

Ansys Fluent Getting Started (New Fluent Experience)

Workshop: Addition of Turning Vanes to a Rectangular Duct

Release 2021 R1




Capability Level

- This tutorial is supported by all licensing capability levels

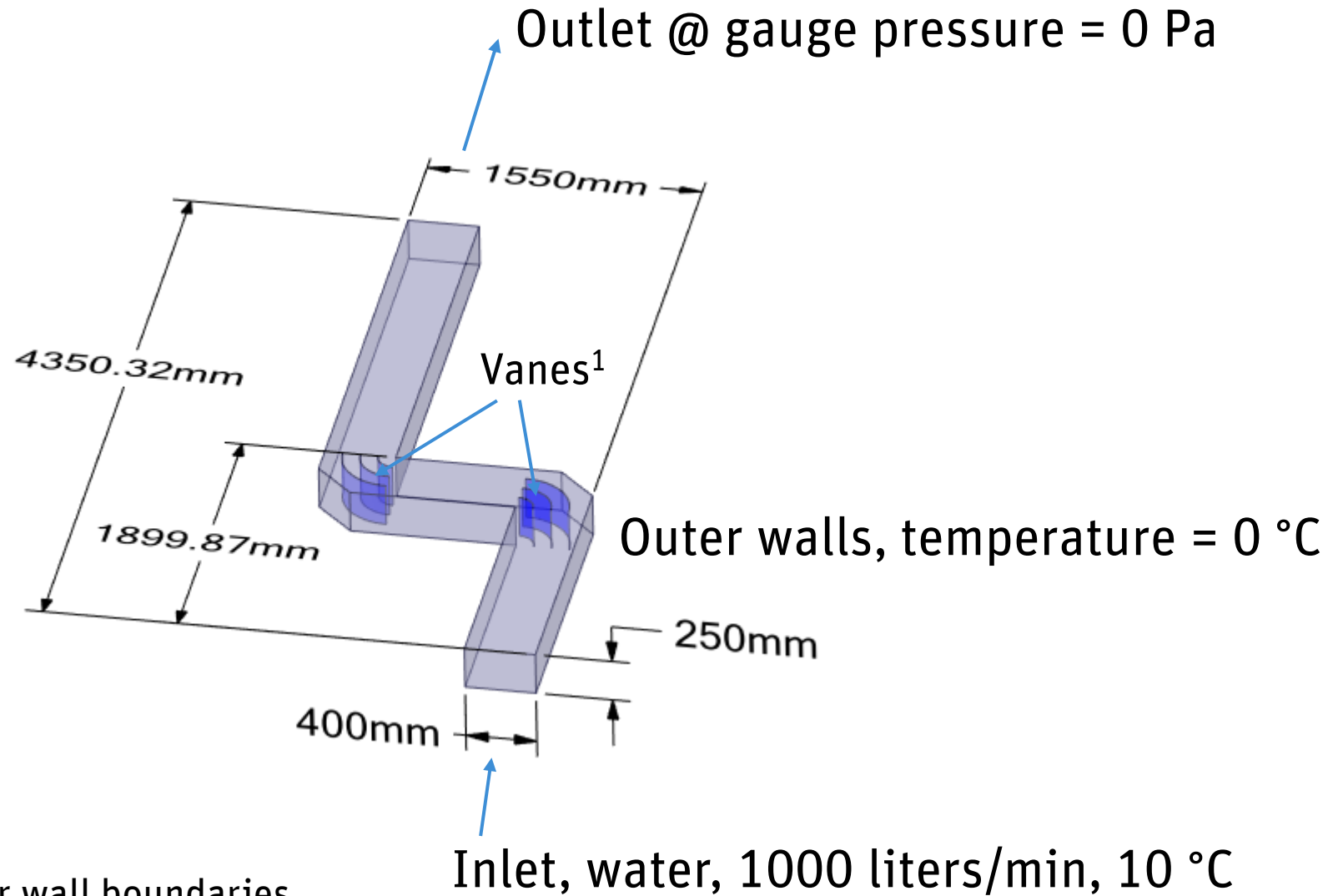
/ Problem Description

- Water flows through a rectangular duct (cross section 250 mm x 400 mm) with two 90° turns
- It is proposed to add turning vanes in each of the turns
- The duct will be simulated both with and without vanes to assess their effect on flow and pressure drop
 - This can be done with a single CAD and single mesh file by changing the boundary condition type of the vanes back and forth between an interior boundary and a wall boundary
 - Flow passes freely through interior boundaries as if there is nothing there
- Additionally, the wall of the duct will be exposed to a cold environment and the simulation will be used to assess whether insulation should be added to limit temperature change in the water as it passes through the duct
- The goal of the simulation is to compare the pressure drop in the duct without vanes to the pressure drop with vanes and to quantify the change in water temperature



Identify the key simulation outcomes at the beginning. You can use these to monitor progress of solution.

Geometry and Operating Conditions for Duct

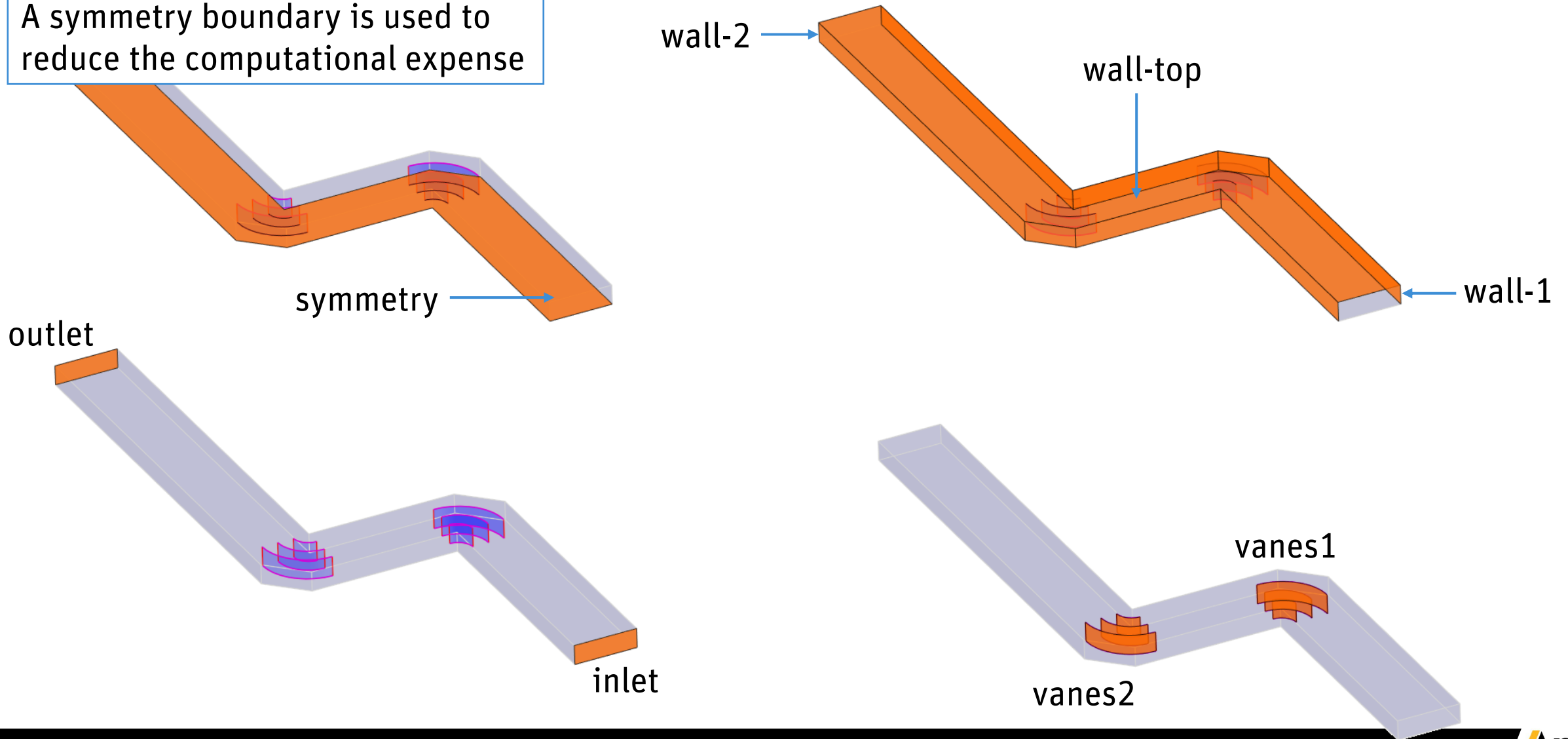


1. Vanes will be either interior or wall boundaries

Inlet, water, 1000 liters/min, 10 °C

Named Selections in Duct

A symmetry boundary is used to reduce the computational expense



Learning Outcomes

Learning Aims:

The emphasis of this workshop is to extend the skills learned in the previous workshops:

- Defining physical models such as energy
- Defining material properties
- Cell zone material assignment
- Changing boundary types

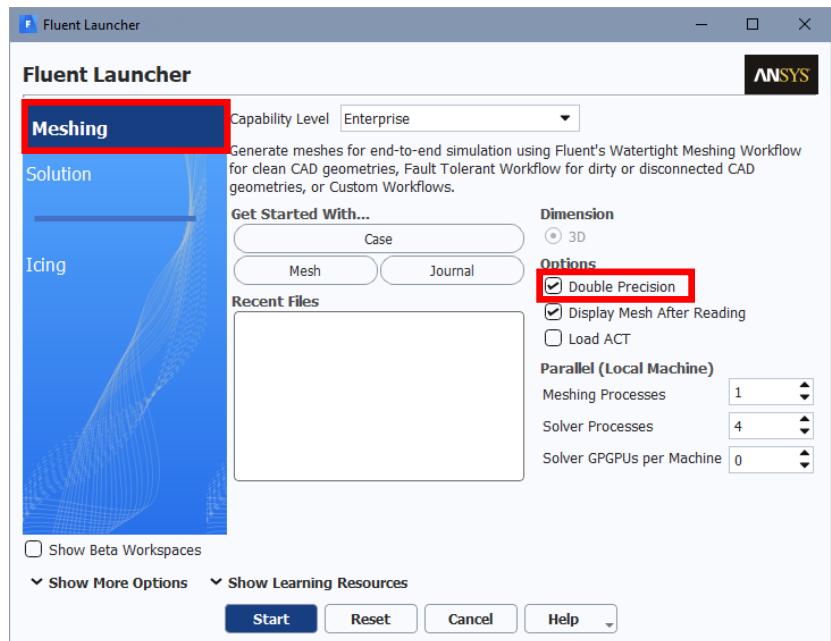
Learning Objectives:

To learn how to activate physical models and define material properties and become familiar with integrating these functions into the ribbon based workflow

Starting Fluent

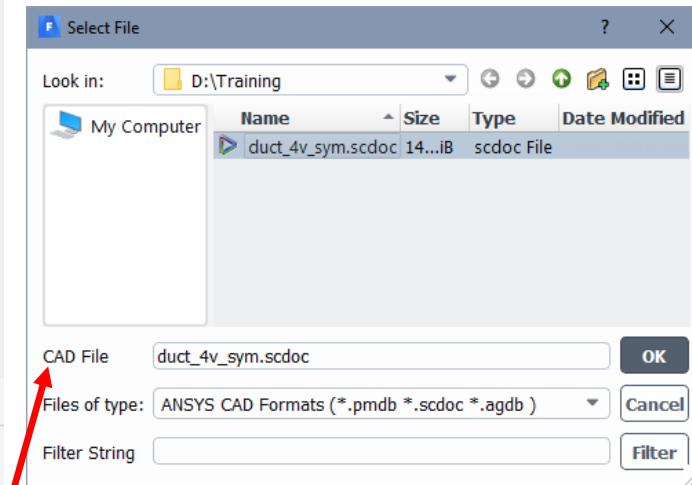
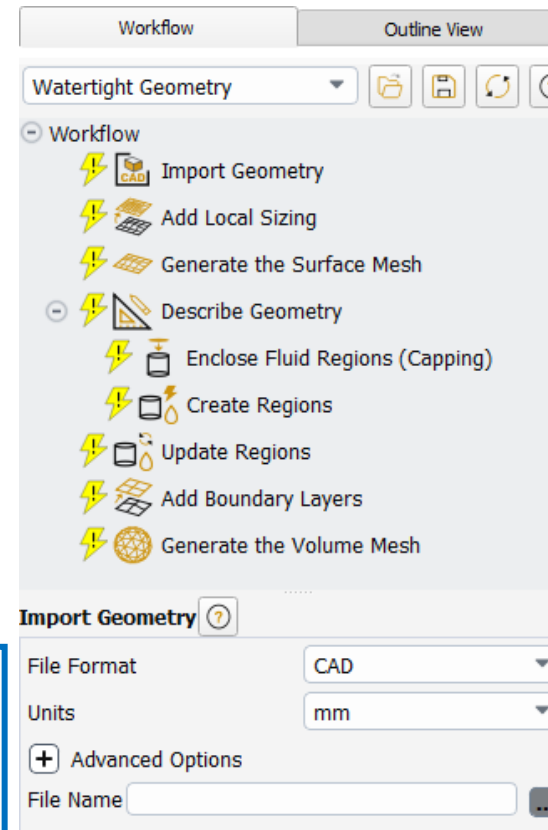
- Open the Fluent Launcher Window and ensure Meshing Mode is selected
 - If necessary, expand "Show More Options" to change the working directory

- Select the Watertight Geometry workflow, set Units to mm and import "duct_4v_sym.scdoc"



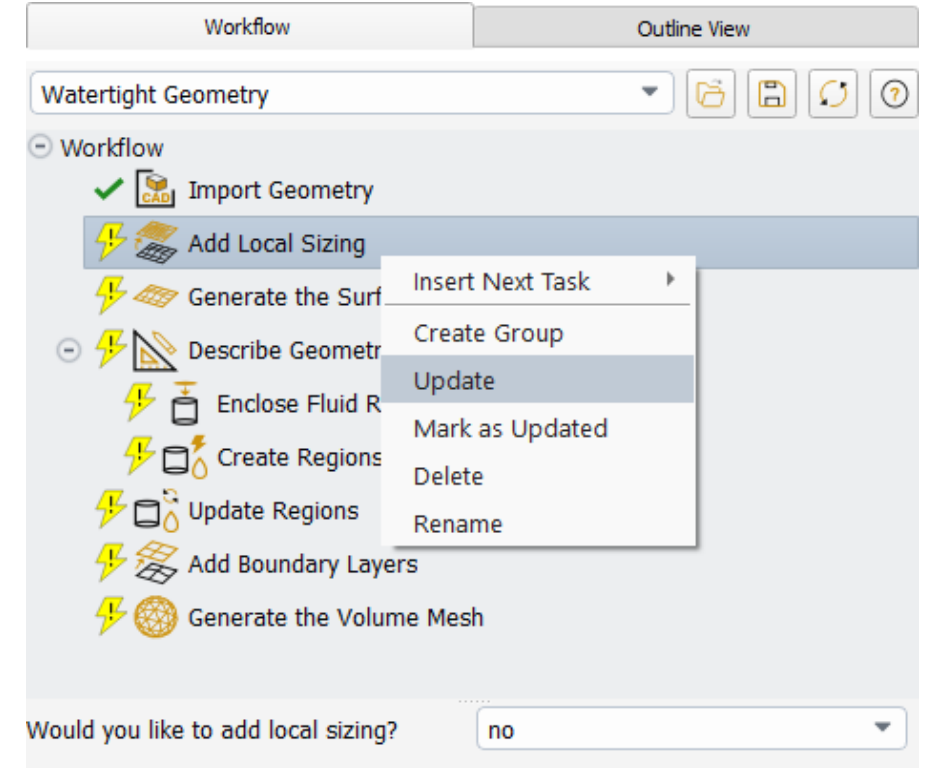
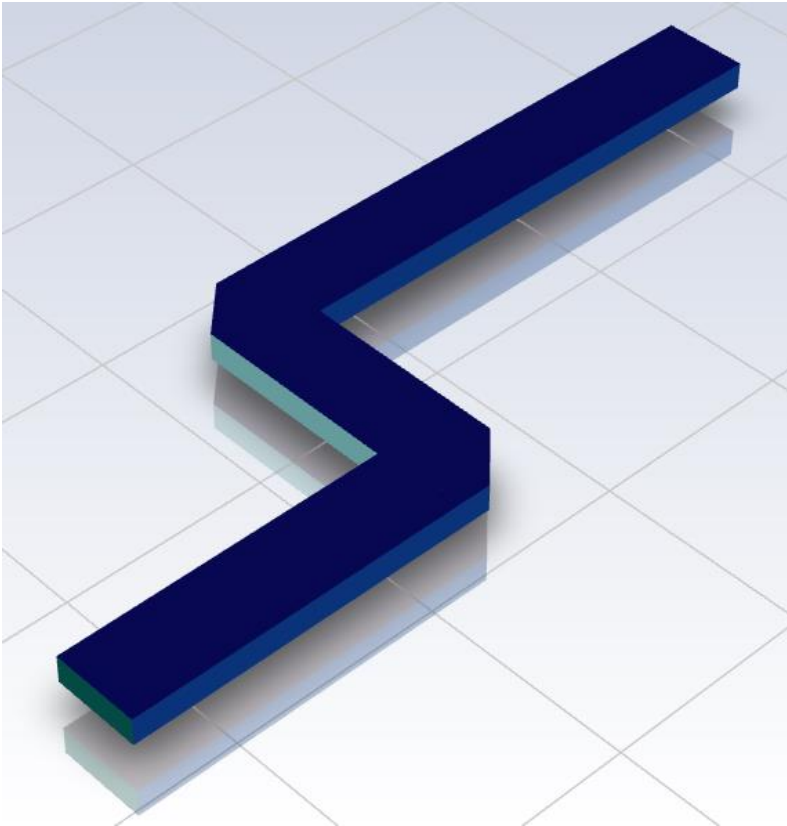
Double Precision often improves convergence for problems with heat transfer.

