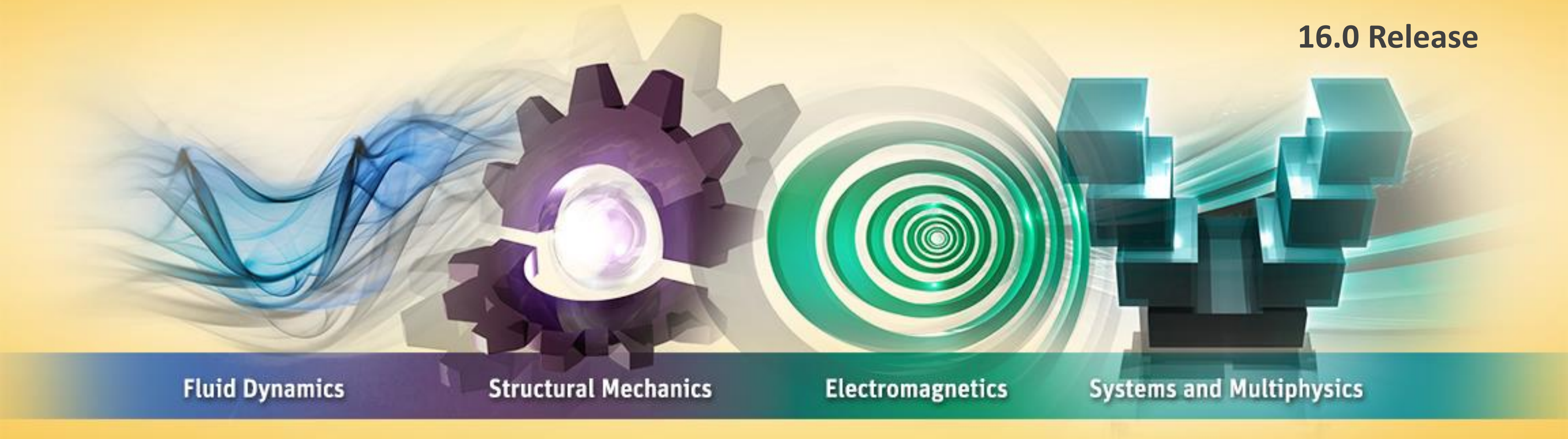
Appendix B
Submodeling

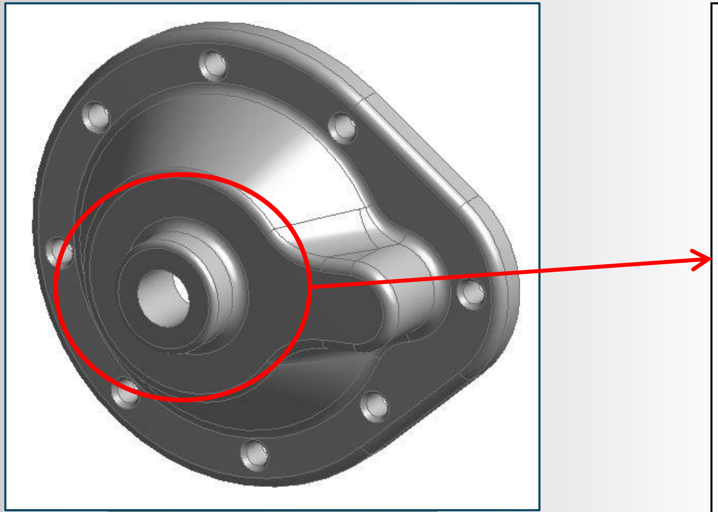
16.0 Release



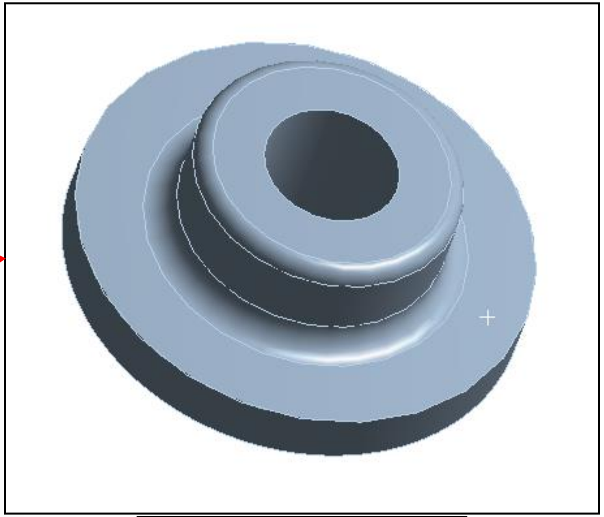
Introduction to ANSYS Mechanical

Workshop AppB – Submodeling

- **Workshop WSAppB – Submodeling**
- **Goal:**
  - **Solve a full model (coarse mesh) and then setup and solve a submodel representing a portion of the full model (fine mesh).**



