

Workshop 3.4

Named Selections + Object generator

16.0 Release

A visualization of fluid dynamics showing blue, wavy, semi-transparent surfaces that resemble smoke or liquid flow, set against a light yellow background.

Fluid Dynamics

A 3D rendering of a purple gear with a glowing white center, surrounded by other faint gears, symbolizing structural mechanics.

Structural Mechanics

A series of concentric green circles with a glowing center, representing electromagnetic fields or wave propagation.

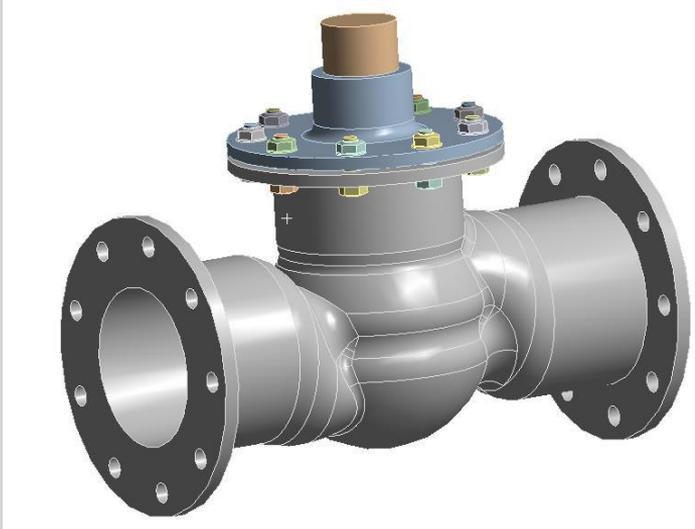
Electromagnetics

A 3D arrangement of teal and black rectangular blocks of varying sizes, some stacked and some floating, representing systems and multiphysics simulations.

Systems and Multiphysics

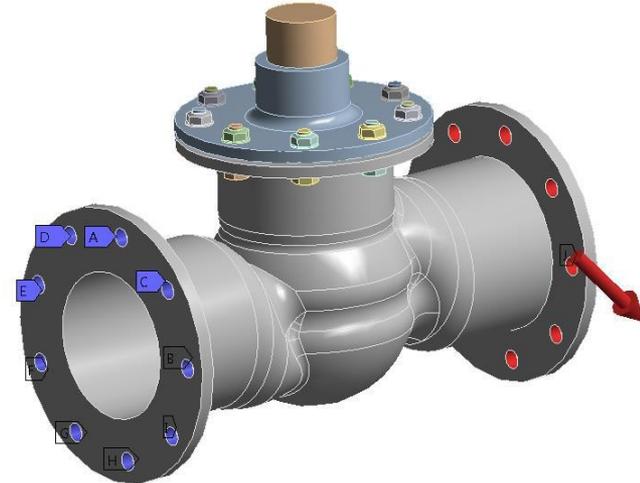
Introduction to ANSYS Mechanical

The goal of this workshop is to use several techniques to create named selections and use the object generator in order to minimize the time spent on preprocessing.

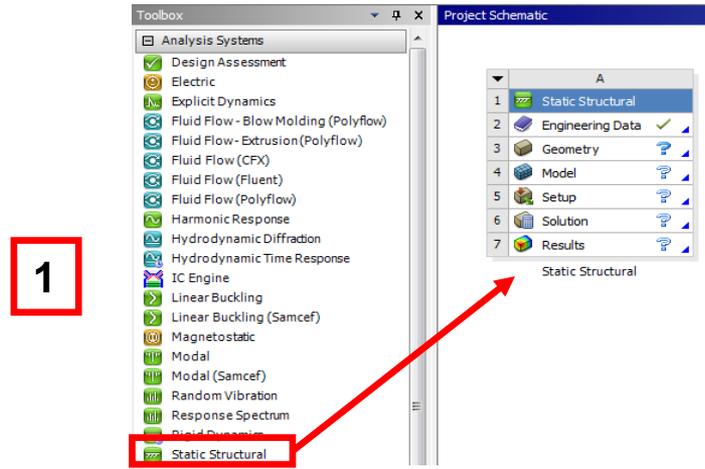


A: Static Structural
Static Structural
Time: 1, s

- A Fixed Support
- B Fixed Support 2
- C Fixed Support 3
- D Fixed Support 4
- E Fixed Support 5
- F Fixed Support 6
- G Fixed Support 7
- H Fixed Support 8
- I Fixed Support 9
- J Force: 1414,2 N



