Ansys Fluent Getting Started (New Fluent Experience)

Workshop: Static Mixer
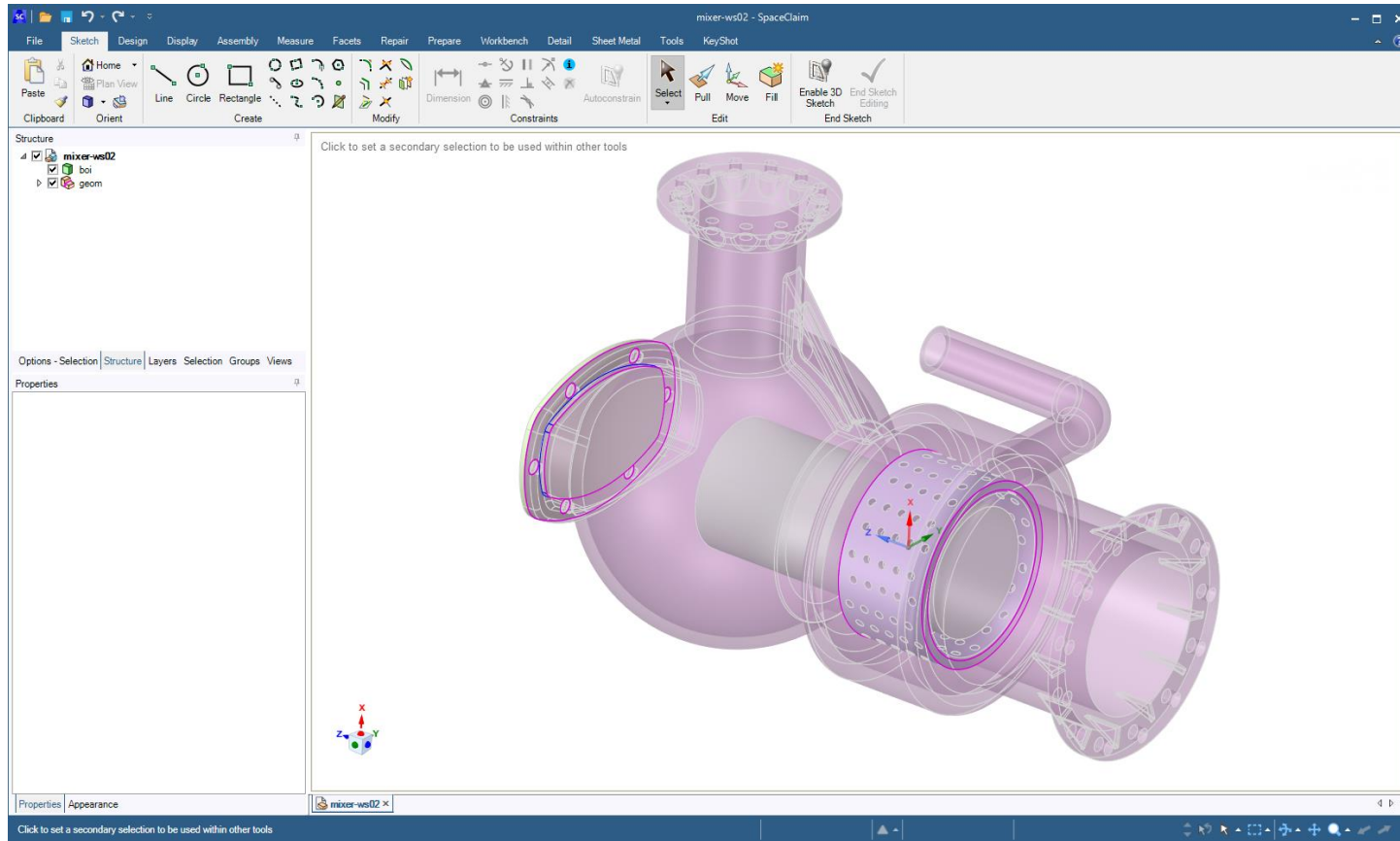
Release 2021 R1

Ansys

Capability Level

- This tutorial is supported by all licensing capability levels

Introduction


- **Workshop Description**

CAD files for a mixer will be imported into the watertight geometry workflow and a volume mesh will be generated. Both global and local size controls will be used along with capping to extract the fluid region
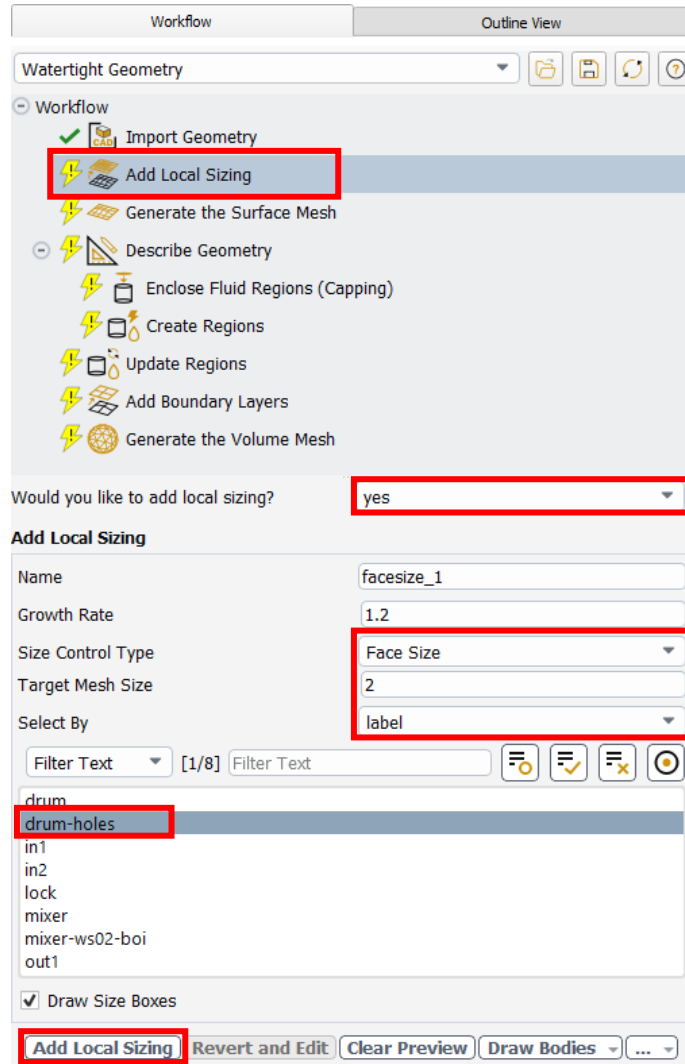
Starting Fluent

