# Ansys Fluent Getting Started (New Fluent Experience)

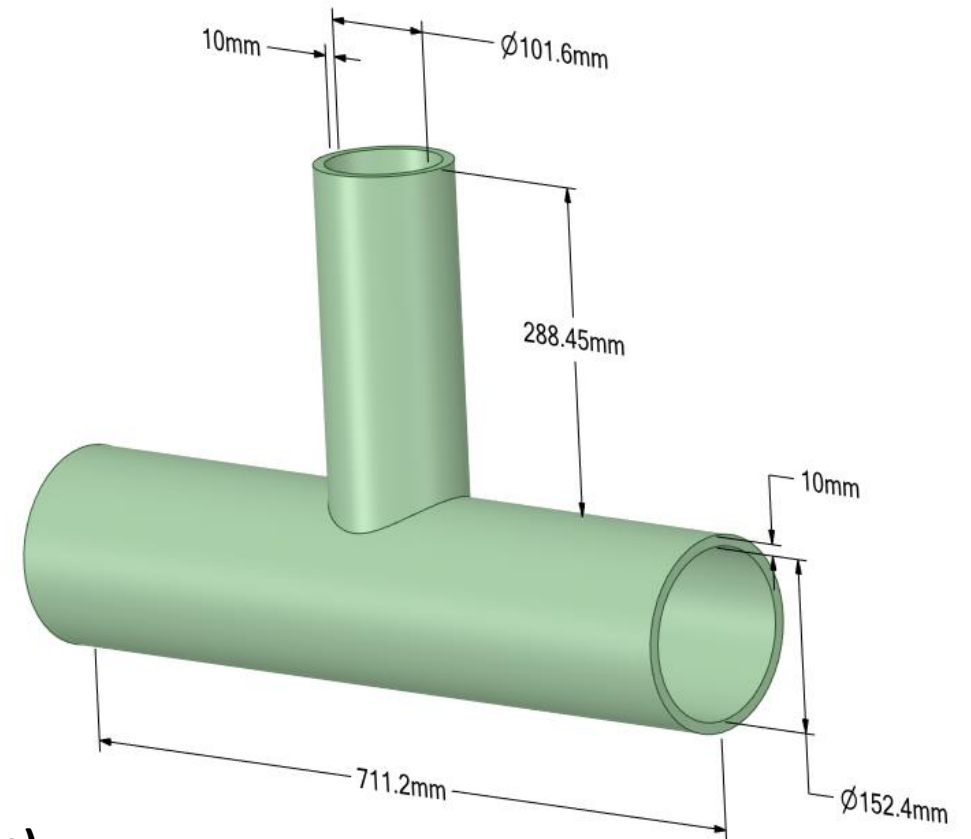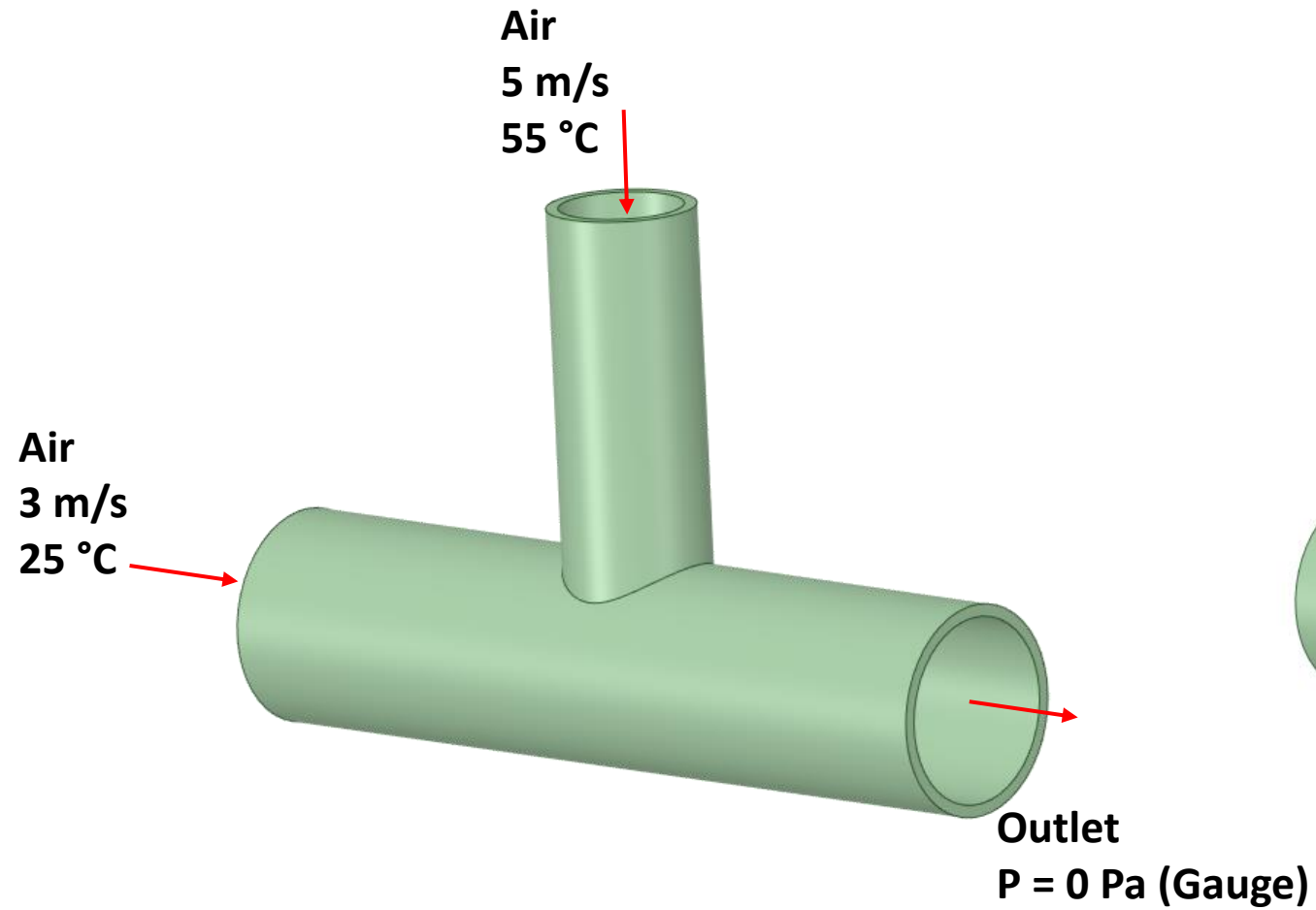## Workshop: Mixing Tee

Release 2021 R1

**Ansys**

# Capability Level

- This tutorial is supported by all licensing capability levels

# Problem Description

- Mixing tees are commonly used for static mixing of two fluid streams

- In this problem, a mixing tee is used to mix two low speed air streams, one at 25 °C and the other at 55 °C

- The goal of the simulation is to evaluate how effectively the two streams are mixed by examining temperature profile on the cross-section of the tee's exit
  - A more uniform profile indicates more effective mixing

# Problem Description



Air
5 m/s
55 °C

Air
3 m/s
25 °C

Outlet
P = 0 Pa (Gauge)

10mm  Ø101.6mm

288.45mm

10mm

711.2mm

Ø152.4mm

# Workshop Overview

- This workshop begins by importing a CAD geometry into Fluent and the geometry is the solid material of the pipes

- CFD simulation requires the geometry of the space inside the solid (known as the fluid domain or fluid region) and that geometry has to be created somehow

- In the main part of this workshop, the solid geometry is imported into Fluent, meshed surfaces are created to cap its openings and a fluid region is created and then filled with a volume mesh, all inside of Fluent
    - It is often advantageous to be able to begin with only the solid geometry

- There is an optional second part to the workshop, beginning on slide 42, in which the CAD geometry is instead opened first in SpaceClaim and the fluid domain is extracted there before being sent to Fluent to be meshed
    - This part can be done if interested in getting more practice with fluid volume extraction in SpaceClaim or comparing the two different approaches, but it is not required
    - In the end, the same mesh is produced and the CFD solver instructions are identical
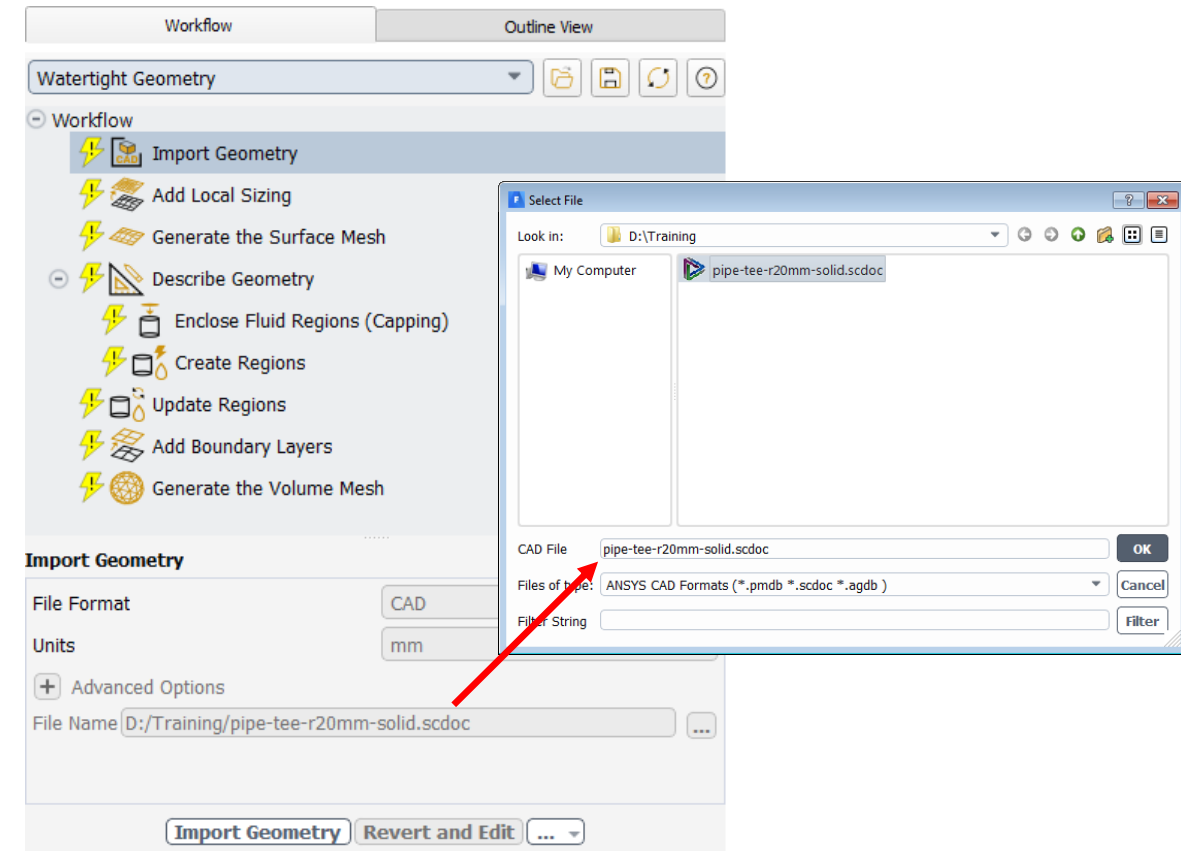
/Ansys
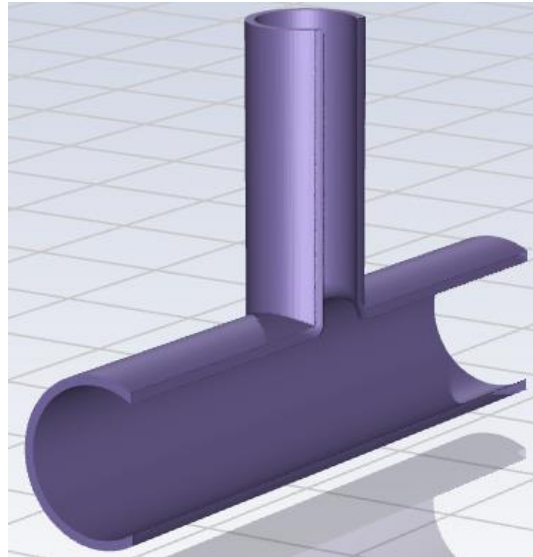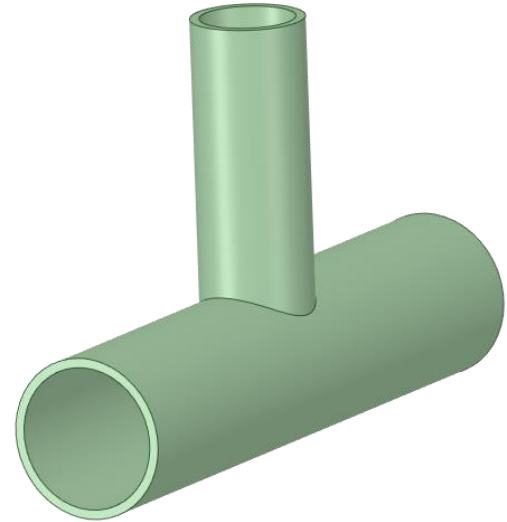
# Starting Fluent

- Open the Fluent Launcher Window and ensure Meshing Mode is selected
  - This workshop is compatible with all Capability Levels
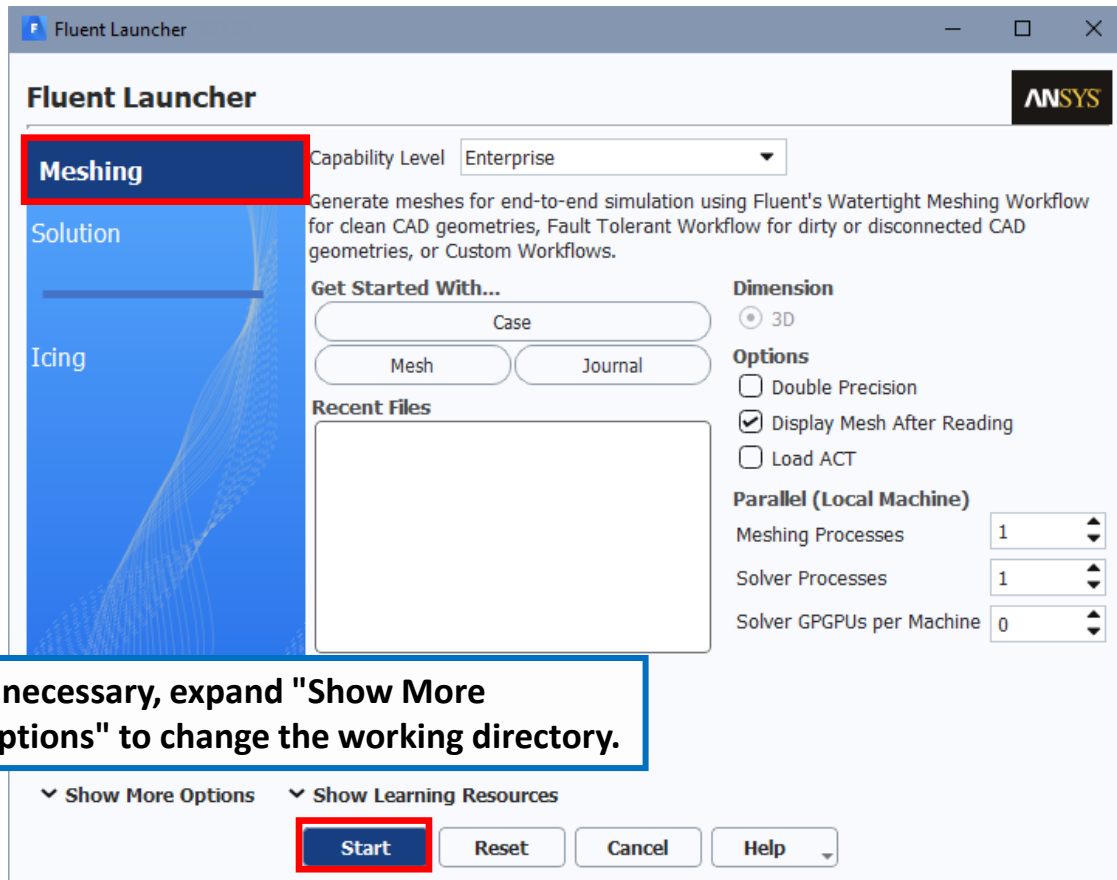
- In the Fluent window, select the Watertight Geometry workflow, and import "pipe-tee-r20mm-solid.scdoc"



**If necessary, expand "Show More Options" to change the working directory.**

# Basic Mouse Functionality

- In this workshop, the default mouse functionality will be used
  - Similar to default SpaceClaim mouse functionality
    - The middle mouse button rotates the model in the display window
    - The mouse scroll wheel can be used to zoom in and out
      - Alternatively, hold down the shift key and use the middle mouse button to zoom in and out
    - Control key plus middle mouse button can be used to drag the model
      - Fluent defaults for Ctrl + MMB and Shift + MMB are the opposite of the SpaceClaim defaults

- Customization of the mouse buttons is possible and will be described later in the training
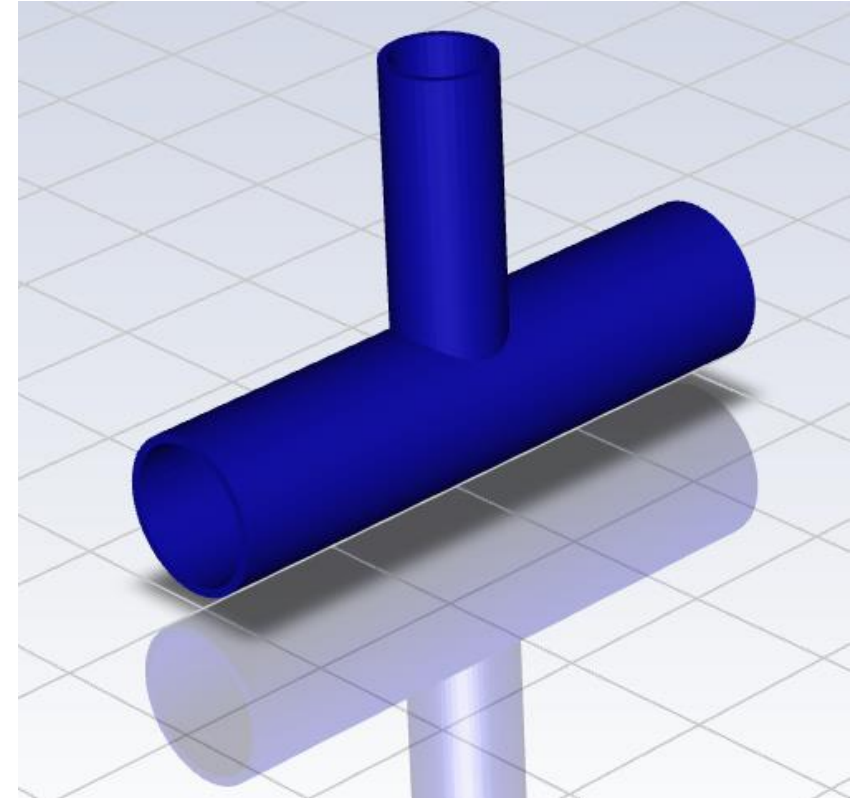
# Add Local Sizing

