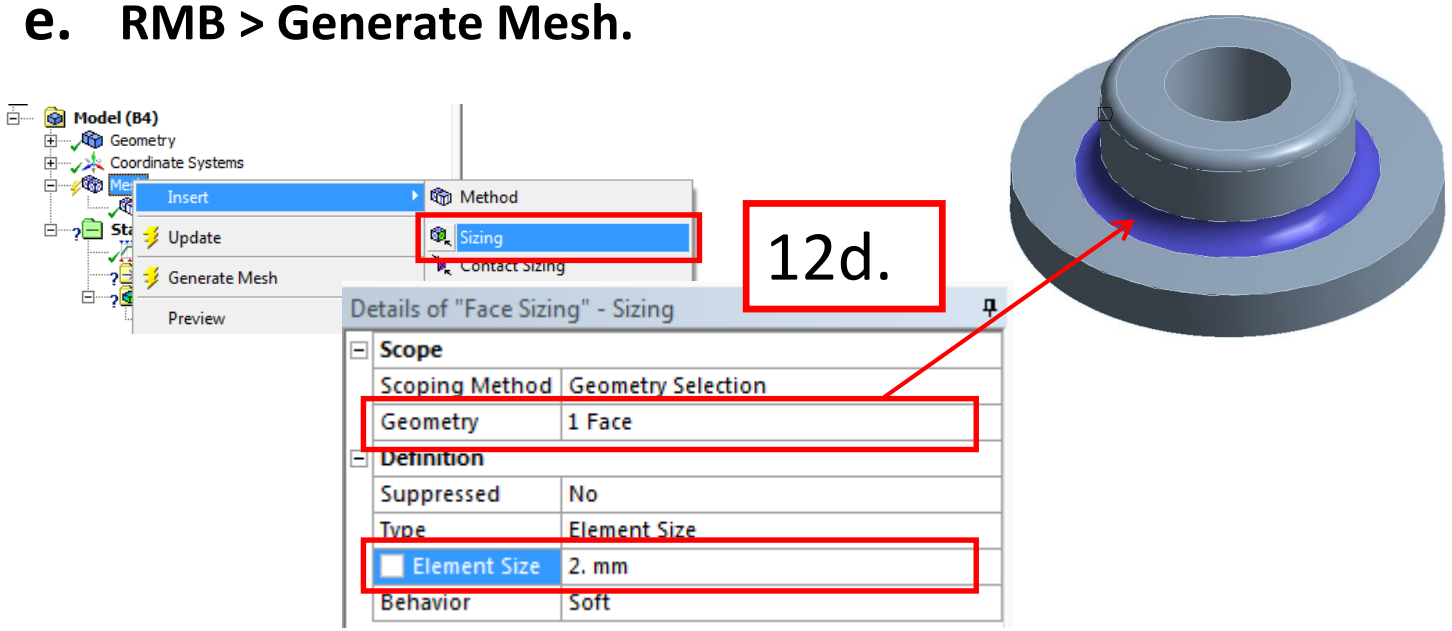


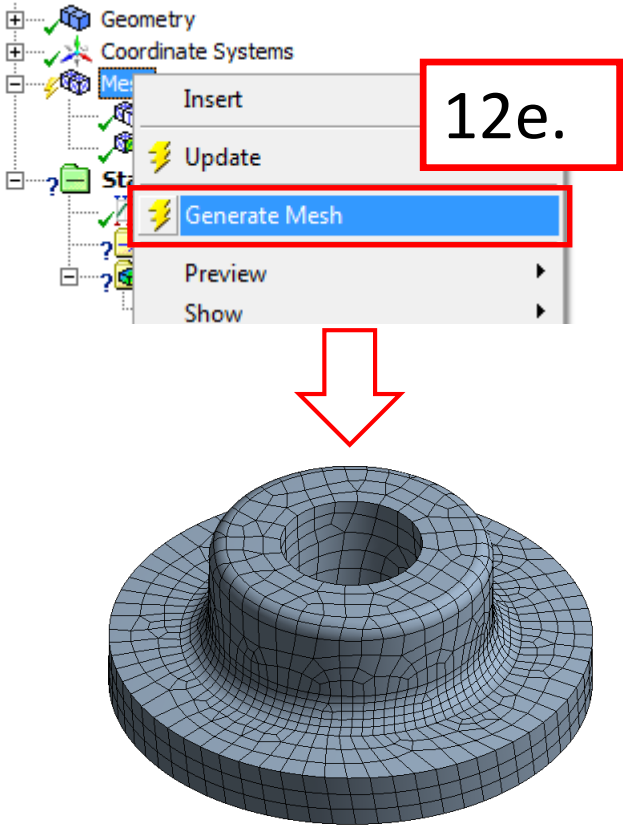
Preprocessing—Submodel

12. Continue …

d. Select the Mesh branch, then RMB > Insert > Sizing, set Element Size to “2 mm,” and scope the face shown.

e. RMB > Generate Mesh.