- Open the Fluent Launcher Window and ensure Meshing Mode is selected
  - If necessary, expand "Show More Options" to change the working directory
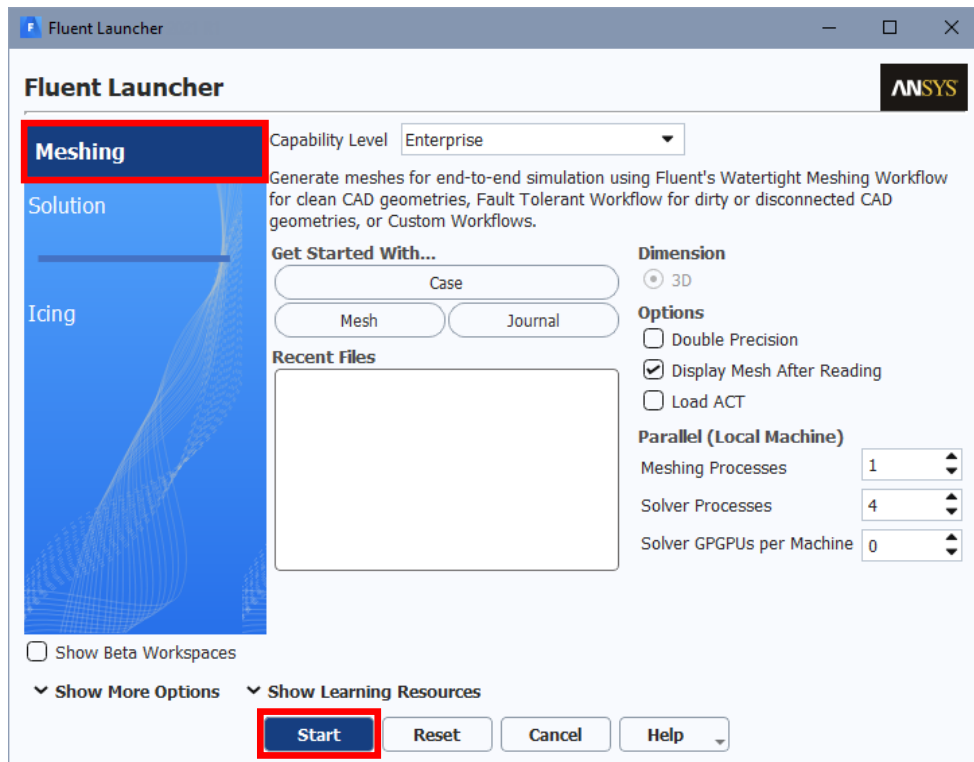
