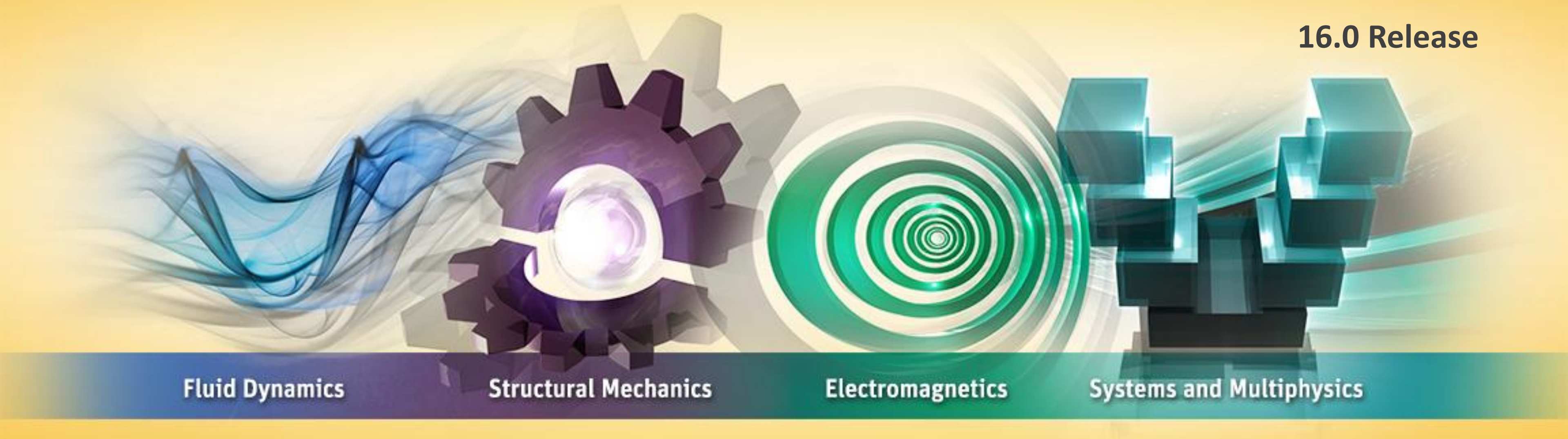
# Workshop 7.1
# Linear Structural Analysis

16.0 Release

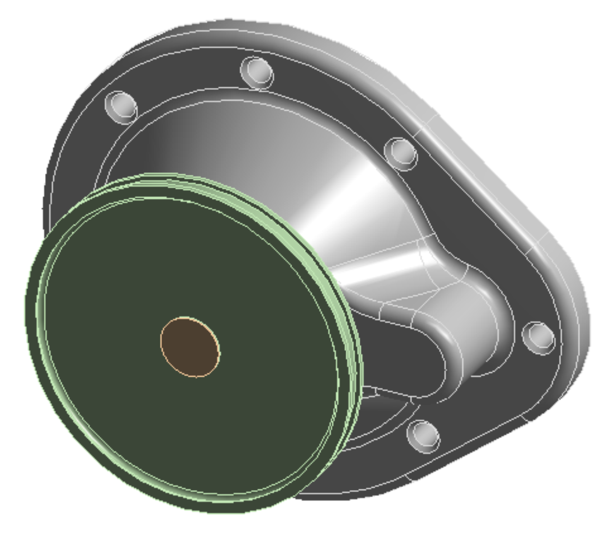



## Introduction to ANSYS Mechanical

# Goals

Workshop 7.1 consists of a 5 part assembly representing an impeller type pump. Our primary goals are to analyze the assembly with a load on the belt of 100N to test:

- That the impeller will not deflect more than 0.075mm with the applied load.
- That the use of a plastic pump housing will not exceed the material's elastic limits around the shaft bore.



# Assumptions

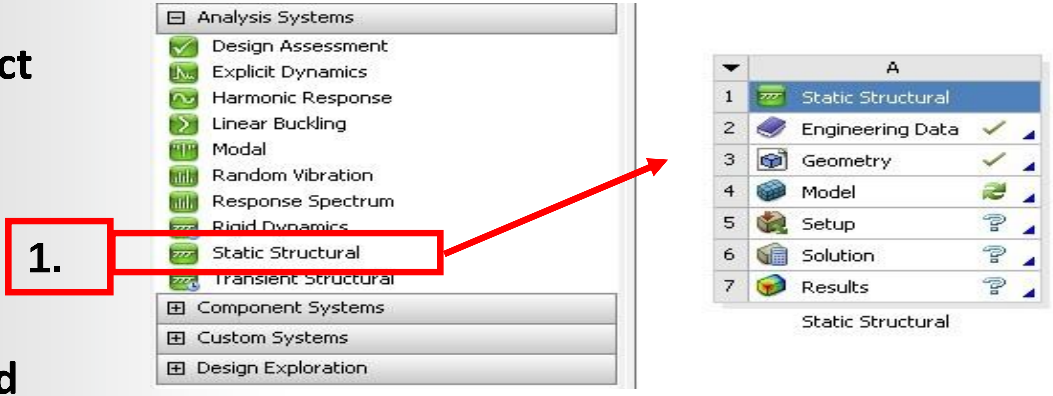
We'll assume the pump housing is rigidly mounted to the rest of the pump assembly.  To simulate this, a frictionless support is applied to the mounting face.

Similarly, frictionless surfaces on the mounting hole counter bores will be used to simulate the mounting bolt contacts.  (Note if accurate stresses were desired at the mounting holes, a "compression only" support would be a better choice).