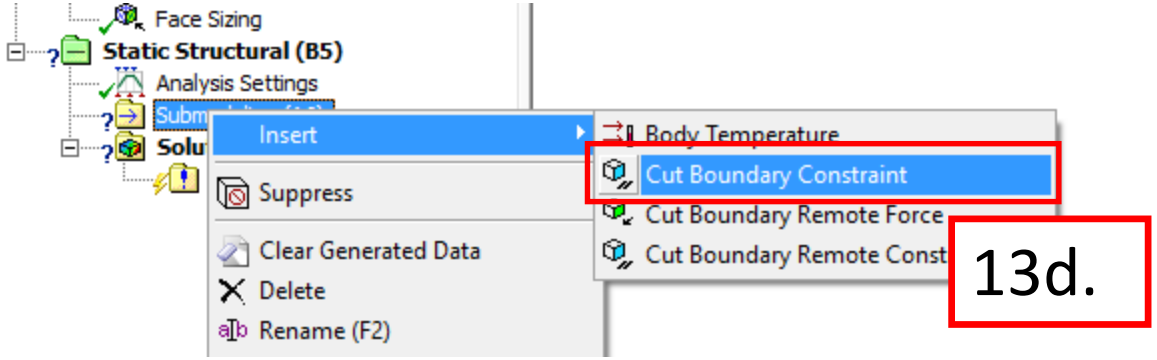


Note: In the interest of time, we have not refined the mesh as much as one might in actual practice.

Preprocessing—Submodel

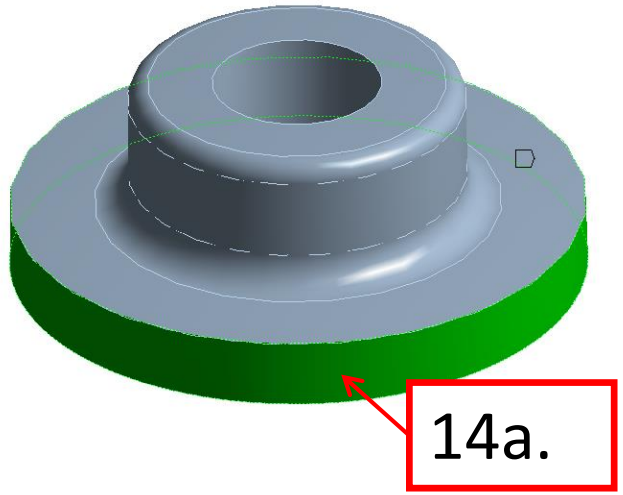
Note that the Outline for the Submodeling analysis contains a “Submodeling” branch.