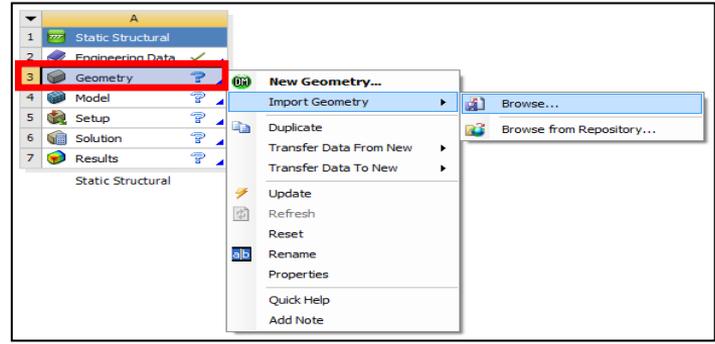
1. Double click “Static Structural” analysis type to add a new system.



1

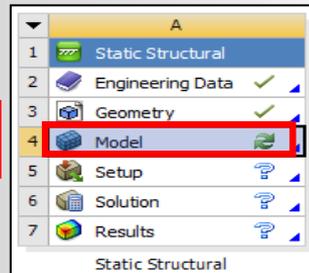
2. From the “Geometry” cell, RMB > “Import Geometry” and browse to: “Valve_RM_20130113.stp”.

2.

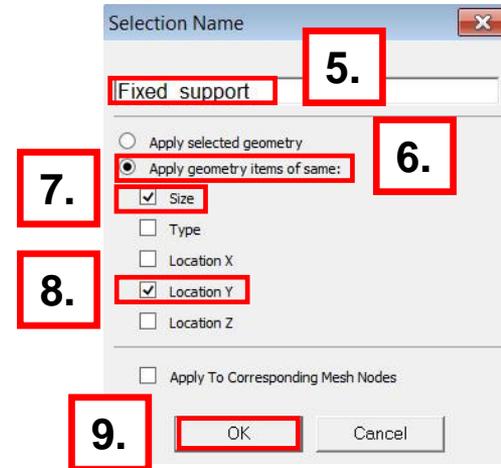
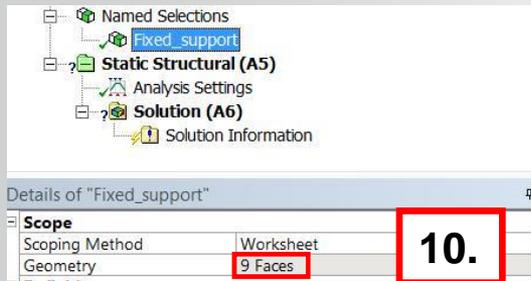
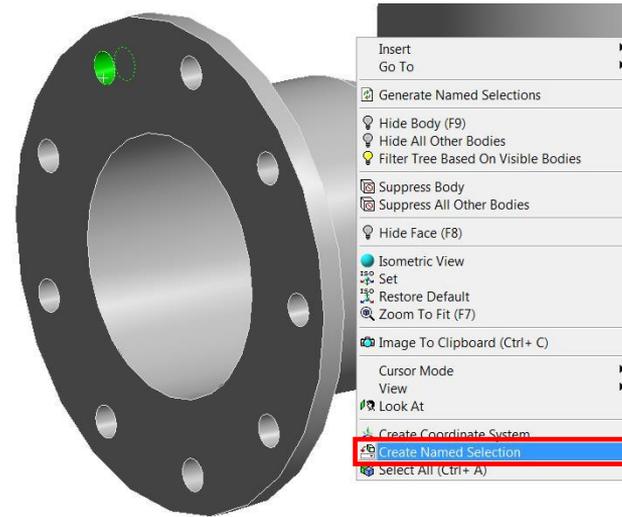


3. Double click the “Model” cell to start Mechanical.

3.