4 Solver Processes recommended for this workshop but you may need to use fewer based on availability and license status.



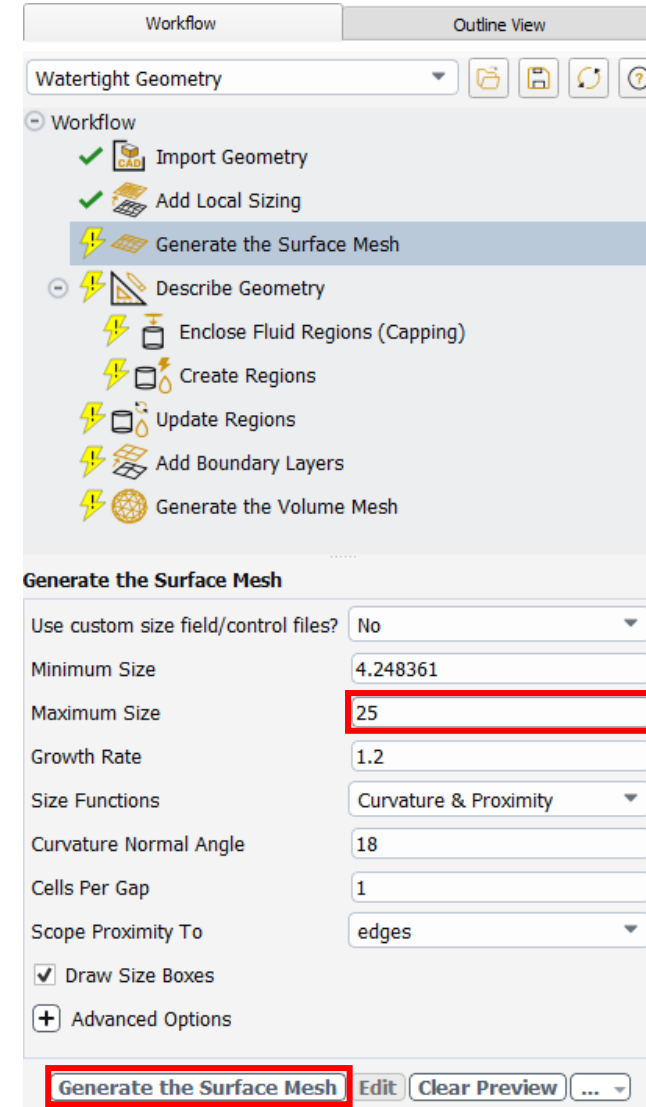
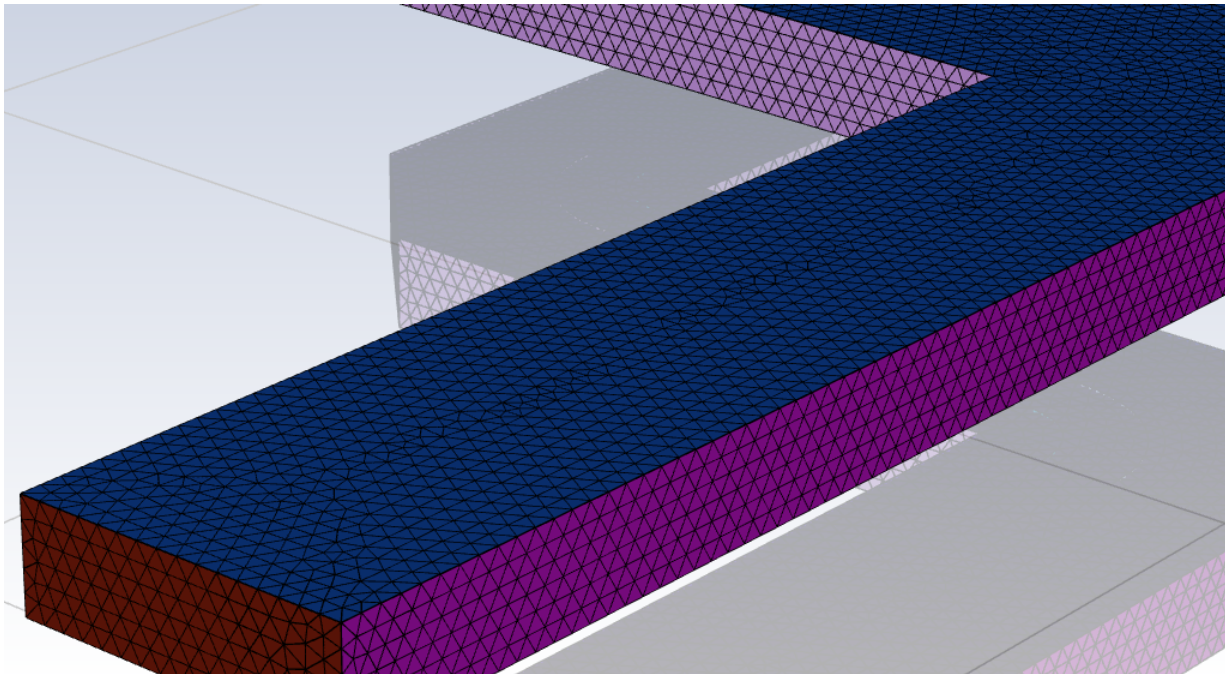
/ Add Local Sizing

- Keep "Would you like to add local sizing?" as no
- Right click on the Add Local Sizing task and select Update



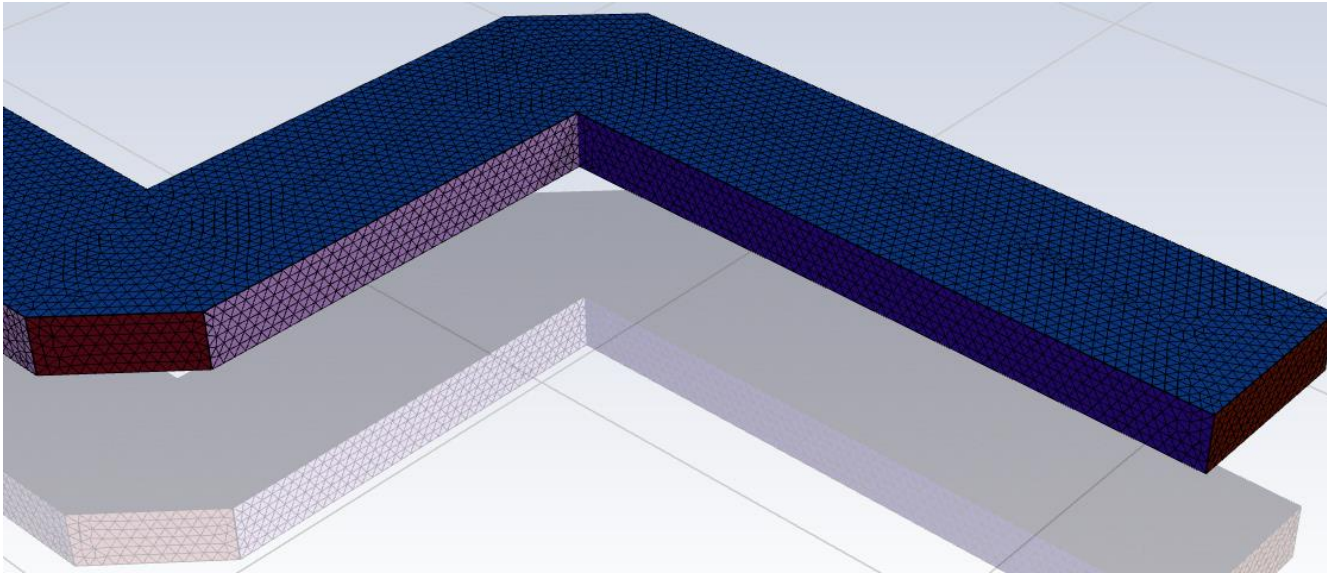
/ Generate the Surface Mesh

- Change Maximum Size to 25 mm and generate the surface mesh
 - The duct cross section is 250 mm x 400 mm so 25 mm is 10% of the smallest dimension
 - Note the maximum skewness value reported in the console window ... values below 0.7 are acceptable



/ Describe Geometry

- The geometry consists of only fluid regions with no voids
- Select Yes under Change all fluid-fluid boundary types from wall to internal
 - The simulation will begin with the vanes defined as 'internal', which is the same as if no vanes are present
 - After the initial solution has been calculated, they will be changed to walls and a second solution will be calculated



Workflow Outline View

Watertight Geometry

Workflow

- ✓ Import Geometry
- ✓ Add Local Sizing
- ✓ Generate the Surface Mesh
- ⊖ Describe Geometry
- ⚡ Update Boundaries
- ⚡ Update Regions
- ⚡ Add Boundary Layers
- ⚡ Generate the Volume Mesh

Describe Geometry

Geometry Type

- ☐ The geometry consists of only solid regions
- ☒ The geometry consists of only fluid regions with no voids
- ☐ The geometry consists of both fluid and solid regions and/or voids

Change all fluid-fluid boundary types from 'wall' to 'internal'?

- ☒ Yes
- ☐ No

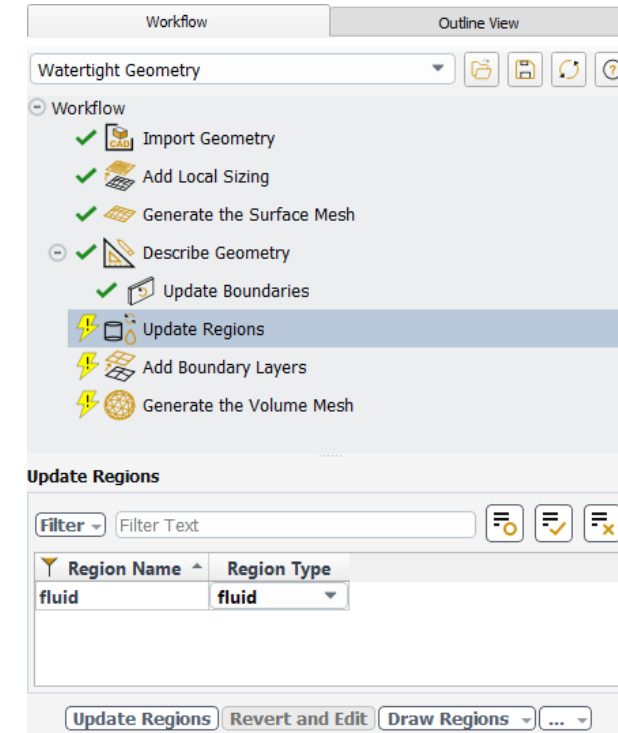
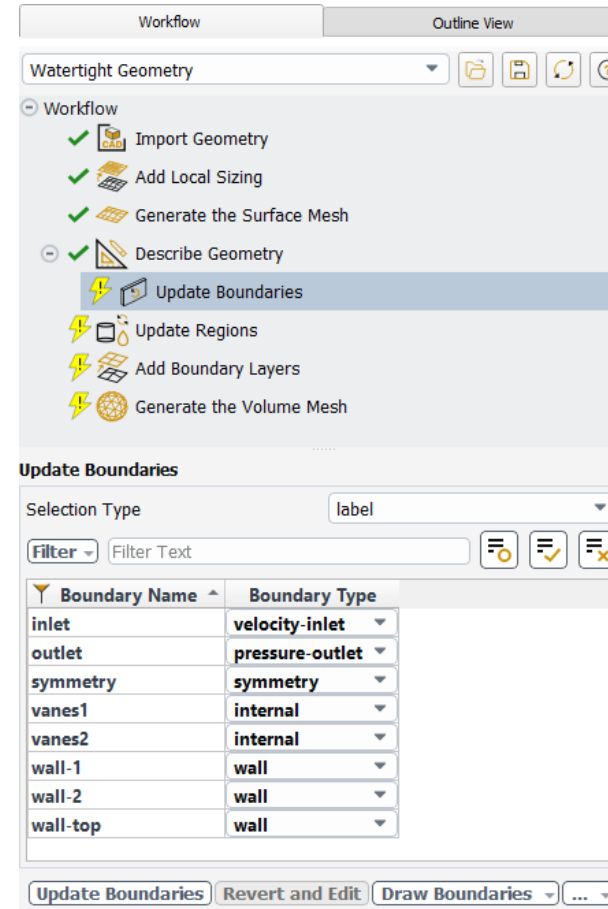
Do you need to apply Share Topology?

- ☐ Yes
- ☒ No

Describe Geometry Revert and Edit ...

/ Update Boundaries and Update Regions

- Review the boundary assignments in the Update Boundaries task and click Update Boundaries
- Review the region assignment in the Update Regions task and click Update Regions



Add Boundary Layers

- In the Add Boundary Layers task, enter 5 for Number of Layers** and click Add Boundary Layers
- In the Add Boundary Layers task, enter 5 for Number of Layers, choose to grow on selected-labels select vanes1 and vanes2, and click Add Boundary Layers
 - Although the vanes will start off as interior zones, boundary layers will be added here so that it is not necessary to remesh when they are later changed to walls in solution mode

Would you like to add boundary layers?

Add Boundary Layers

Name:

Offset Method Type:

Number of Layers:

Transition Ratio:

Growth Rate:

Add in:

Grow on:

Would you like to add boundary layers?

Add Boundary Layers

Name:

Offset Method Type:

Number of Layers:

Transition Ratio:

Growth Rate:

Add in:

Grow on:

[2/8]

** The default value of 3 is intended to reduce the possibility of skewed meshes in the case of poor surface geometry or very small gaps. In clean geometry such as the duct, this can be increased to 5 for improved boundary layer resolution. In many aerodynamics applications where high fidelity resolution of the boundary layer is required, as many as 20-30 layers or more might be used.

/ Generate the Volume Mesh

- In the Generate the Volume Mesh task, select poly-hexcore and generate the mesh

Generate the Volume Mesh

Fill With: poly-hexcore

Buffer Layers: 2

Peel Layers: 1

Min Cell Length: 4.248361

Max Cell Length: 67.97378

☐ Region-based Sizing

☒ Enable Parallel Meshing

+ Advanced Options

+ Global Boundary Layer Settings

Generate the Volume Mesh Revert and Edit Clear Preview Draw Mesh

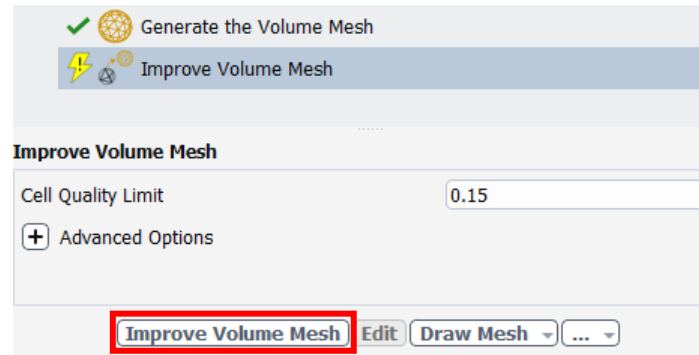
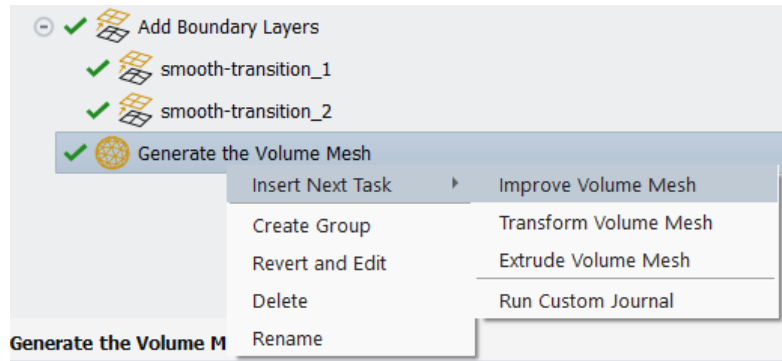
The orthogonal quality reported in the console window will vary slightly depending on how many meshing processes were used. If the value reported upon completion of the volume mesh is less than 0.10, perform the instructions on the next slide. If the value is greater than 0.1, move forward two slides.