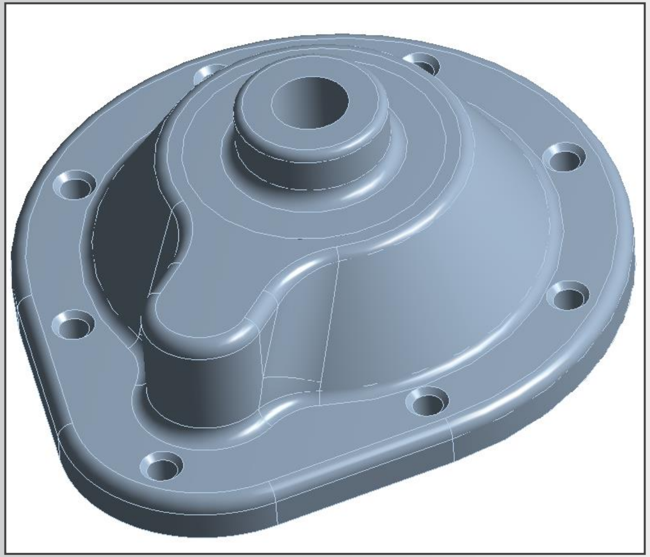
Full Model



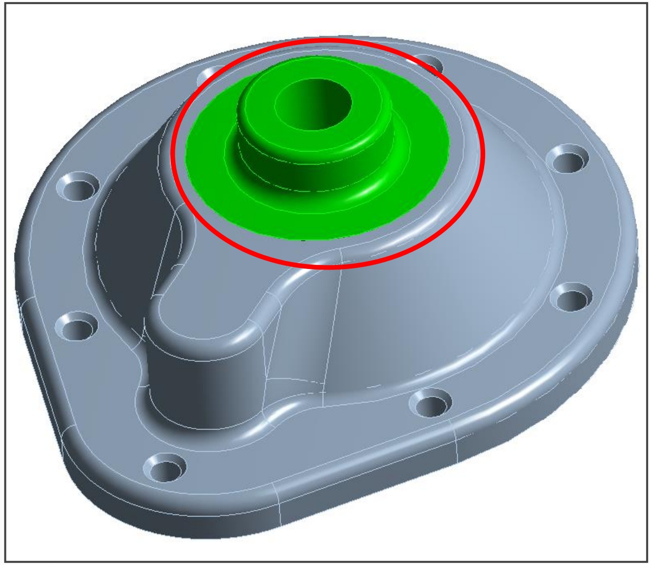
Submodel

Approach

Submodeling requires the use of 2 geometry models. One model to represent the full geometry and another representing a portion of the full model. For this exercise we used the ANSYS DesignModeler application to slice a piece from the full model.



Full Model

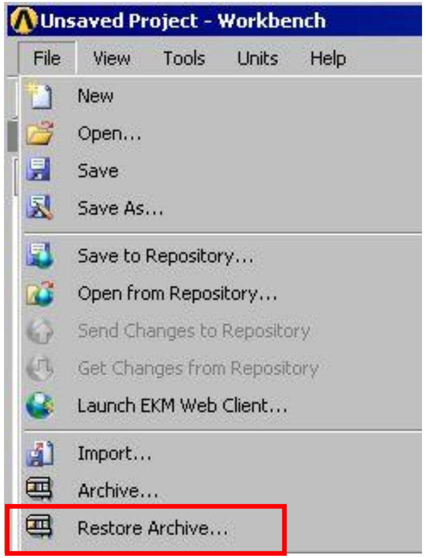


Submodel

Project Schematic

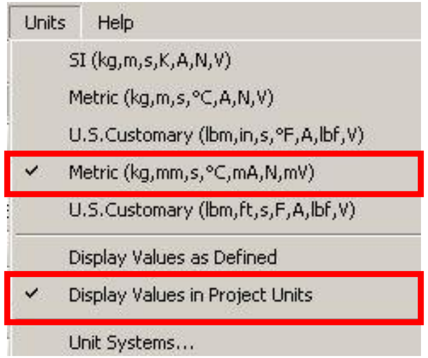
Begin a new Workbench session and, from the Project page, choose “Restore Archive . . . “ and browse to the file “Submodeling_WS_APPXB.wbpz” and Open (location provided by instructor).

When prompted, “Save” using the default name and the same location.



From the “Units” menu verify:

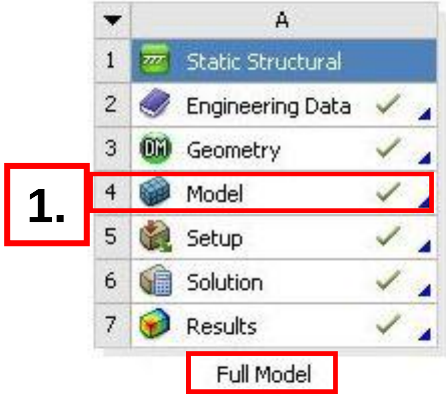
- Project units are set to “Metric (kg, mm, s, ºC, mA, N, mV).
- “Display Values in Project Units” is checked (on).



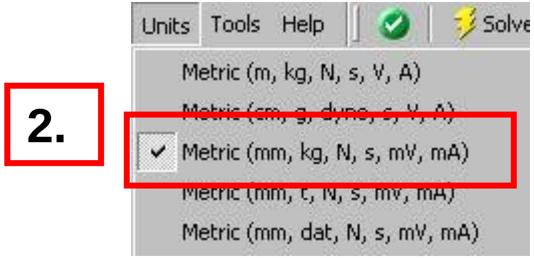
. . . Project Schematic

When the archive is opened note the existing static structural system has been renamed "Full Model".