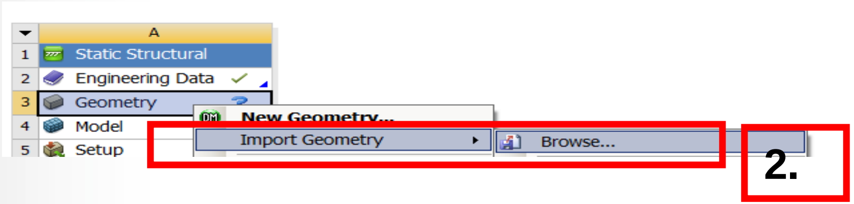
Finally, a bearing load (X = 100 N) is used on the pulley to simulate the load from the drive belt.  The bearing load will distribute the force over the face of the pulley only where the belt contact occurs.

# Project Schematic

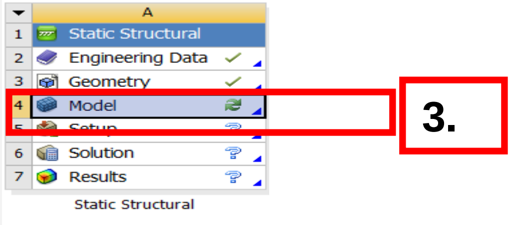
1. From the Toolbox insert a “Static Structural” system into the Project Schematic.




2. From the Geometry cell, RMB and “Import Geometry > Browse”. Import the file “Pump_assy_3.stp”.




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



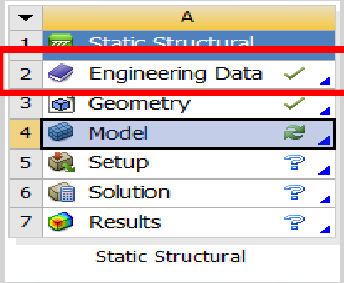

  

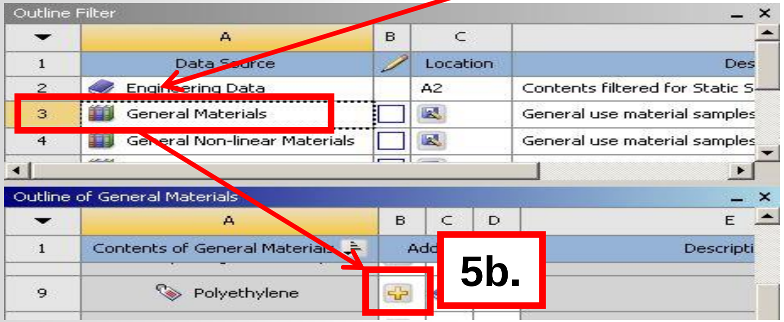
# Preprocessing

4. On the toolbar within the ANSYS Mechanical environment, select Units, and change the system's unit to
   - Metric (mm, kg, N, s, mV, mA)
   - (Note – material data is not being checked but it is good practice to confirm units)

5. Add "Polyethylene" the Engineering Data (<u>return to Workbench window</u>):
   a. Double click the Engineering Data cell.
   b. Activate the Engineering Data Sources toggle and highlight General Materials and click the + sign next to "Polyethylene".
   c. Return to Project by closing down Engineering Data.

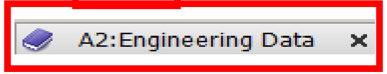

5a.



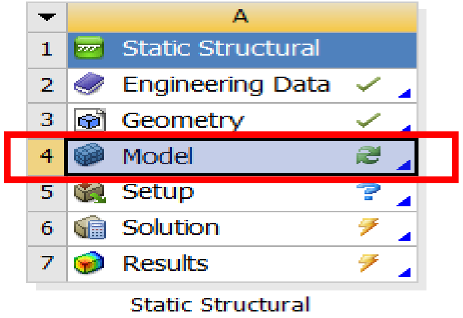

5b.

5c.



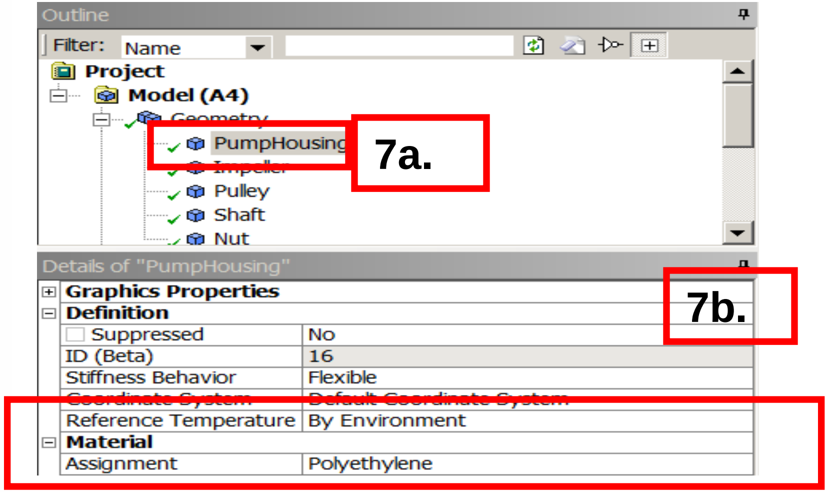
# . . . Preprocessing

6. **Refresh the Model cell:**
   a. RMB > Refresh.