- Select the Watertight Geometry workflow, set Units to mm and import "mixer-ws02.scdoc"
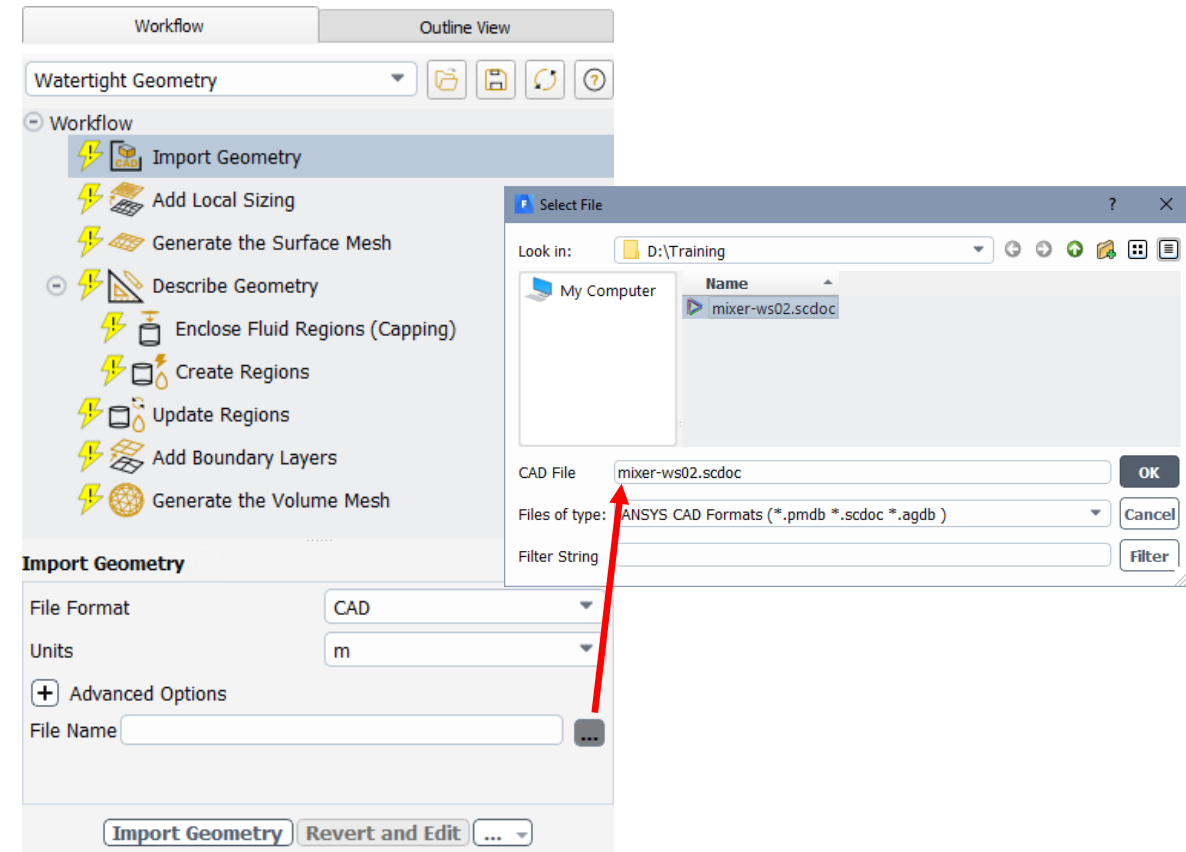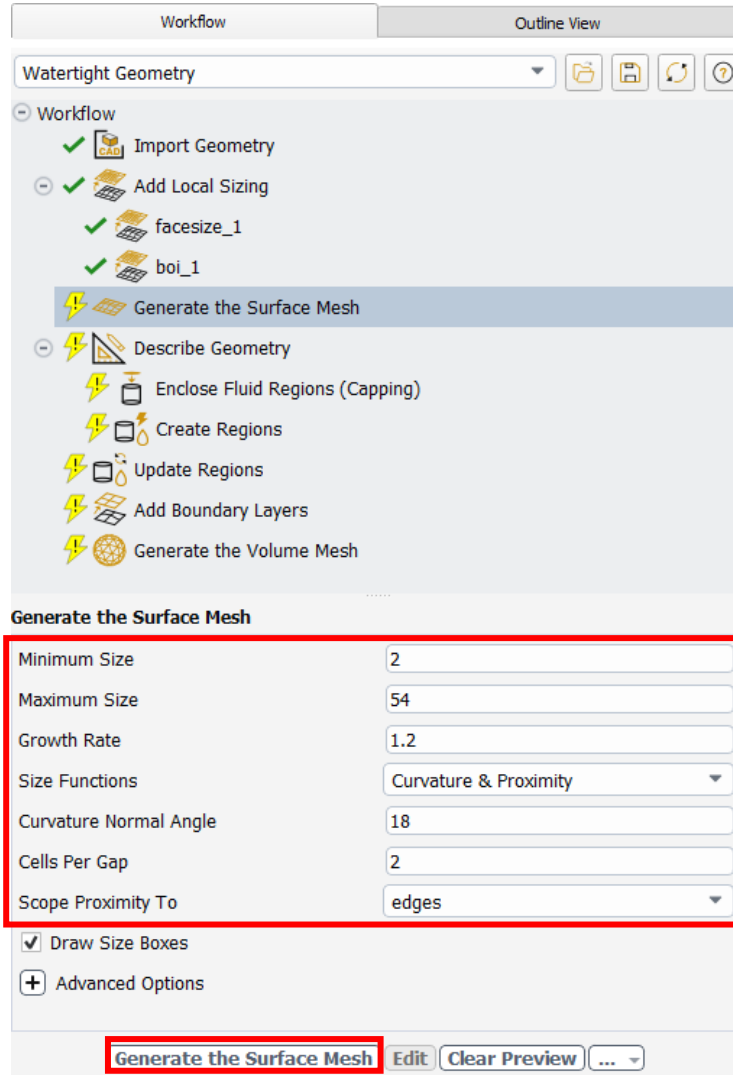
Create Size Controls