1. **From the Static Structural system double click (or RMB > Edit) the "Model" cell.**



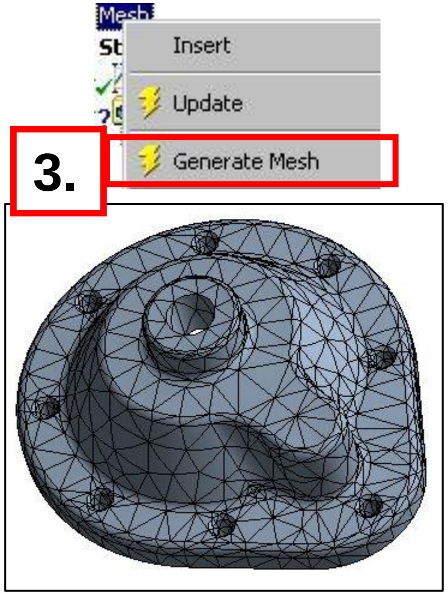
2. **When Mechanical opens, verify the units are set to "Metric (mm, kg, s, mV, mA)".**

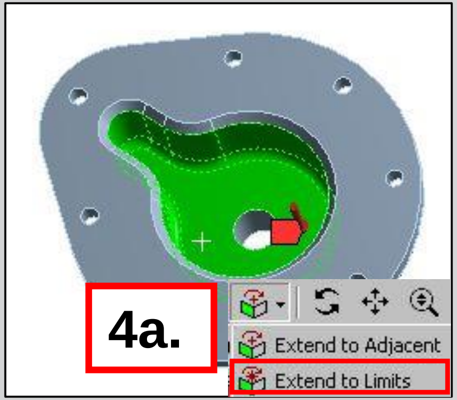


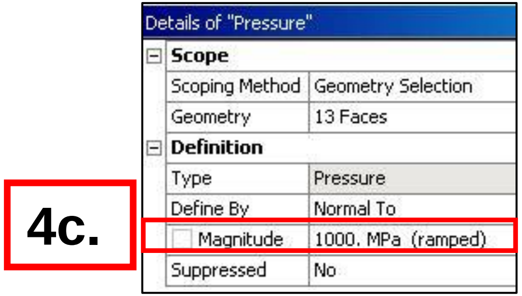
Preprocessing

3. Highlight the mesh branch, RMB > Generate Mesh.

4. Apply a pressure load:
   a. With the static structural branch highlighted, select one of the interior surfaces of the housing and choose "Extend to Limits" (should result in 13 faces).
   b. "RMB > Insert > Pressure".
   c. Enter Magnitude "1000" MPa in the details.





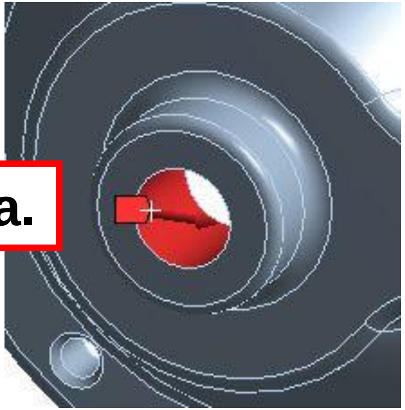


. . . Preprocessing

5. **Add a force to the housing:**
   a. Select the cylindrical face of the center hole in the housing.
   b. "RMB > Insert > Force".
   c. Define by components and enter 200 N X component.

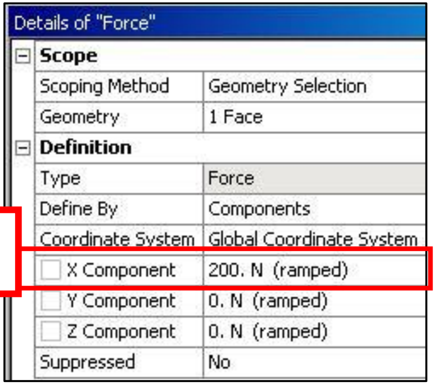
6. **Add a compression only support:**
   a. Select the planar surface on the back of the housing.
   b. "RMB > Insert > Compression Only Support".

5a.



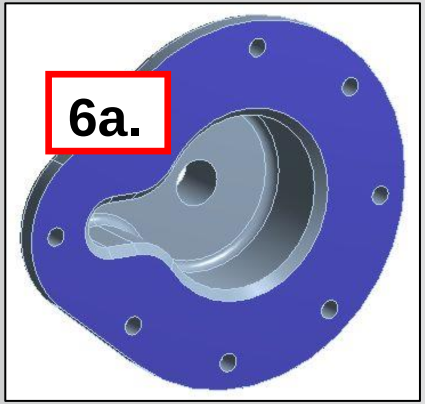
5b.

5c.

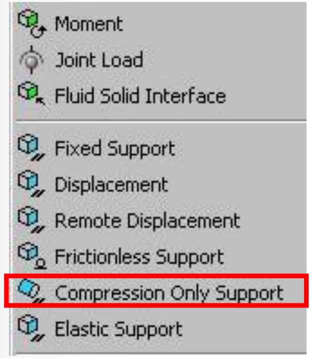


Details of "Force"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	200. N (ramped)
Y Component	0. N (ramped)
Z Component	0. N (ramped)
Suppressed	No

6a.