- Left click Add Local Sizing in the task list and keep "Would you like to add local sizing?" as no

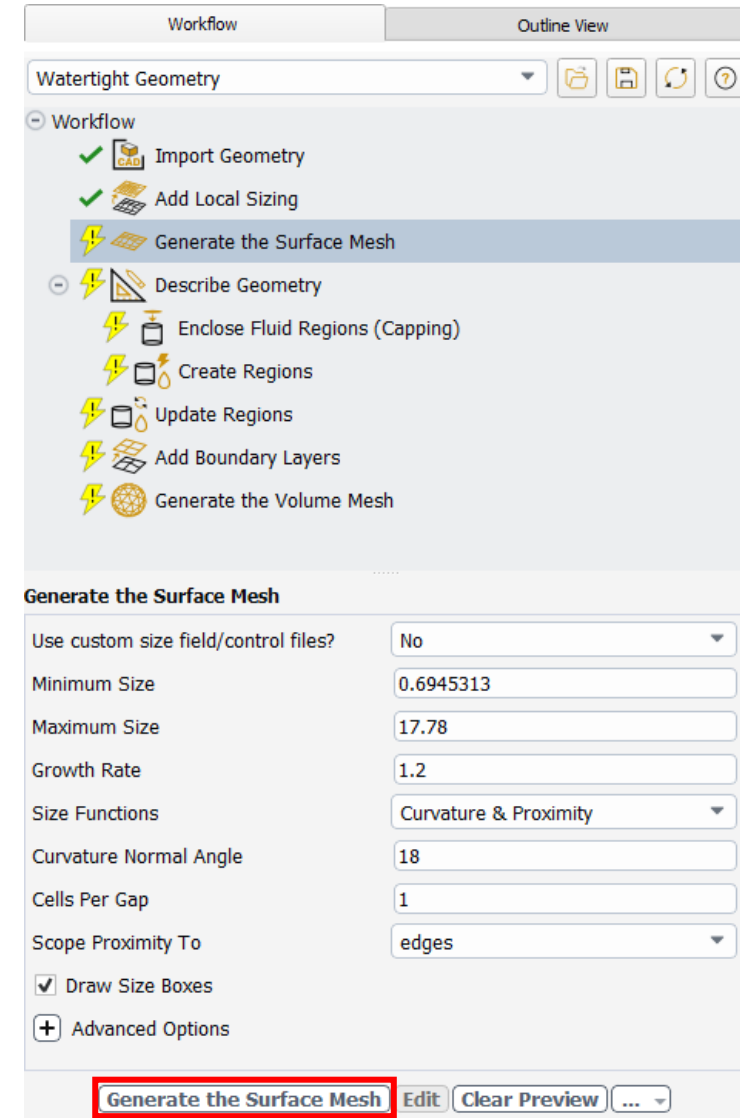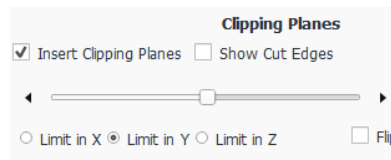- Right click on the Add Local Sizing task and select Update
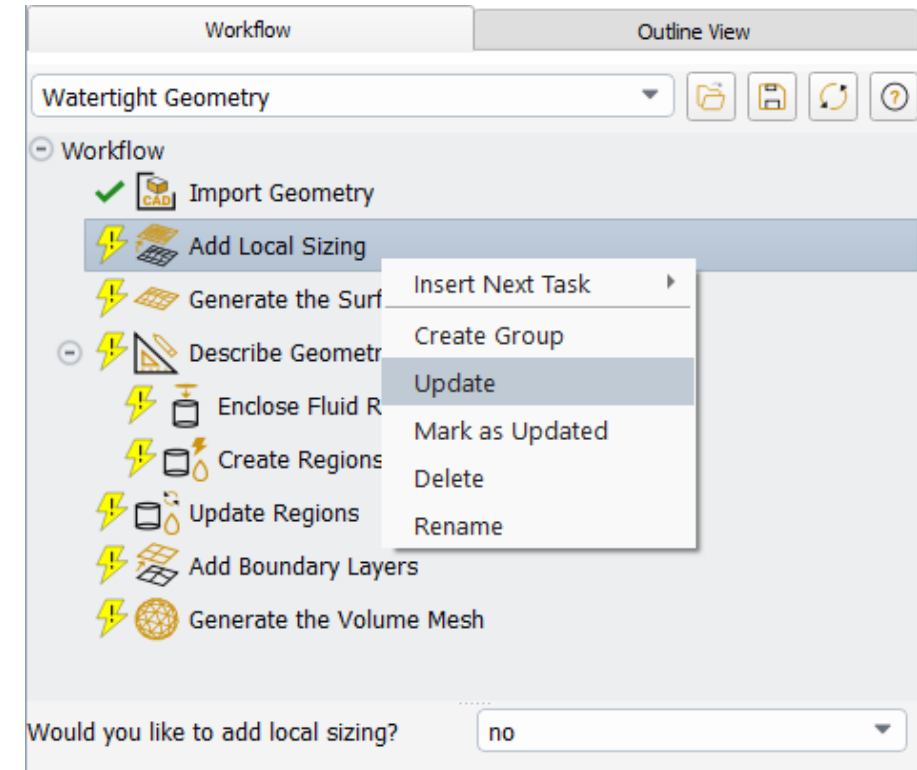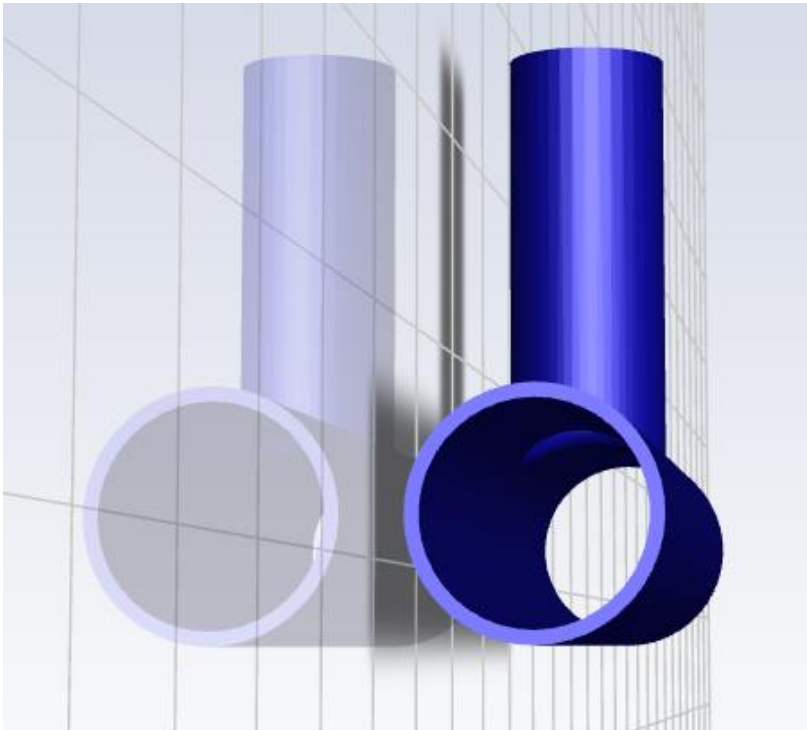
# Generate the Surface Mesh

- Keep default entries for the Generate the Surface Mesh task and create the mesh
  - Note the maximum skewness value reported in the console window ... values below 0.7 are acceptable
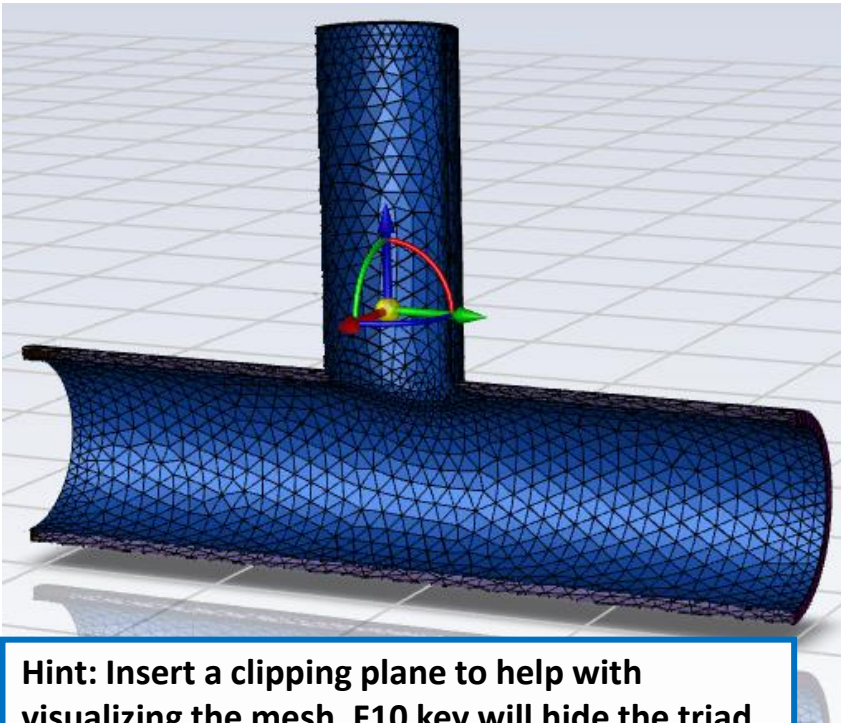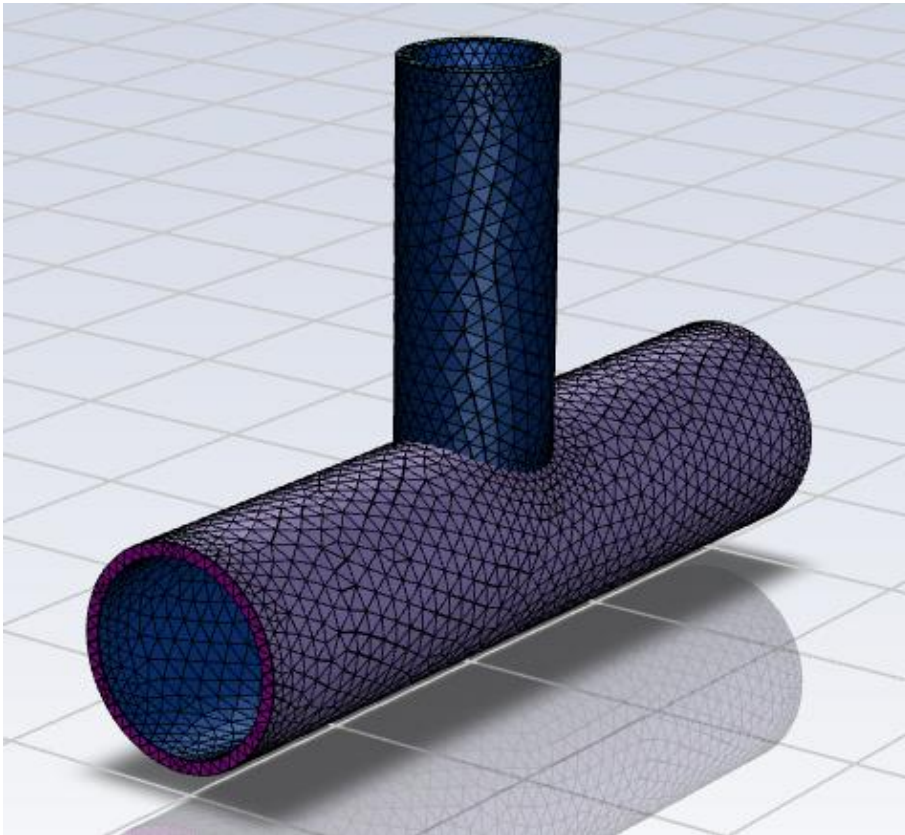


**Hint: Insert a clipping plane to help with visualizing the mesh. F10 key will hide the triad**

# Describe Geometry

- The geometry consists of only solid regions and in the next step you will cap openings and extract the fluid region
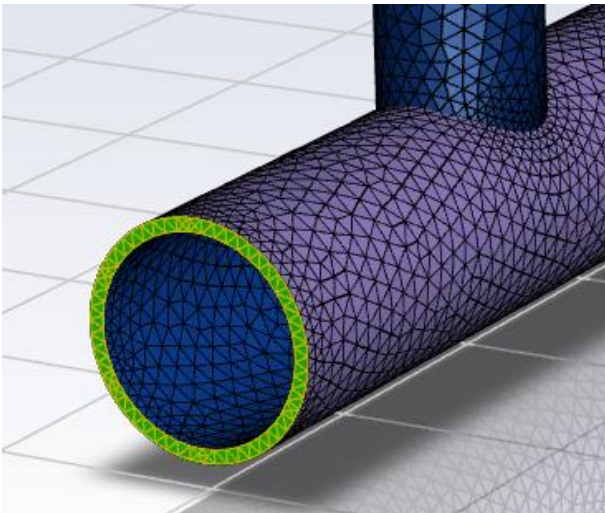
# Enclose Fluid Regions

- Capping surfaces can be selected by zone if named selections were not defined in SpaceClaim
  - The easiest way is to right click on the surface in the graphics window, which will cause it to be highlighted in the zone list
    - (this selection is not shown on this slide, proceed to next slide)

**There are no Named Selections in the CAD file, so automatic naming is applied to all of the surfaces. It is not possible to tell which surface is which by looking at the names, but instead of having to choose from the panel, you can make selections directly on the model in the graphics window, as shown in the next slide.**
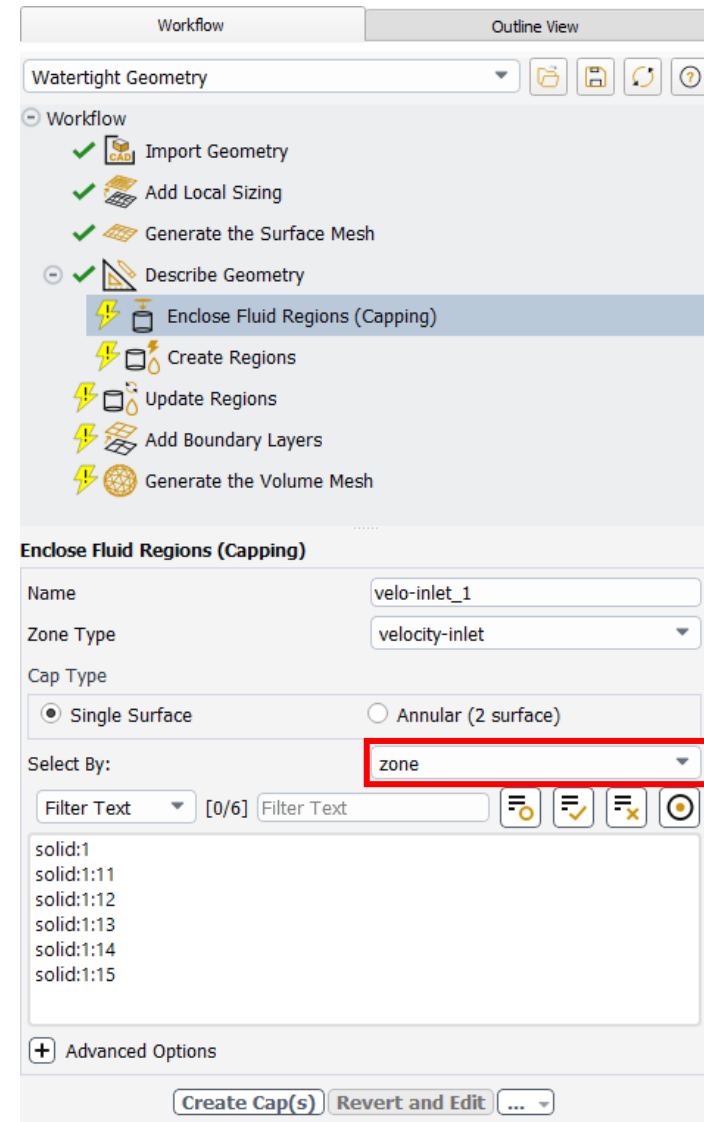
# Capping Surfaces: Inlets

- Right click on the surface bounding the large opening on the high-X side to select it
  - In case of accidental selection of a different surface, use the F2 key to clear the incorrect selection

- Change the name to "inlet-large" in the input panel and leave the type as "velocity-inlet"

- Click Create Cap(s)



**Enclose Fluid Regions (Capping)**

| | |
|---|---|
| Name | inlet-large |
| Zone Type | velocity-inlet ▾ |

Cap Type

◉ Single Surface          ○ Annular (2 surface)

Select By:          zone ▾

Filter Text ▾  [1/6] Filter Text

solid:1
solid:1:11
solid:1:12
solid:1:13
solid:1:14
solid:1:15

⊞ Advanced Options

Create Cap(s)   Revert and Edit   ... ▾

**Automatically generated surface names could be different on different computer systems. Rely on the selection in the graphics window, not the numbers shown in the figure above.**
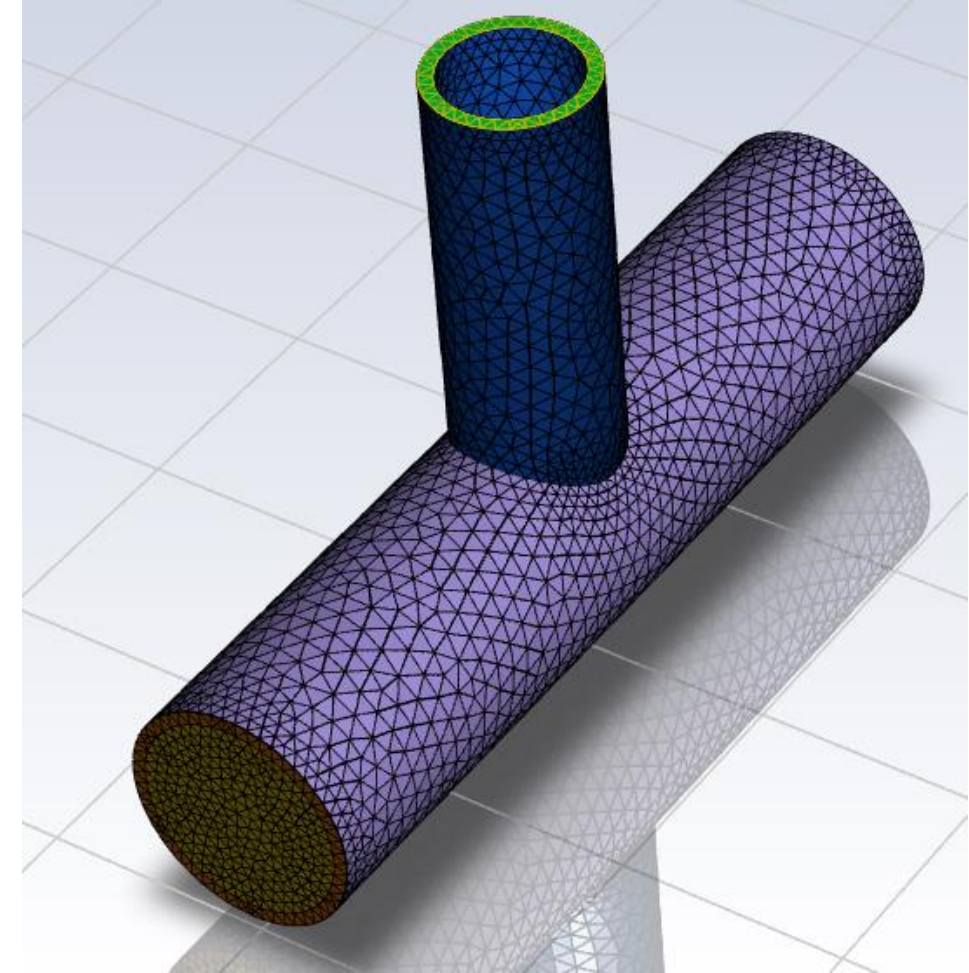
# Capping Surfaces: Inlets