- **Under Add Local Sizing**
- **Select yes under Would you like to add local sizing?**
- **Set Size Control Type to Face Size and Select By label**
- **Select drum-holes from the label list**
- **Set Target Mesh Size = 2 mm and click Add Local Sizing**

- **Add another sizing but this time set Size Control Type to Body Of Influence**
  - **Not shown on this page**

- **Select mixer-ws02-boi from the label list**
- **Set Target Mesh Size = 15 mm and click Add Local Sizing**

The local sizing of 2mm ensures good resolution of small flow passages through the drum holes. The circumference of the drum hole openings is around 47 mm, so 2 is approximately 1/20th of the circumference.

The BOI ensures good resolution of the mesh in the region where the strongest mixing takes place and finer mesh is needed to resolve high gradients in the flow. Here, 15 mm is a guess based on the inner drum diameter, which is 290 mm. So with 15 mm there will be ≈ 20 mesh elements across the mixing region. In the end, a mesh refinement study (covered in the Best Practices lecture) is needed for any given problem to confirm that the mesh resolution is sufficient, but making an educated guess like this will increase the likelihood that the initial mesh is reasonable.
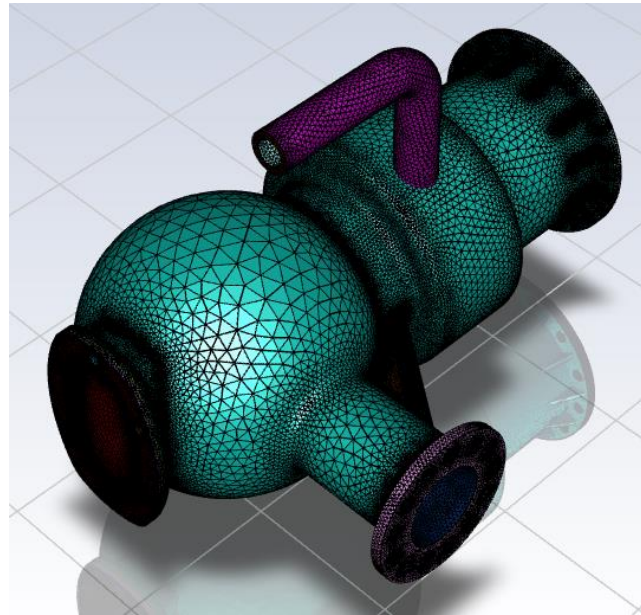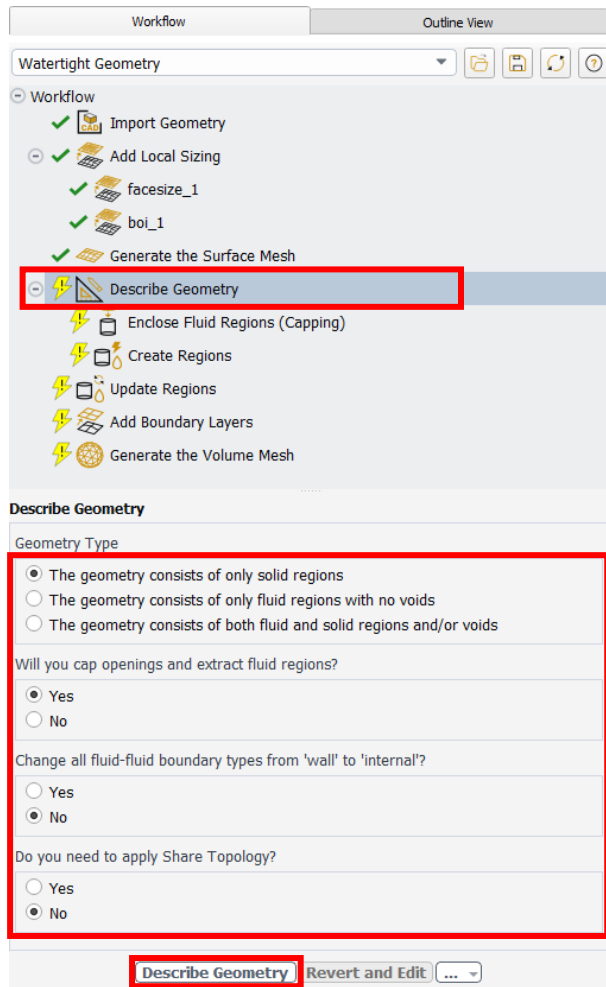
Generate the Surface Mesh