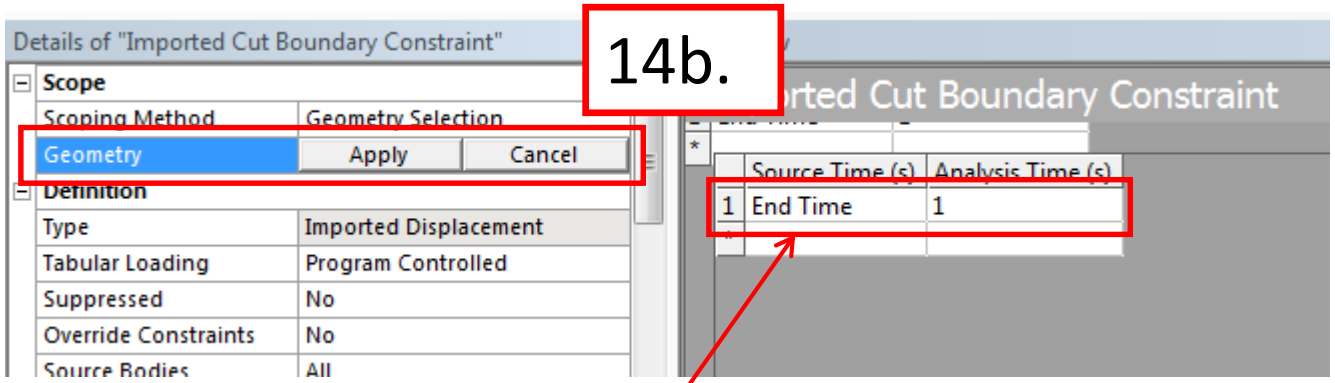
13. Import displacements from the full model: select the Submodeling branch, then RMB > Insert > Cut Boundary Constraint.

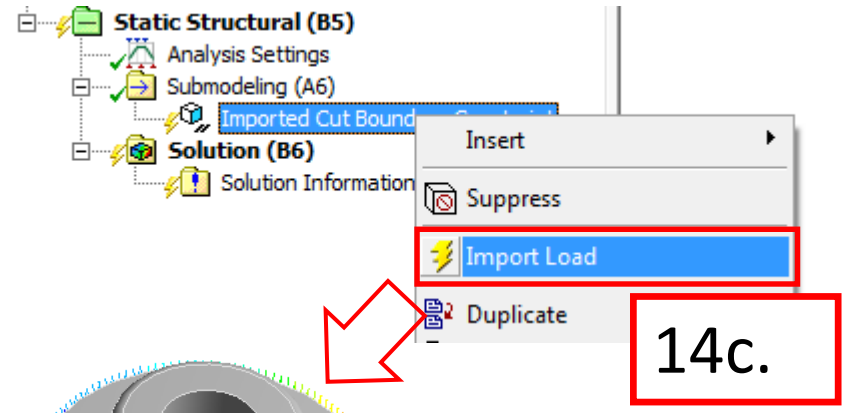


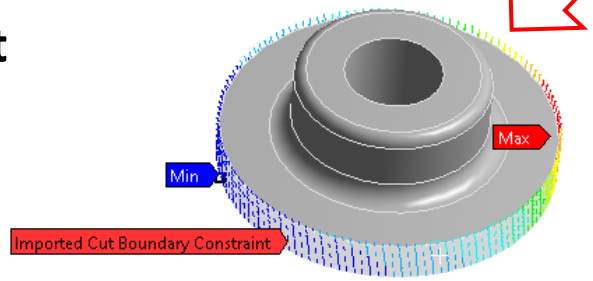
Preprocessing—Submodel

14. Map displacements from the full model onto the submodel:
    a. Select the face on the submodel that represents the cut boundary.
    b. In the Imported Cut Boundary Constraint Details view, click “Apply” to complete the geometry selection.
    c. Select Imported Cut Boundary Constraint, then RMB > Import Load.









Note: Since the solution of the full model was static, the default import is from the “End Time.” If the full model had been a multistep or transient analysis, we could have chosen from any of the solution points to map from.

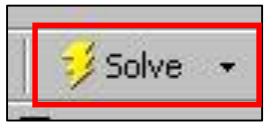
Preprocessing—Submodel

Add boundary conditions to match the full model.

15. Select the Static Structural branch and add a force:
    a. Select the face shown, then RMB > Insert > Force.
    b. In the Details view, set Define By to Components and set X Component to 200 N.