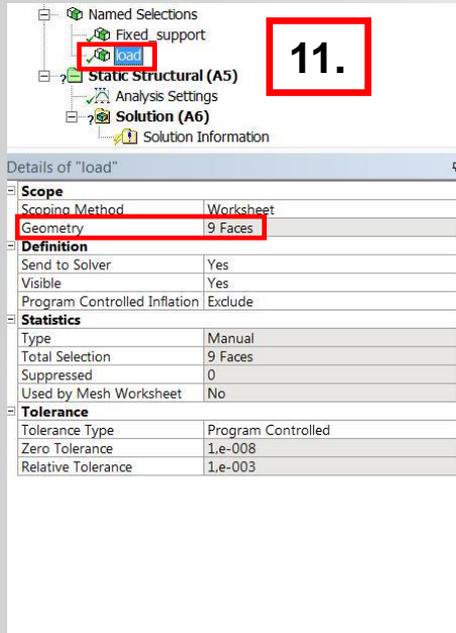


4. Select one of the holes in the –Y direction
RMB > Create Named Selection.
5. In the Selection dialog enter the name “Fixed_support”.
6. Choose “Apply geometry items of same:”.
7. Check the box “Size”.
8. Check the box “Location Y”.
9. Click the “Ok” button.
10. In the tree, highlight the new named selection “Fixed_support” and note the scope of the selection is 9 faces.



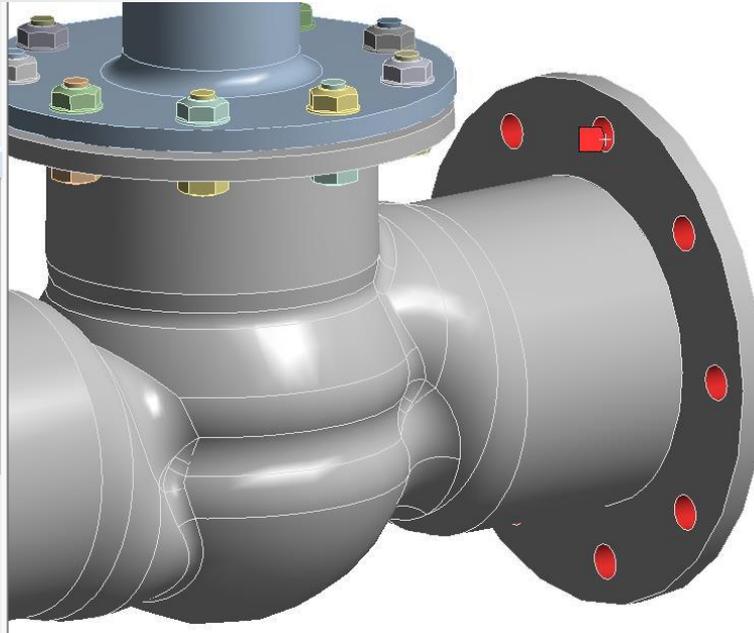
We will create the different named selections.