- Right click on the surface bounding the small opening to select it

- Enter a name of "inlet-small" in the input panel and leave the type as "velocity-inlet"

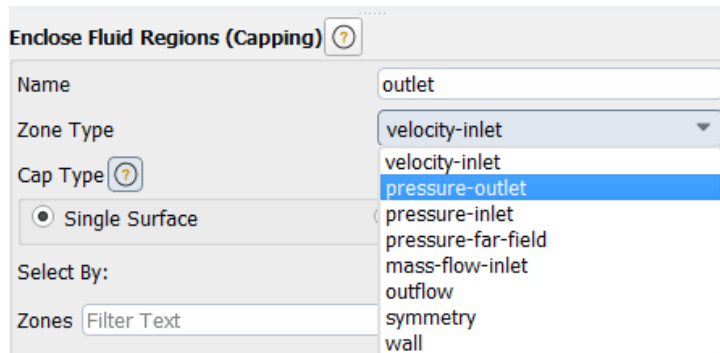- Click Create Cap(s)

**Enclose Fluid Regions (Capping)**

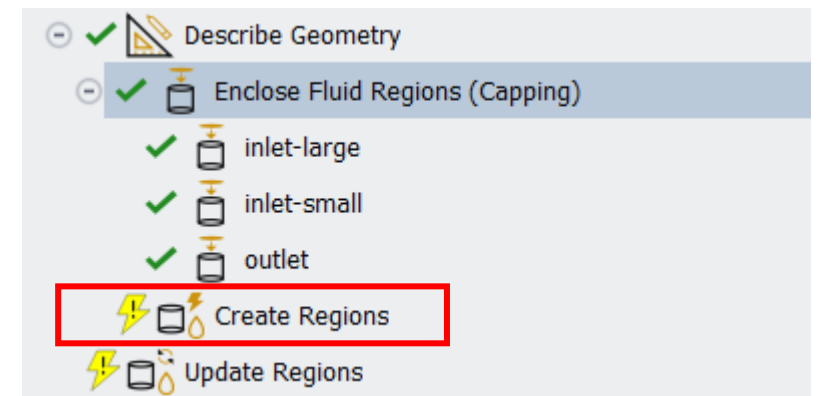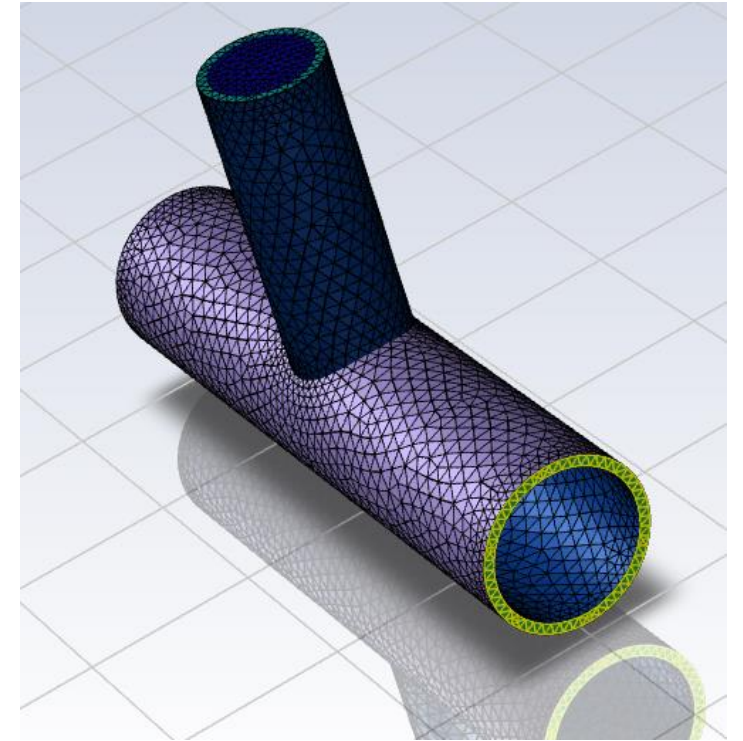| | |
|---|---|
| Name | inlet-small |
| Zone Type | velocity-inlet ▼ |

[ **Create Cap(s)** ]

# Capping Surfaces: Outlet

- Right click on the remaining large opening

- Enter a name of "outlet" in the input panel and make sure to change the zone type to "pressure-outlet"

- Click Create Cap(s)





**You have now fully capped the fluid region and ready to extract the fluid zone. Click on the "Create Regions" Task in the Workflow**
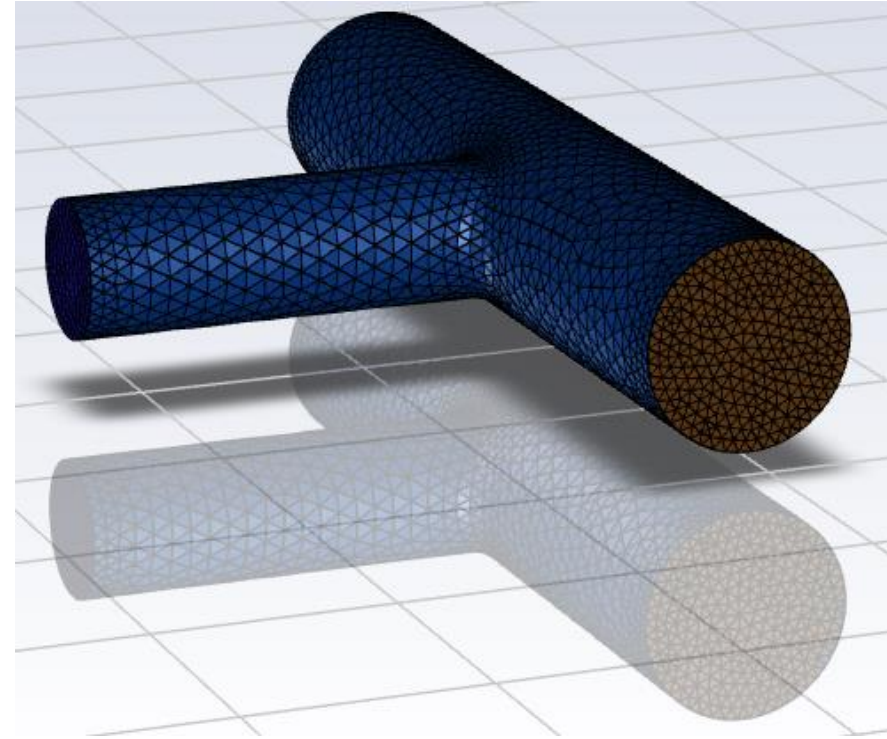
# Regions

- In Create Regions, the estimated number of fluid regions is 1, which is correct, so just click Create Regions, which will create the fluid zone of the domain

- In Update Regions, change the name of the fluid region to "fluid" by clicking on the corresponding Region Name

- Click Update Regions to complete the task
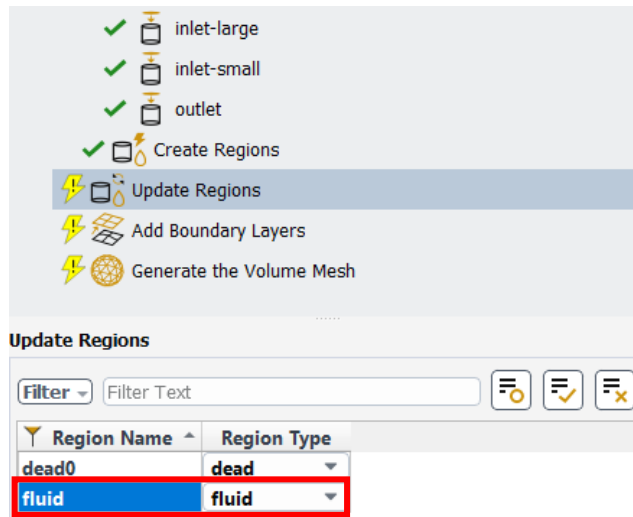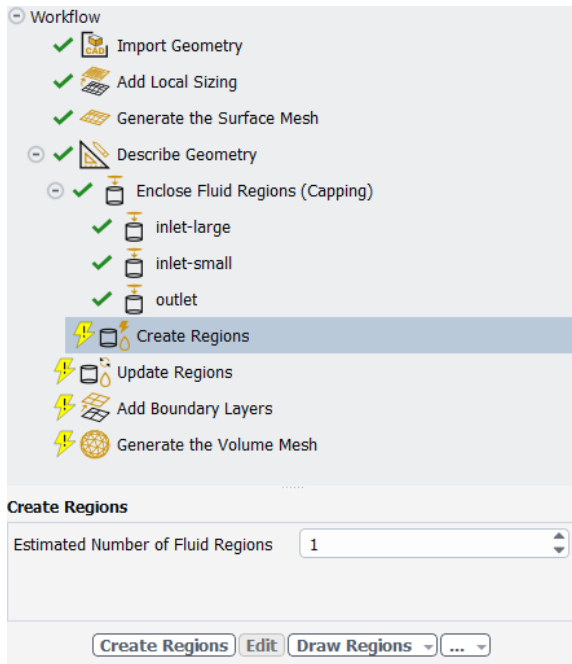


**The solid region from the CAD import has been misidentified as a dead region. However, we only need the fluid region for this problem, so leave the dead region as it is.**

©2021 ANSYS, Inc. Unauthorized use, distribution, or duplication is prohibited.

# Add Boundary Layers and Generate the Volume Mesh

- Complete the Add Boundary Layers and Generate the Volume Mesh tasks using the default settings



**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 "mixing-tee-volume-mesh.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
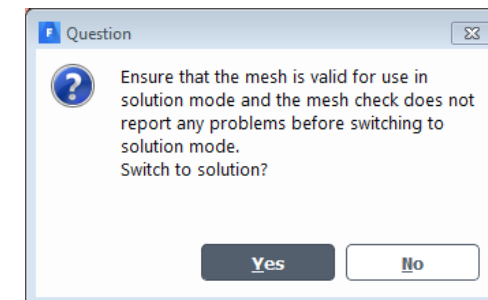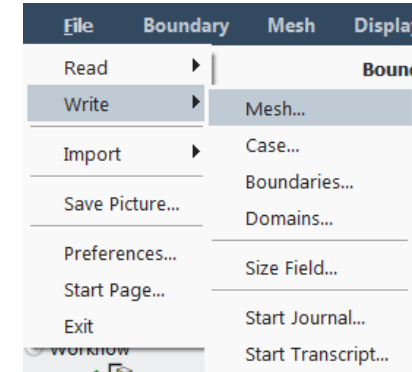
# Fluent Workflow (Solution Mode): Ribbon
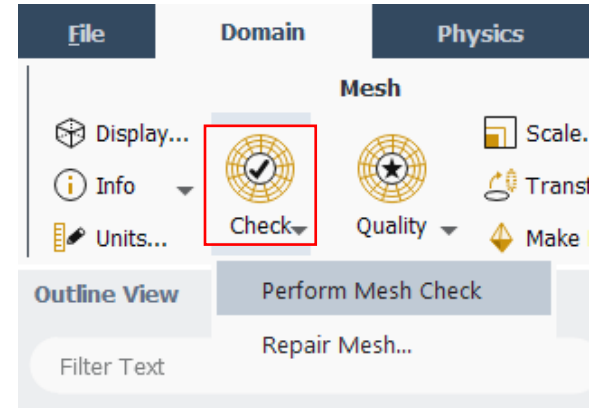
- The Ribbon is used to guide the basic Fluent workflow
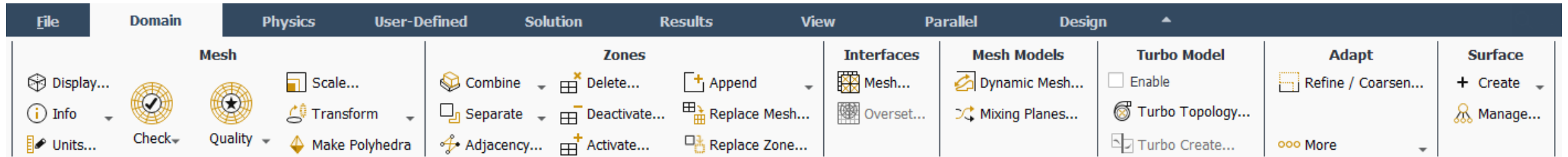


- The four primary tabs used in every simulation are
  - Domain
  - Physics
  - Solution
  - Results

- For this case (and probably most other cases too), you will use them going in order from left to right

# Domain: Mesh Check

- In the Domain tab, in the Mesh group, click Check then Perform Mesh Check and examine the output in the Console
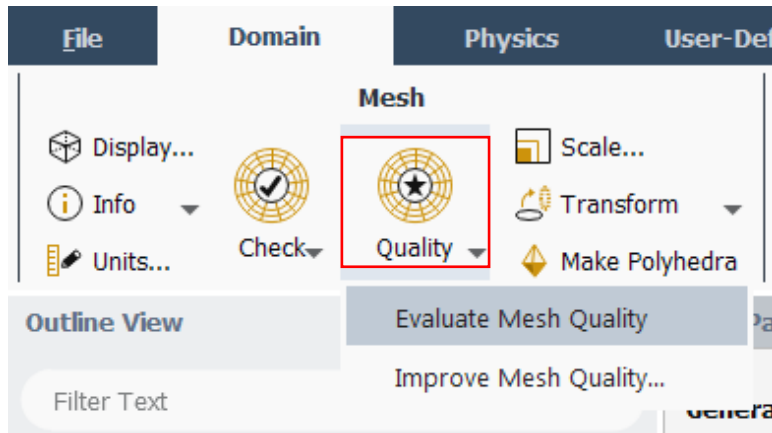


**Console**

```
Domain Extents:
   x-coordinate: min (m) = -3.556000e-01, max (m) = 3.556000e-01
   y-coordinate: min (m) = -7.608395e-02, max (m) = 7.609602e-02
   z-coordinate: min (m) = -7.609520e-02, max (m) = 3.746500e-01
Volume statistics:
   minimum volume (m3): 2.659074e-08
   maximum volume (m3): 8.652588e-06
     total volume (m3): 1.523446e-02
Face area statistics:
   minimum face area (m2): 4.243395e-07
   maximum face area (m2): 3.899928e-04
Checking mesh...............................
Done.
```

The mesh check ensures that each cell is in a correct format and connected to other cells as expected. It is recommended to check every mesh immediately after reading it. Failure of any check indicates a badly formed or corrupted mesh which will need repairs prior to simulation.

You only have to look for *error messages* in the output. If "Done" appears, as it does here, without any errors or warnings, the mesh check was successful.

# Domain: Mesh Quality

- In the Domain tab, in the Mesh group, click Quality then Evaluate Mesh Quality, and examine the output in the Console



Mesh quality is very important for getting a converged, accurate solution. The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1.

Ideally, the minimum orthogonal quality for all types of cells should be more than 0.1, with an average value that is significantly higher (0.2), but sometimes it is possible for the minimum to be as low as 0.01 (Source: ANSYS Fluent User's Guide).

This value was also reported in meshing mode upon creation of the volume mesh and in general it is not necessary to repeat the quality check in solution mode if it has already been done in Meshing mode.

The maximum aspect ratio is within an acceptable range for meshes with boundary layers.