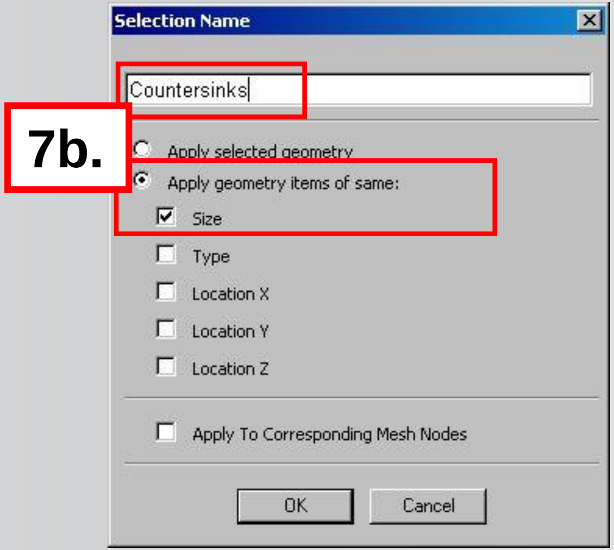


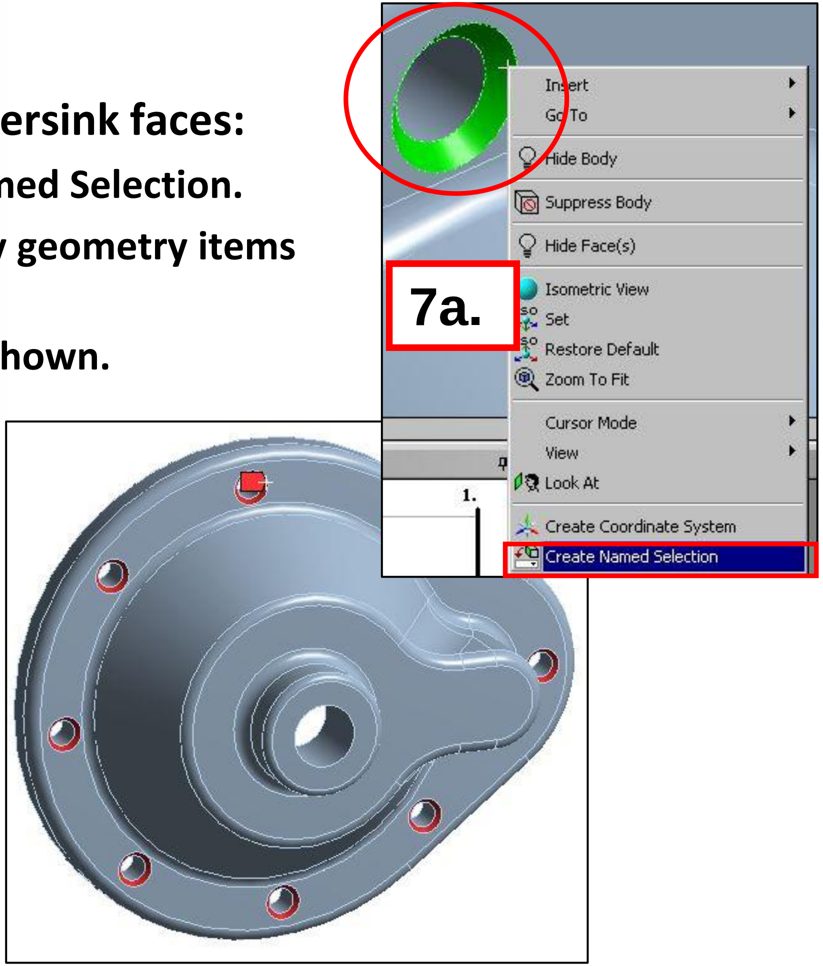
6b.



. . . Preprocessing

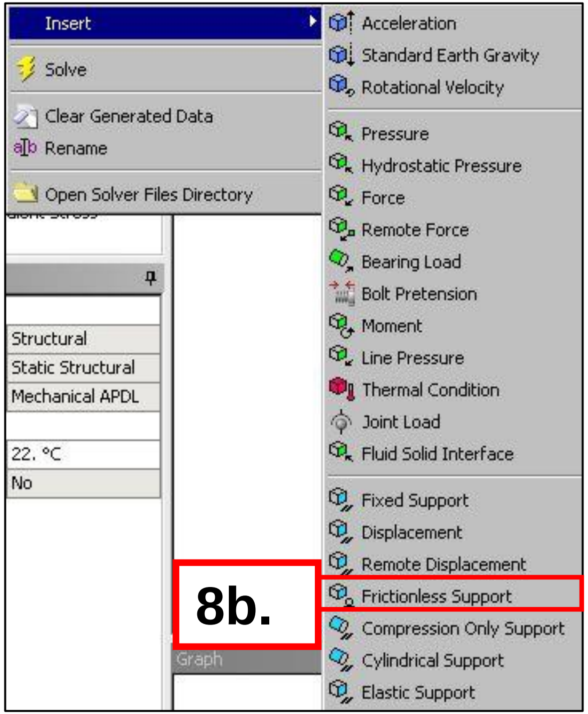
7. Create a named selection containing the countersink faces:
   a. Highlight one countersink face, RMB > Create Named Selection.
   b. In the dialog box enter “Countersinks” and “Apply geometry items of same: size” and OK.
      – The resulting NS should contain 8 faces as shown.