Console

```
----- The mesh has a minimum Orthogonal Quality of: 0.05
----- The volume meshing of duct_4v_sym is complete.
```

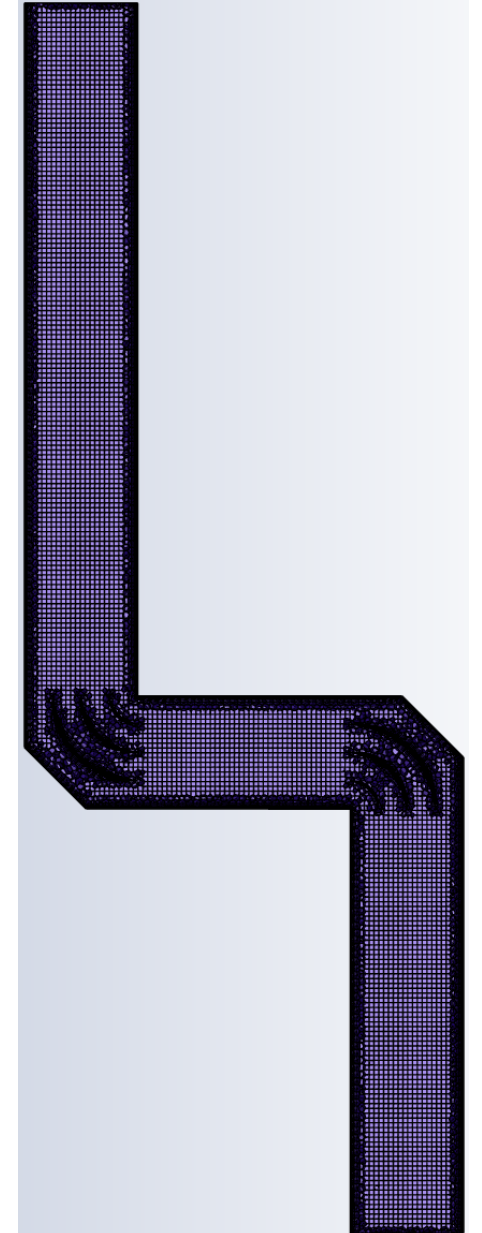
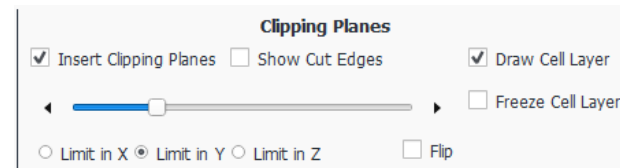

Improve Volume Mesh

- Right click Generate the Volume Mesh and insert an Improve Volume Mesh task
- Leave the default entry for the Improve Volume Mesh task and click Improve Volume Mesh



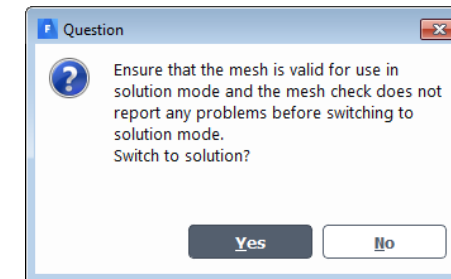
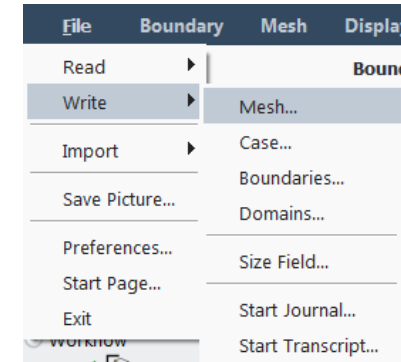
In the console window it will be reported that the final minimum orthogonal quality is 0.15

Clipping Planes settings used in graphic to the right. Additionally, it may be necessary to untick "All Faces" in Display



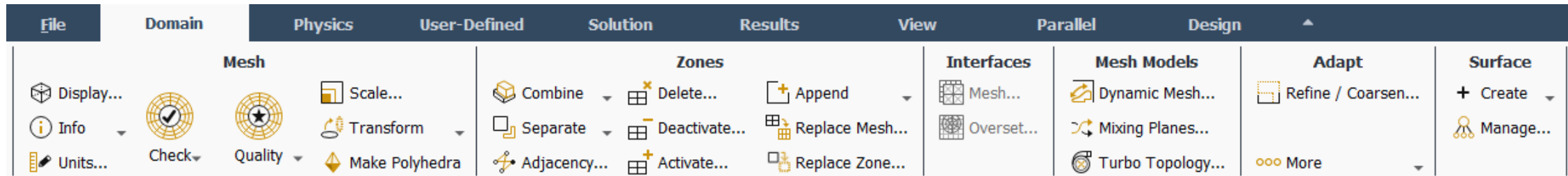
/ Write Mesh and Switch to Solution Mode

- Go to File > Write > Mesh and save the mesh as "duct-volume-mesh.msh.gz"
 - 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

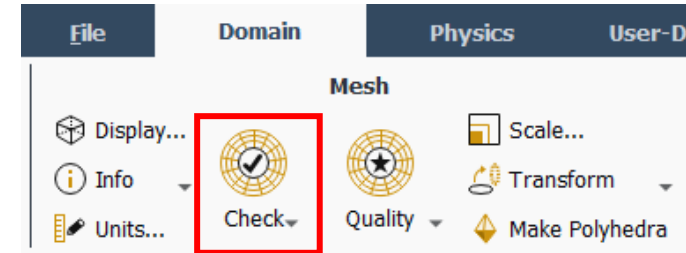


- The four primary tabs used in every simulation are
 - Domain
 - Physics
 - Solution
 - Results
- For this case (and probably most other cases too), you will use them going in order from left to right

Domain: Mesh Check

- In the Domain tab, click on **Check** and **Perform Mesh Check**

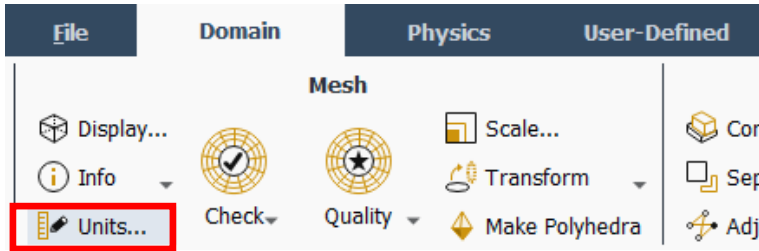
```
Console
>
Domain Extents:
  x-coordinate: min (m) = -1.550000e+00, max (m) = 6.223999e-18
  y-coordinate: min (m) = 1.250000e-01, max (m) = 2.500000e-01
  z-coordinate: min (m) = -4.350322e+00, max (m) = 8.805246e-16
Volume statistics:
  minimum volume (m3): 4.188825e-09
  maximum volume (m3): 1.451233e-05
  total volume (m3): 2.690981e-01
Face area statistics:
  minimum face area (m2): 1.202313e-06
  maximum face area (m2): 8.192069e-04
Checking mesh.....
Done.
```



The mesh check returns no error messages.

Domain: Units

- Click **Units** to open the **Set Units** panel

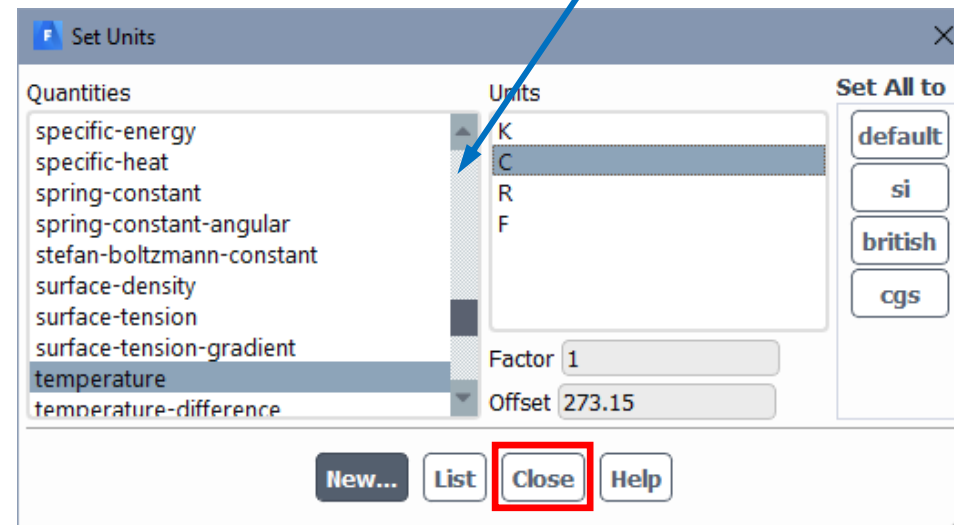


- Select **temperature** on the left, set the units to Celsius (C) and close the panel

Fluent always uses the SI unit system in the background, but you can enter values and postprocess results using whatever units are most convenient.

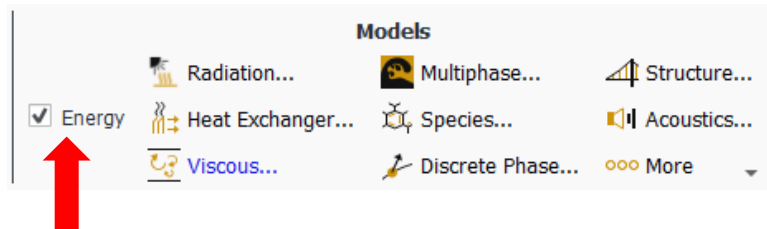
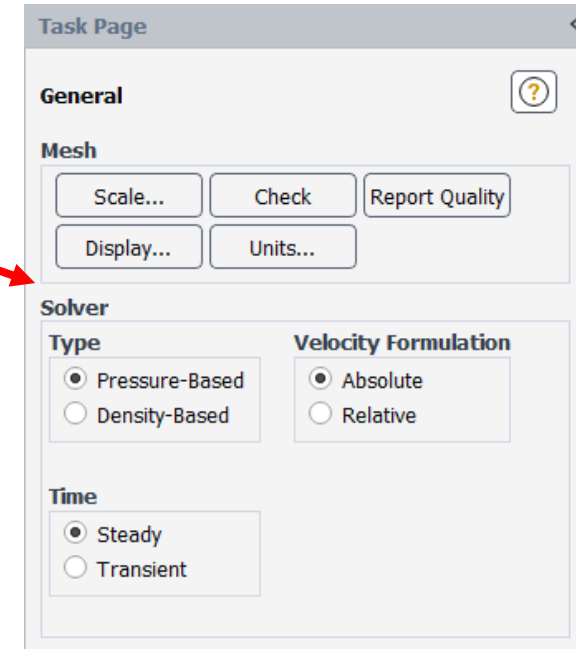
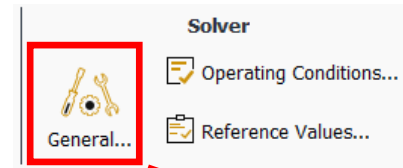
Units can be changed at any time in a Fluent session, even after the solution has been calculated.

Tip: click the mouse anywhere on the slider bar and type 't' on your keyboard to go directly to list entries beginning with 't'. This works on many different panels in Fluent

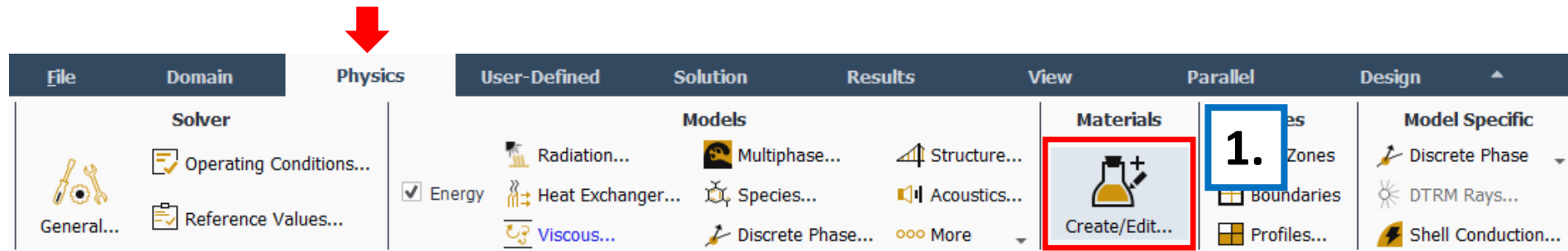


Physics: Solver and Models

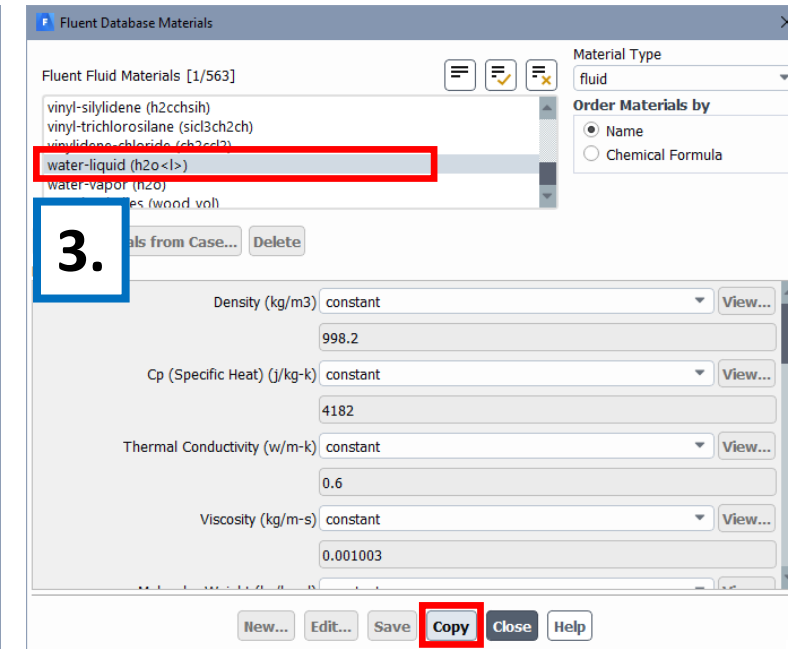
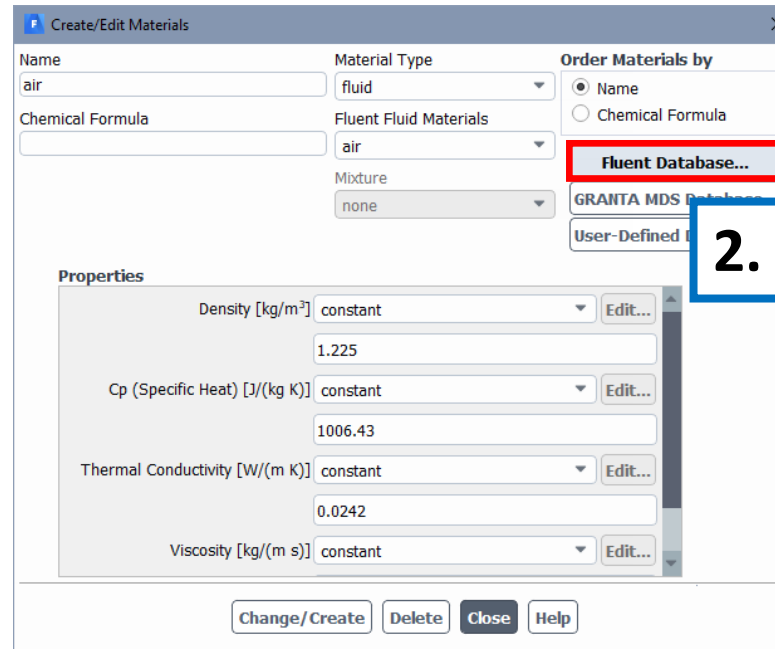
- The default settings in Solver are Time = Steady and Type = Pressure-Based.
 - Inputs for these are found by clicking General
 - Use these settings for this tutorial
- In Models, select Energy
 - This problem involves heat transfer with the duct walls



Physics: Materials



1. Click **Create/Edit** to open the materials panel
2. Air is currently the only fluid material so click **Fluent Database**
3. In the Fluent Database Materials panel select *water-liquid* under the list of materials, click **Copy** and then close the panel