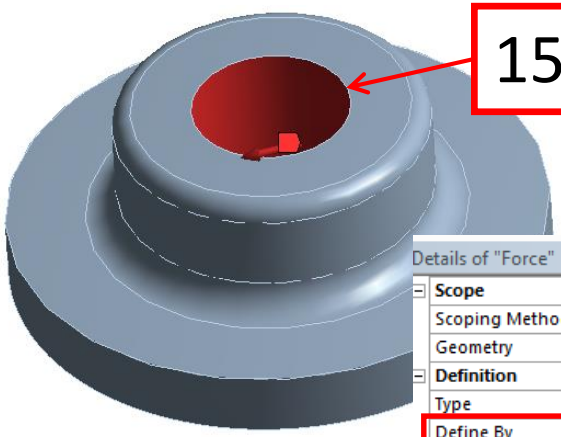
16. Select the Static Structural branch and add a pressure:
    a. Select the bottom face of the submodel shown, then RMB > Insert > Pressure.
    b. In the Details view, set Magnitude to “1000 MPa.”

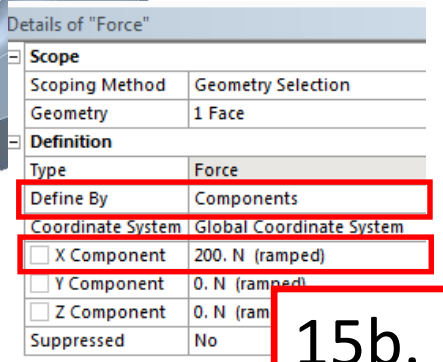
17. Solve.



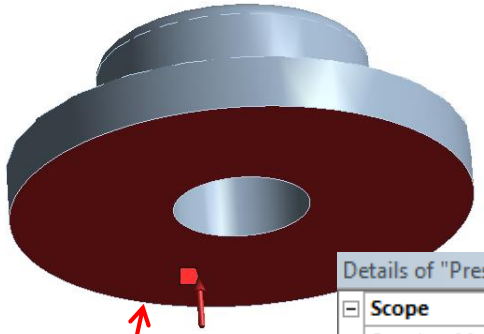
17.

Note: The full model contained both compression only supports and frictionless supports. Since no part of the submodel contains regions where these supports were applied, we do not include them. Their effect on the submodel is communicated via the displacement mapping.

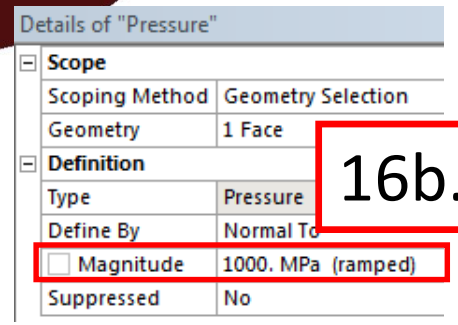




15b.



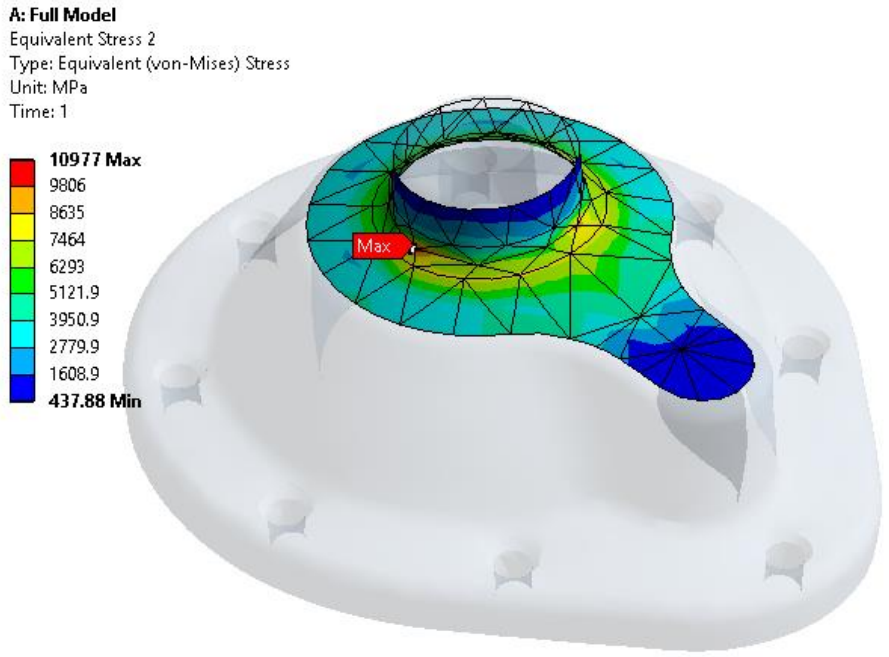
16a.