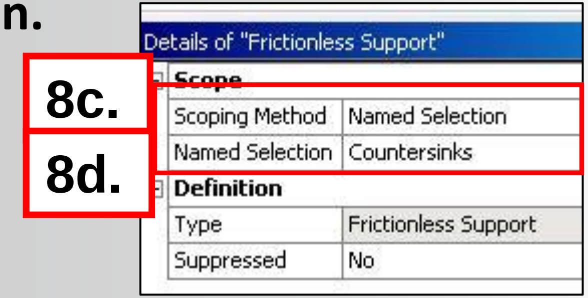




. . . Preprocessing

8. Add frictionless supports to countersink faces:
   a. Highlight the Static Structural branch.
   b. RMB > Insert > Frictionless Support.
   c. In the details change scoping method to “Named Selections”.
   d. Select the “Countersinks” named selection.

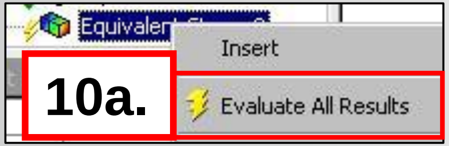


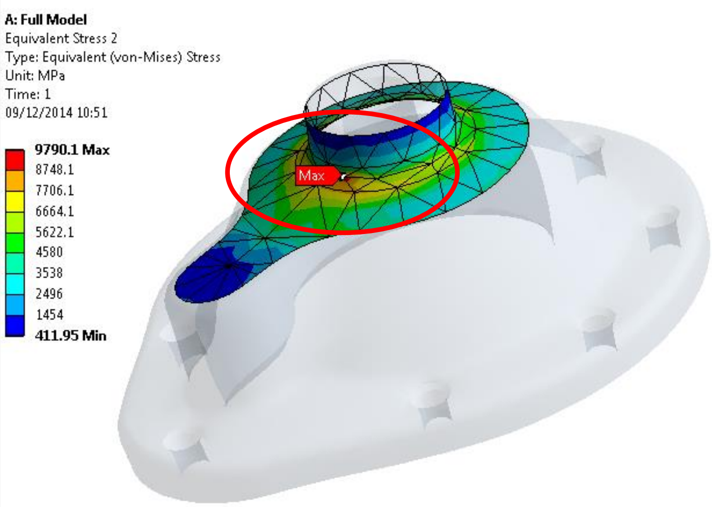


9. Solve

Full Model Solution

10. When the solution completes highlight the Solution branch:
    a. Insert an Equivalent Stress object, RMB > Evaluate All Results.





As the plot shows the potential problem areas are around the blend on the top of housing. An efficient approach to investigate these areas in more detail is to create a submodel of this part of the geometry. In this example we have used ANSYS DesignModeler geometry application to slice out a portion of the model which we will use next.

ANSYS

Submodeling Schematic

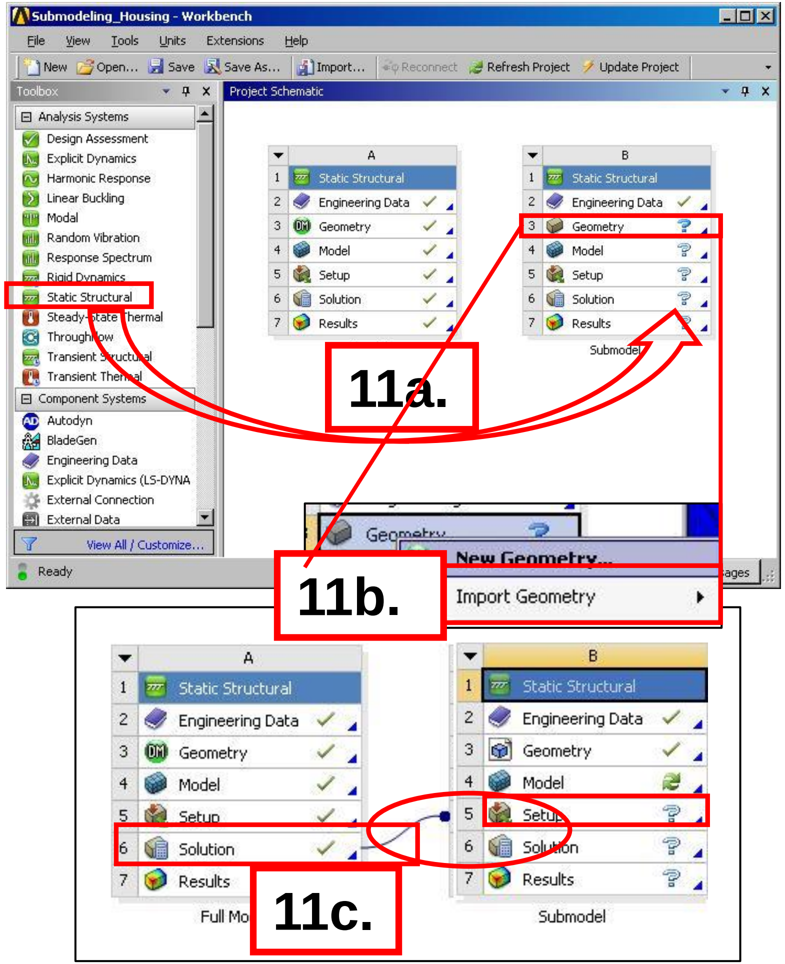
11. Set up the Submodel in the project:

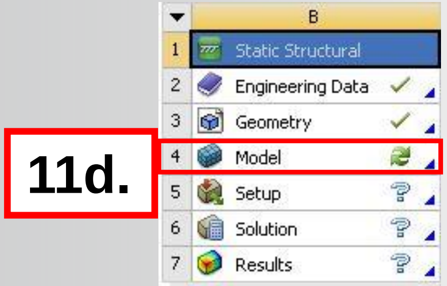
   a. Drag & Drop a new standalone Static Structural system into the project and rename “Submodel”.

   b. From the geometry cell in the new system RMB > Import Geometry > browse to the file “Submodelv150.stp”.

   c. Drag & Drop the Solution cell from the full model onto the Setup cell in the submodel.

   d. Double click the Model cell to start Mechanical.