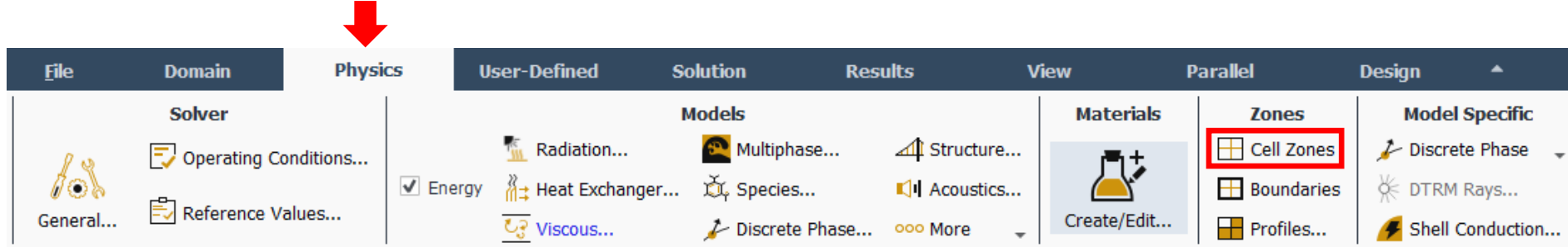


Physics: Edit Materials

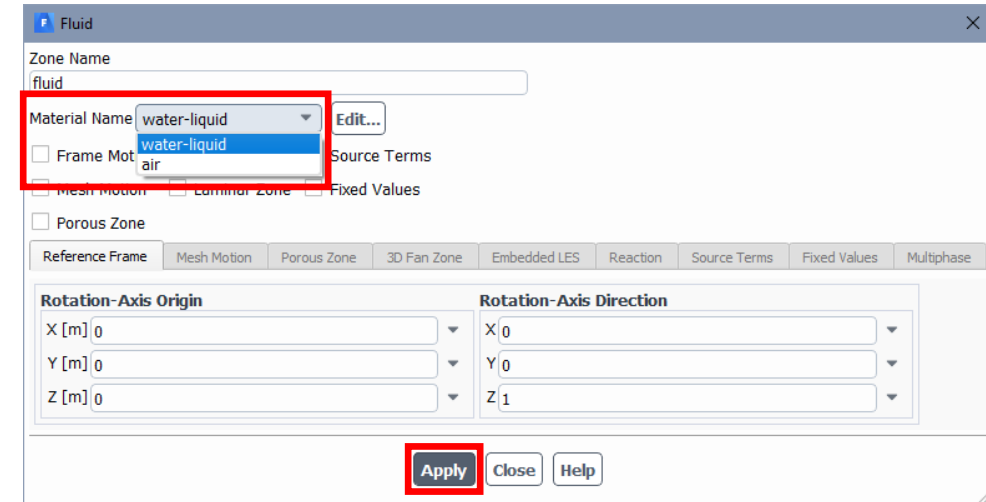
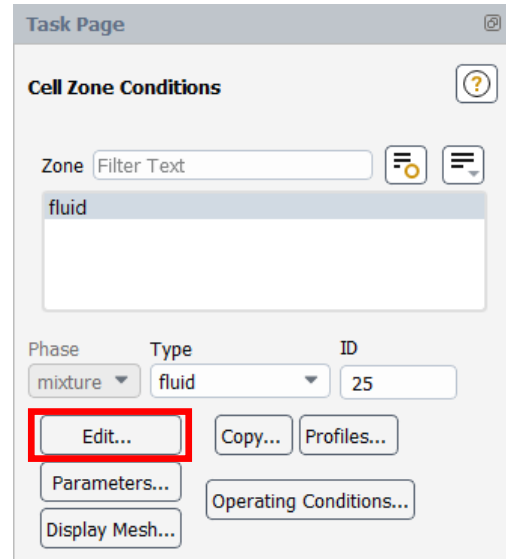
- In the Create/Edit Materials panel, ensure the fluid material is water-liquid
- Change the density to 1000 kg/m³ and change the viscosity to 0.001 kg/m-s
 - Although in general, accurate results require accurate definition of material properties, changes as minor as this are highly unlikely to affect the results, however 1000 kg/m³ may look nicer in a report than 998.2 kg/m³ and similarly, there will be no reason to explain why the extra .000003 kg/m-s is included in the definition of viscosity
- Click Change/Create to update the property values and then close the panel

The screenshot shows the 'Create/Edit Materials' dialog box in ANSYS. The 'Name' field is 'water-liquid', 'Material Type' is 'fluid', and 'Chemical Formula' is 'h2o<I>'. The 'Fluent Fluid Materials' dropdown is set to 'water-liquid (h2o<I>)' and 'Mixture' is 'none'. On the right, 'Order Materials by' is set to 'Name', and there are buttons for 'Fluent Database...', 'GRANTA MDS Database...', and 'User-Defined Database...'. The 'Properties' section lists: Density [kg/m³] as 'constant' with a value of 1000; Cp (Specific Heat) [J/(kg K)] as 'constant' with a value of 4182; Thermal Conductivity [W/(m K)] as 'constant' with a value of 0.6; and Viscosity [kg/(m s)] as 'constant' with a value of 0.001. Each property has an 'Edit...' button. At the bottom, the 'Change/Create' button is highlighted with a red rectangle, along with 'Delete', 'Close', and 'Help' buttons.

Physics: Cell Zones

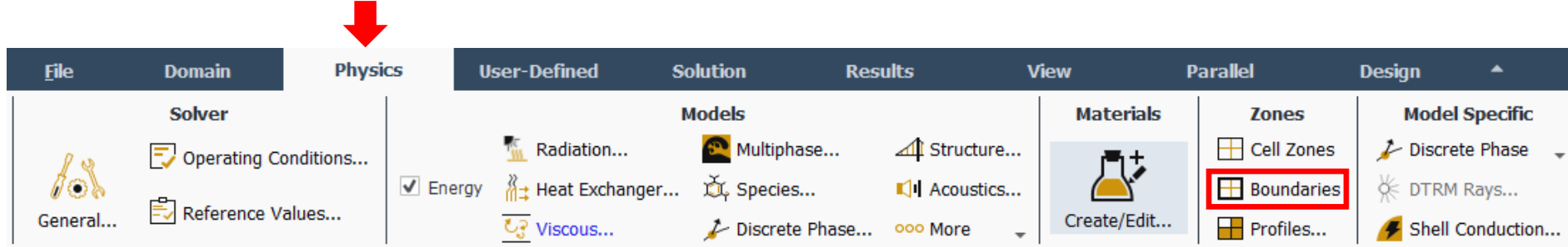


- Click **Cell Zones** to open the Cell Zone Conditions task page
- Select the zone in the task page and click **Edit**
- Change the material from air to water-liquid
 - This is important – unless you do this it will use air to calculate the solution
- Click **Apply** and then **Close**



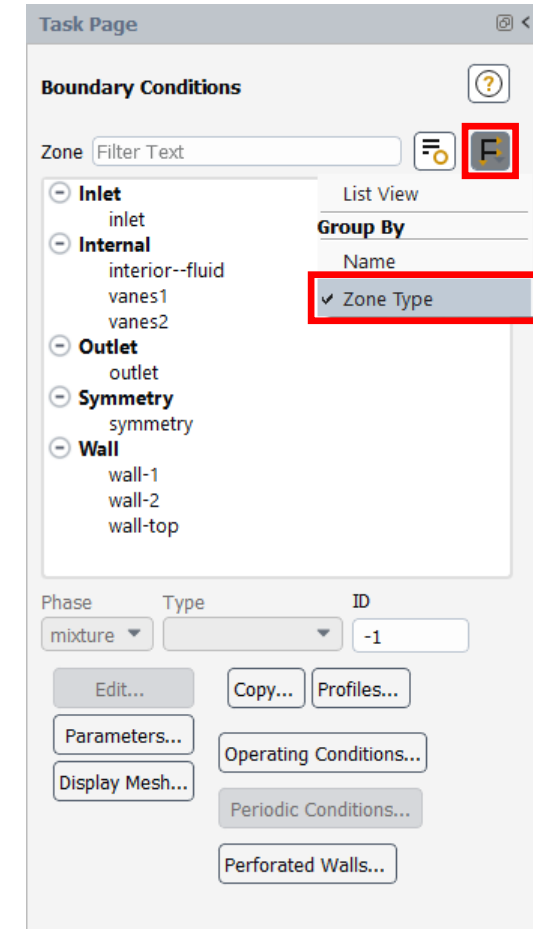
The order of fluids in the materials panel shown in the previous slide is unimportant. The solver always uses the fluid that has been selected in the cell zone panel, so it is important to make the right selection here if your fluid is not air.

Physics: Boundary Conditions



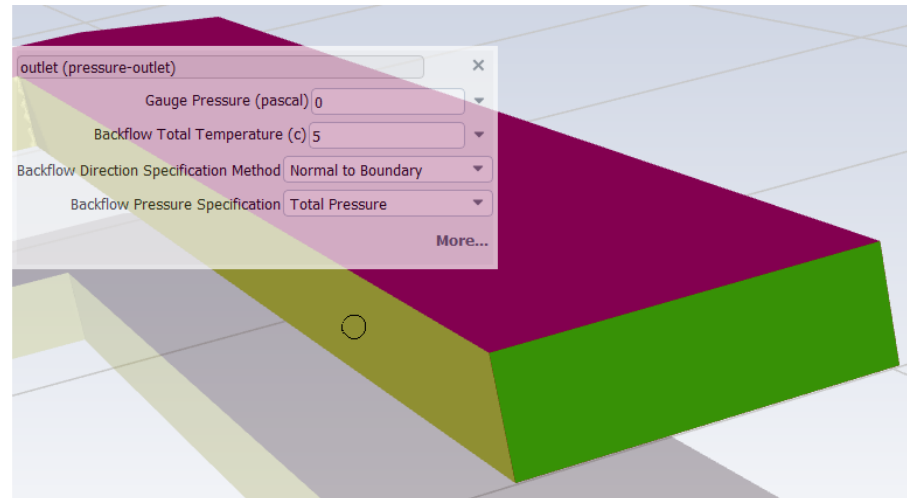
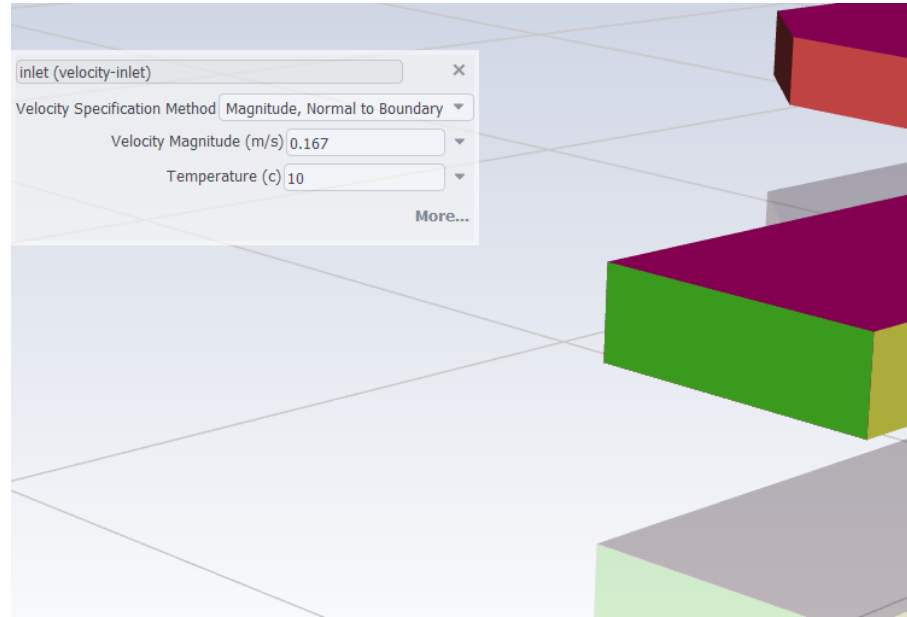
- Click **Boundaries**
 - This opens the Boundary Conditions Task Page
- In the Boundary Conditions task page, display the boundaries by type
 - This opens the Boundary Conditions Task Page
- There are 9 boundary zones in this case
 - No input needed for internal boundaries
 - No input needed for symmetry boundary

inlet	velocity and temperature need to be defined
outlet	gauge pressure = 0, set backflow temperature
wall-1	temperature boundary condition will be defined
wall-2	temperature boundary condition will be defined
wall-top	temperature boundary condition will be defined



Physics: Boundary Condition Assignment with Quick Property Editor

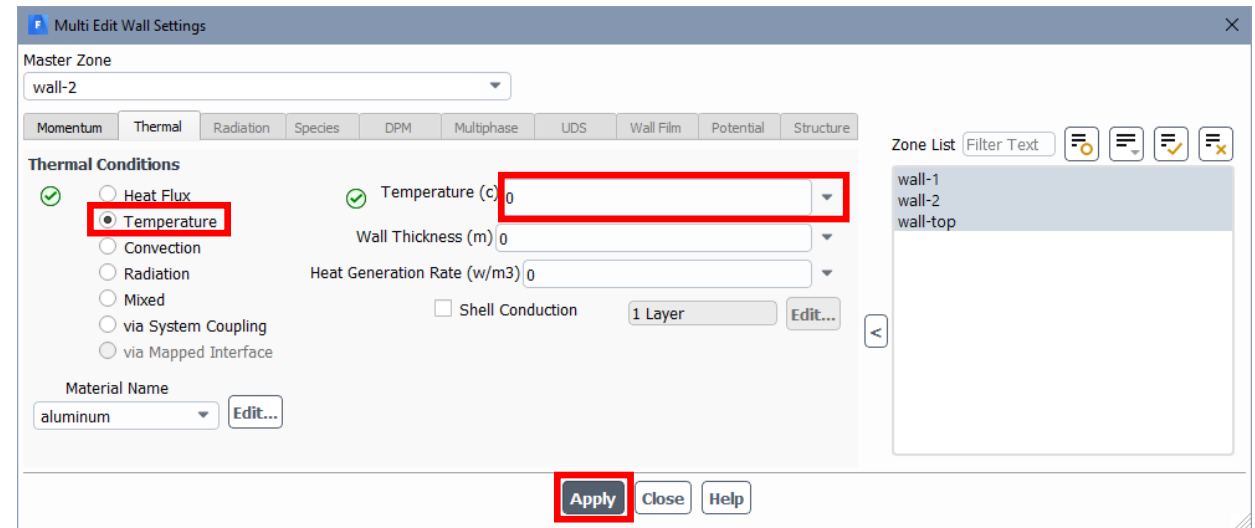
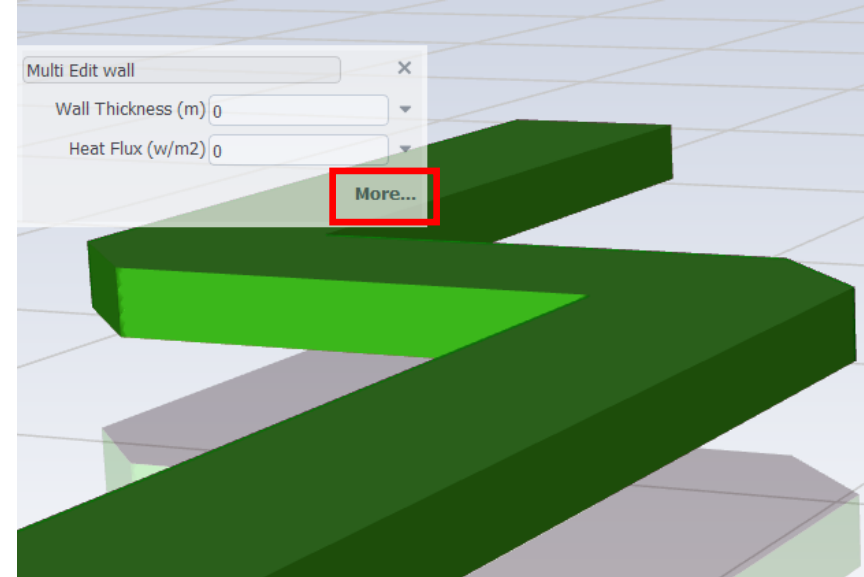
- Left click the inlet boundary in the graphics window
 - Selected boundary turns green
 - A quick property editor appears in the graphics window
- Enter 0.167 m/s for velocity and 10 °C for temperature
 - Entries in the quick property editor panel are saved automatically
 - Calculation of the precise velocity value can be found in the appendix
- Left click the outlet boundary in the graphics window
 - Leave the Gauge Pressure at the default value of 0 pascal and enter 5 °C for the backflow total temperature
 - The backflow temperature will only be used if there is reverse flow and while no reverse flow is expected here, it is good practice to enter a value in the expected temperature range