**Return to the Mechanical window.**

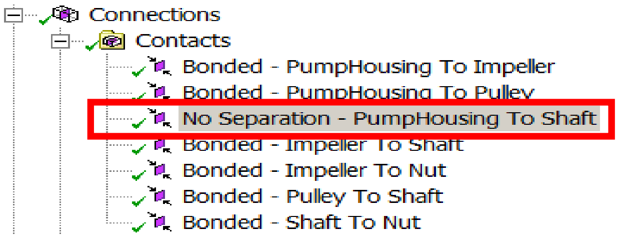
7. **Change the material on the pump housing:**
   a. Highlight "PumpHousing" under geometry.
   b. From details change the material assignment to "Polyethylene".

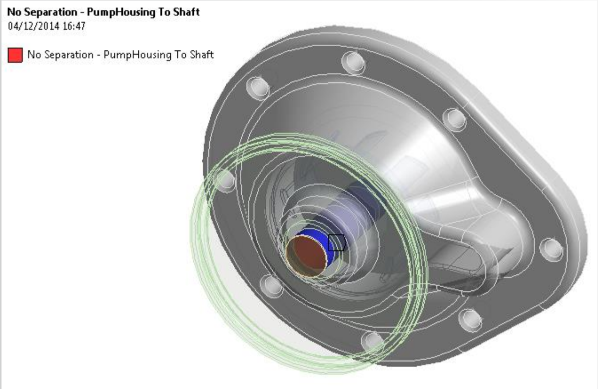


# . . . Preprocessing

8. Change the contact region behavior for one of the contact regions (shown below):
   a. Hold the shift key and highlight all the contacts, and RMB>Rename Based on Definition.
   b. From the detail window change the contact type to “no separation” for the contact PumpHousing to Shaft
      - The remainder of the contacts will be left as “bonded” (all default contact condition is set to bonded).

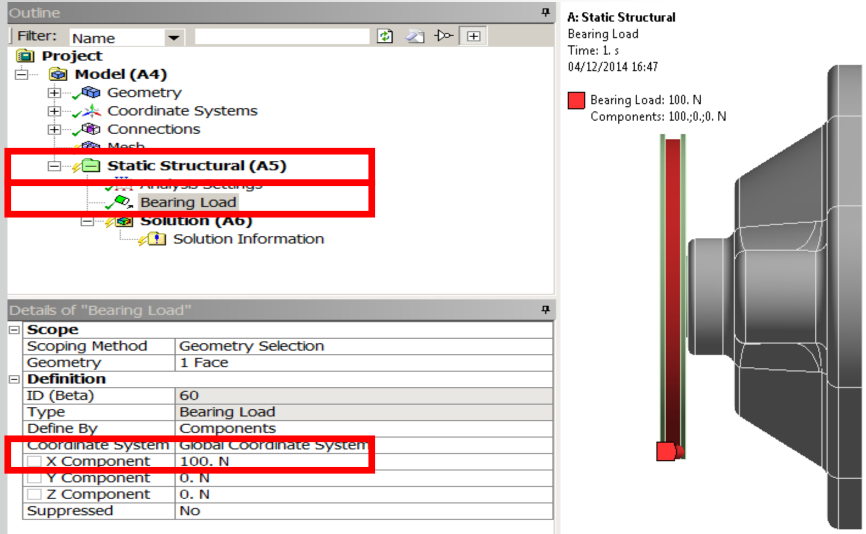


# Environment

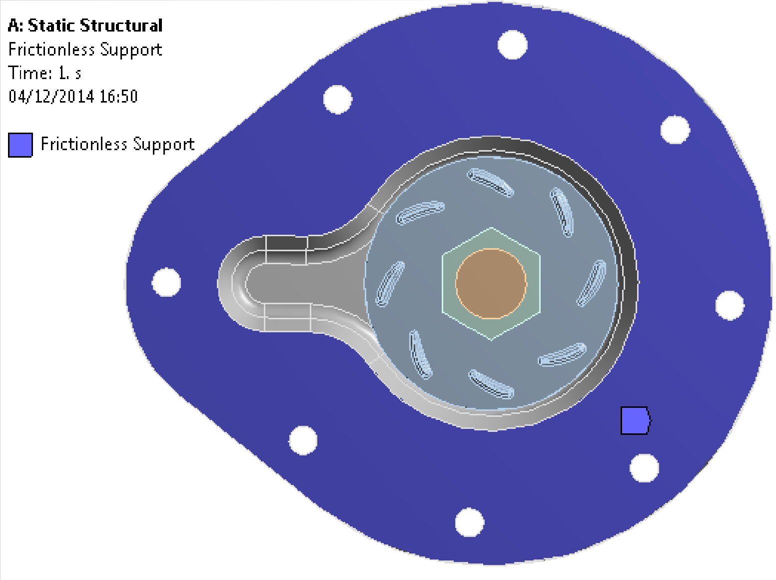
9. Apply the bearing load to the pulley:
   a. Highlight the "Static Structural" branch.
   b. Highlight the pulley's groove surface.
   c. RMB > Insert > Bearing Load".
   d. From the detail window change to "Components" and "X = 100 N"



# . . . Environment

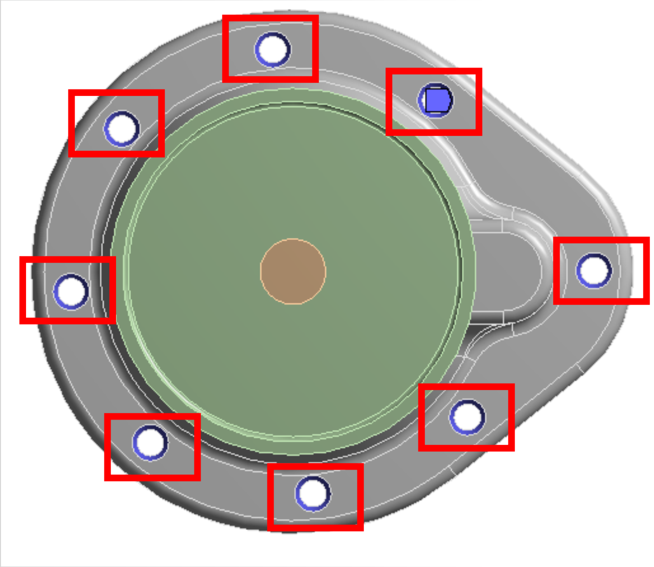
10. Apply supports to the assembly:
    a. Highlight the mating face on the pump housing (part 1).
    b. “RMB > Insert > Frictionless Support”.