. . . Preprocessing

12. When Mechanical opens, mesh the submodel:

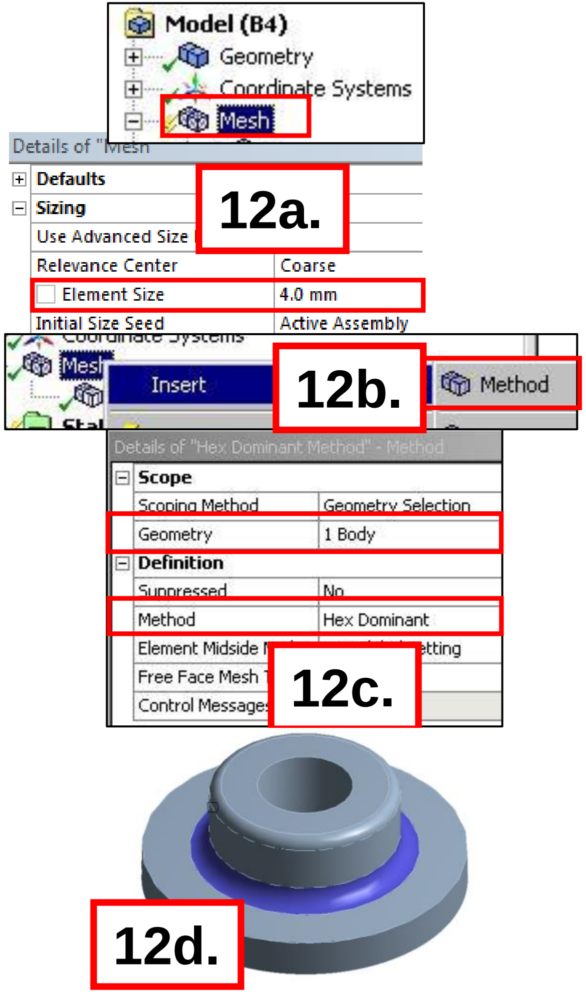
a. Highlight the Mesh branch and, in the Sizing section of the details enter Element Size = 4 mm.

b. From the Mesh branch RMB > Insert > Method and scope it to the body of the geometry.

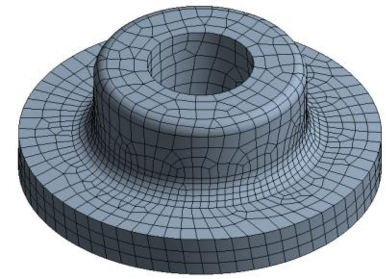
c. Change the method to "Hex Dominant".

d. From the Mesh branch RMB > Insert > Sizing, define an element size = 2mm and scope the face as shown on the screenshot

e. From the mesh branch RMB > Generate Mesh.



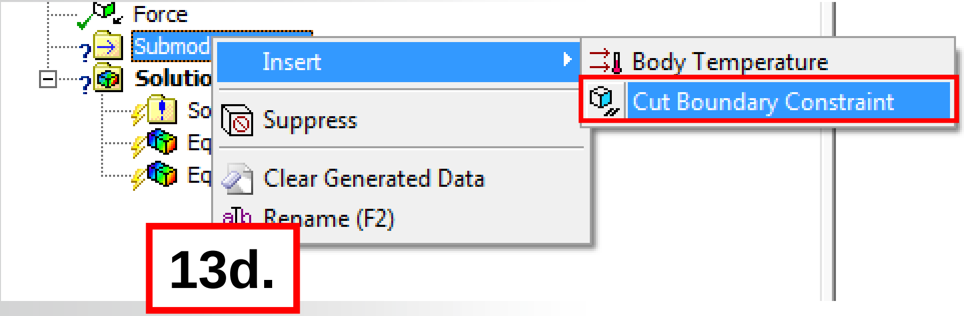
Note that in the interest of time we have not refined the mesh as much as one might in actual practice.



Importing Displacements

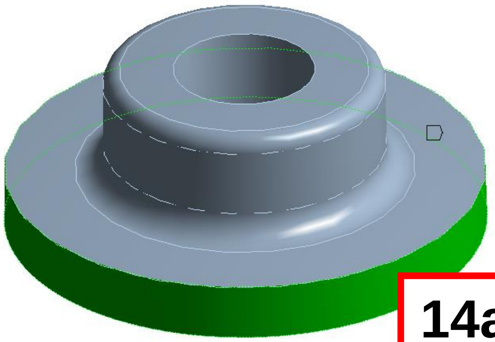
In the new Mechanical session you will see a “Submodeling” branch.