# Domain: Units

- In the Mesh group, click Units



- Scroll down to "temperature", select C to change the units to Celsius, then close the panel

In the background, Fluent converts all units to SI when it calculates the solution. The units panel allows you to choose familiar units to be used for problem definition and results post-processing.
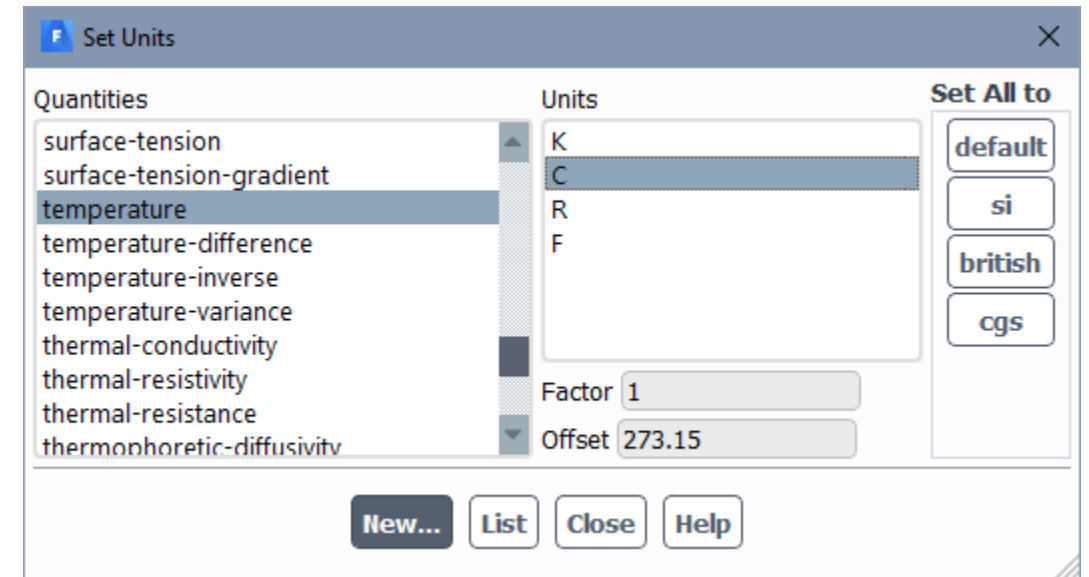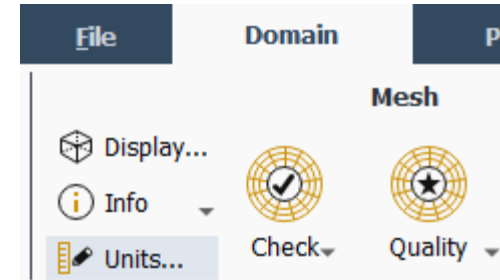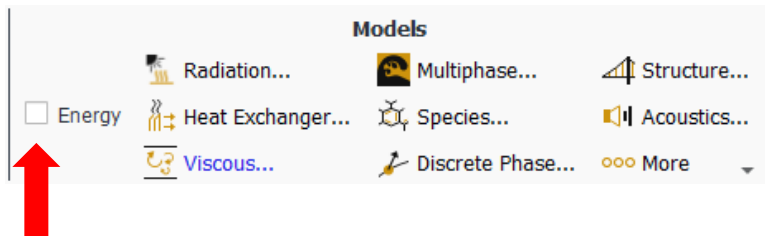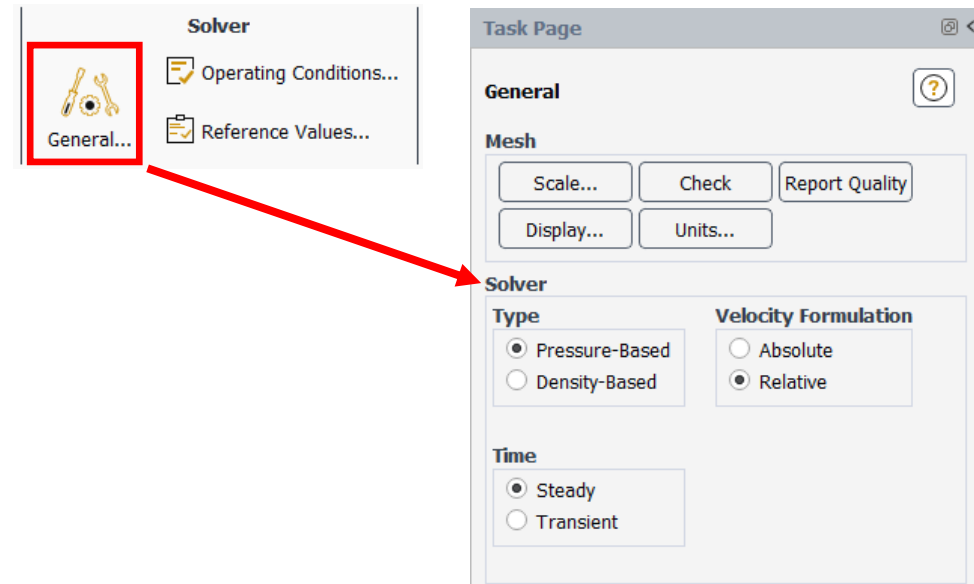
# Physics

- Remember, the Ribbon is used to guide the basic Fluent workflow
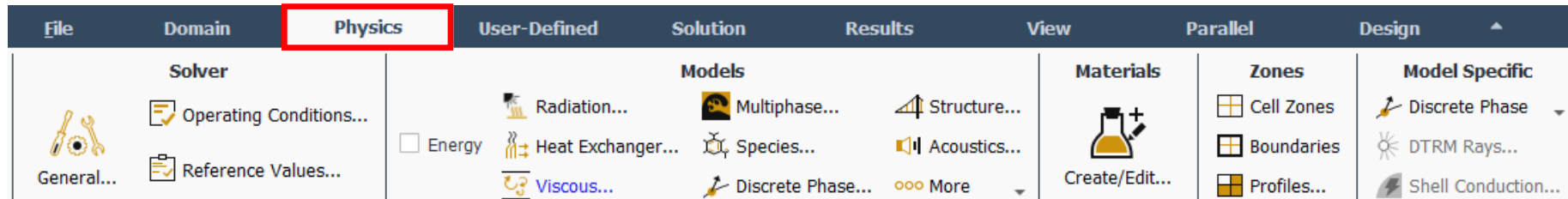


- The four primary tabs used in every simulation are
  - Domain
  - Physics
  - Solution
  - Results

  - You normally go through them from left to right – click the Physics tab in the Ribbon

  - In Physics, you will almost always need to provide input or make choices for entries in the Solver, Model, Materials and Zones sections
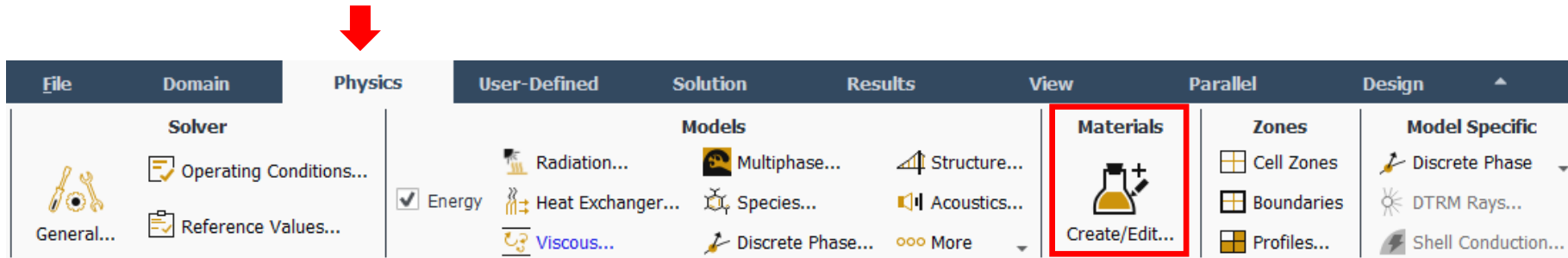
# Physics: Solver and Models

- The default settings in Solver are Time = Steady and Type = Pressure-Based.
    - Inputs for these are found by clicking General
    - Use these settings for this tutorial



- In Models, select Energy
    - This tells the solver to solve for the temperature as the hot and cold streams mix

# Physics: Materials

- Click on Create/Edit… in the Materials section



- This will open the Materials panel
  - The default material is air, so no changes are needed
    - The properties correspond to atmospheric pressure and 15° C (59° F)
  - If you were simulating a gas other than air, or simulating water, or some other liquid, it would have to be defined using this panel, which you will learn in one of the later workshops
  - Click Close to close the panel.
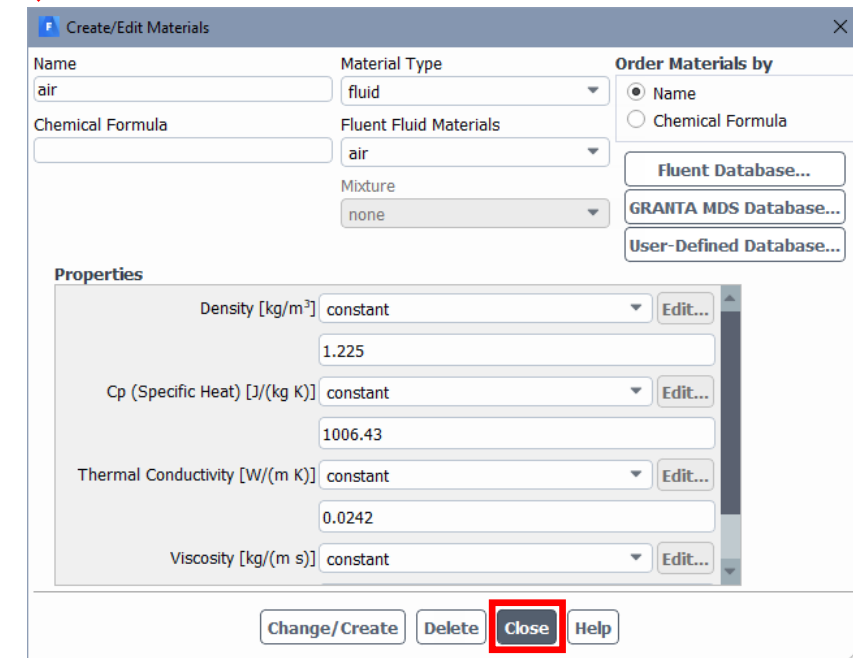
# Physics: Zones
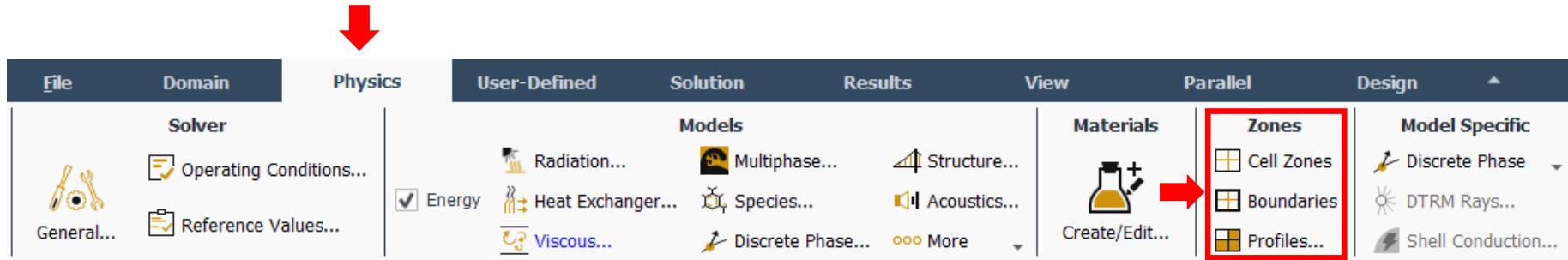


- In Fluent, the cells in the mesh are grouped into one or more cell zones

- Each cell zone is bounded by one or more boundary zones

- You will define boundary conditions for boundaries and cell zone conditions for cell zones

- Default cell zone settings apply for this problem – you will learn about defining cell zone conditions in other workshops

- Boundary conditions need to be defined
  - Click Boundaries to bring up the Task Page for Boundary Conditions
    - (continued on the next page)
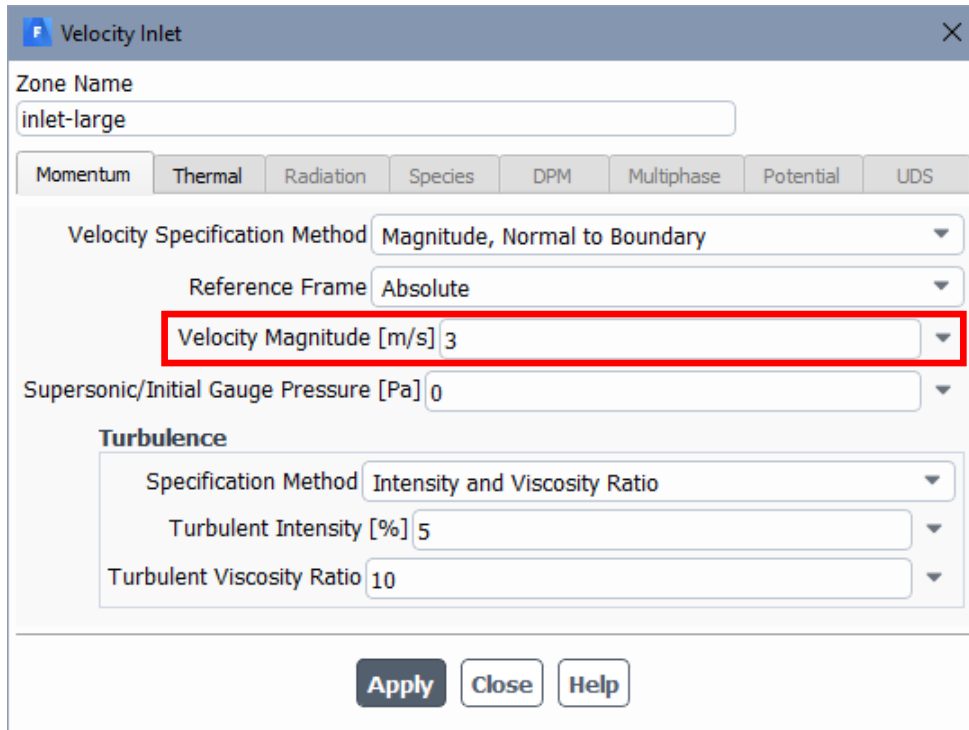
# Physics: Boundary Conditions (1)
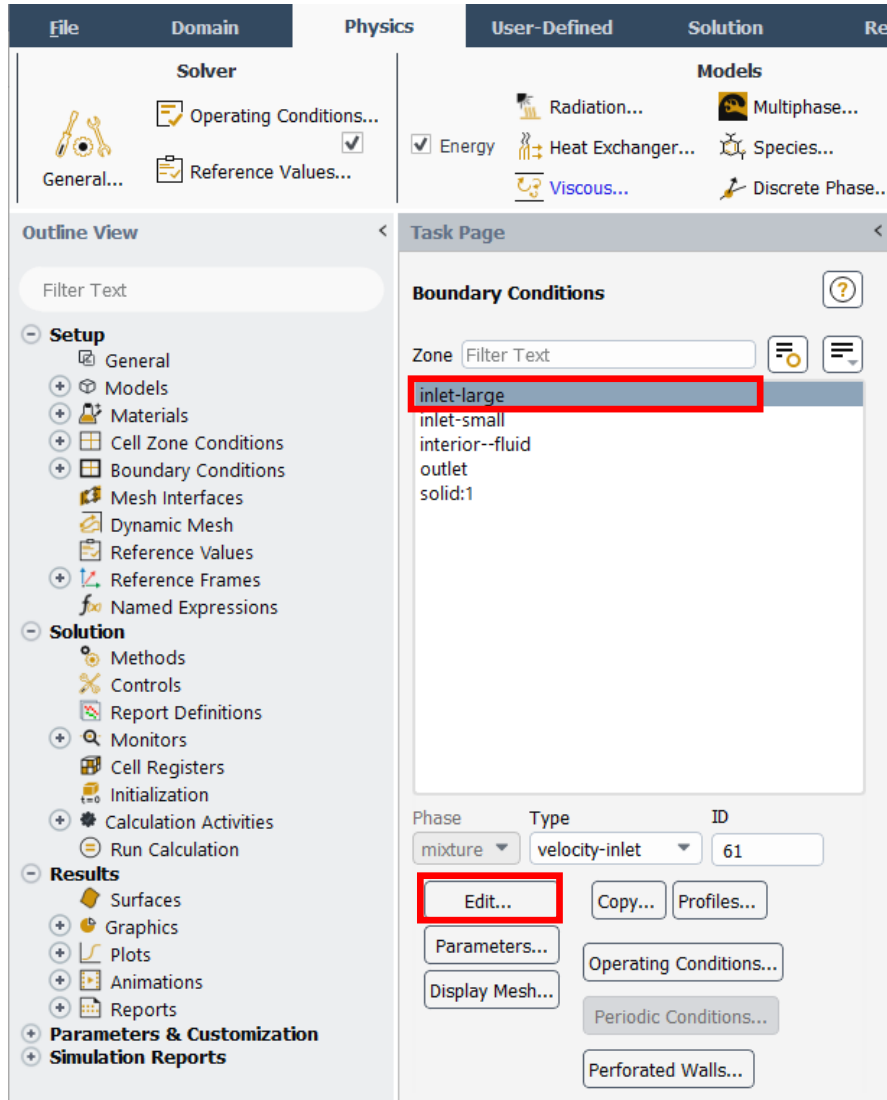


- Clicking on Boundaries in the Ribbon (previous page) opens the Boundary Conditions task page
- There are five zones listed
  - inlet-large, inlet-small, and outlet come from the capping surfaces defined in the Watertight Geometry workflow
  - solid:1 is the bounding surfaces (e.g. walls) of the fluid domain
  - The other zone in the list, interior--fluid, is the collection of all the internal faces of all the mesh cells
    - Fluent needs this for its internal data structure but no condition is applied on these faces
    - You will have zones such as this in all your Fluent models and can simply ignore them
- Select inlet-large in the list and click Edit...
  - Or just double click on the name

# Physics: Boundary Conditions (2)

- Enter a value of 3 m/s for the velocity

- Click the Thermal tab and enter a value of 25 °C

- Click Apply, then close the panel
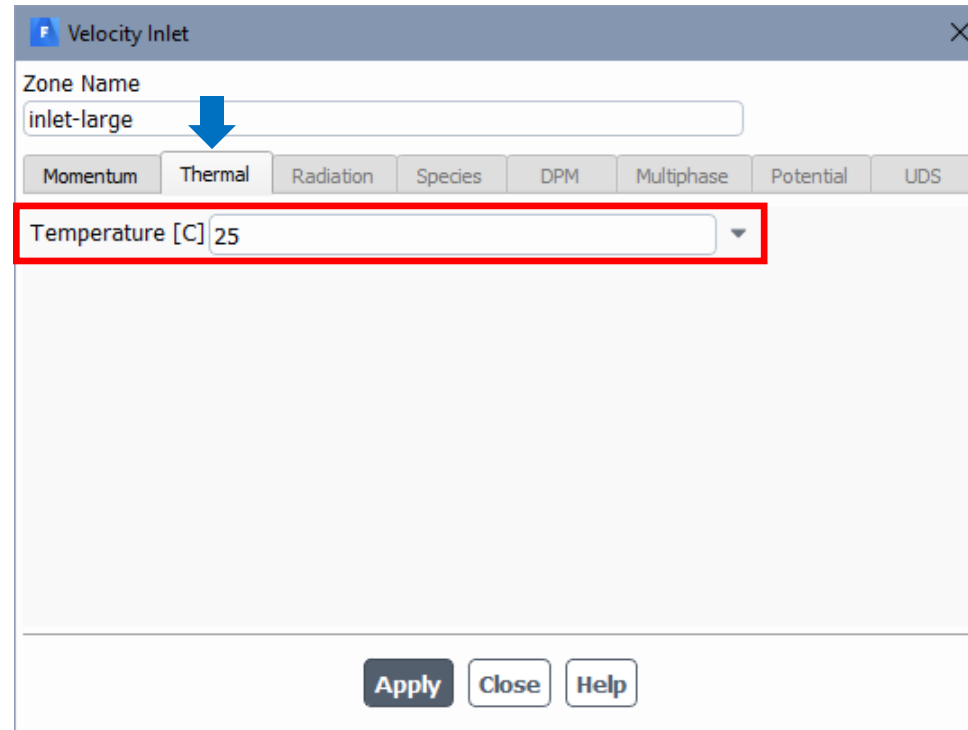
# Physics: Boundary Conditions (3)