# . . . Environment

Now we will add the frictionless supports to the 8  countersink portions of the mounting holes (shown here).

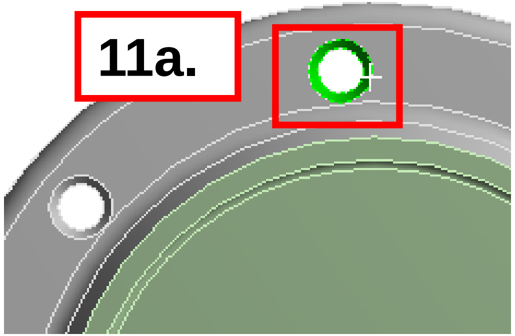
Each of the required surfaces could be selected individually while holding the CTRL key. However, this could be accomplished more swiftly with the use of either Named Selections or macro

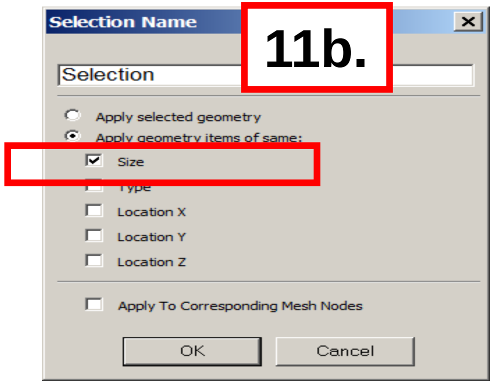


# . . . Environment

11. Select any one of the countersunk holes:
    a. RMB, choose “Create Named Selection”.
    b. Under “Apply geometry items of same”, check box for “size”
    c. Click “ok”
    d. RMB on Static Structural and insert a Frictionless Support
    e. In the details window switch “Scoping Method” to Named Selection
    f. Under “Named Selection” switch to “Selection”




11f.

Details of "Frictionless Support 2"

| | |
|---|---|
| **Scope** | |
| Scoping Method | Named Selection |
| Named Selection | Selection |
| **Definition** | |
| ID (Beta) | 67 |
| Type | Frictionless Support |
| Suppressed | No |

If the user used this approach, go directly to step 13 (slide 14). Alternative, if the user decides to take the macro approach, continue to the next slide.

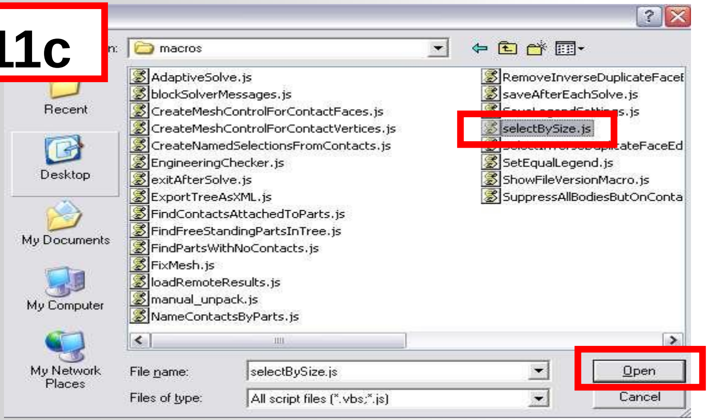
  

11. Rather than using Named Selections, the selection can be accomplished by size macro:
    a. Highlight 1 of the countersink surfaces (arbitrary).
    b. Choose “Tools > Run Macro . . .” and browse to:

**C:\Program Files\ANSYS Inc\v160\aisol\DesignSpace\DSPages\macros**

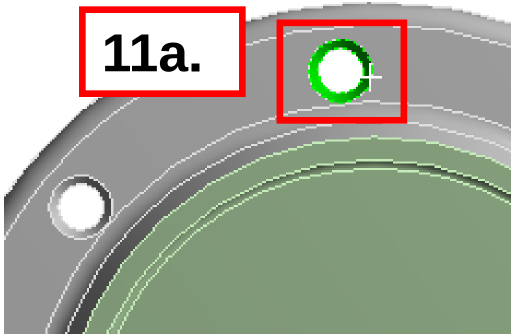
    c. In the browser choose “selectBySize.js”
    d. Click “Open”

11c




11d

11a.