13. Import displacements from the full model:
    a. Highlight “Submodeling” RMB > Insert > Cut Boundary Constraint.

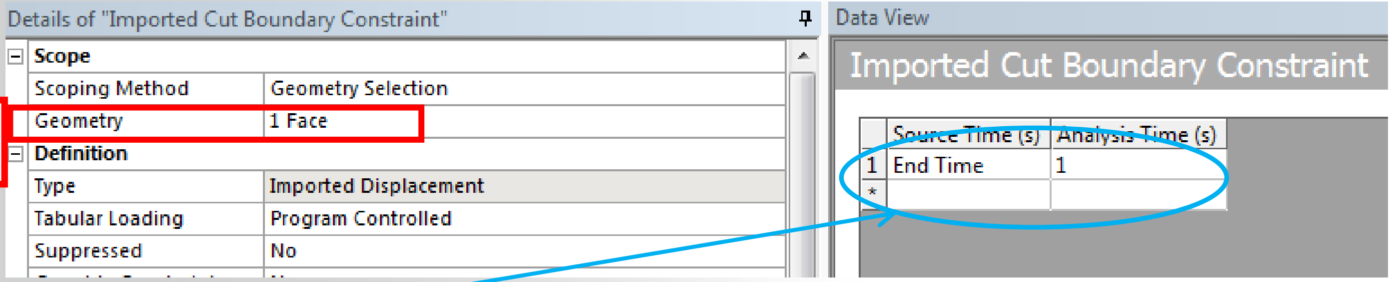


. . . Importing Displacements

14. Map displacements from the full model onto the submodel:
    a. Select the face on the model representing the cut boundaries.
    b. In the details of the Imported Cut Boundary Constraint "Apply" the selected geometry.
    c. Highlight "Imported Cut Boundary Constraint " RMB > Import Load.

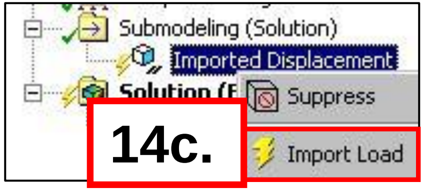


14a.



14b.

Since the solution of the full model was static, the default import is from the "End Time". If the full model had been a multi-step or transient analysis we could have chosen any solution points to map from.



14c.

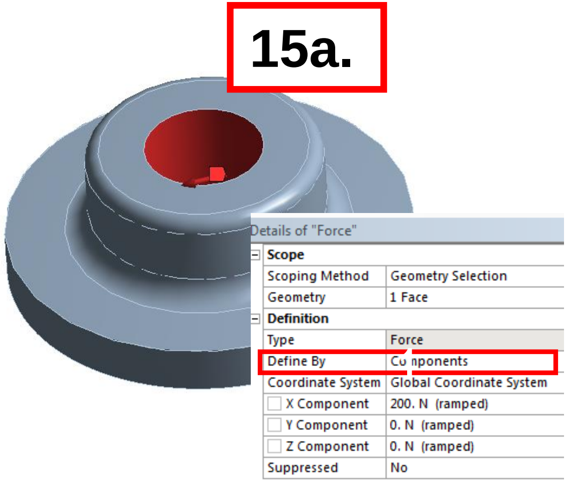
  February 27, 2015

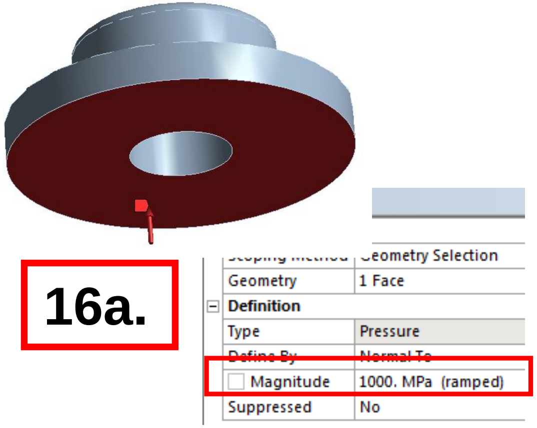
Adding Boundary Conditions

Add boundary conditions to match the full model.

15. **Highlight the static structural branch and add a force:**
    a. Select the face, RMB > Insert > force
    b. Define by components and enter 200 N (X component)

16. **Highlight the static structural branch and add a pressure:**
    a. Select the bottom face of the submodel, RMB > Insert > Pressure.
    b. Enter Magnitude "1000" Mpa in the details.



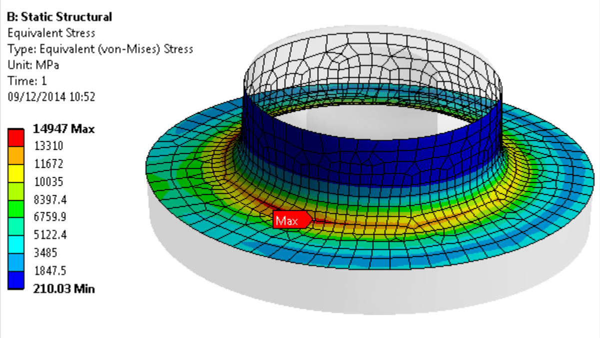


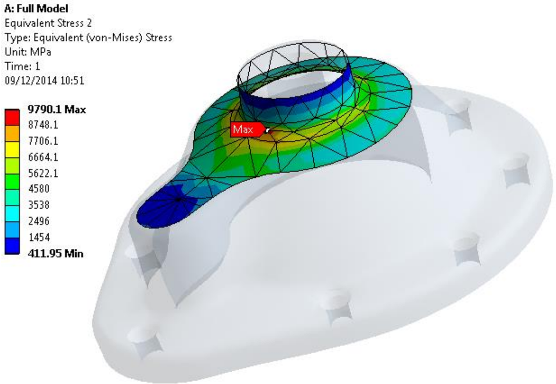
Recall that the full model contained both compression only support and frictionless support. Since no part of the submodel contains regions where these supports were applied we do not add them. Their effect is seen in the displacements mapped to the cut boundaries.