11. Repeat the operations 4 -> 10 for the other side of the geometry and name the new named selection "load"

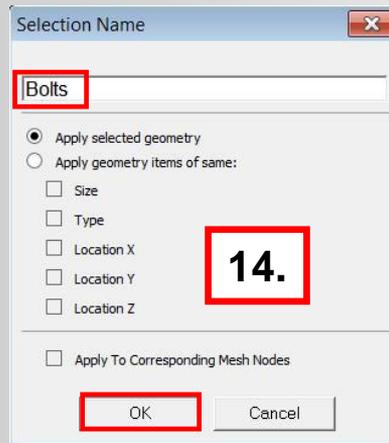
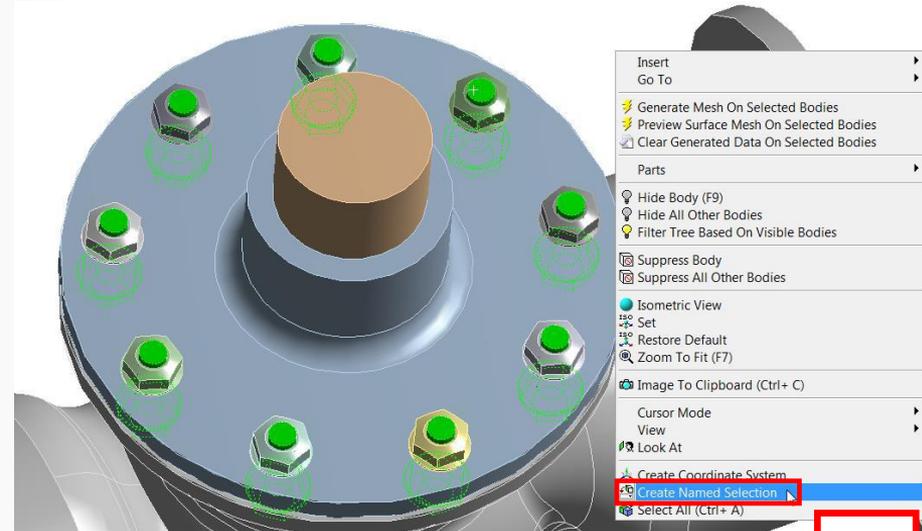
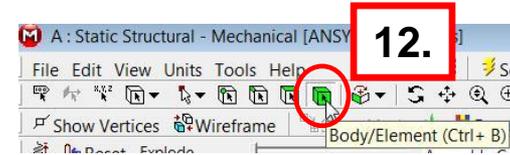


The screenshot shows the ANSYS software interface. In the Named Selections tree, the 'load' selection is highlighted with a red box. A red box with the number '11.' is overlaid on the tree. Below the tree, the 'Details of "load"' dialog box is open, showing the following information:

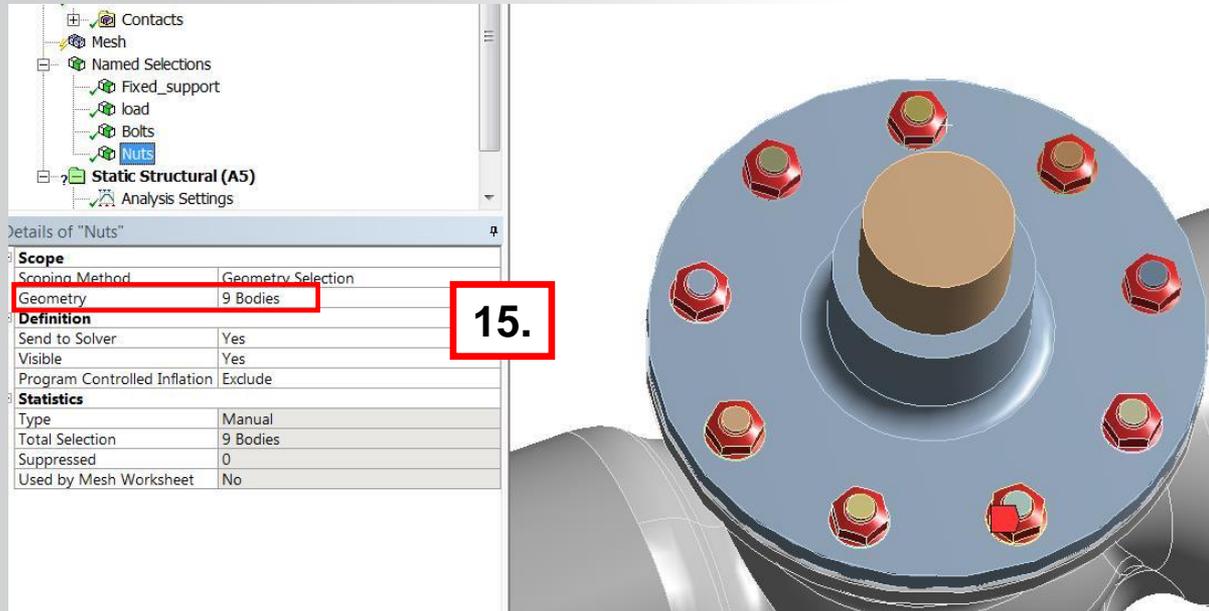
Details of "load"	
Scope	
Scoping Method	Worksheet
Geometry	9 Faces
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Manual
Total Selection	9 Faces
Suppressed	0
Used by Mesh Worksheet	No
Tolerance	
Tolerance Type	Program Controlled
Zero Tolerance	1.e-008
Relative Tolerance	1.e-003