- **Under Generate the Surface Mesh**
- **Add the global size controls as shown in the screenshot on the left**
- **Click Generate the Surface Mesh**
- **Note the maximum skewness value reported in the console window upon completion of the surface mesh – values below 0.70 are desirable**

Note how changing the minimum and maximum values changes the mesh size preview boxes in the graphics window. The minimum size, 2 mm, is selected to ensure that the surface mesh will be no finer anywhere in the model than what was defined by the face sizing on the previous slide. The maximum value is taken from the spherical chamber located just before the outlet. Its diameter is 540 mm, so 54 mm is 10% of that.
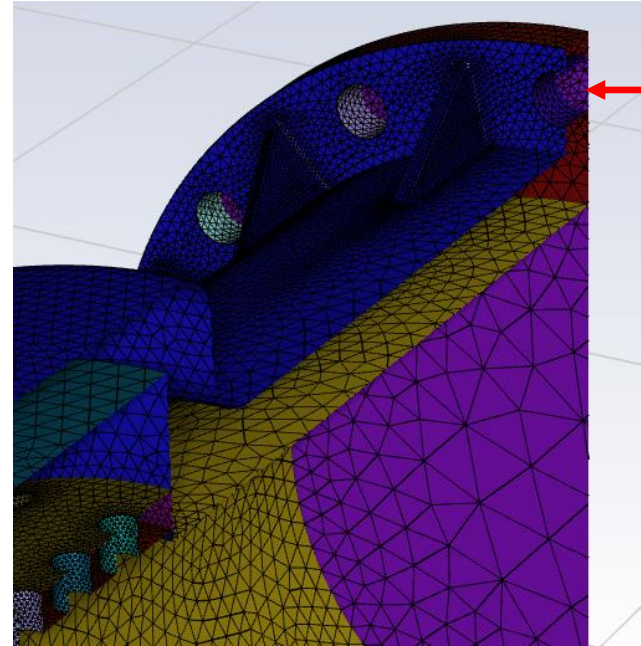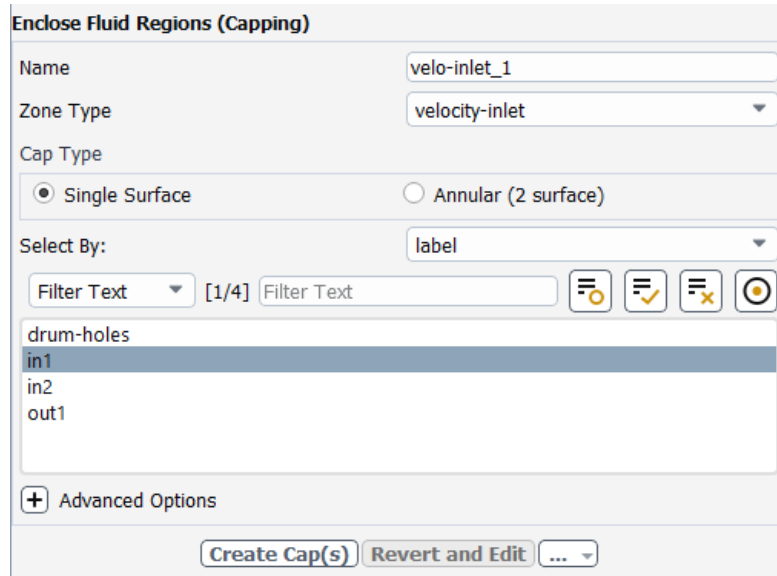
Increasing the number of cells per gap produces finer mesh resolution in regions where there might be fine geometrical details. There are a number of small details in the geometry which is the reason for the increase. Proximity is usually scoped to edges only unless a conjugate heat transfer calculation will be performed with the solid regions, in which case a finer surface mesh is needed to ensure the thickness of the solids are meshed with an adequate number of cells.