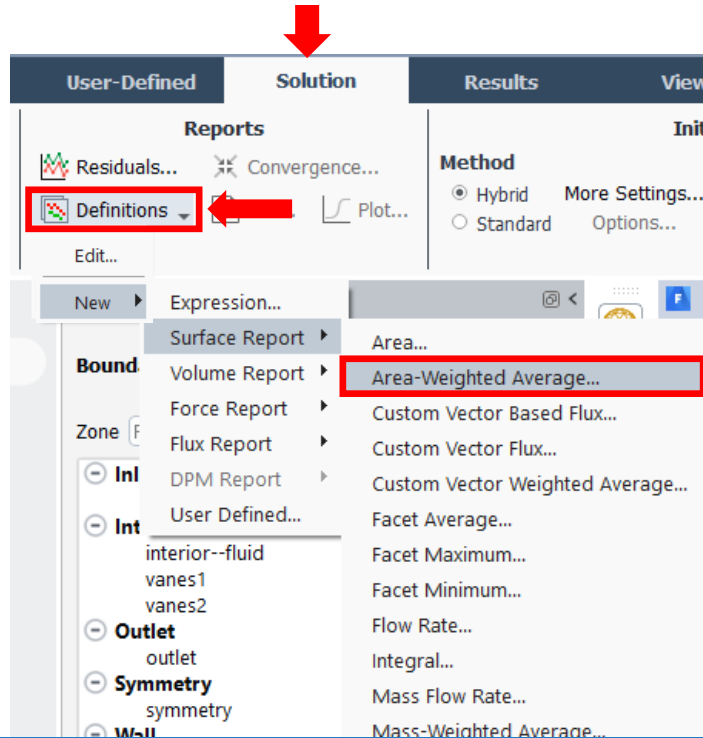
Physics: Boundary Conditions Using Multi-Edit

- Use ctrl + left click to select all three walls in the graphics window
- The quick editor panel shows heat flux but here the temperature needs to be assigned so click More...
- Under Thermal Conditions select Temperature
 - A green check mark appears, indicating that if the Apply button at the bottom of the panel is clicked, a Temperature condition will be applied to all boundaries selected in the zone list on the right of the panel
- Set the value of temperature to 0 °C
 - A second green check mark appears, indicating that if the Apply button at the bottom of the panel is clicked, the temperature will be set to 0 °C (or whatever value is entered in the panel) for all boundaries selected in the zone list on the right of the panel
- Click Apply

It is not required to use multi-edit to assign boundary conditions to the walls. They could have been assigned one at a time or the conditions for one wall could have been defined and then copied to the other two walls.

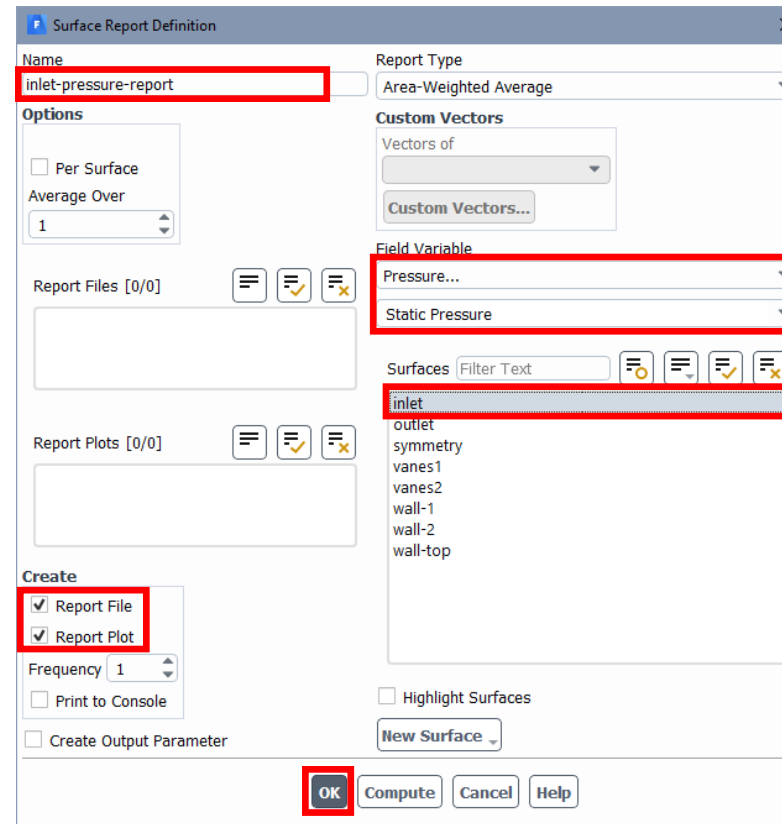


Solution: Report Definition (Inlet Pressures)



As Fluent calculates the solution, the pressure at the inlet will adjust until it corresponds to the value that would be needed to maintain flow at the prescribed inlet velocity. This makes inlet pressure a useful quantity for solution monitoring.

- In the Solution tab, click on **Definitions** in the Reports section and choose **New > Surface Report > Area-Weighted Average**



Enter the following in the definition panel and click OK:

Name = inlet-pressure-report

Variable = Static Pressure

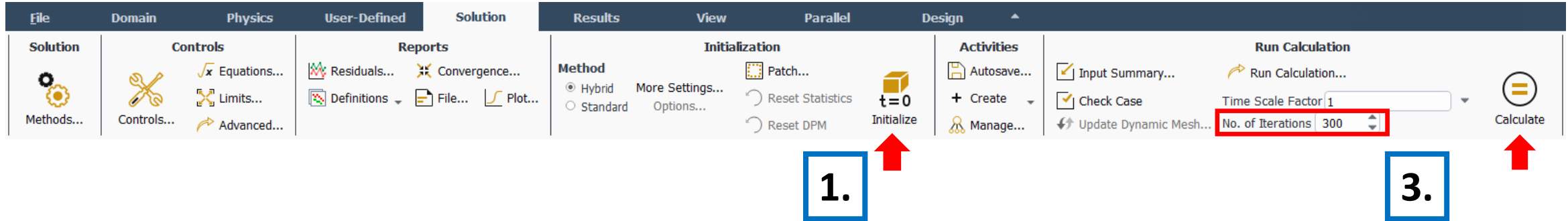
Surface = inlets

Report File = check

Report Plot = check

/ Solution: Initialize and Calculate

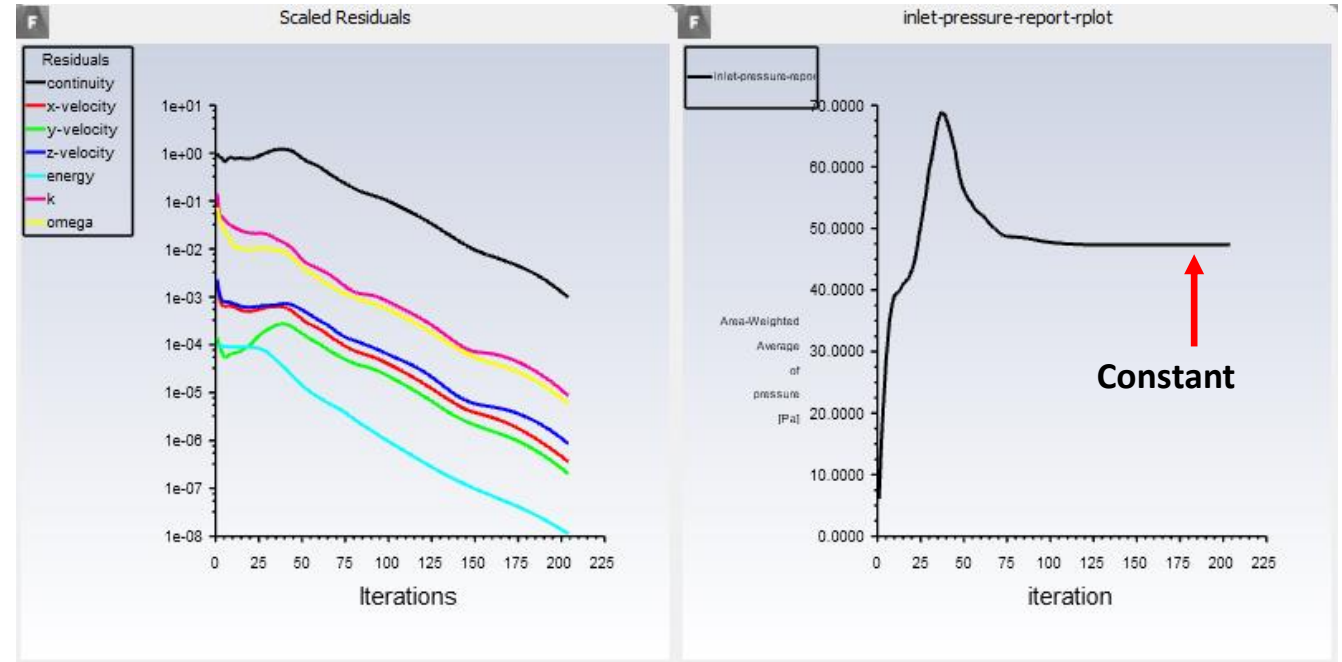
2.



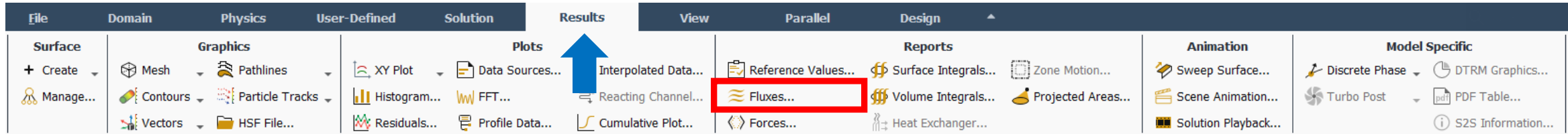
1. Click Initialize
2. File > Write Case
 - It is good to develop the habit of saving after setup has been completed
3. Set the number of iterations to 300 and click Calculate

/ Solving: Solution

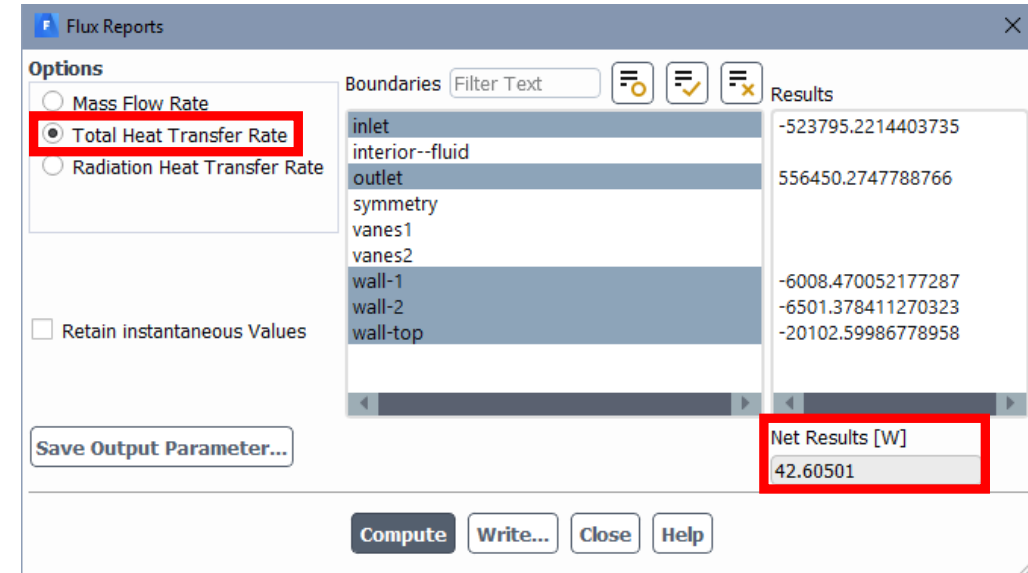
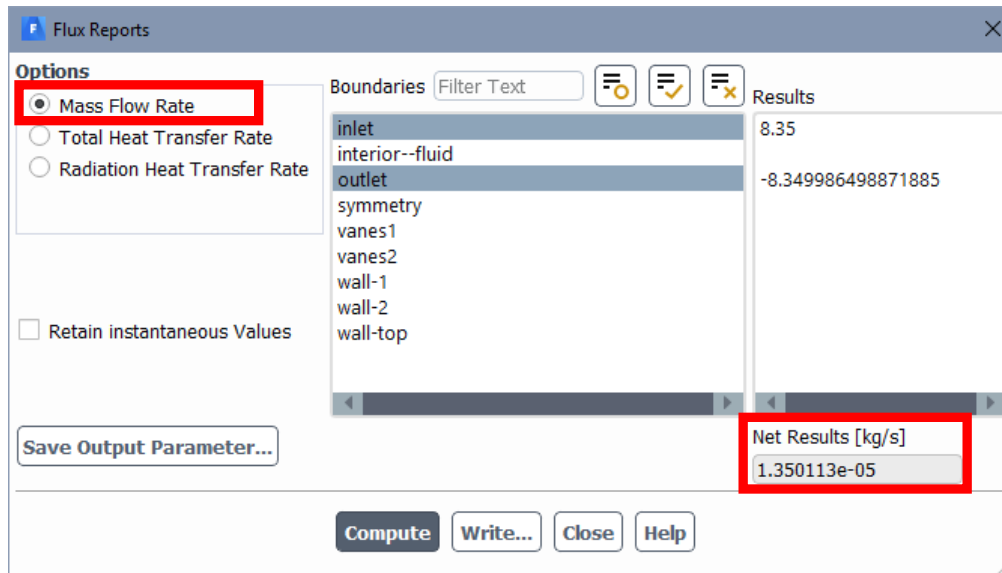
- Right click on the window tab and choose **SubWindow View** to show both the residuals and the report definition plot
 - You can do this while the solution is iterating
 - If you want to go back, just right click and select Tabbed View
- The iterations will automatically stop when the residuals reach convergence
- Notice in the plot on the right that the variable in the report plot has reached a constant value
- Go to File > Write > Case and Data



Results: Fluxes

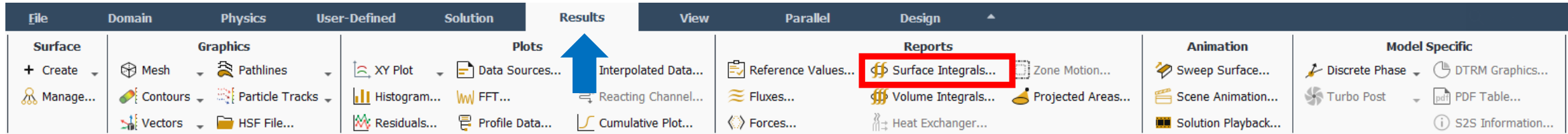


- In the Reports section, click on **Fluxes** and compute mass and energy balances

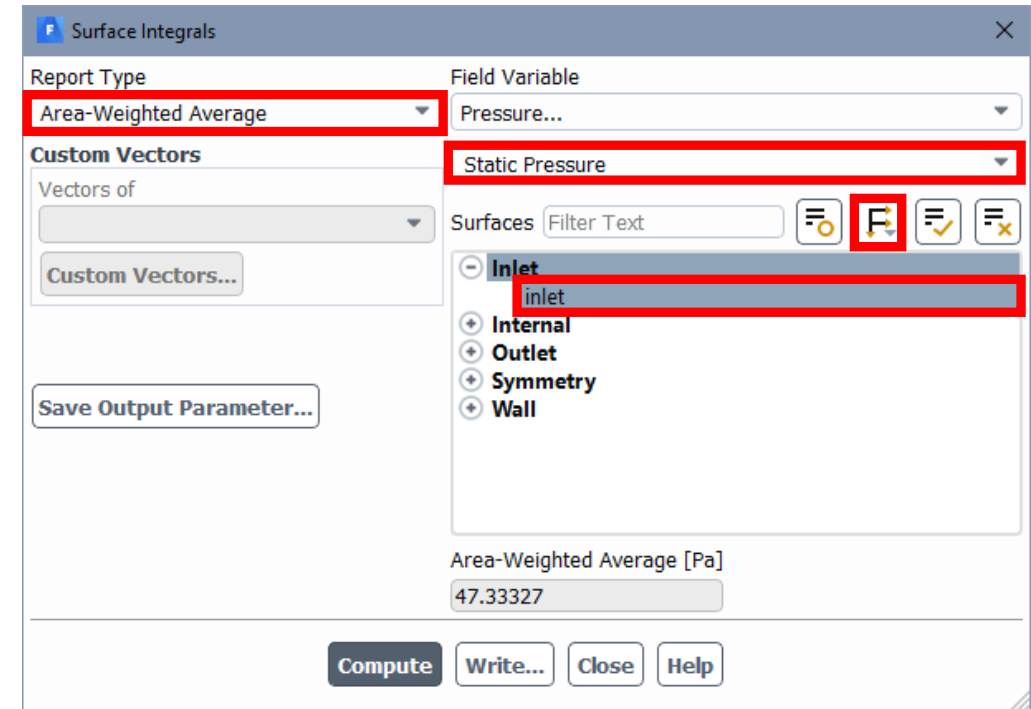


For Mass Flow Rate, the net value of $\sim 1e-5$ kg/s is negligible compared to the inlet flow rate (8.35 kg/s). For Total Heat Transfer Rate, the net value of 42.6 W is negligible compared to the value at any of the boundaries, for instance it is only around 0.12% of the sum of the walls (~ 32 kW). The precise value in Net Results in this case may be slightly different on different computers or different versions of Ansys Fluent, but it should still be a very small percentage.