12. Choose the Body selection mode
13. Select all the bolts > RMB > Create Named Selection
14. In the Selection dialog enter the name "Bolts" and click "OK"



15. Repeat the operations 12, 13, 14 for the other nuts name the new named selection “Nuts”



The screenshot displays the ANSYS software interface. On the left, the 'Details of "Nuts"' dialog box is open, showing the following information:

Scope	
Scoping Method	Geometry Selection
Geometry	9 Bodies
Definition	
Send to Solver	Yes
Visible	Yes
Program Controlled Inflation	Exclude
Statistics	
Type	Manual
Total Selection	9 Bodies
Suppressed	0
Used by Mesh Worksheet	No

The 'Geometry' field in the 'Scope' section is highlighted with a red box, and a red box with the number '15.' is overlaid on the dialog box. The right side of the image shows a 3D model of a blue flange with a central brown cylindrical protrusion and nine red hexagonal nuts arranged in a circular pattern around the flange.

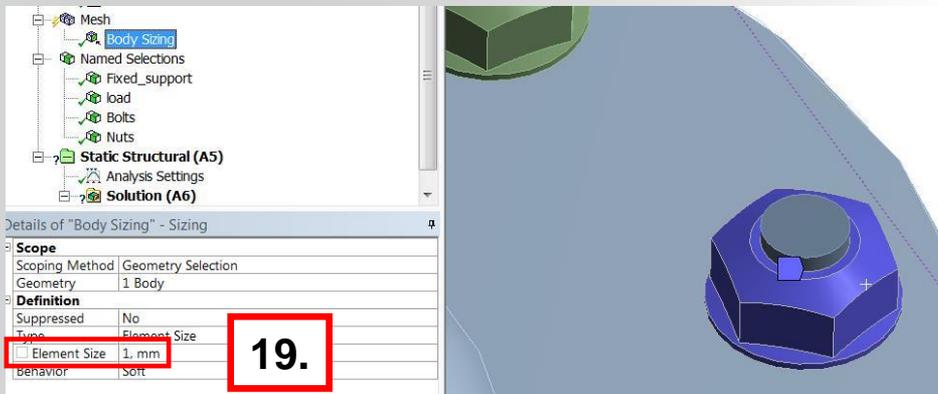
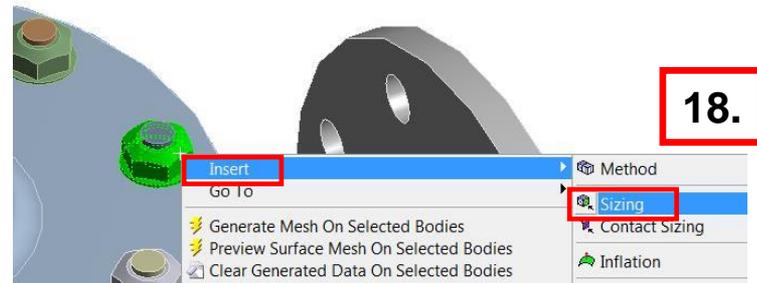
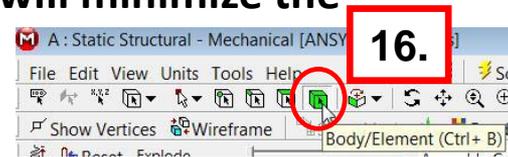
With the help of the named selection and the object generator we will minimize the time of the mesh setup.