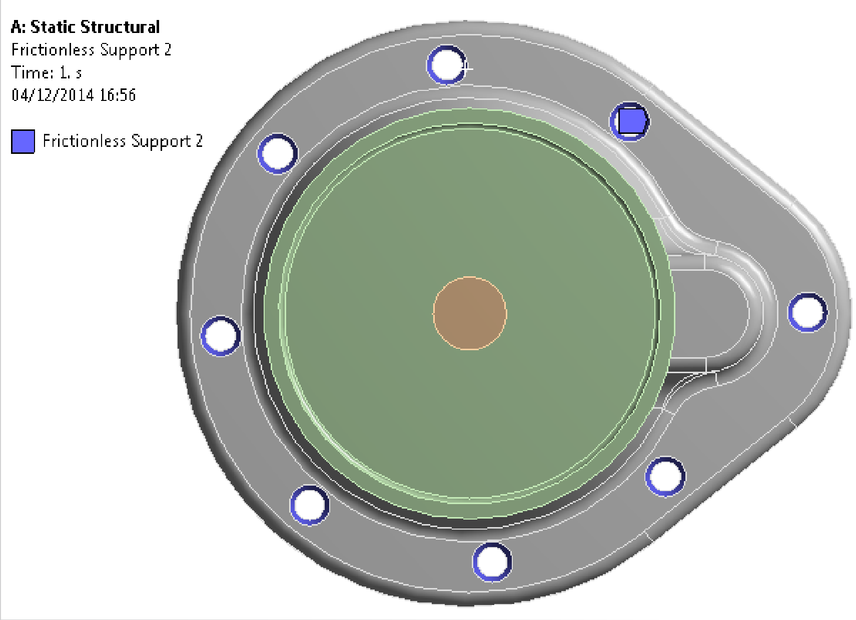


# . . . Environment

**Constrain the countersunk hole surfaces:**

**12. From the context menu, click on "Supports" and choose "Frictionless Support" or "RMB > Insert > Frictionless Support"**



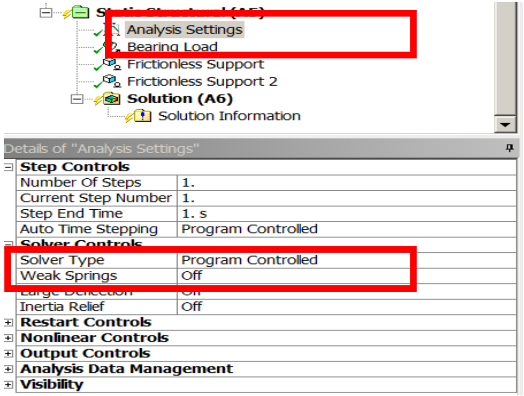

  

# Solution

13. Highlight the "Analysis Settings" and from the details window change "Weak Springs" from "Program Controlled" to "Off".

*Note : Because of the presence of frictionless supports and non bonded contact, Workbench-Mechanical will trigger the use of weak springs during the solution. If we know the model is fully constrained we can turn off this function.*

13.




14. Solve the model:

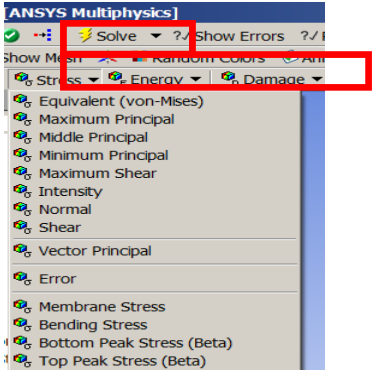
- Choose solve from the tool bar or RMB Solution branch and choose "Solve".

# Postprocessing

15. Add results to solution:
    a. Highlight the solution branch:
    b. From the context menu, choose Stress > Equivalent (von-Mises) or RMB > Insert > Stress > Equivalent (von-Mises)
    c. Repeat the step above, choose Deformation > "Total Deformation"

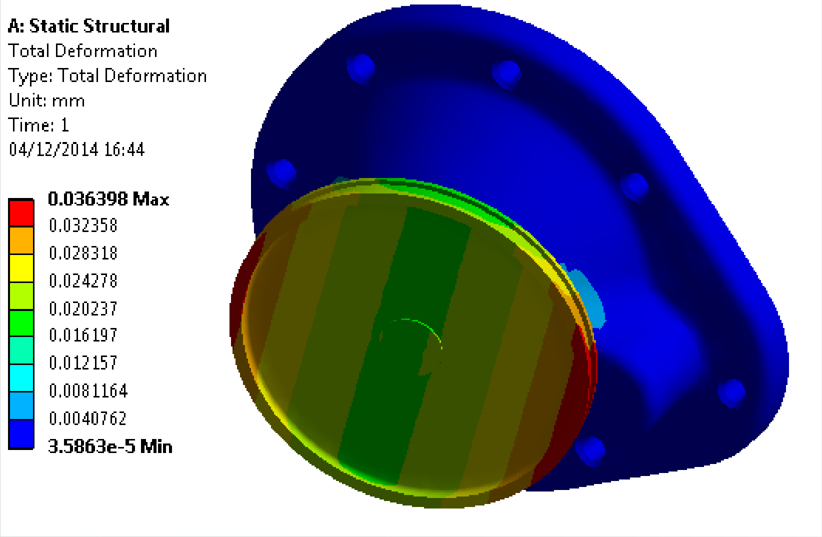
- Solve again.
  - *Note: adding results and re-solving the model will not cause a complete solution to take place. Requesting new results requires only a re-read of the results file.*
  - *Alternatively, the requested results can be process by RMB on Solutions and pick Evaluate All Results options*