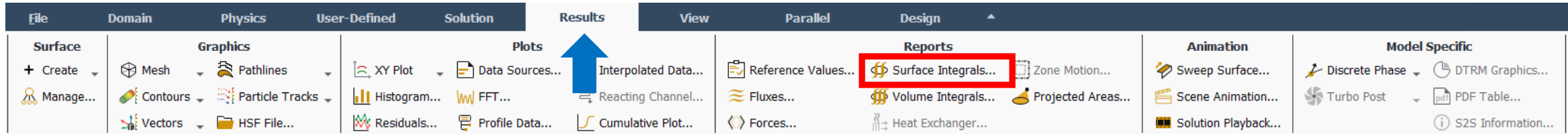
Results: Surface Integrals



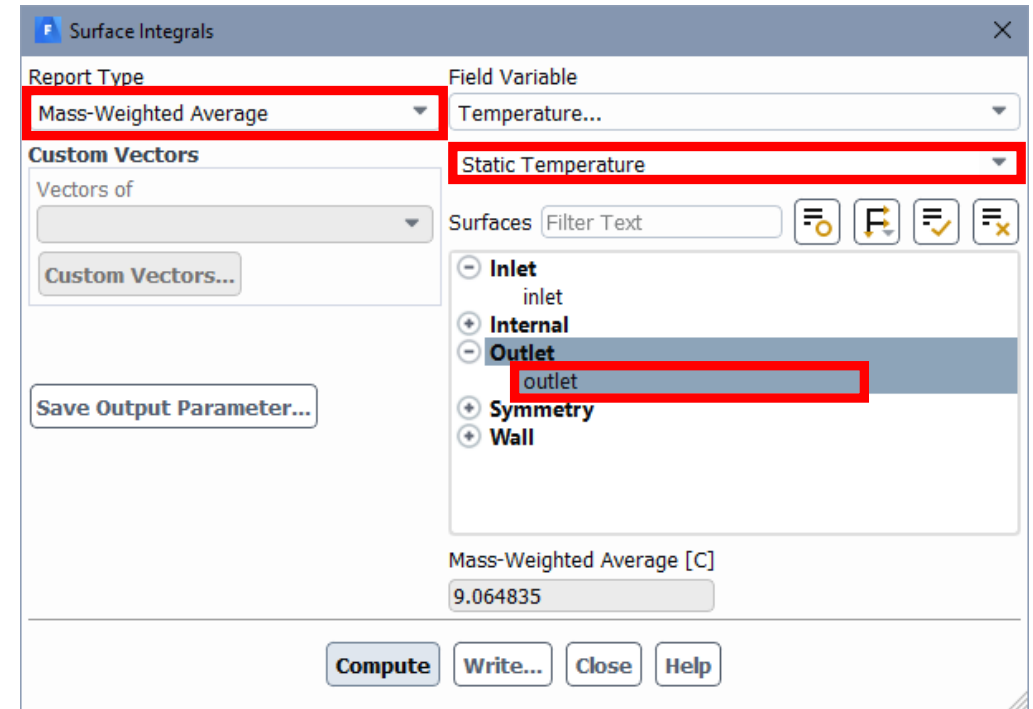
- In the Reports section, click on **Surface Integrals**
- One quantity of interest in the simulation is the pressure at the inlet
- Select Area-Weighted Average for Report Type, Static Pressure for Field Variable and the inlet surface, then click Compute
 - Grouping surfaces by zone type may be convenient
- Because the outlet pressure was defined to be 0 Pascals (gauge), the inlet pressure represents the pressure drop in the duct
 - This value will be compared with the pressure drop when the vanes are changed from interior zones to walls
 - Because the vanes are interior zones right now, the result on this page represents the pressure drop in through an empty duct



Results: Surface Integrals

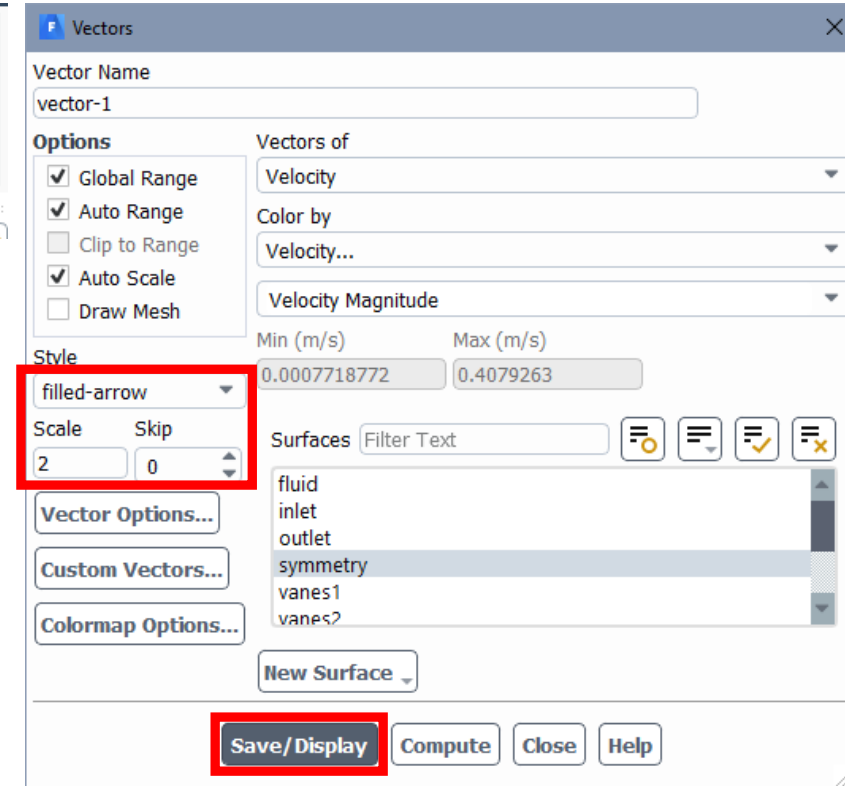
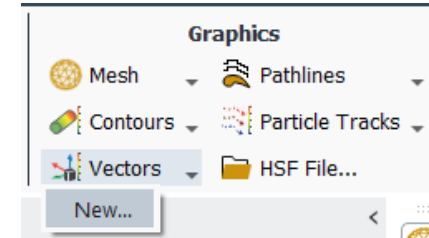
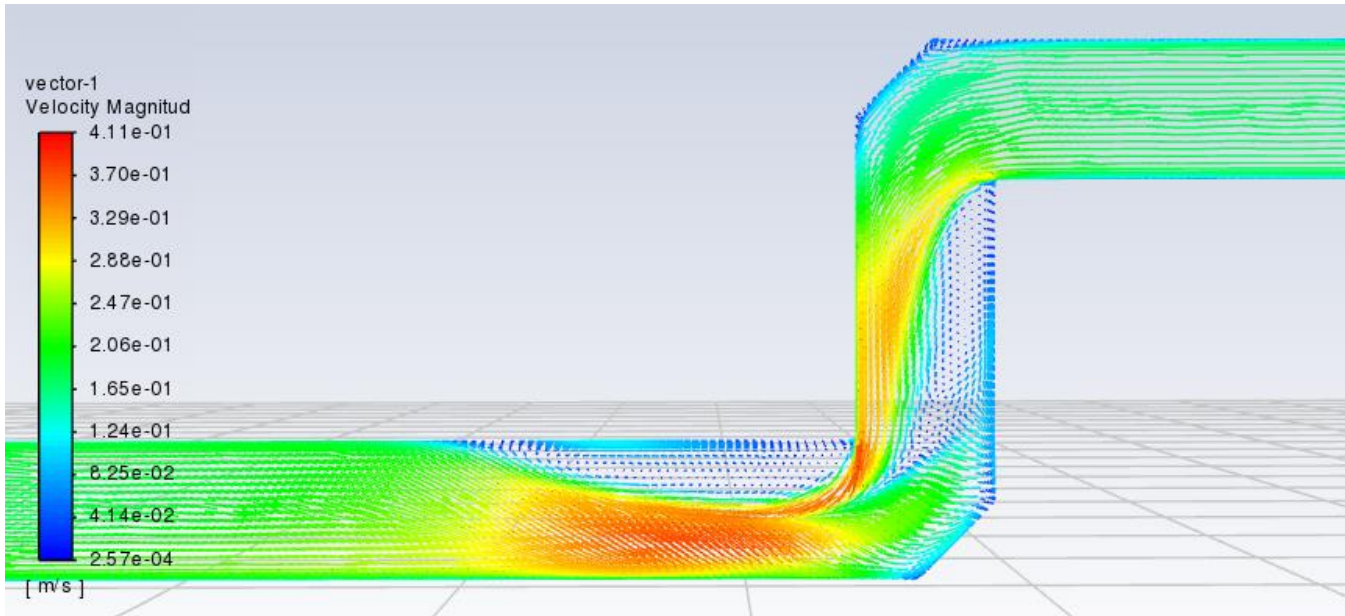


- Another quantity of interest in the simulation is the temperature at the outlet
- In the Surface Integrals panel, select Mass-Weighted Average for Report Type, Static Temperature for Field Variable and the outlet surface
- The result indicates that the average temperature has decreased by around 1 °C
 - Depending on circumstances, this may be significant, or it may not be significant, but either way, the equipment operator would be able to use this information to determine whether or not the duct walls should be insulated



Results: Vectors

- Click on Vectors and choose New
- Select the symmetry surface and click Save/Display
- Experiment with selecting various options under Style, Skip and Scale



Results: Pathlines

- Right click in the graphics window and create a new Pathline
- Select Time under Color by and release from the inlet
- Click Save/Display

- Note the console window reports many of the pathlines that were tracked are incomplete (exact number may vary slightly)

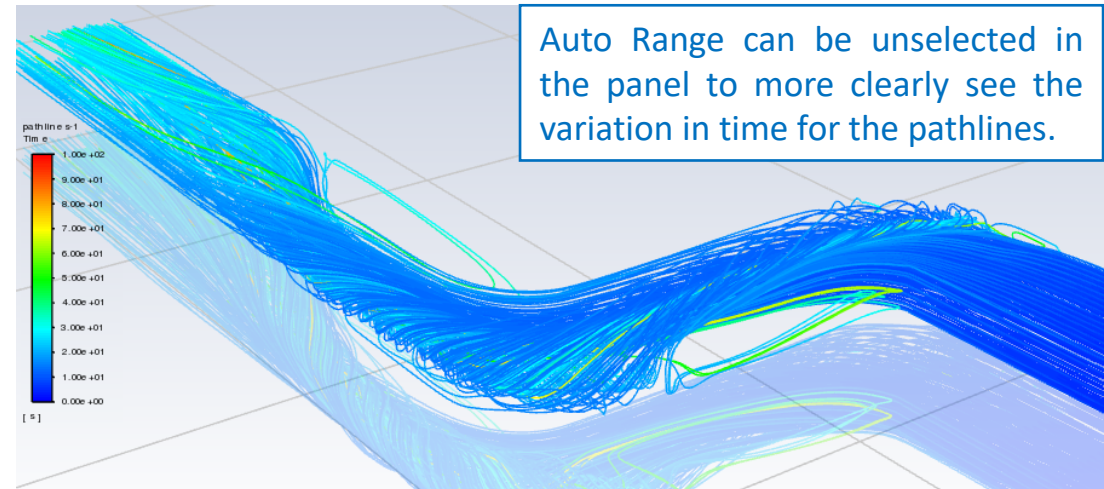
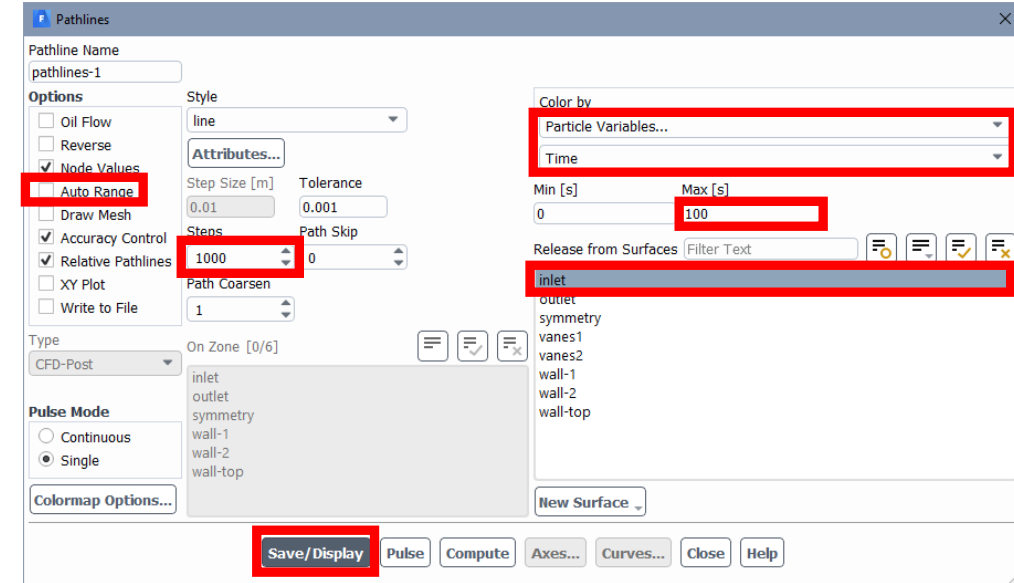
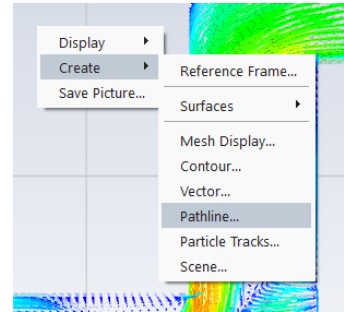
```
number tracked = 242, escaped = 33, incomplete = 209
```

- Increase the number of steps to 1000 and click Save/Display again
 - Most of the pathlines are complete (escaped)

```
number tracked = 242, escaped = 241, incomplete = 1
```