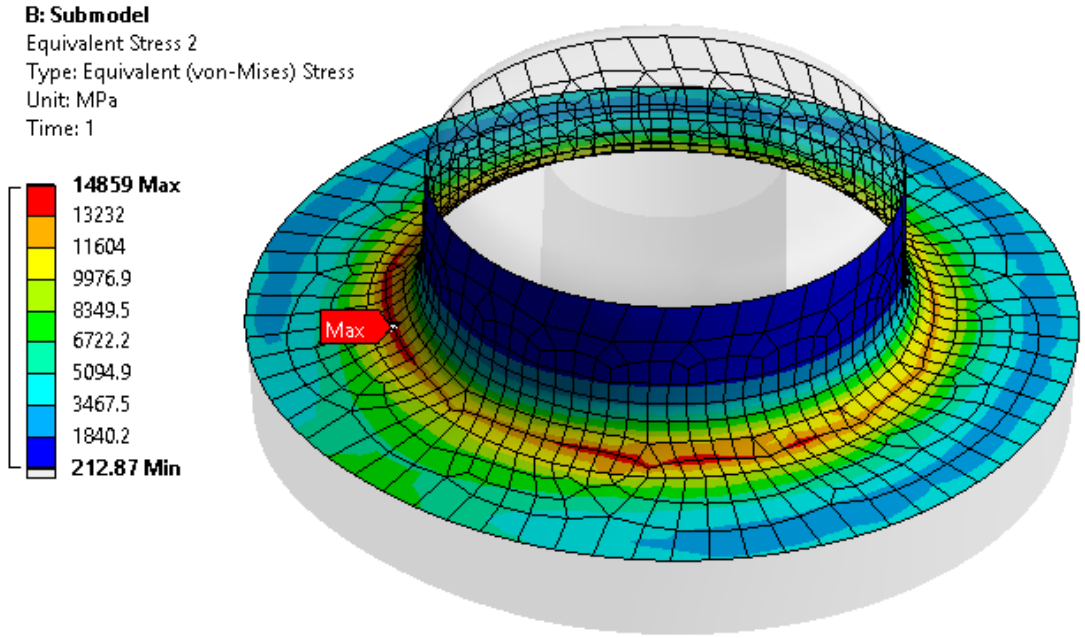


16b.

Postprocessing—Submodel

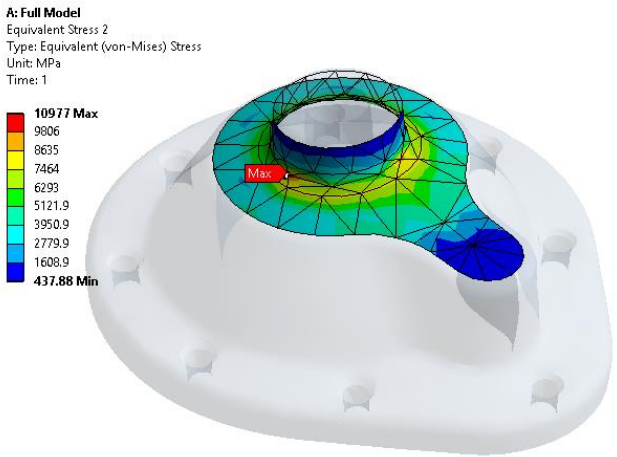
When the solution completes, select the Solution branch, insert an Equivalent Stress object, evaluate, and compare it to the full model results. A significant change can be seen: the maximum reported stress increases by approximately 35%.