16. Choose the Body selection mode

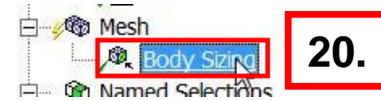
17. Go on Mesh section by clicking on the mesh branch in the tree

18. Select one of the nuts > RMB > insert > Sizing

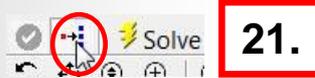
19. Choose 2 mm for the size of the element



20. Select the Body sizing



21. Select the object generator tools

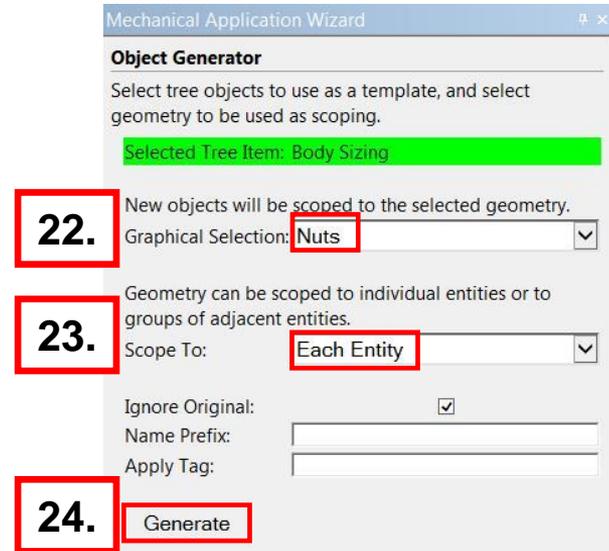
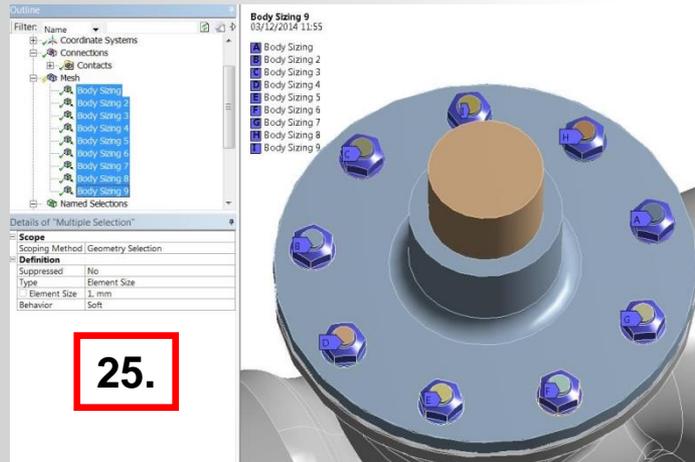


22. Change “Graphical Selection” to “Nuts”

23. Change “Scope to” to “Each Entity”

24. Click on Generate

25. The result is an automatic creation of a “Body sizing” for each nut



26. Repeat the operations 16 -> 25 for the mesh of the bolts

The screenshot displays the ANSYS software interface. On the left, the **Outline** panel shows a list of **Body Sizing** operations from 6 to 18, all with green checkmarks. Below it, the **Details of "Multiple Selection"** panel is open, showing the following information:

Scope	
Scoping Method	Geometry Selection
Definition	
Suppressed	No
Type	Element Size
<input type="checkbox"/> Element Size	1, mm

In the center, a red box contains the number **25.**

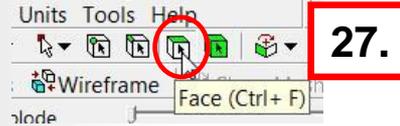
On the right, the **Body Sizing 18** panel shows a list of selection points:

- A Body Sizing 10
- B Body Sizing 11
- C Body Sizing 12
- D Body Sizing 13
- E Body Sizing 14
- F Body Sizing 15
- G Body Sizing 16
- H Body Sizing 17
- I Body Sizing 18

The 3D model on the right shows a bolted flange assembly with a central cylindrical part. The flange has several bolts. Selection points A through I are marked on the bolts and the flange surface.

We will minimize the time of the boundaries setup with the help of the named selection and the object generator.