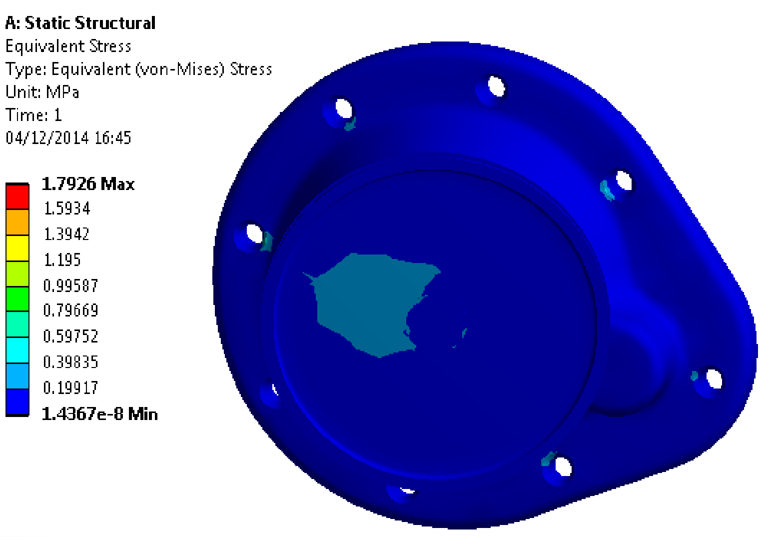


# . . . Postprocessing

While the overall plots can be used as a reality check to verify our loads, the plots are less than ideal since much of the model is displayed in few colors. (your results may vary slightly due to meshing differences).




To improve the quality of results available we will “scope” results to individual parts”.

# . . . Postprocessing

16. **Scope the results to individual bodies/surfaces:**

    a. Highlight the "Solution" branch and switch the selection filter to "Body" select mode.

    b. Select the impeller (part 2)

    c. "RMB > Insert > Stress > equivalent (von- Mises)"

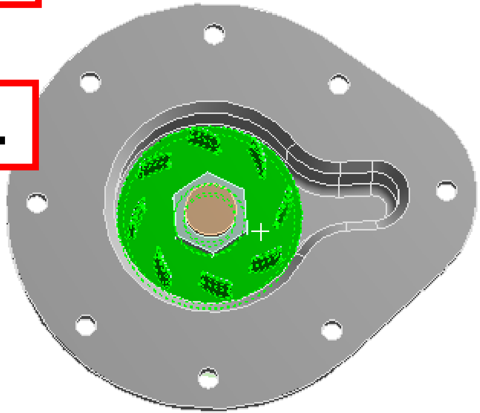
    - *Notice the detail for the new result indicates a scope of 1 Body.*
    - *Now the local variation in the results are more visually distinct.*

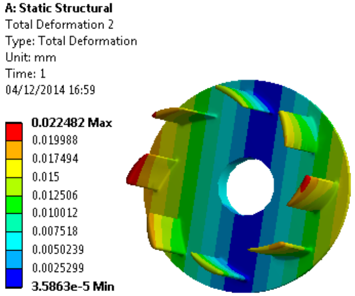
17. **Repeat the procedure above to insert "Total Deformation" results for the impeller part.**

18. **Repeat to add individually scoped stress and total deformation results to the pump housing (part 1).**

16a.

16b.





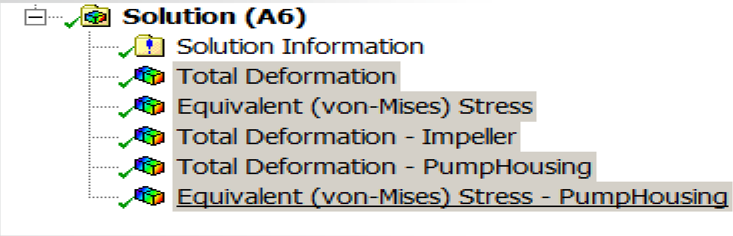

  

# . . . Postprocessing

19. To give physically meaningful names to the result objects, it is useful to rename the new results:
    a. Highlight all results > RMB > Rename Based on Definition. The result objects should now contain more informative names