Completed Surface Mesh

- **After completion of the surface mesh choose the following options under Describe Geometry**
- **The geometry consists of only solid regions**
- **Yes under Will you cap …..?**
- **No under Change all fluid-fluid boundary ….?**
  - **Just leave the default option since there are no fluid-fluid boundaries in this model**
- **No under Do you need to apply Share Topology?**
- **Click Describe Geometry**

Create Caps

Enclose Fluid Regions (Capping)

| Name | velo-inlet_1 |
| Zone Type | velocity-inlet |

Cap Type
- ● Single Surface   ○ Annular (2 surface)

Select By: label

Filter Text   [1/4] Filter Text

drum-holes
in1
in2
out1

⊞ Advanced Options

Create Cap(s)   Revert and Edit   ...

Notice that bolt holes are also capped when they are in the same plane as the capping surface. This is expected. During the Create Regions task, Fluent will recognize that the caps over the bolt holes do not enclose any region and it will simply discard them.

- **Under Enclose Fluid Regions (Capping)**
- **Create 3 caps using labels**
  - **Name = velo-inlet_1 Zone Type = velocity-inlet Label = in1**
  - **Click Create Cap(s) to create the cap**
  - **(not shown) Name = velo-inlet_2 Zone Type = velocity-inlet Label = in2**
    - **Click Create Cap(s)**
  - **(not shown) Name = pres_outlet_1 Zone Type = pressure-outlet Label = out1**
   - **Click Create Cap(s)**

/Ansys

Create and Update Regions
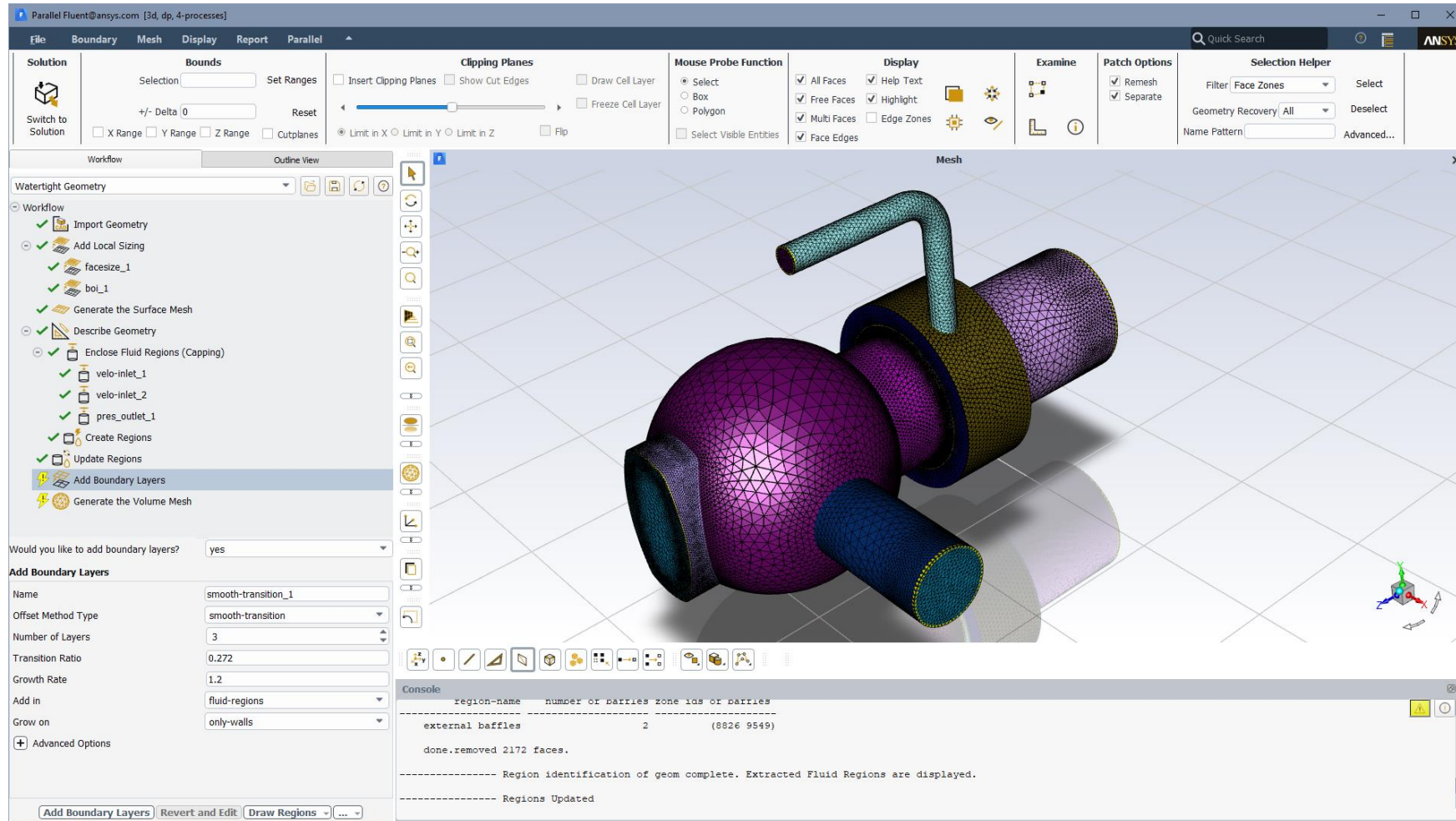
Click Create Regions and Update Regions with default values as the number of expected fluid regions =1 and region assignments are correct