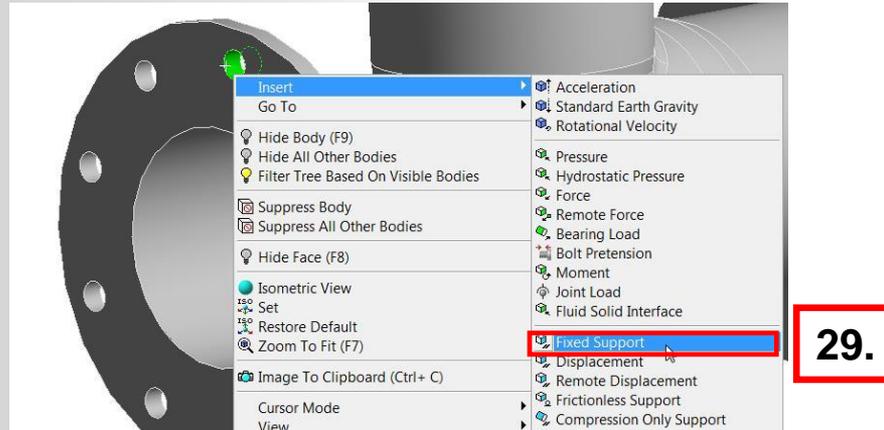
27. Choose the Face selection mode



28. Go on Loads section by clicking on the static structural branch in the tree



29. Select one hole on the -Y direction > RMB > insert > Fixed support



30. Select the fixed support

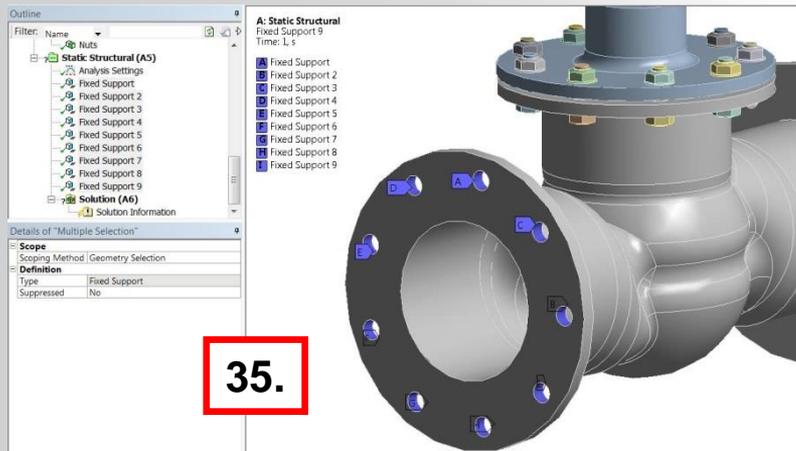
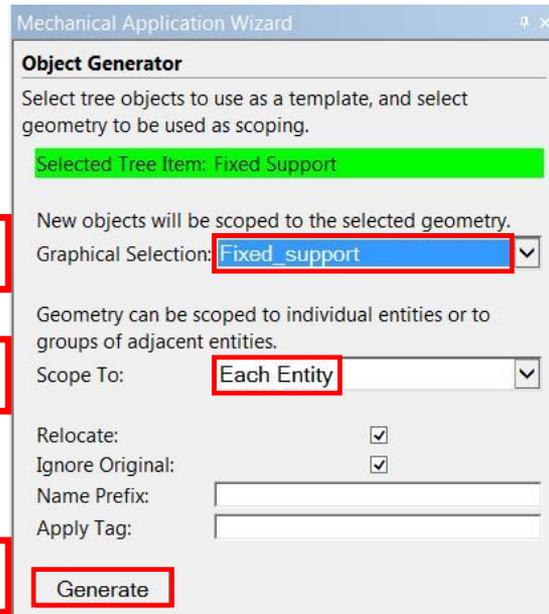
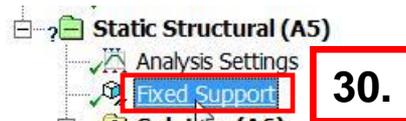
31. Select the object generator tools

32. Change “Graphical Selection” to “Fixed_support”

33. Change “Scope to” to “Each Entity”

34. Click on Generate

35. The result is an automatic creation of a “fixed support” for each hole, which can help on post processing for reaction forces