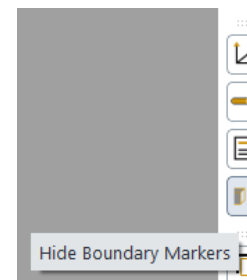
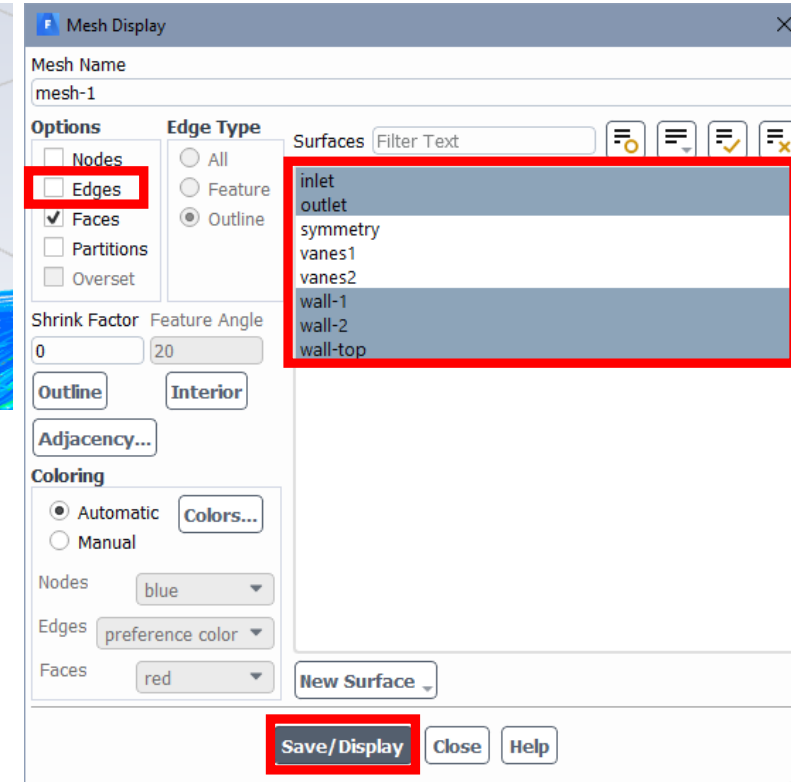
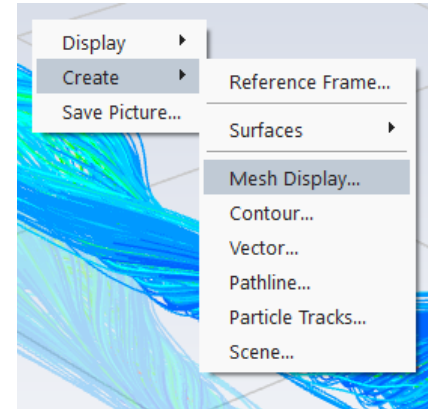
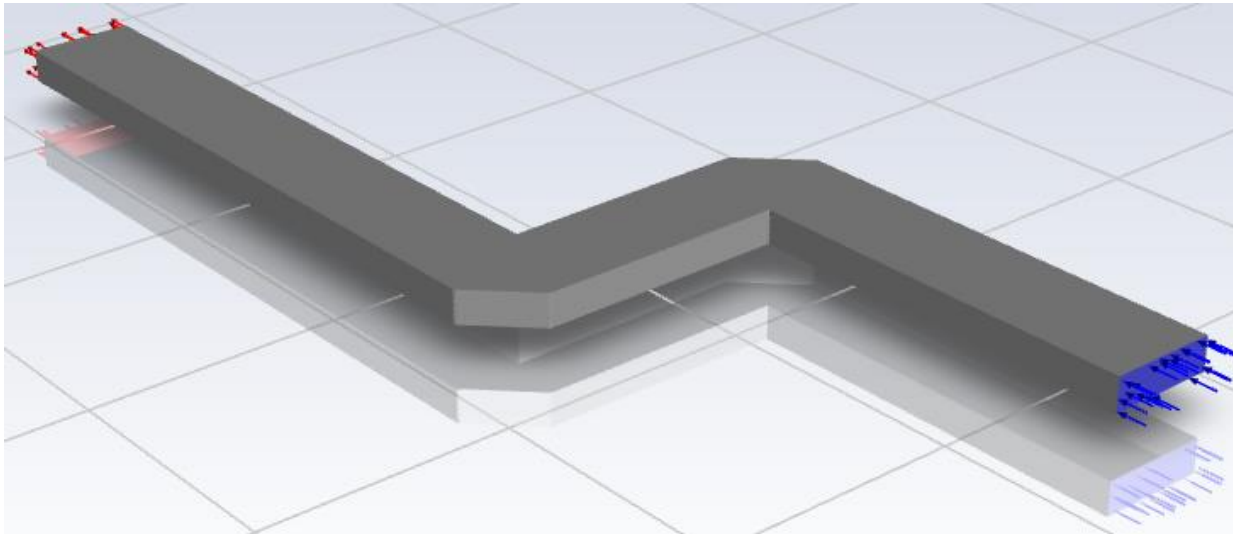
- The number of steps can be increased until all pathlines escape, but that offers little or no advantage in terms of flow visualization

Although the duct geometry is simple, a complex flow develops through the two 90° bends



Results: Mesh Display

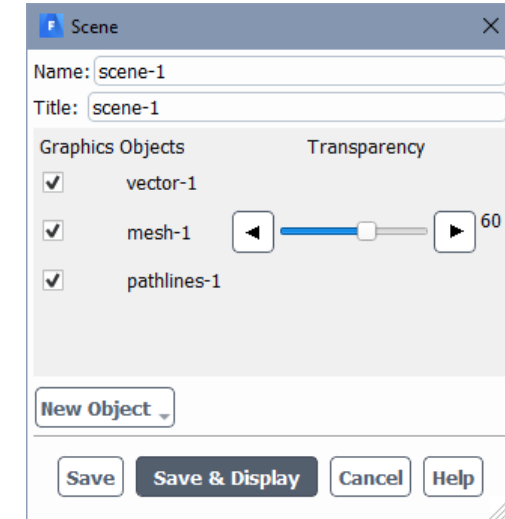
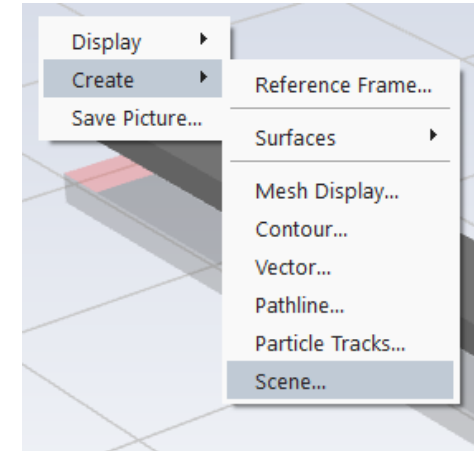
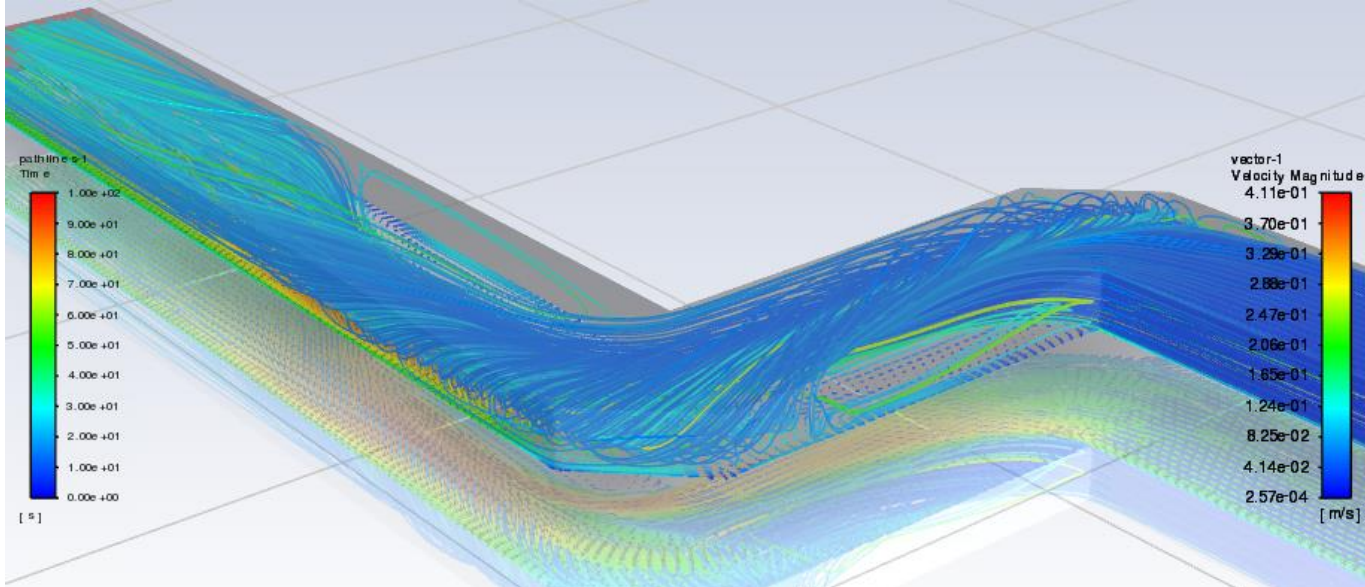
- Right click in the graphics window and select Create > Mesh Display
- Unselect Edges under Options and select the inlet, outlet and wall surfaces
- Click Save/Display



If desired, boundary marker visibility can be toggled on and off from the toolbar on the graphics window border.

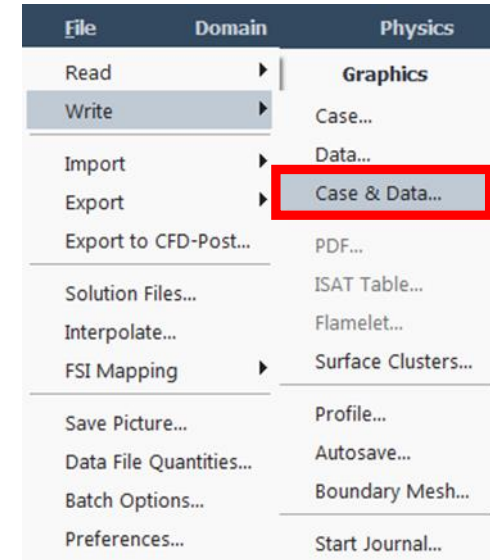
Results: Scene

- Right click in the graphics window and select Create -> Scene
- In the Scene panel, select all graphics objects, set Transparency as shown and click **Save & Display**
 - Transparency does not need to be exactly as shown to the right, just something reasonably close, or even experiment with different values



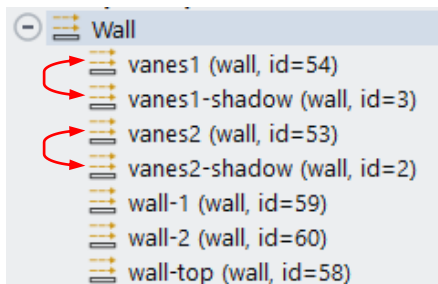
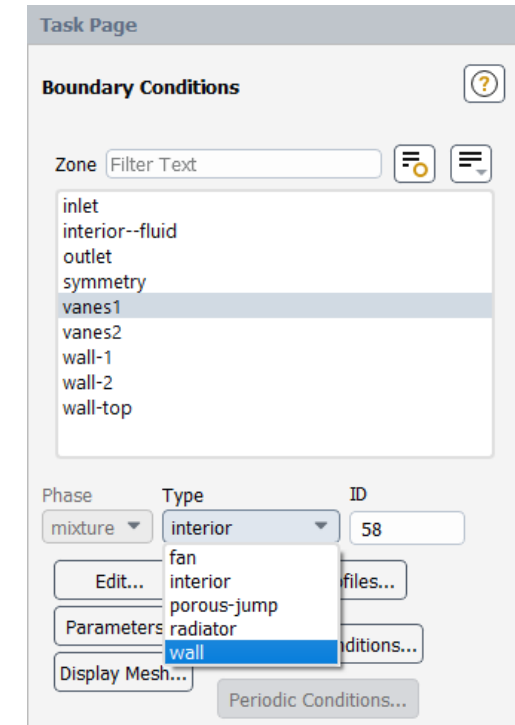
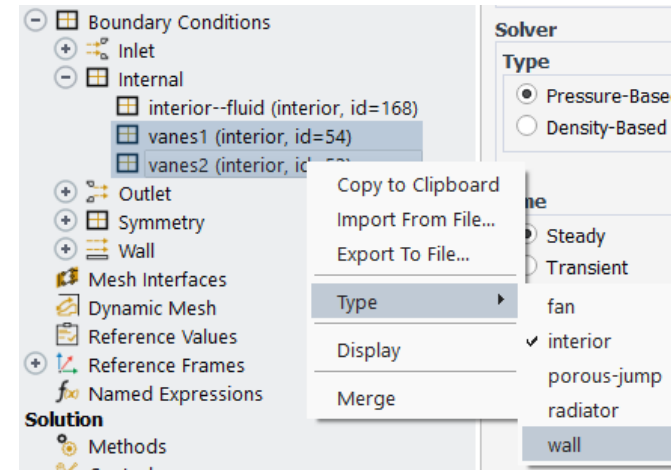
/ Save Case and Data

- Write the case and data file as duct_4v-sym-solved
- In the next step, the boundary type of the vanes will be changed from interior to wall and the solution will be re-calculated to assess the effect of inserting vanes into the duct bends



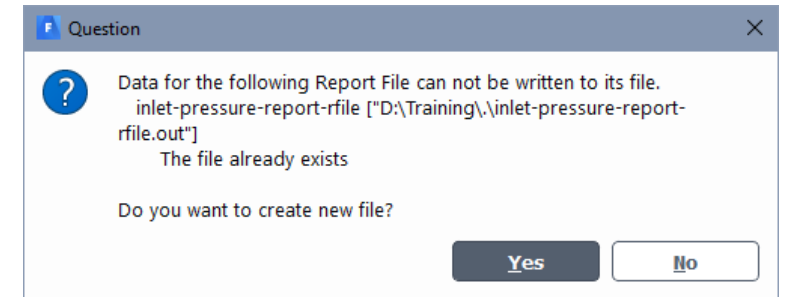
Physics: Change Boundary Condition Types

- Use the Ctrl key to select both vanes boundaries in the outline view
- Right click, select Type and then wall
 - Both will be changed to walls at the same time
 - Boundary condition type can also be changed one at a time in the Boundary Conditions Task Page
- Because the newly created walls are not external boundaries (e.g. there are mesh cells on both sides of the wall), shadow wall zones are created
 - This is explained further in the heat transfer lecture, effectively it means the wall zone is connected to cells on one side and the shadow wall zone is connected to cells on the other side



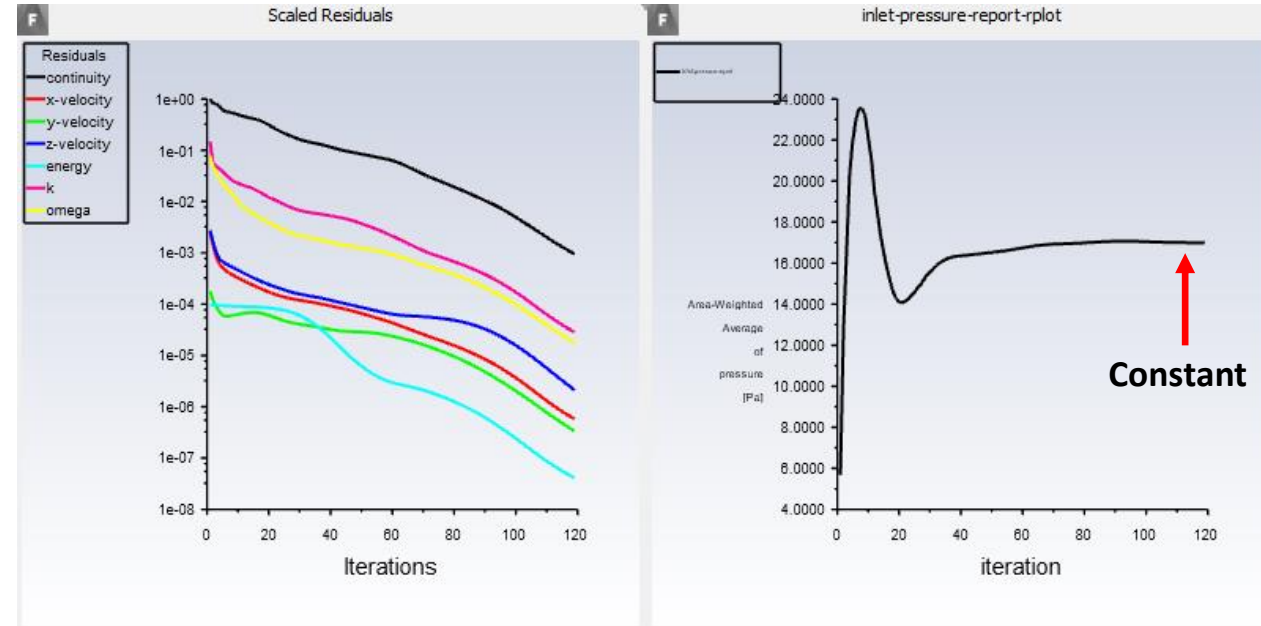
File: Write and Re-Read Case File

- Under File > Write > Case, save the case file as duct_4v-sym-vanes.cas.h5
- Go to the Solution tab, click Initialize and then Calculate
 - The flow field with the vanes is expected to be very different from the flow without the vanes that was just calculated and therefore it is expected that re-initializing the solution will lead to better convergence behavior
 - When prompted about the Report File, click Yes to create a new file