Add Boundary Layers and Generate the Volume Mesh

- Complete the Add Boundary Layers task using the default settings and Generate the Volume Mesh task using poly-hexcore and unticking Mesh Solid Regions
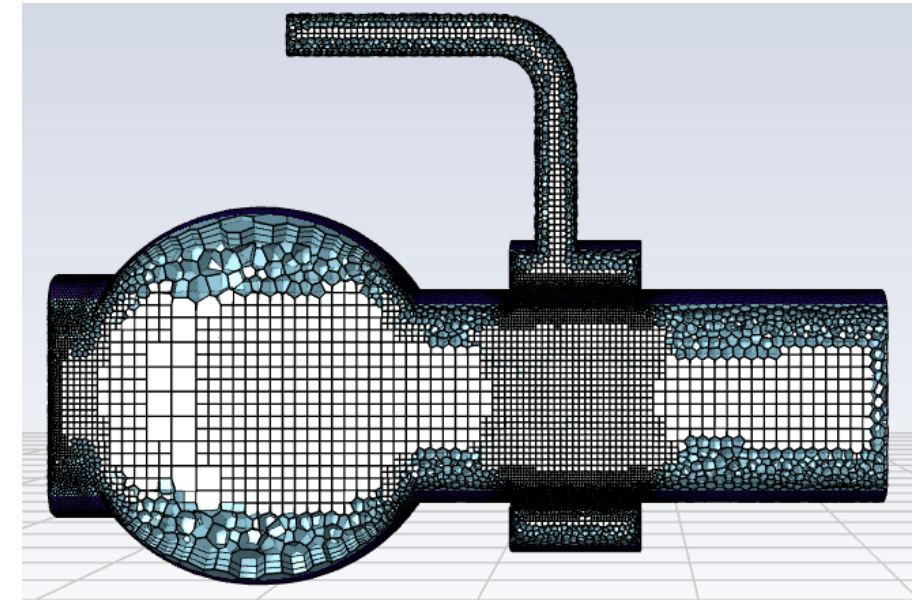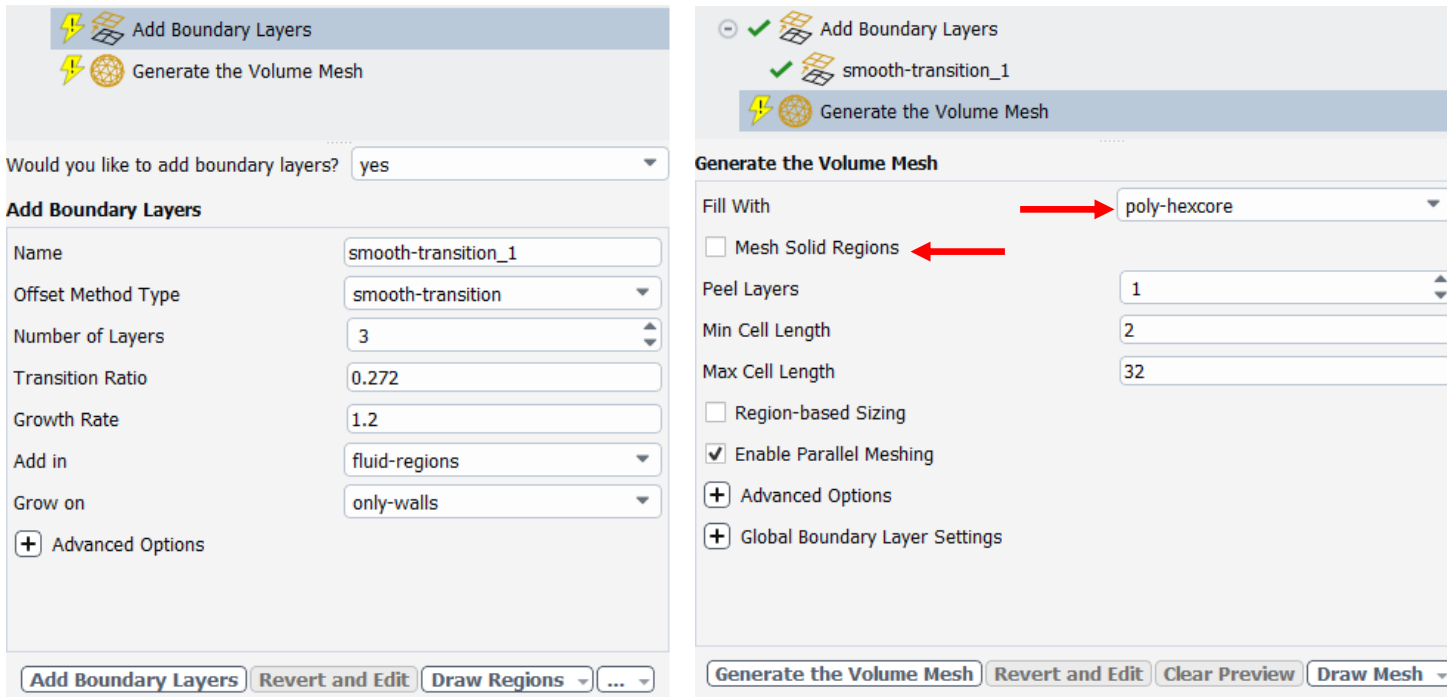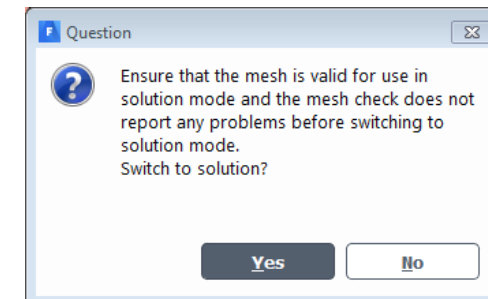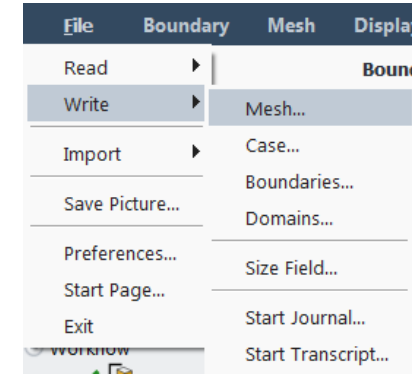

The F10 key can be used to hide the clipping plane triad.

On completion of the volume mesh, Fluent reports the minimum orthogonal quality of the mesh in the console window. Review the console output to ensure this value is 0.1 or higher.

Write Mesh and Switch to Solution Mode

- Go to File > Write > Mesh and save the mesh as "mixer-ws02-volume.msh.gz"
  - While it is not required to save the mesh file before switching to solution mode, the workflow inputs are stored with the mesh, so in case it is desired to make changes in the future, it is easy to do so after reading the mesh into a new Fluent Meshing session

- Click on Switch to Solution
  - Click Yes in the question panel that appears
  - Mesh information is transferred to the solver and the GUI changes from meshing mode to solution mode

End of presentation