20. Evaluate All Results

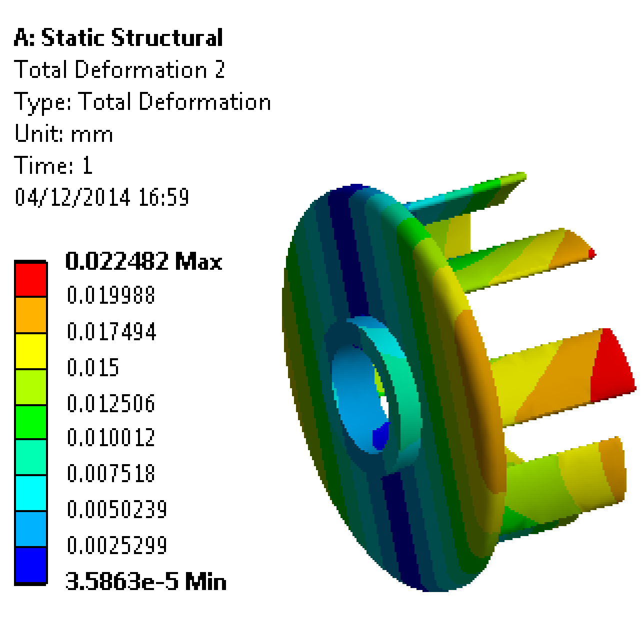
19



# . . . Postprocessing

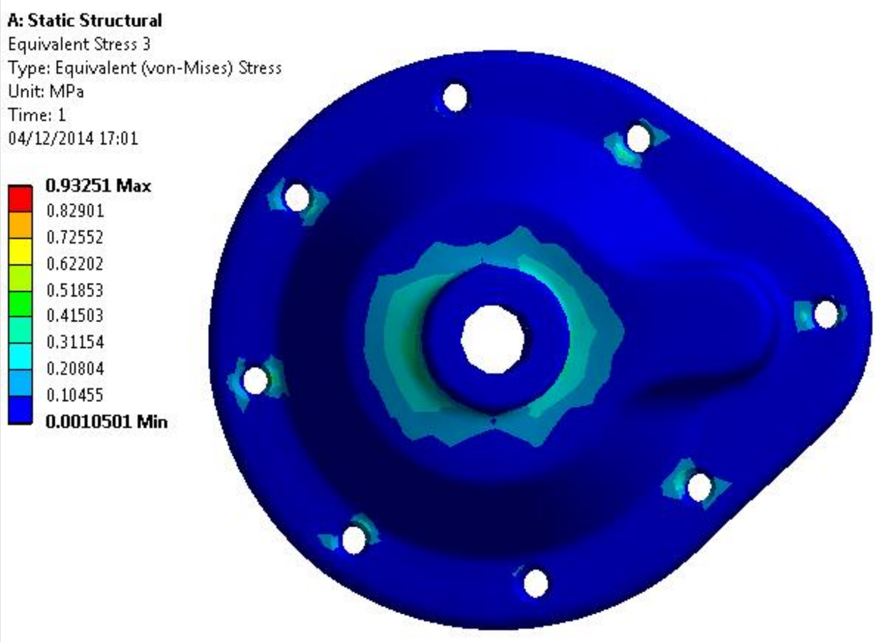
**By checking the impeller deformation we can verify that one of our goals is met. The maximum deformation is approximately 0.028mm (goal < 0.075mm).**



# . . . Postprocessing

**Inspection of the housing stress shows that, overall, the stress levels are below the material's elastic limit (tensile yield = 25 MPa). We could again use scoping to isolate the results in the area of interest.**



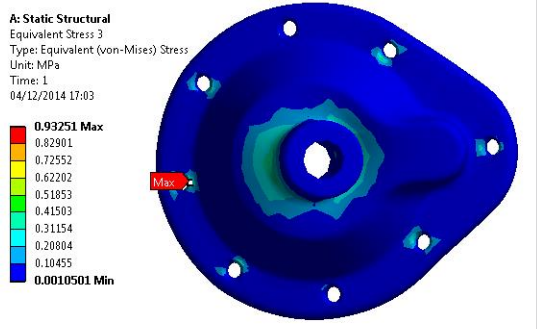

  

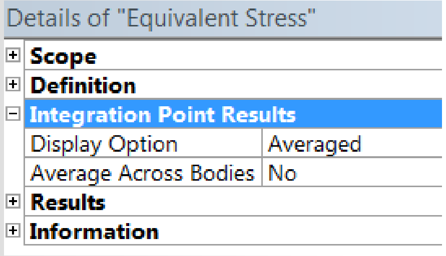
# Examining stress results

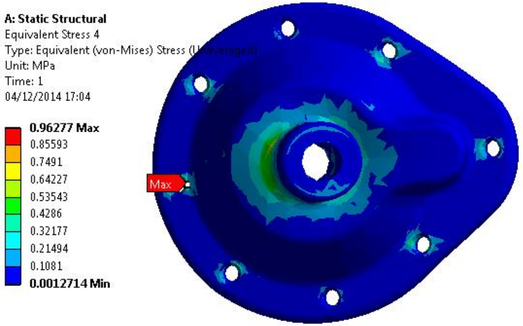
**The stress results are often not as "smooth" as the deformation results, this is due to the fact that the displacement field is continuous across the nodes whereas the stress is not. Therefore the stress results can "fluctuate".**

**By default ANSYS takes the averaged stress result to reduce the "fluctuation"**

**Averaged** **VS** **Un-averaged**



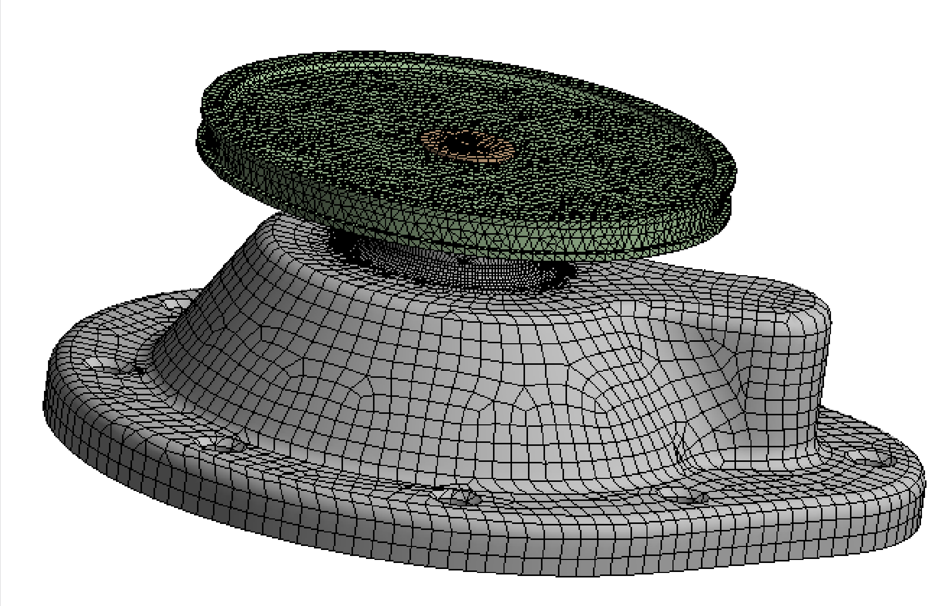

  