- Open the boundary conditions panel for inlet-small

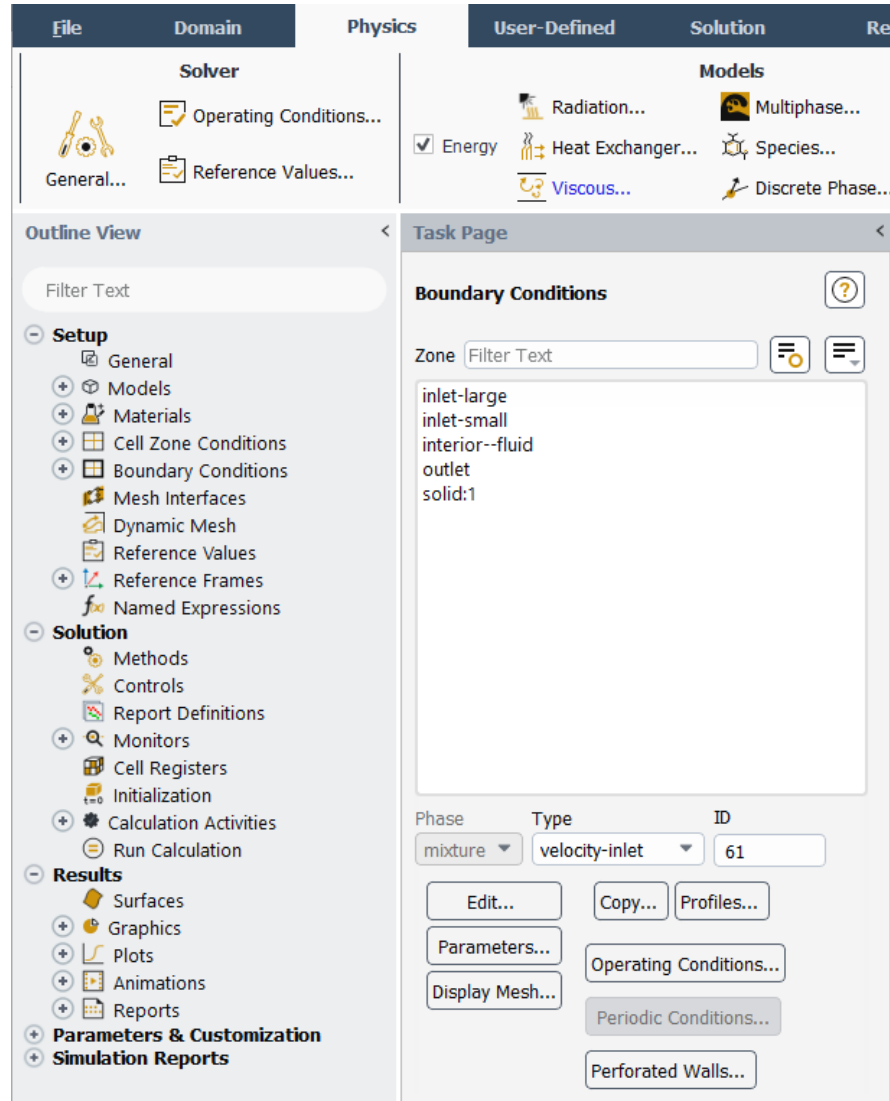- Enter values of 5 m/s for the velocity and 55 °C for the temperature

- Click Apply, then close the panel
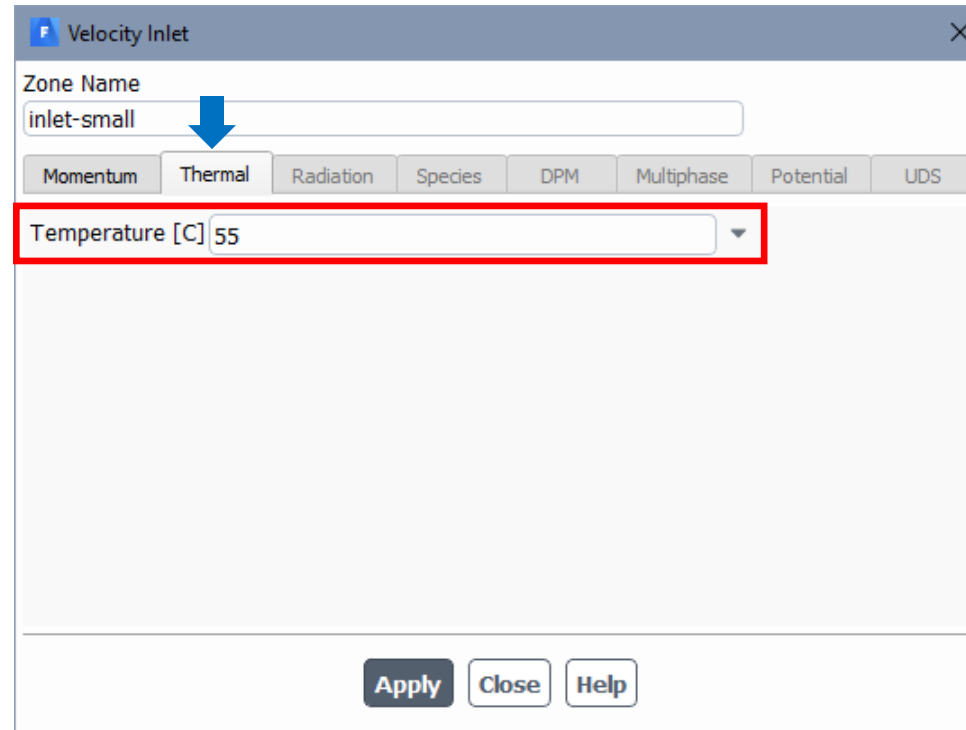
# Physics: Boundary Conditions (4)

- Open the boundary conditions panel for outlet

- Note that the default setting for outlets is Gauge Pressure = 0
  - By coincidence, this is the desired value for this problem, so just close the panel

# Physics: Boundary Conditions (5)



- Two zones remain
  - "solid:1" zone
    - The default momentum boundary condition for walls in Fluent is the no-slip condition
    - The default thermal boundary condition is adiabatic (heat flux = 0)
      - In this problem, we are assuming the wall is adiabatic so it is not necessary to open the panel
  - "interior--fluid" zone
    - Interior zones such as interior--fluid require no inputs
    - It is never necessary to open the boundary condition panel for interior zones

# Solution

- Remember, you are using the Ribbon to guide the workflow
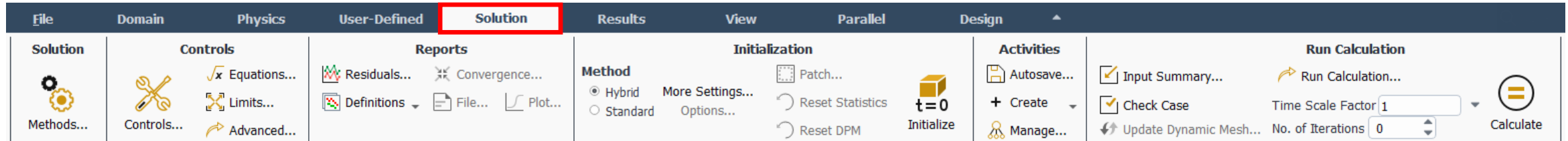


- The four primary tabs used in every simulation are
  - Domain
  - Physics
  - Solution
  - Results

- You normally go through them from left to right, so now click the Solution tab

- In Solution, you will first create a report definition to help monitor the solution progress, then initialize and calculate the solution

# Solution: Report Definition
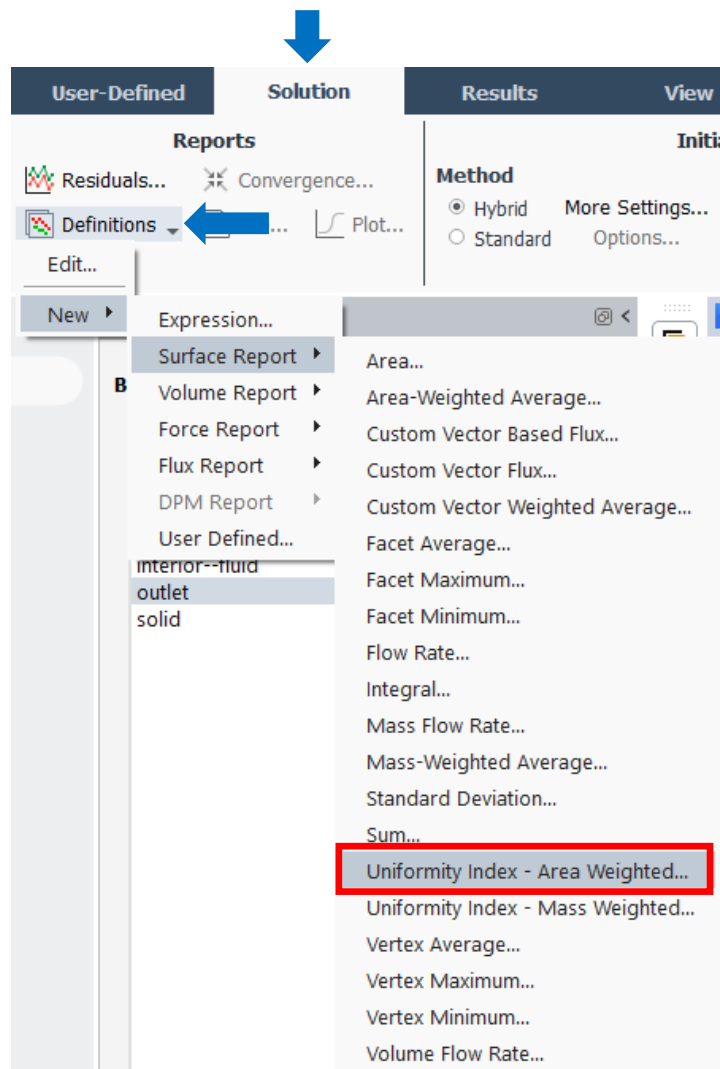


- In the Solution tab, click on Definitions in the Reports section and choose New > Surface Report > Uniformity Index – Area Weighted…
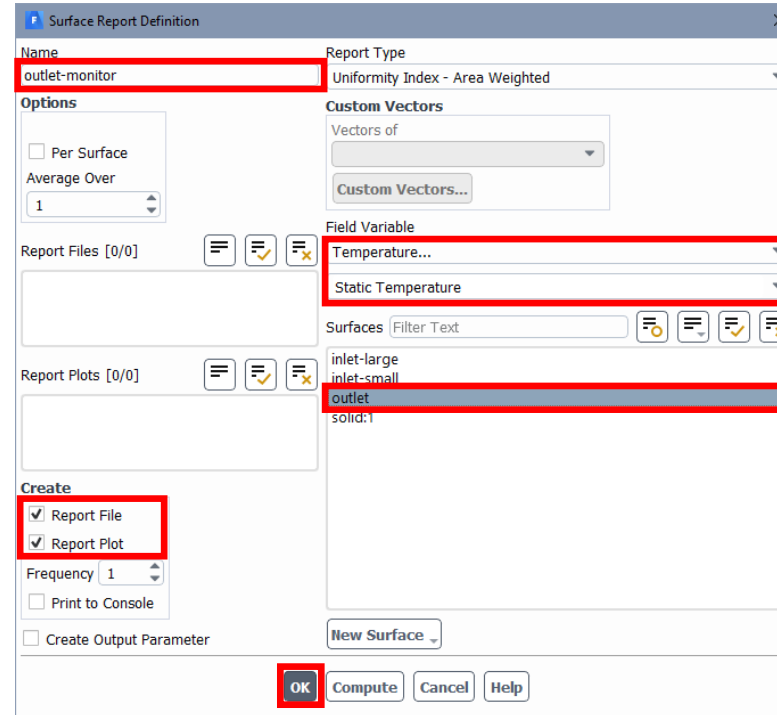
**Enter the following in the definition panel:**
Name = outlet-monitor
Variable = **Static Temperature**
Surfaces = **outlet**
Report File = **check**
Report Plot = **check**

Click **OK** to finalize the report definition and OK in the Information panel (not shown)

# Solution: Residuals and Report Definitions

- You will soon see that by default Fluent plots values of the residuals, which are indications of errors in the current solution

- Residuals should decrease during the calculation and there are guidelines on the level of reduction needed for the solution to be considered converged
  - This will be covered in the class – defaults work fine for the majority of cases and will be used here

- It is also recommended to observe other important solution quantities, which is the role of Report Definitions
  - We are interested in mixing and temperature distribution at the outlet, so for this problem it makes sense to monitor temperature uniformity at the outlet
  - We want to see that the values have stopped changing by the time the residuals converge

# Solution: Initialize and Calculate
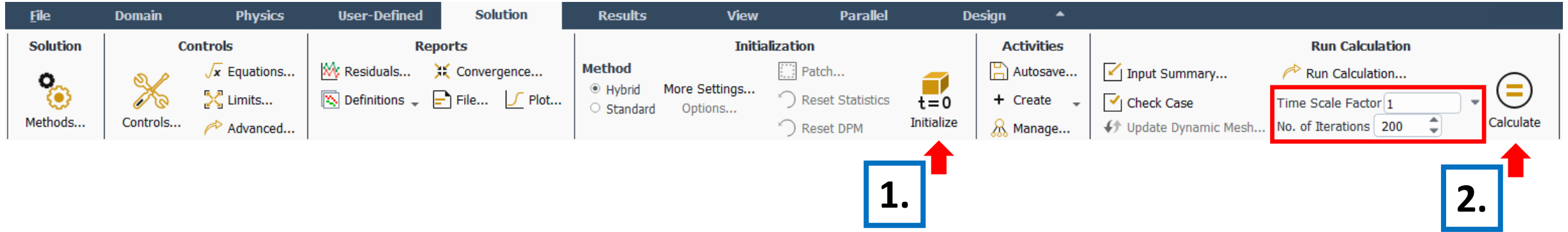


1. **Click Initialize**
   - The iterative method used by Fluent to calculate the flow solution requires each of the cells to be assigned an initial value for all solution variables, which is what Initialize does

2. **Set the number of iterations to 200 and click Calculate**

# Solution

- Right click on the window tab and choose SubWindow View to show both the residuals and the report definition plot
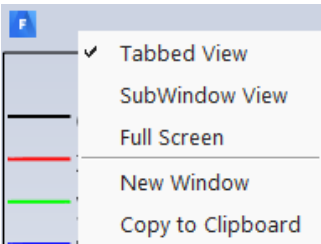


- You can do this while the solution is iterating
- If you want to go back, just right click and select Tabbed View

- The iterations will automatically stop when the residuals reach convergence
- Notice in the plot on the right that the uniformity has reached a constant value
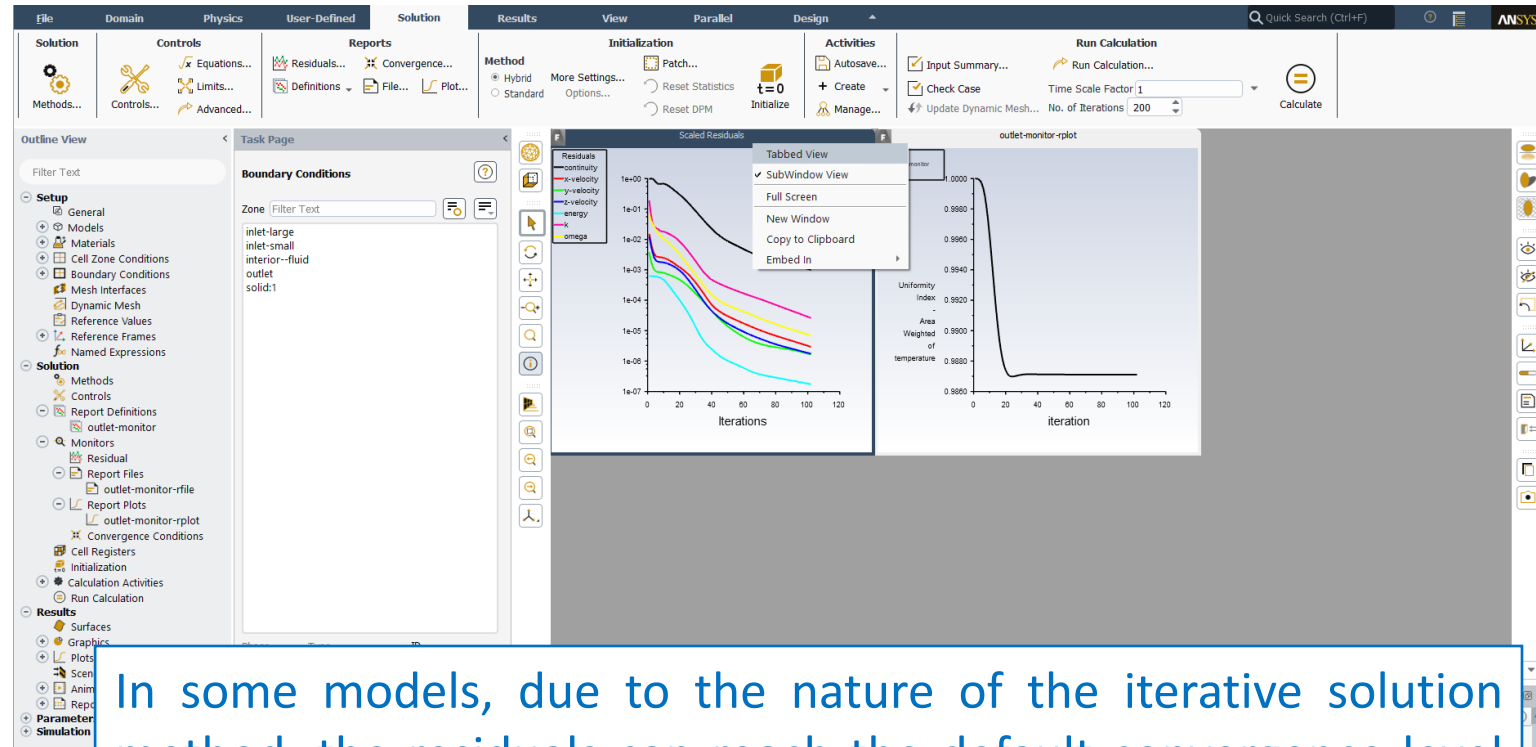