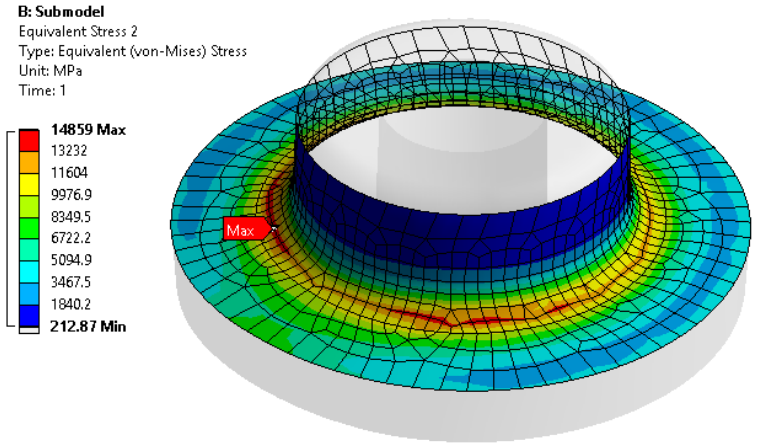




Verification

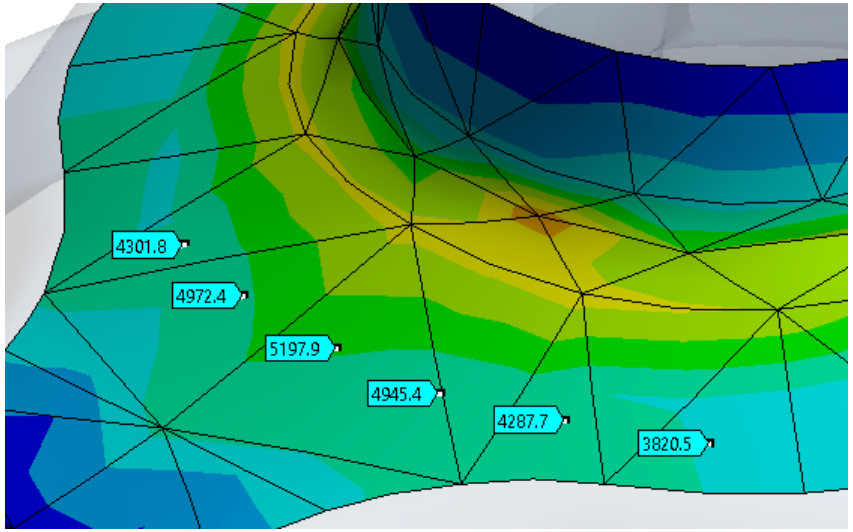
Submodeling introduces an additional approximation into the finite element solution process, and good practice dictates that this approximation be checked as part of results verification. One of the essential assumptions of the submodeling approach is that solution field in the region of the cut boundaries should be the same in both the full model and the submodel. Our verification goal, then, is to evaluate the results of both models on or near the cut boundaries to make sure they are in reasonable agreement. If they are not in agreement, it is an indication that the cut boundaries are too close to the stress concentration that we are trying to resolve.