# Go further!

**The Mesh of the bodies is very coarse…**

**In order to get more physical results, the user must change the mesh.**

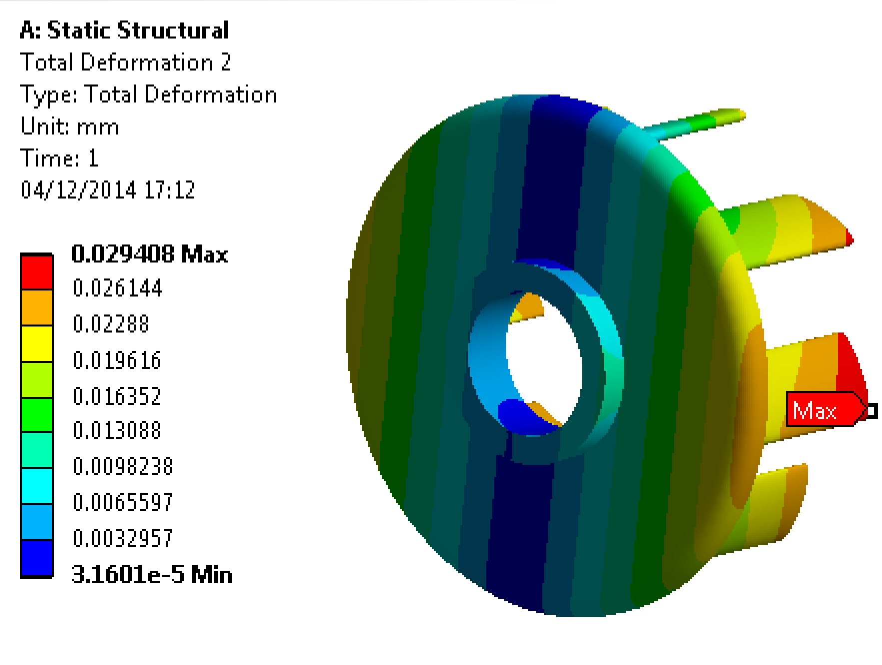
**Here's a suggestion of mesh:**



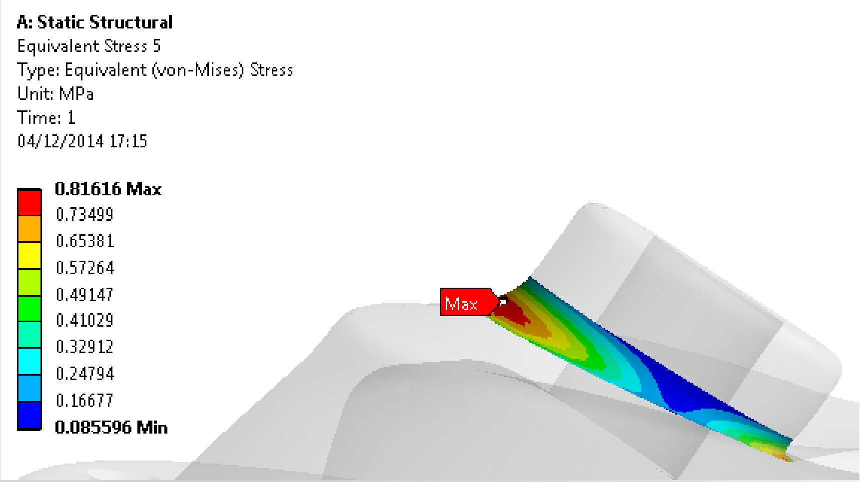
# . . . Go further!

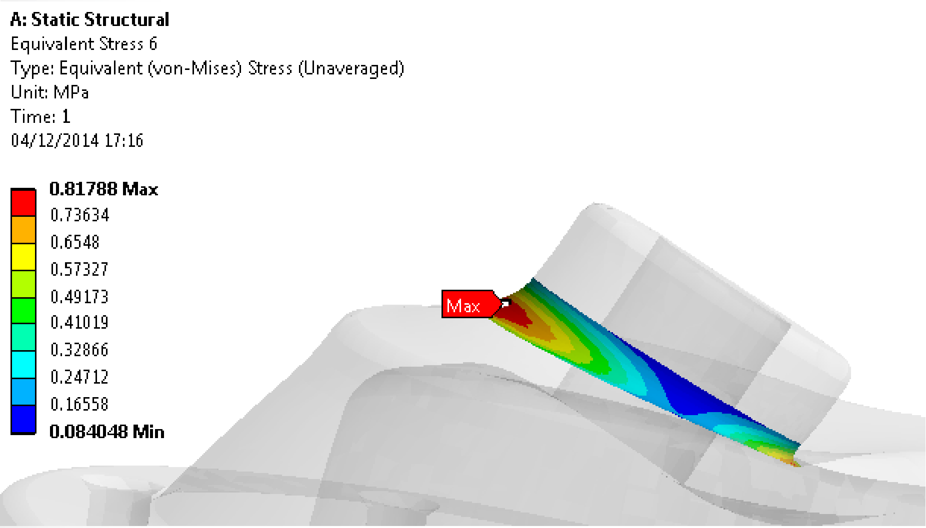
**The user can now re-examine the results:**



# . . . Go further!


A: Static Structural
Equivalent Stress 5
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
04/12/2014 17:15
0.81616 Max
0.73499
0.65381
0.57264
0.49147
0.41029
0.32912
0.24794
0.16677
0.085596 Min
Max


A: Static Structural
Equivalent Stress 6
Type: Equivalent (von-Mises) Stress (Unaveraged)
Unit: MPa
Time: 1
04/12/2014 17:16
0.81788 Max
0.73634
0.6548
0.57327
0.49173
0.41019
0.32866
0.24712
0.16558
0.084048 Min
Max