In some models, due to the nature of the iterative solution method, the residuals can reach the default convergence level before the solution values become constant.  This is undesirable and later in the course you will learn what to do if that happens.  Here everything looks good.  Save the case and data files in File > Write > Case & Data
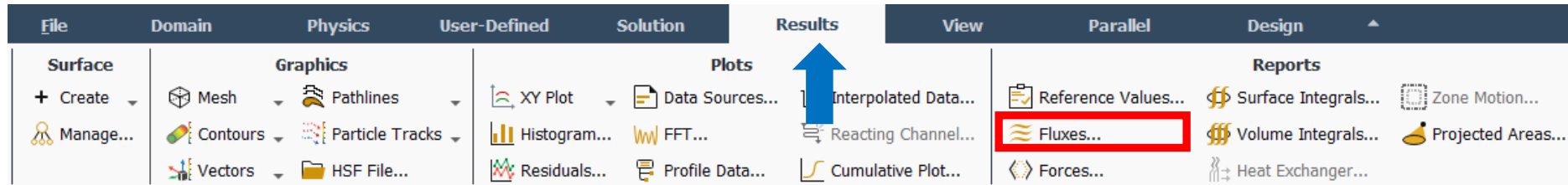
# Results

- Remember, use the Ribbon to guide your workflow



- The four primary tabs used in every simulation are
  - Domain
  - Physics
  - Solution
  - Results
- You have been going through them from left to right, so now click the Results tab
- In Results, you will check for conservation of mass and view temperature contours on the inlets, outlets and wall

# Results: Reports



- In the Reports section, click on Fluxes

- Select the Mass Flow Rate option

- Select the two inlets and the outlet in the list of boundaries

- Click Compute
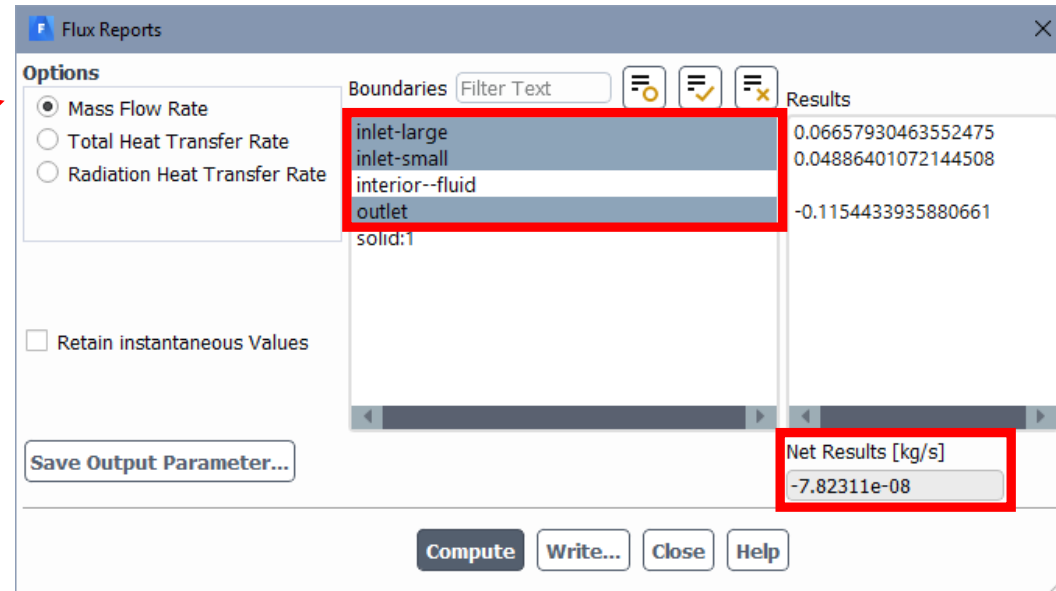
In the Flux Reports panel, a positive value of mass flow rate means that flow enters the domain and a negative value means it leaves the domain. Therefore, according to conservation of mass, if the values from all inlets and outlets are added together, they should sum to zero. Due to the nature of the numerical method, the Net Results value will never be exactly zero, but it should be small compared to the inlet value. Here it is several orders of magnitude lower than the value from any individual boundary, which is very low. The normal expectation is that it should be less than 1%.
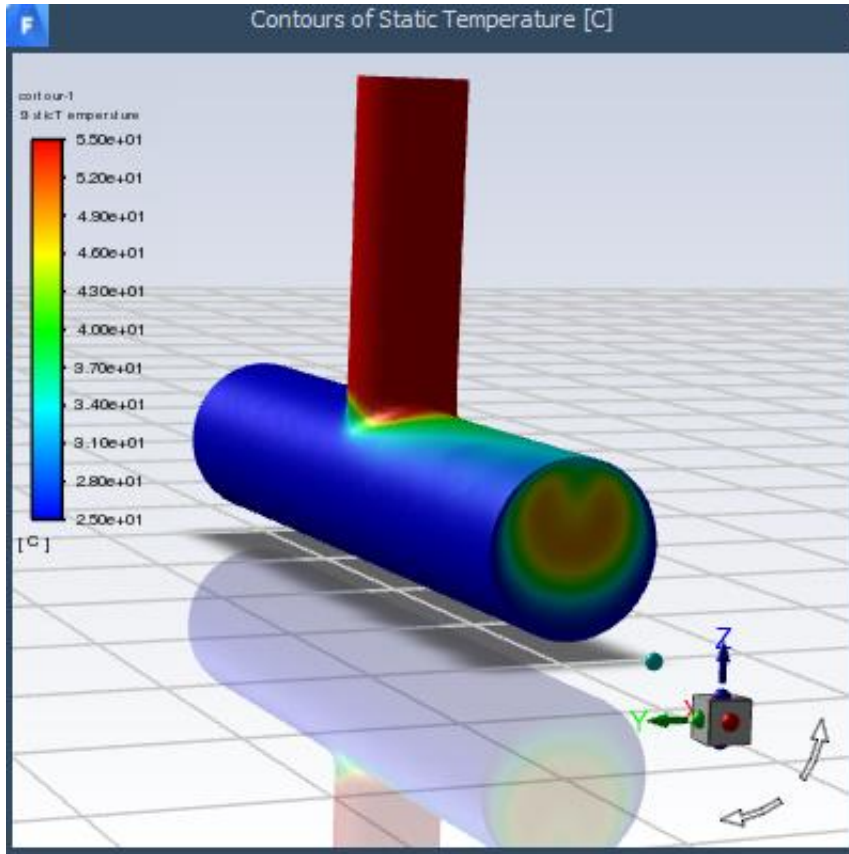
# Results: Contours
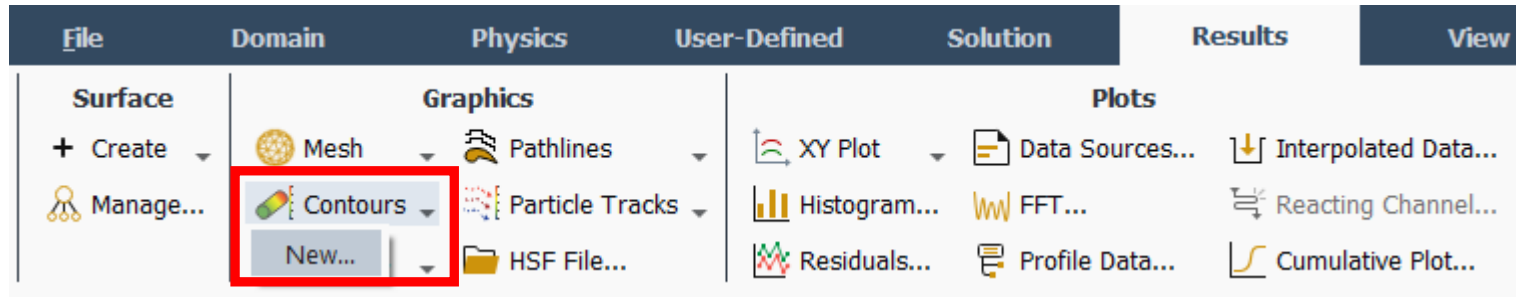


- In the Graphics section, click Contours and New…

- In the Contours panel, select all surfaces

- Select Static Temperature for the variable

- Select the Filled option

- Click Save/Display and then Close
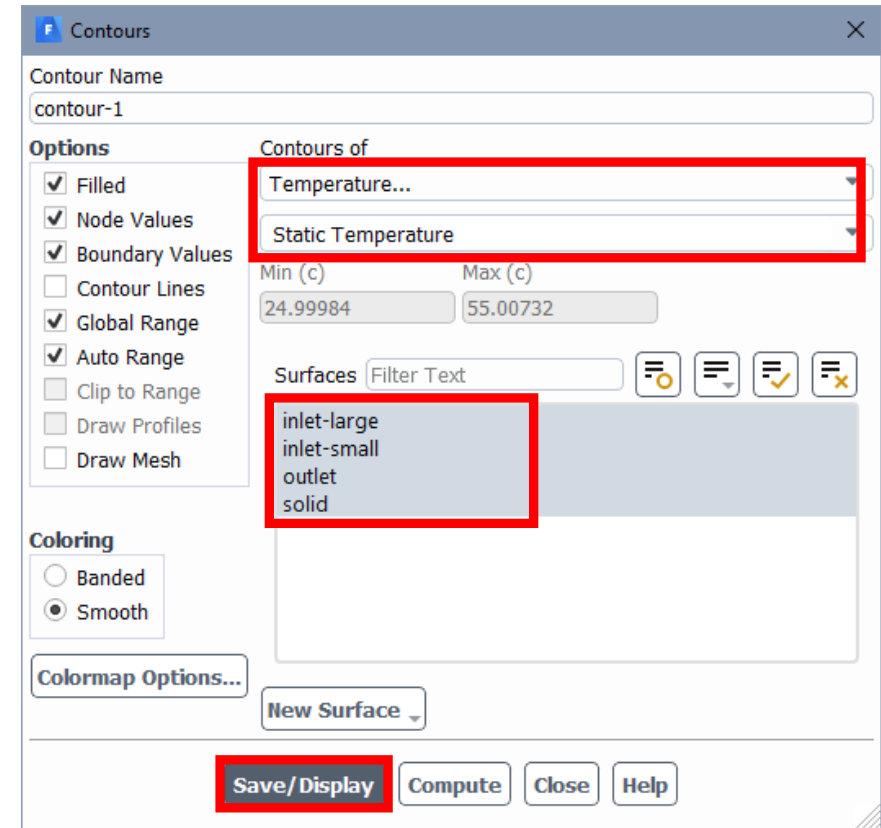
# Results: Contours



Contours of Static Temperature [C]

- Contours appear in the graphics window after Save/Display was clicked

- The temperature distribution on the outlet shows the streams are not completely mixed at the outlet position

Remember you can toggle the graphics between tabbed view and sub-window view by right clicking on the title bar of the tab or the window.

# Close Fluent and Save Case and Data

- Write the case and data file and exit Fluent
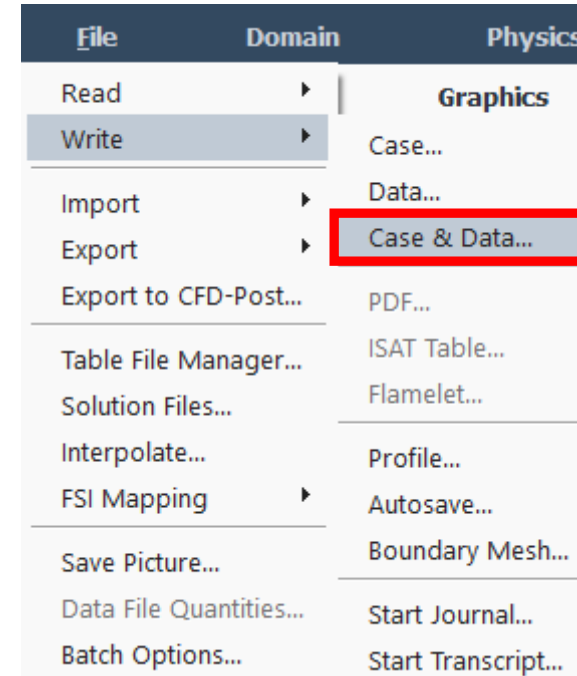
# Summary

- In this tutorial you have learned the basic steps for CFD analysis with ANSYS Fluent

- Start with a CAD file of your geometry

- Import the CAD file and create the mesh using the Watertight Geometry workflow in Fluent Meshing
  - In this case, the CAD was a model of solid parts and capping surfaces were defined to extract a fluid region

- When the volume mesh has been created, switch to the solution mode
  - This all occurs in the same window with mesh transfer from meshing to solver taking place in the background

- Set up, solve and post-process the problem in Fluent solution mode
  - The solution mode workflow is guided by the ribbon

# Optional

- Some people may prefer extracting a flow volume in SpaceClaim to using capping and region extraction in Fluent

- In this optional exercise, you can read the CAD file into SpaceClaim, perform the fluid volume extraction, and send the fluid volume directly to Fluent to be meshed
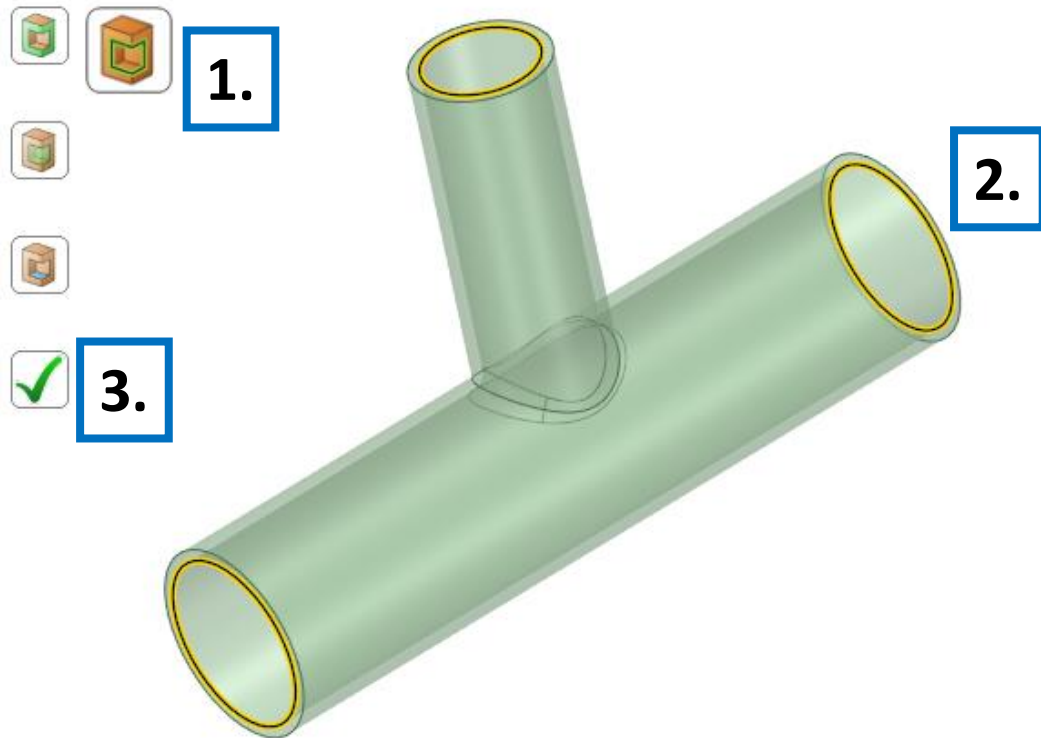
# Launch SpaceClaim and Import File

- Start SpaceClaim and open pipe-tee-r20mm-solid.scdoc
- Click the Prepare tab and select Volume Extract
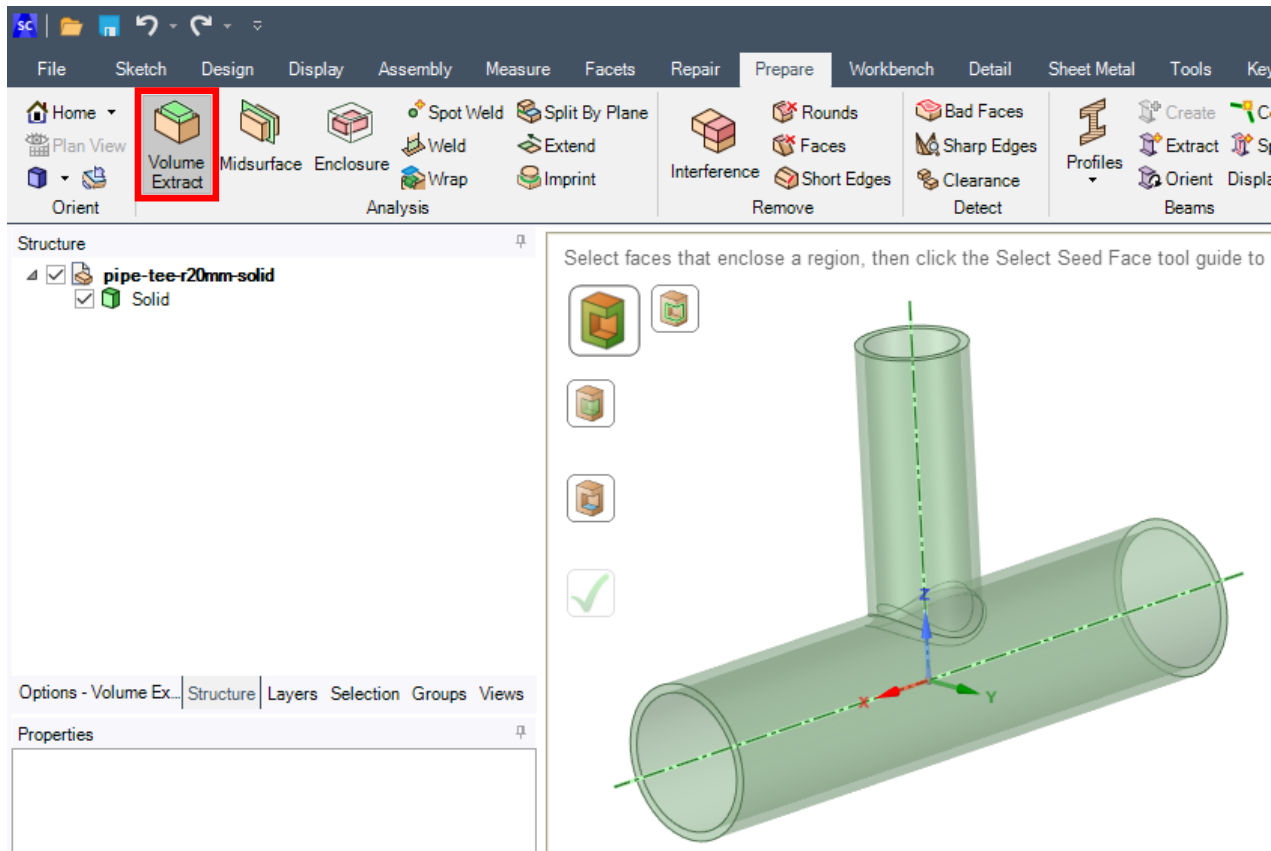
# Volume Extract

- Click the icon for edge loop selection
- Select the three edges shown in the figure
- Click the Complete tool guide
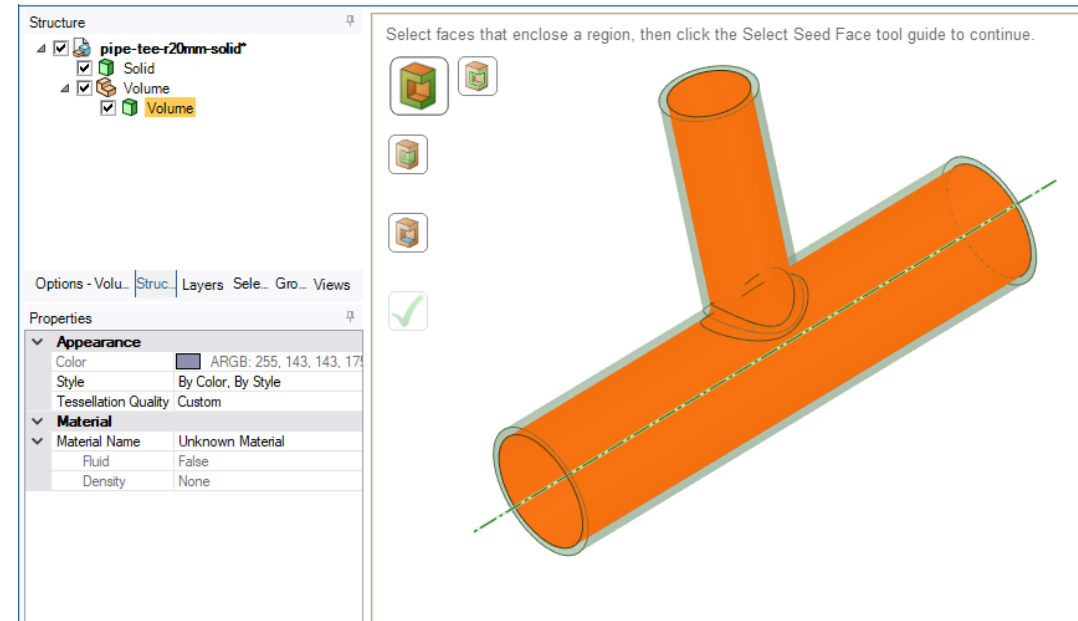


The fluid volume is created successfully.