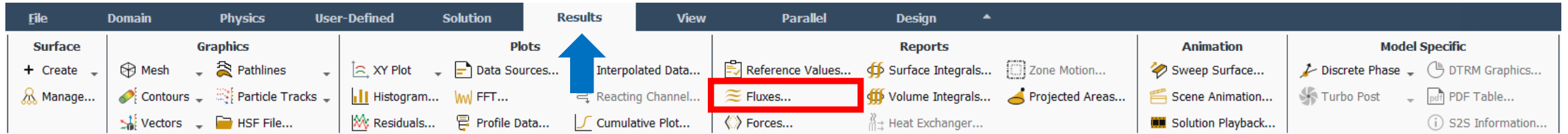


/ Solving: Solution

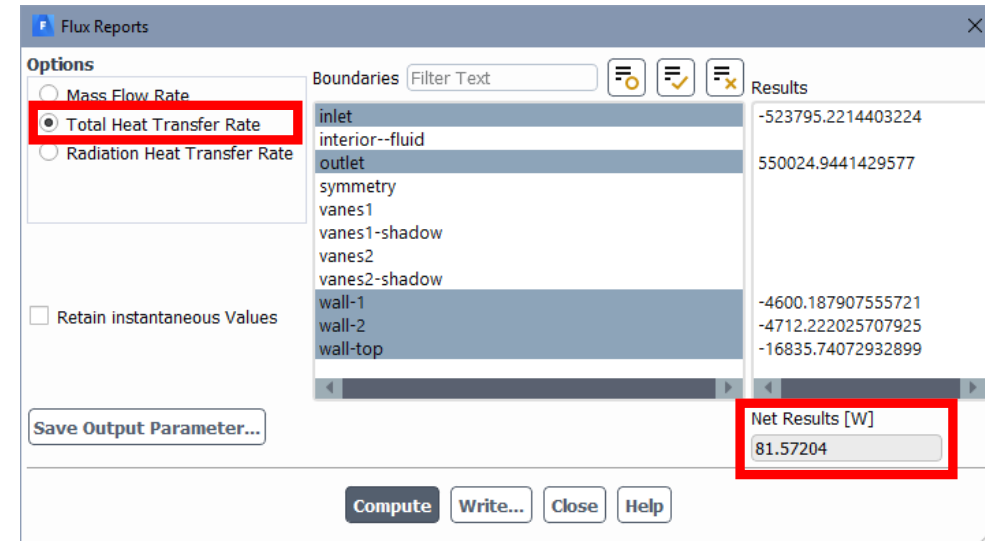
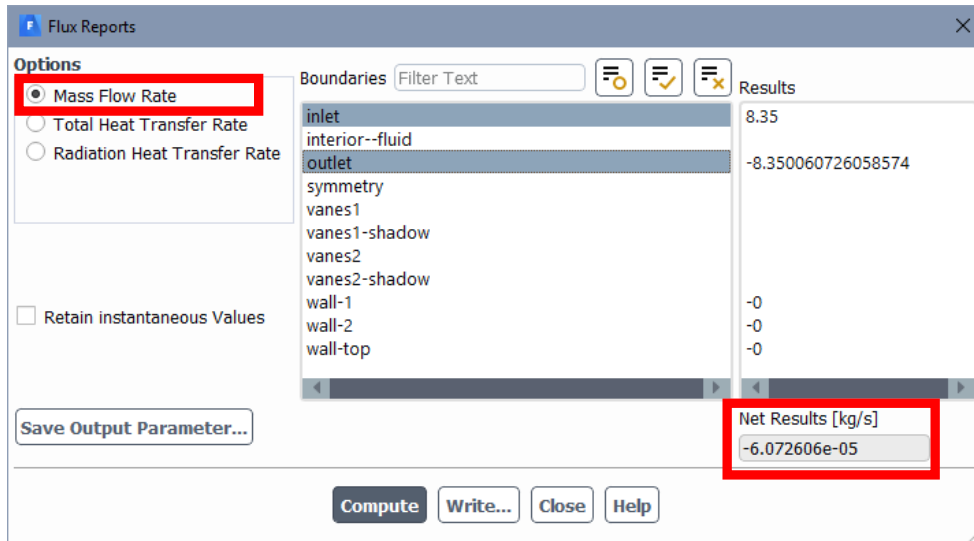
- Right click on the window tab and choose SubWindow View to show both the residuals and the report definition plot
 - You can do this while the solution is iterating
 - If you want to go back, just right click and select Tabbed View
- Good convergence is obtained
- Go to File > Write > Case and Data



Results: Fluxes

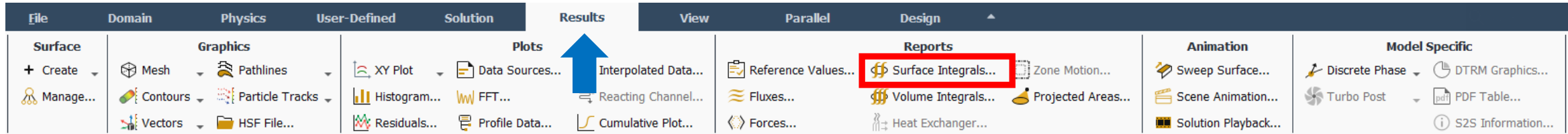


- In the Reports section, click on **Fluxes** and compute mass and energy balances

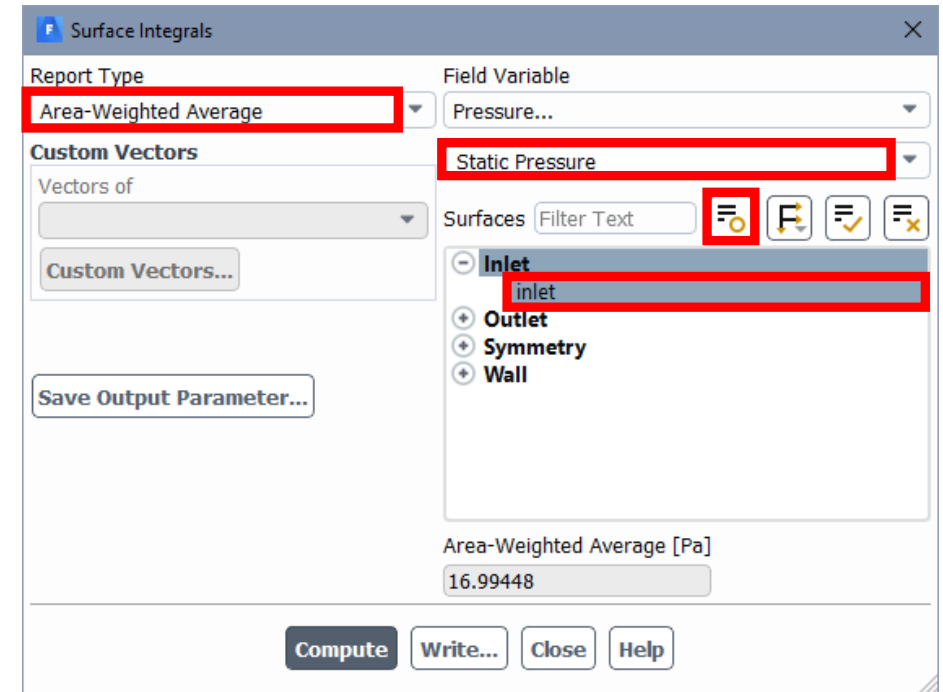


For Mass Flow Rate, the net value of $\sim 6e-5$ kg/s is negligible compared to the inlet flow rate (8.35 kg/s)
For Total Heat Transfer Rate, the net value of 81.6 W is much less than the ~ 26 kW combined rate of heat transfer from the walls.
The precise value in Net Results may be system dependent, but it should still be a very small percentage.

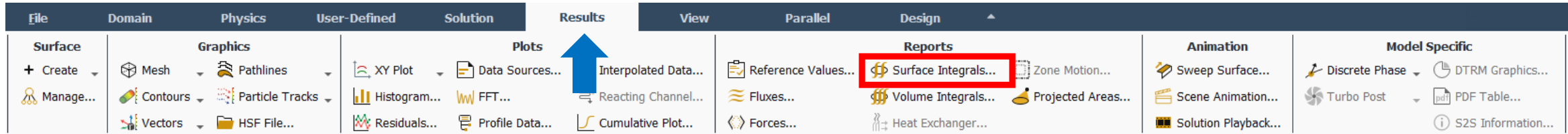
Results: Surface Integrals



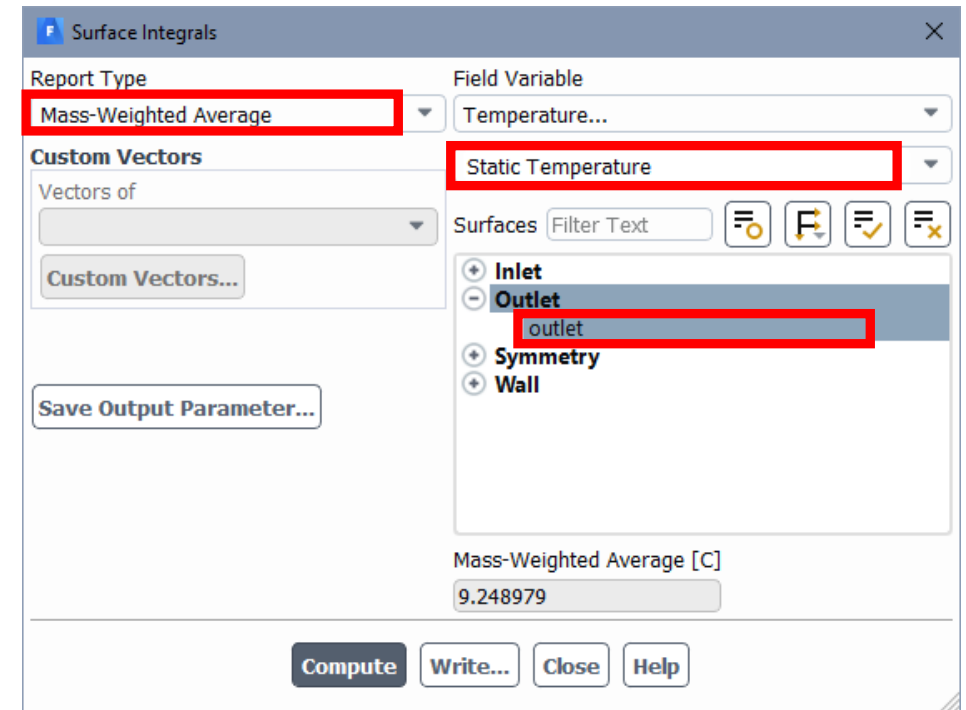
- In the Reports section, click on [Surface Integrals](#)
- One quantity of interest in the simulation is the pressure at the inlet
- Select Area-Weighted Average for Report Type, Static Pressure for Field Variable and the inlet surface
 - Grouping surfaces by zone type may be convenient
- The result of 17.0 Pa is 64% lower than the pressure drop (47.3 Pa) without vanes



Results: Surface Integrals

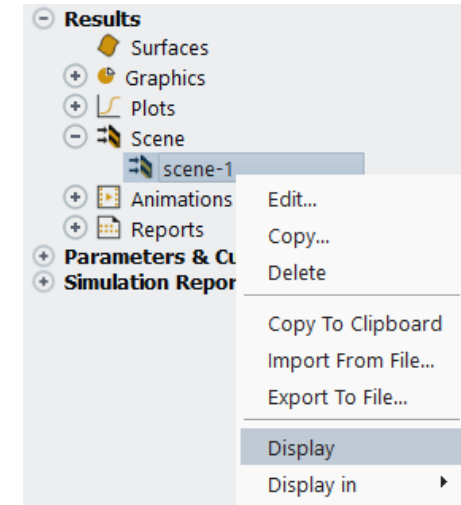
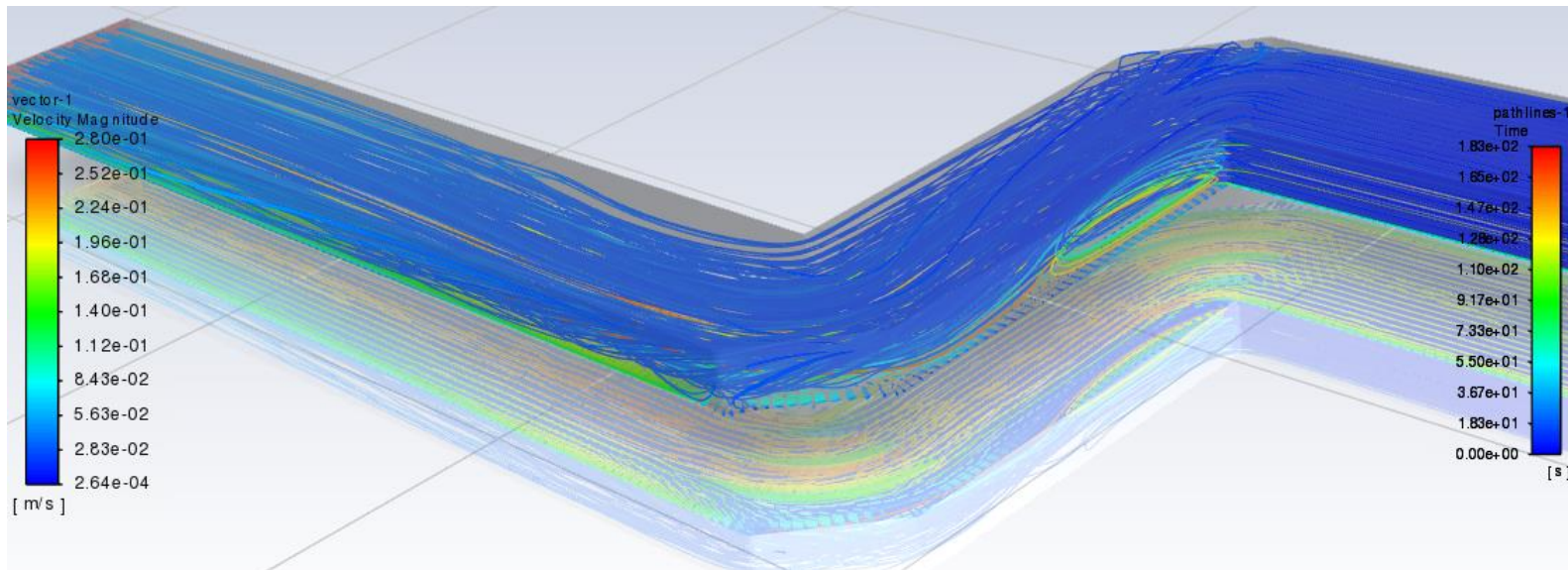


- Another quantity of interest in the simulation is the temperature at the outlet
- In the Surface Integrals panel, select Mass-Weighted Average for Report Type, Static Temperature for Field Variable and the outlet surface
- The result indicates that the average temperature has decreased by slightly less than without vanes but the decrease is still around 1 °C
 - Depending on circumstances, this may be significant, or it may not be significant, but either way, the equipment operator would be able to use this information to determine whether or not the duct walls should be insulated



Results: Scene

- Right click on scene-1 in the outline view and select Display
- The pathlines indicate the flow is much more aligned with the duct and the recirculation zones at the corners are much smaller than they were in the open duct



/ Wrap-Up

- Remember when using Fluent, the Ribbon guides the workflow of your Fluent session. In almost every session, you use 4 main tabs going from left to right, and in this session you did the following:

1. Domain

- Mesh Check
- Units

2. Physics

- Solver (defaults used for most problems)
- Models (Energy)
- Materials (added water-liquid as a material)
- Cell zone conditions (assigned water-liquid as the fluid in the cell zone conditions panel)
- Boundary conditions (entered boundary conditions for inlet (velocity and temperature), outlet (pressure and backflow) and wall (temperature) using quick editor and multi-edit features, changed boundary type for vanes)

3. Solution

- Create report definitions to monitor solution variables
- Initialize the solution
- Enter the number of iterations and calculate

4. Results

- Use Report Fluxes to check mass and energy balances
- Use Surface Integrals to check pressure drop and outlet temperature
 - Addition of vanes reduced pressure drop by 64%
- Use vectors and pathlines to visualize flow
- Use scene for attractive post-processing displays

/ Summary

- In this tutorial you have learned further details about Setting Up Physics
 - Physical Models
 - How to activate energy equation for heat transfer
 - Materials
 - How to add a material, in this case water-liquid but same process applies for any material, to your Fluent model
 - Zones
 - How to assign the fluid material in the cell zone panel
 - Additional practice defining boundary conditions
- Additionally, you have further experience with solving and post-processing
 - Solving
 - Create report definitions for important solution quantities to help monitor convergence
 - Post-processing
 - Use Flux Reports and Surface Integrals
 - Create graphics objects (vectors, pathlines, mesh display)
 - Use a scene to combine multiple post-processing objects in the graphics display

Appendix



/ Hand Calculations

Geometry and Fluid Properties				
rho	1000	kg/m3	density	
mu	0.001	kg/m-s	viscosity	
h	0.25	m	duct height	
w	0.4	m	duct width	
A	0.1	m2	Area (cross section)	
Dh	0.307692	m	Hydraulic Diameter	
Flow Rate and Velocity				
Q	1000	l/min	Volumetric Flow	
Q	16.66667	l/s	Volumetric Flow	
Q	0.016667	m3/s	Volumetric Flow	
V	0.167	m/s	Average Velocity	
Re	51282	-	Reynolds #	



End of presentation