Solving the Submodel

17. Solve.

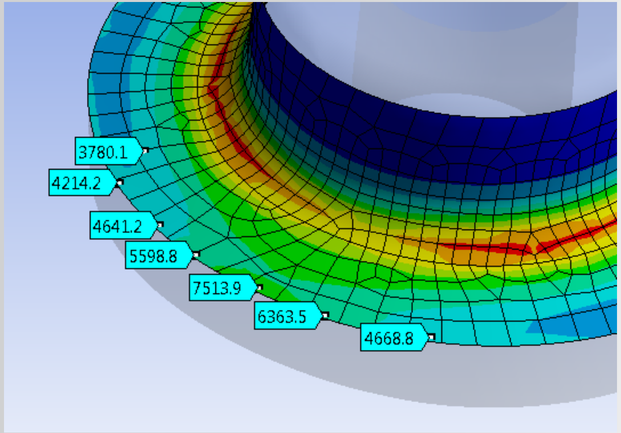
If we add equivalent stress to the submodel and compare it to the full model, a significant change can be seen (> 30%). A part of any submodeling solution should include some form of verification regarding the location of the cut boundaries. The goal is to evaluate the results of both models on or near the cut boundary to make sure they are in reasonable agreement. If they do not agree it indicates the cut boundaries are too close to the stress concentration.

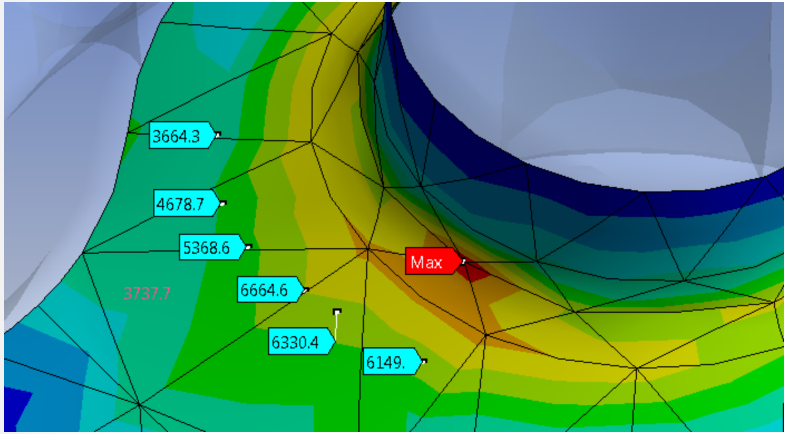




Cut Boundary Verification

One technique might be to simply query the model using probes to get a feel for how well the 2 results agree. While this is quick and easy, a drawback is it relies on an "eyeball" location for the probes. Another technique is to compare path plots.

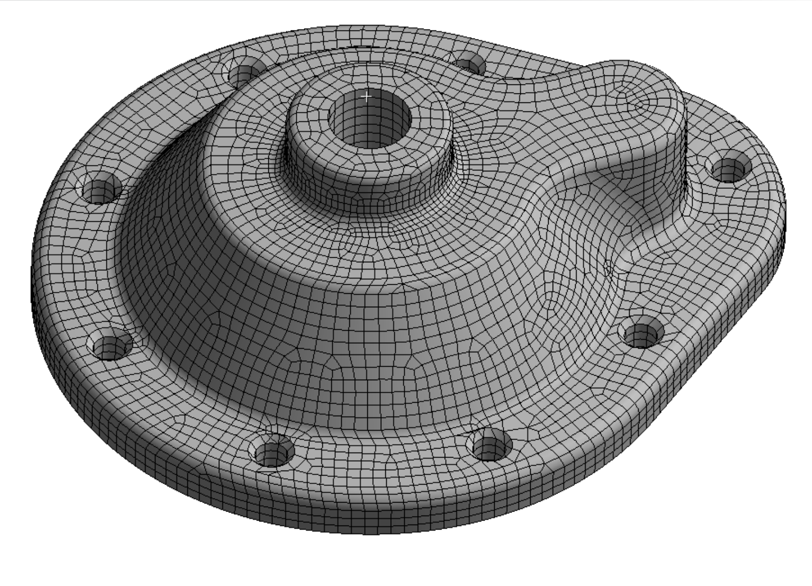




Although not part of the workshop, if time permits, you may complete the verification techniques shown on the remainder of the pages.

…To go further

Try to have similar results on the full model and compare results and elapsed time. To do that, for example, you can define meshing options on the full model as on the submodel



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

*----------------------------------------------------------------------------*
|                                                                            |
|                          ANSYS RUN COMPLETED                               |
|                                                                            |
|----------------------------------------------------------------------------|
|                                                                            |
|          Release 15.0           UP20131014         WINDOWS x64             |
|                                                                            |
|----------------------------------------------------------------------------|
|                                                                            |
| 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       Time  =  18:38:18          |
|       Elapsed Time (sec) =        310.000       Date  =  11/14/2013        |
|                                                                            |
*----------------------------------------------------------------------------*
```