# Suppress Solid for Physics and Send to Fluent

- Right click the solid body in the tree and select Suppress for Physics

- In the Workbench tab, click Fluent and choose Watertight Geometry Workflow

- Click Start



**1.**

**2.**

**3.**
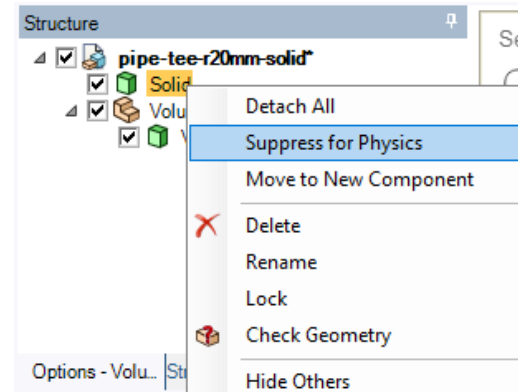
The symbol across the part icon indicates suppressed bodies.
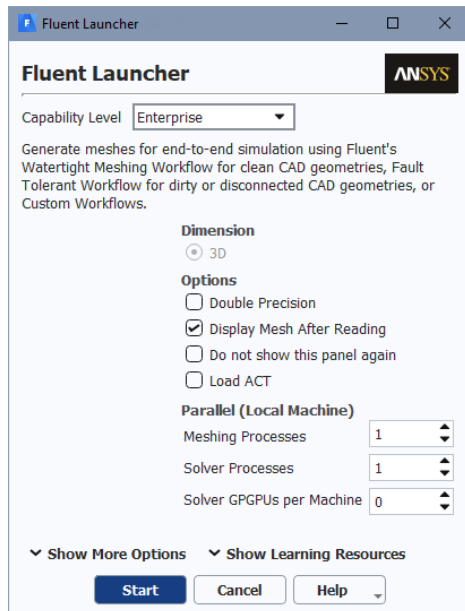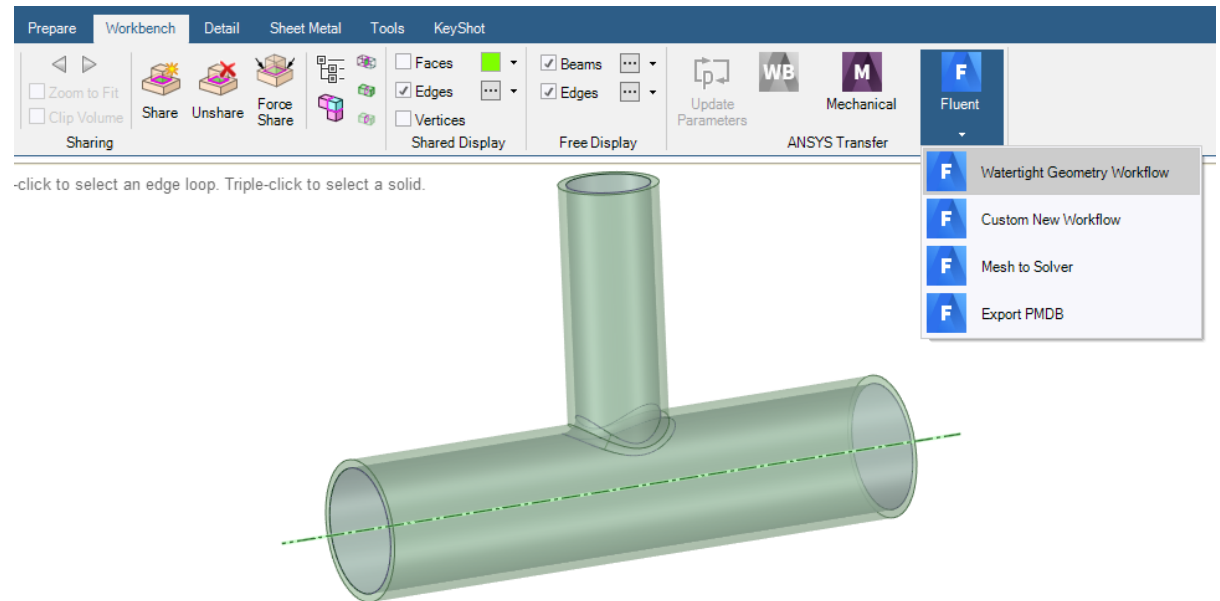
# Import Geometry and Add Local Sizing

- If necessary, geometry units can be changed, but here they should be left as mm

- Keep "Would you like to add local sizing?" as no

- Right click on the Add Local Sizing task and select Update

# Generate the Surface Mesh

- Keep default entries for the Generate the Surface Mesh task and create the mesh
  - Note the maximum skewness value reported in the console window … values below 0.7 are acceptable



**Hint: Insert a clipping plane to help with visualizing the mesh.**
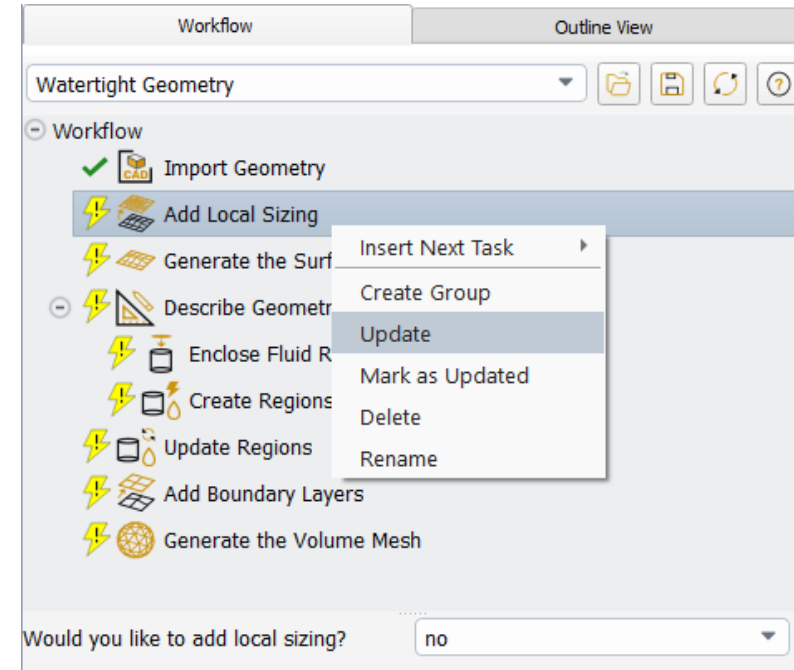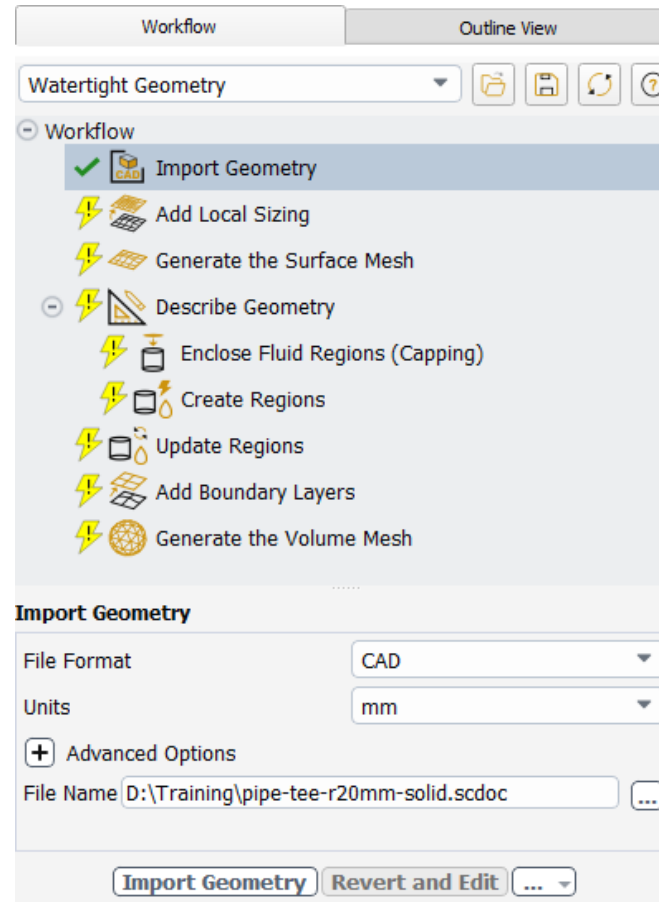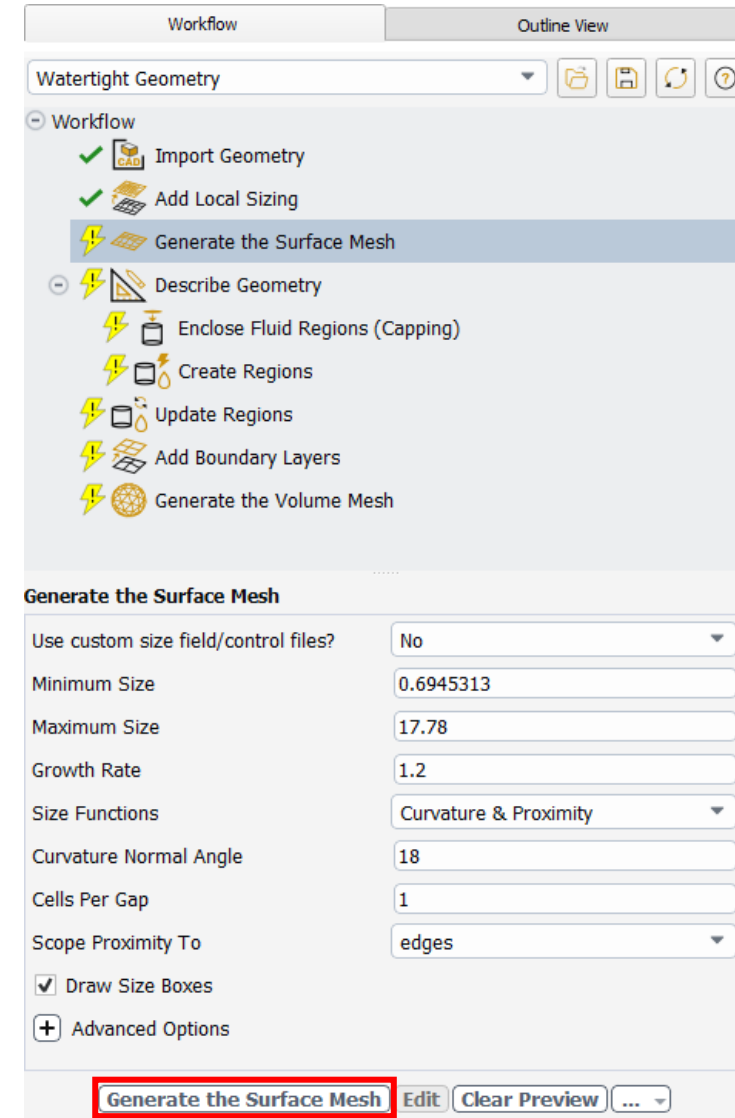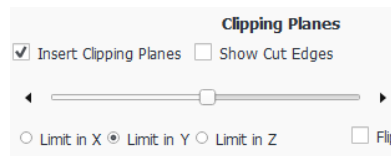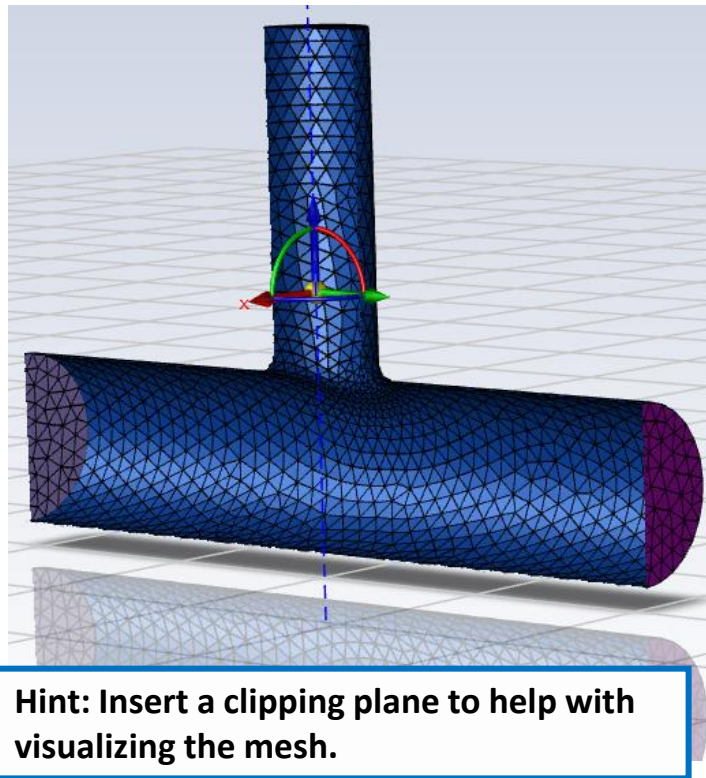
# Describe Geometry