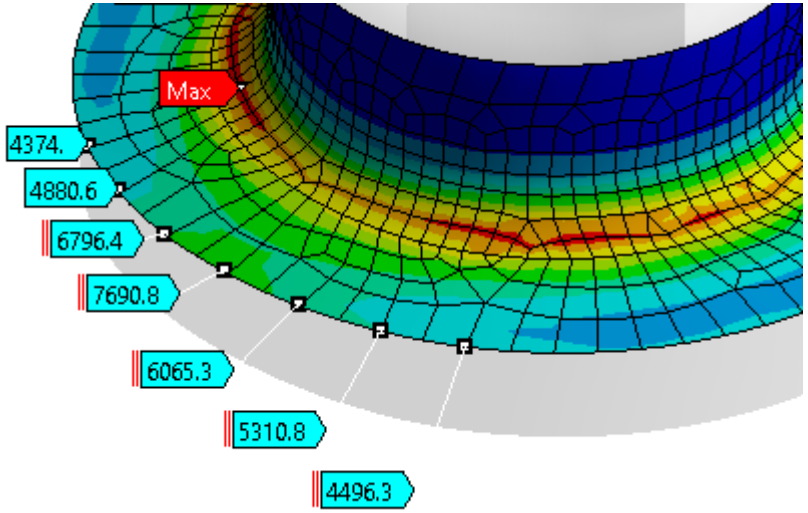




Verification

The most common verification technique is simply to query both models using probes to assess agreement of stress results near the cut boundaries. While this is quick and easy, a drawback is that it relies on “eyeball” location for many of the probes. A more precise technique would be to define construction paths along the locations of the cut boundaries in the two models and create path plots for comparison.

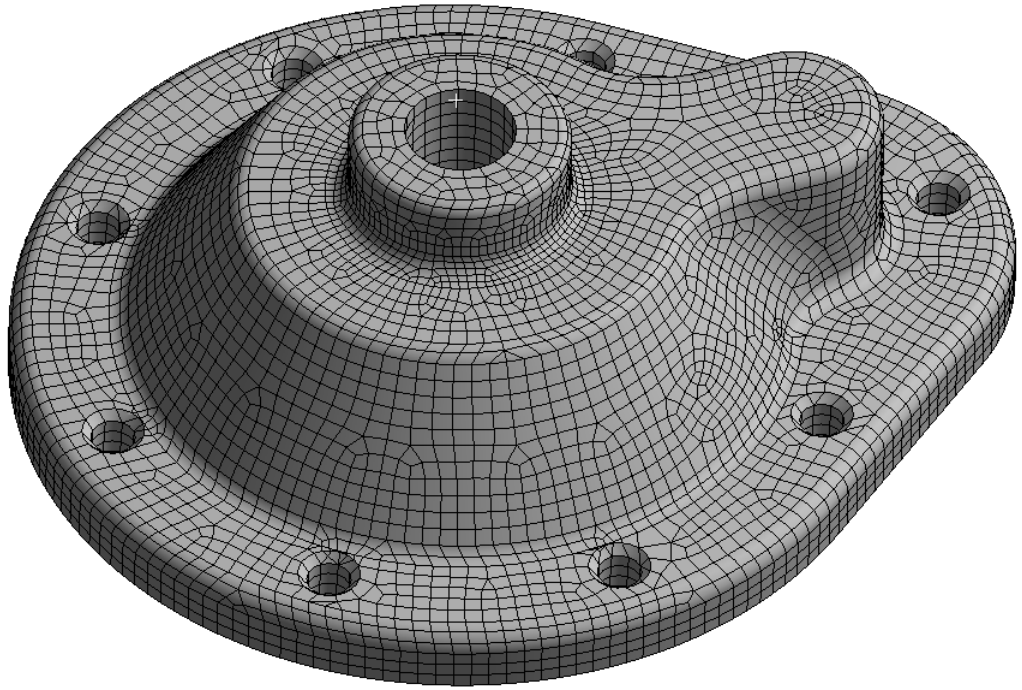




Go Further!

If you find yourself with extra time, try the following:

1. **Refine the full model mesh in successive steps with the objective achieving the same stress results as reported by the submodel. Compare elapsed time for the various approaches.**



```
+------------------ E N D   A N S Y S   S T A T I S T I C S ------------------+

*-------------------------------------------------------------------------*
|                                                                         |
|                          ANSYS RUN COMPLETED                            |
|                                                                         |
|-------------------------------------------------------------------------|
|                                                                         |
|                                                                         |
|                                                                         |
|-------------------------------------------------------------------------|
|                                                                         |
| Database Requested(-db)    512 MB    Scratch Memory Requested      512 MB |
| Maximum Database Used       65 MB    Maximum Scratch Memory Used   341 MB |
|                                                                         |
|-------------------------------------------------------------------------|
|                                                                         |
|        CP Time      (sec) =         713.985                             |
|        Elapsed Time (sec) =         310.000                             |
|                                                                         |
*-------------------------------------------------------------------------*
```

ANSYS®

Workshop 08.2: Submodeling

END