36. RMB on static structural > insert > Force

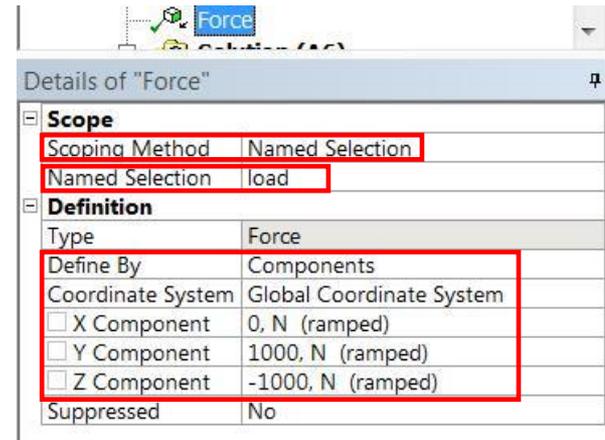


36.

37. In the details view choose for “Scoping method”:
“Named selection”

38. Choose for “Named selection”: “Load”

39. Change “Defined by” to “component” and put
1000 N for Y direction and -1000N for Z direction



37.

38.

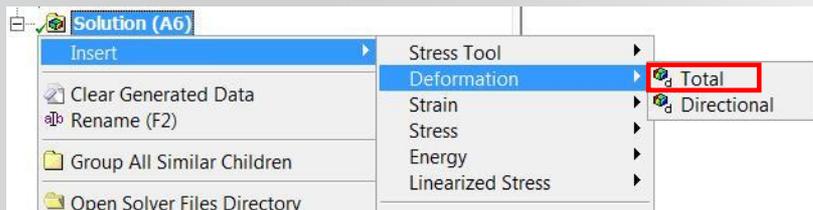
39.

40. Launch the solution by clicking on solve



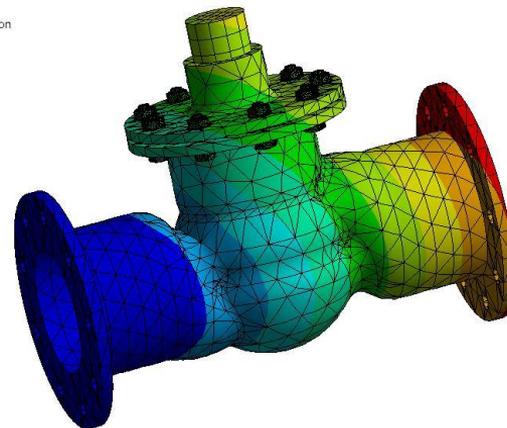
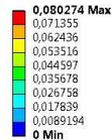
40.

41. Highlight the Solution branch, RMB > Insert > deformation > Total

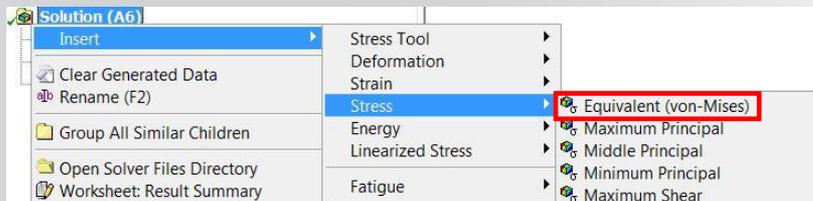


41.

A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1

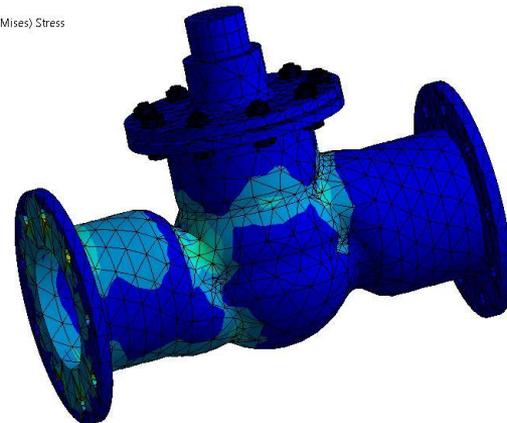
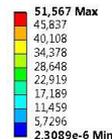


42. Highlight the Solution branch, RMB > Insert > stress > Equivalent



42.

A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1



If you finish this workshop and find yourself with extra time, you could try the following steps:

- 1. Try to use named selection and object generator on the post processing in order to be quicker on result demands.**