- The geometry consists of only fluid regions with no voids

- Click Describe Geometry and in the next step you will update the boundaries
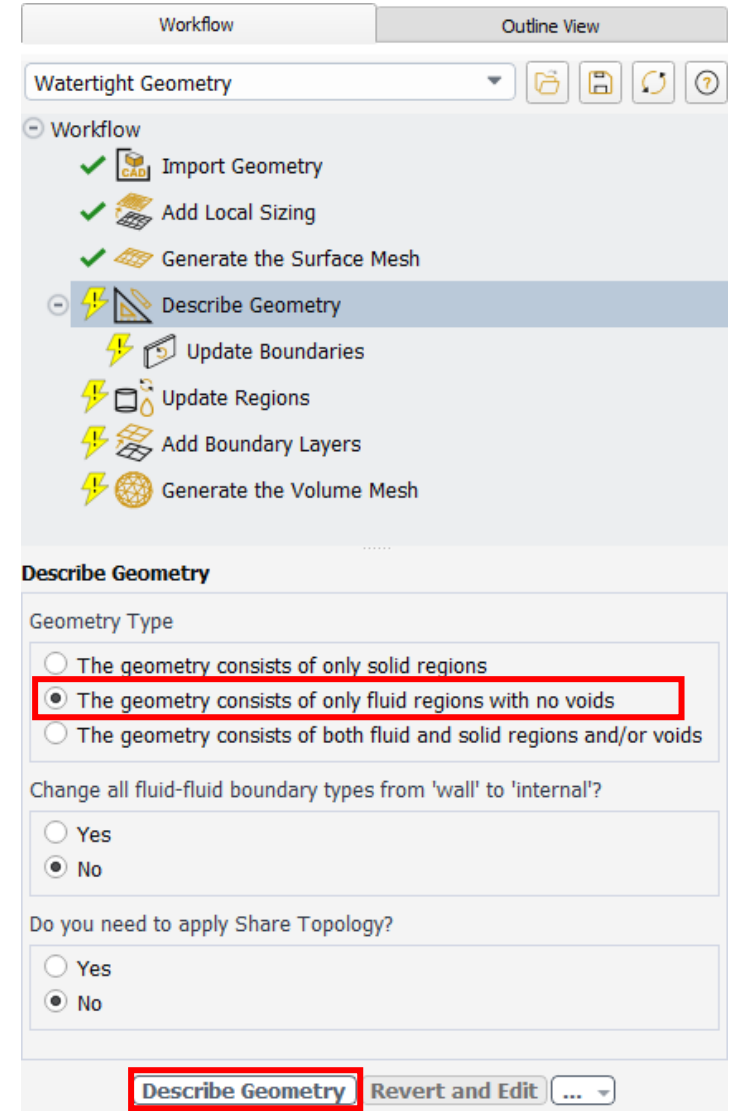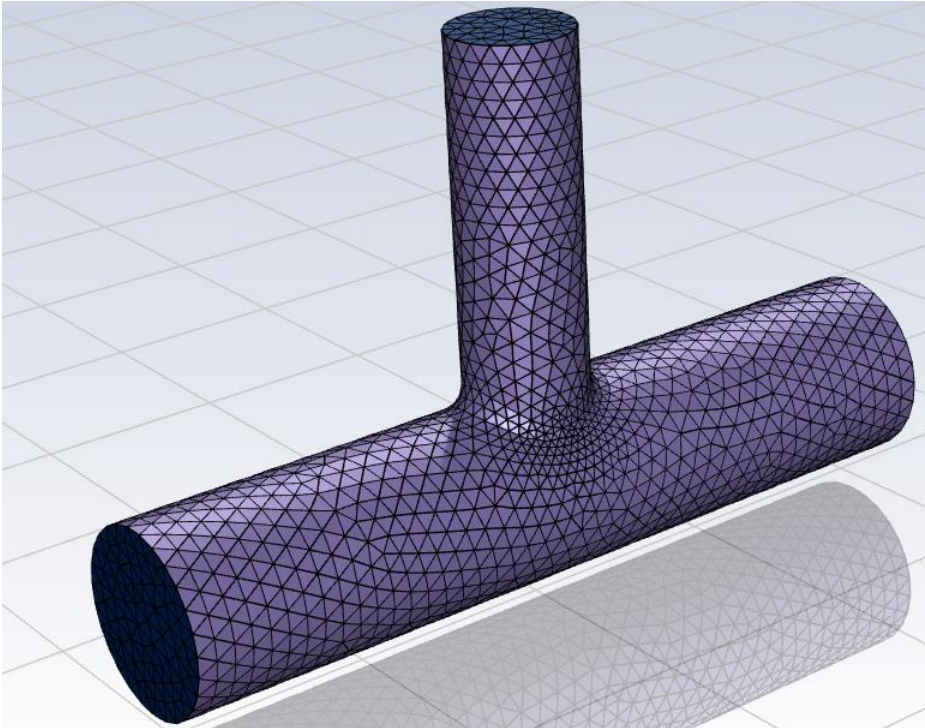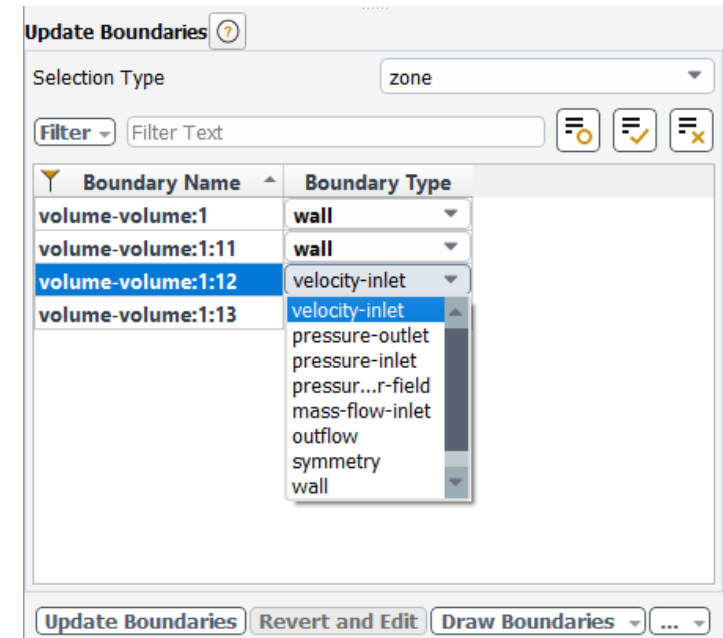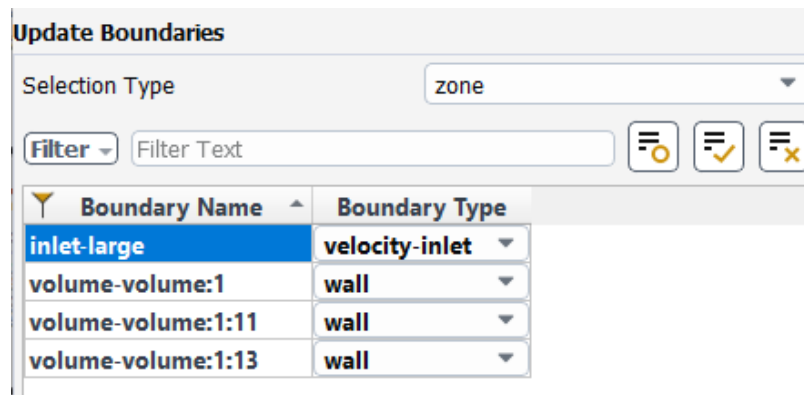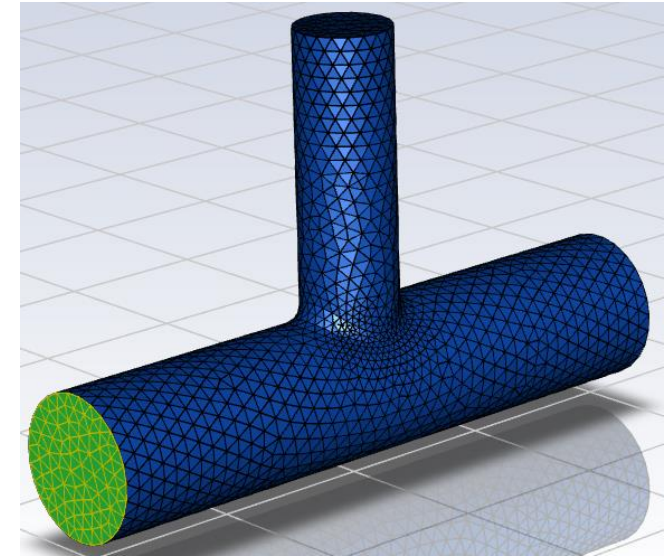
# Updating Boundaries

- The geometry file did not have any named selections, so the surfaces appear in the panel but with non-descriptive, automatically generated names

- In such a case, it is easier to select surfaces directly in the graphics window, as shown in the next slide
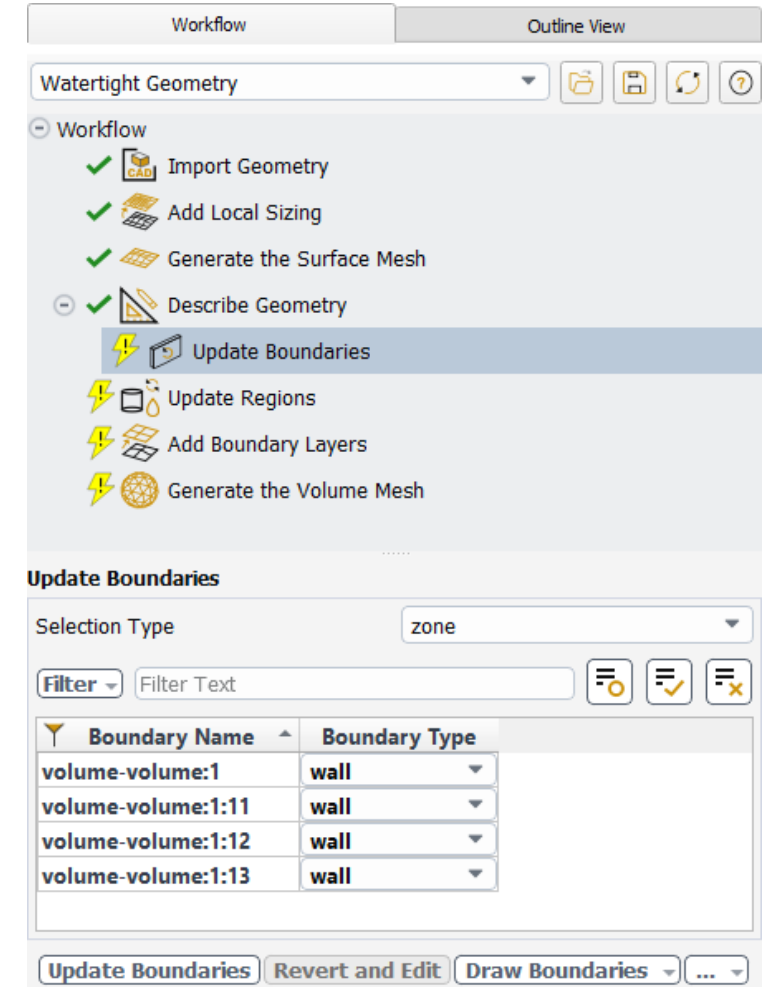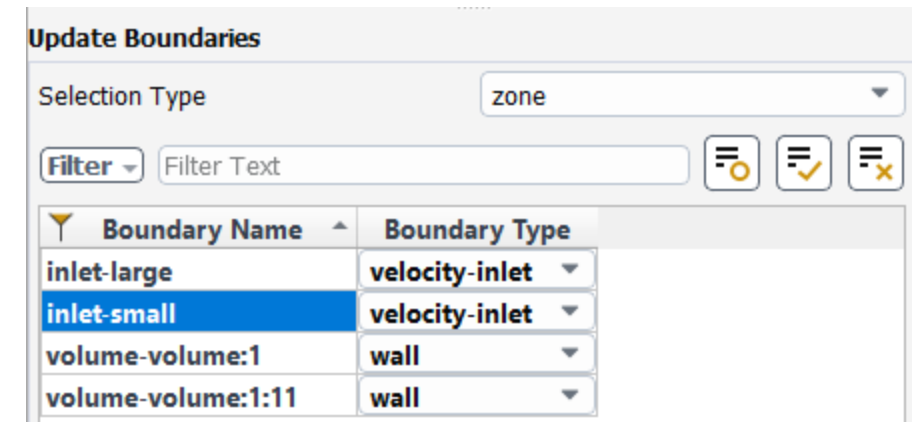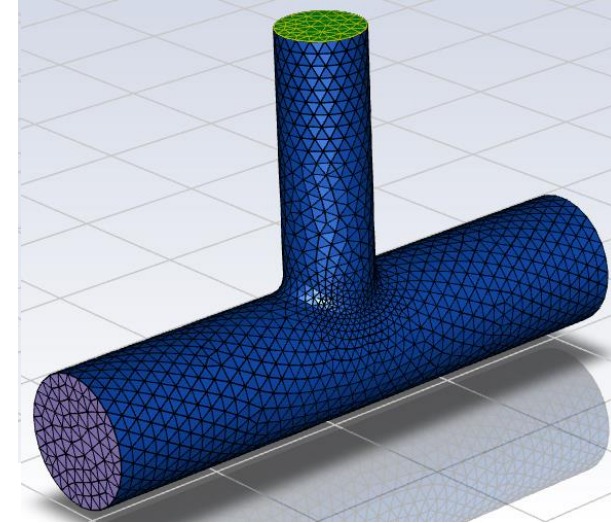
# Update Boundaries: Large Inlet



- Right click on the inlet surface
  - It will be highlighted in green
- On selection, the zone will also become selected in the zone list in the panel
- First, change the Boundary Type to "velocity-inlet"
  - The initial boundary name, here "volume-volume:1:12", might be different on your system so be sure to use whatever boundary turns blue in the panel when the inlet is selected in the graphics window, even if it has a different name
- Second, change the name to "inlet-large"
  - Do this second because as soon as the name changes the list automatically reorders itself
- Do not click Update Boundaries yet – wait until all the boundaries have been defined

# Update Boundaries: Small Inlet

- Click the F2 key to unselect the large inlet
- Right click on the small inlet surface
  - It will be highlighted in green
- On selection, the zone will also become selected in the zone list in the panel
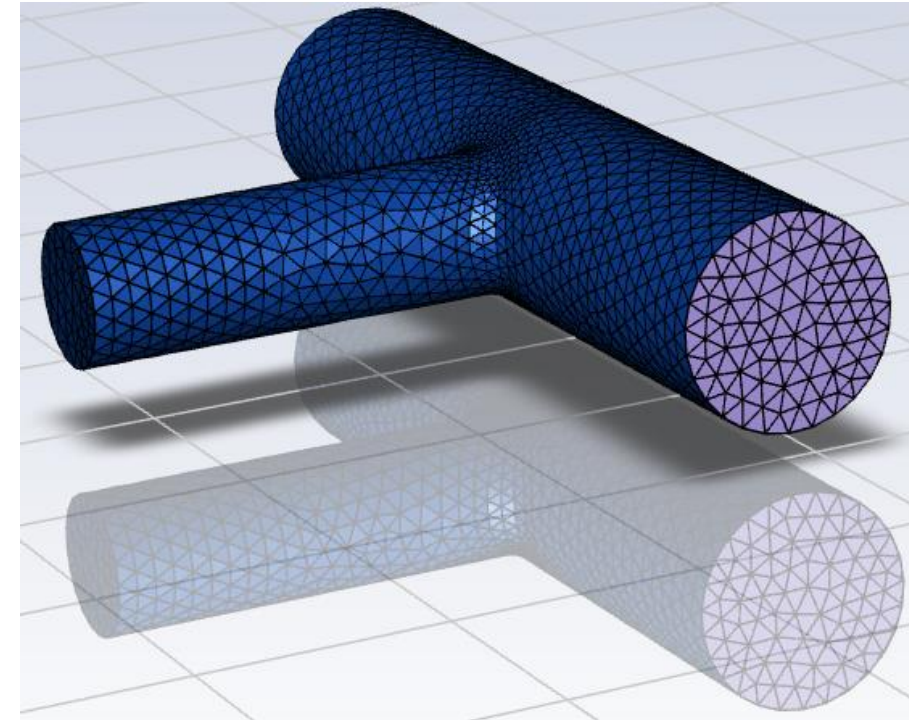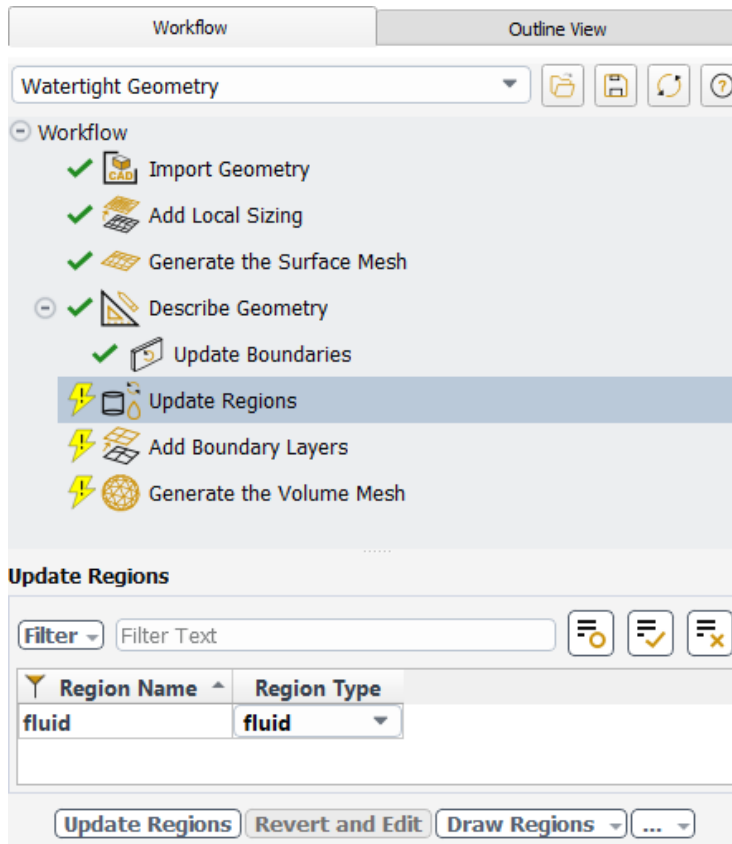- As before, change its type to velocity-inlet and then its name to "inlet-small"

# Update Boundaries: Outlet and Wall

- Click the F2 key to unselect the small inlet

- Add a clipping plane in y to more easily select the outlet

- In the panel, first change its type to pressure-outlet, then change its name to "outlet"

- In the panel, change the name of the other boundary to "wall" and click Update Boundaries

  - Changing the name of the wall is not strictly necessary but in solution mode it will be nicer if it has a descriptive name
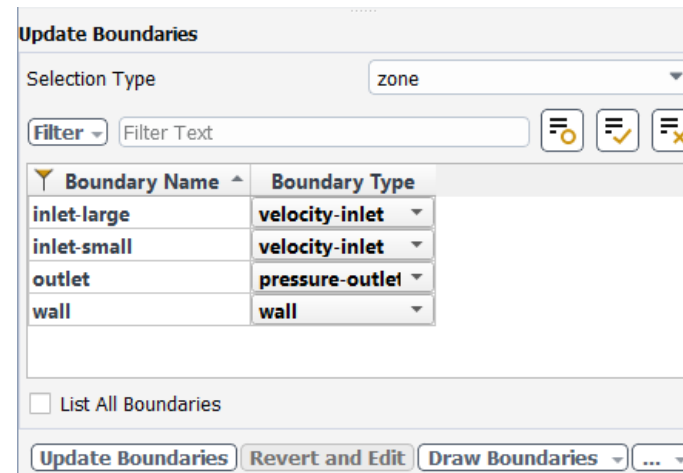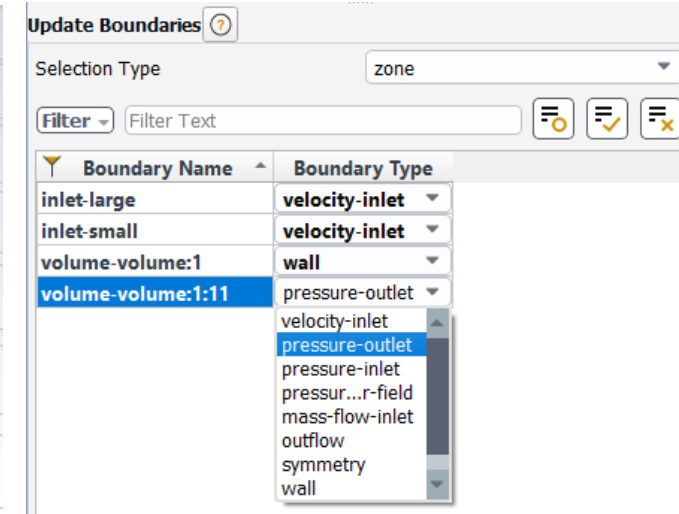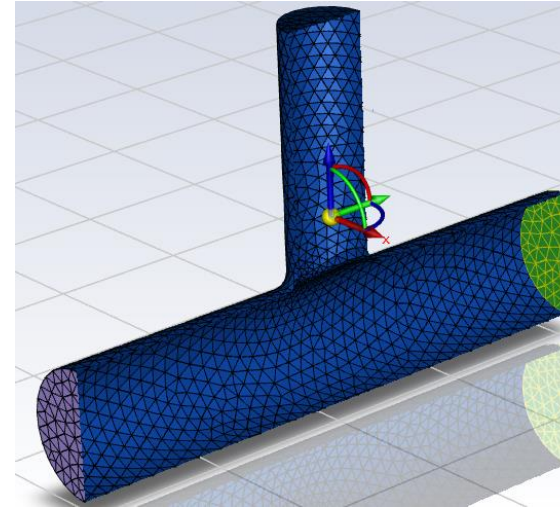
# Update Regions

- In Update Regions, change the name of the fluid region to "fluid" by clicking on the corresponding Region Name

- Click Update Regions to complete the task

# Add Boundary Layers and Generate the Volume Mesh

- Complete the Add Boundary Layers and Generate the Volume Mesh tasks using the default settings



**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

- At this point, write the mesh and switch to solution mode, as on slide 17

- The solution mode instructions are identical to those in the main part of the workshop, beginning with slide 18

/Ansys

**End of presentation**