STAR-CCM+ Introduction Tutorial Part 2: Laminar Flow in a Rectangular Duct

R1.3

Bronwyn Rempel
Ethan Axdal
Scott J. Ormiston

Department of Mechanical Engineering
University of Manitoba
Canada

2022-10-12

1 Introduction

1.1 Overview

This tutorial is a continuation of the *StarCCM+ Introduction Tutorial Part 1: A Simple Duct Mesh*. The mesh created in *Part 1* is used in this tutorial to set up and run an isothermal single-phase laminar internal flow simulation. The main objective of this tutorial is to familiarize a new user with the StarCCM+ workflow and interface.

1.2 Geometry Nomenclature

The rectangular duct domain created in *Part 1* is shown schematically in Figure 1.

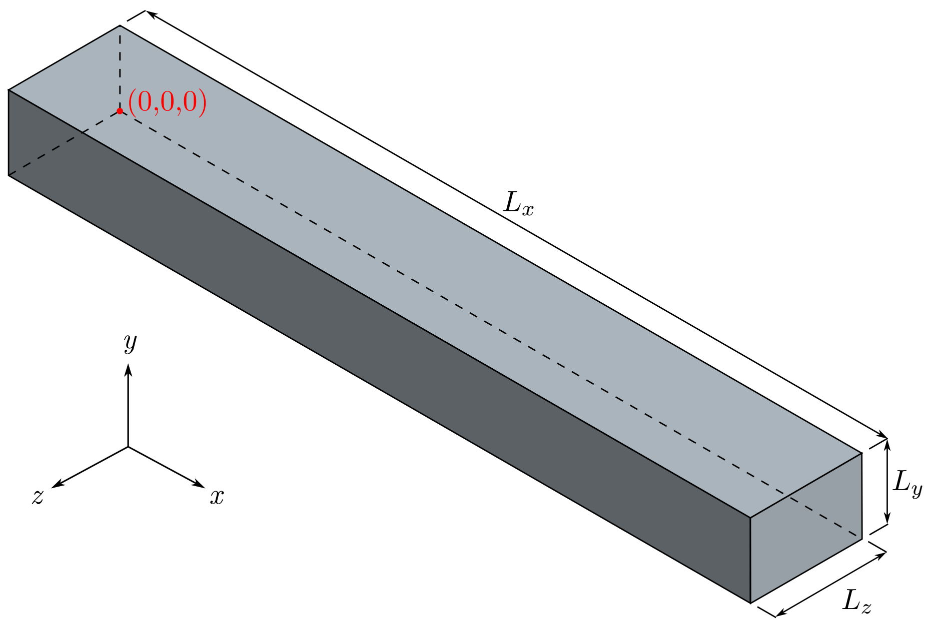


Figure 1: Rectangular Duct Geometry

The duct has length L_x, height L_y, and depth L_z. The flow is assumed to be symmetric about an x-z plane that bisects the duct, and therefore only half of the full duct is modelled. One corner of the duct is defined to lie at the origin, (0,0,0). All faces of the domain were defined during the modelling process and are named as follows: the plane described by $x = 0$ is denoted as Inlet, $x = L_x$ as Outlet, $y = 0$ as Symmetry Plane, $y = L_y$ as Top Wall, $z = 0$ as Right Wall, and $z = L_z$ as Left Wall. Pertinent geometric parameters are given in Table 1.

1.3 Problem Definition

In this tutorial, an isothermal, incompressible, constant property flow of water through a rectangular duct is modelled. There is no heat transfer model engaged; alternative approaches such as using an isothermal flow model or adding heat transfer could be explored as extensions to this tutorial.
The flow is modelled using a rectilinear uniform grid for half the domain, using symmetry in the y direction. All physical parameters required for the simulation are given in Table 1 below.

Table 1: Geometric and Physical Parameters

Parameter	Symbol	Value	Units	Comments
Mass Flow	$\dot{m}$	1.981×10^{-2}	kg/s	Half of the total mass flow of 3.962×10^{-2} kg/s
Density	ρ	997.0	kg/m^3	
Dynamic Viscosity	μ	8.899×10^{-4}	kg/m·s	
Duct Length	L_x	2	m	
Duct Height (Half)	L_y	0.2	m	Half of the total duct height of 0.4 m
Duct Depth	L_z	0.3	m	
Hydraulic Diameter	D_h	0.34286	m	Used to calculate the Reynolds number
Reynolds Number	Re_{D_h}	127.2	–	Indicates a laminar flow regime

1.4 Objectives

This tutorial is intended to demonstrate how to:

1. Import a mesh into the STAR-CCM+ environment (or start from the previous simulation)
2. Select physics models
3. Review and apply initial conditions and understand their importance in solution convergence
4. Specify boundary conditions
5. Set stopping criteria
6. Run a simulation
7. Perform some post-processing of results within the STAR-CCM+ environment
8. Export solution data for further post-processing

2 Setting Up the Simulation

STAR-CCM+ solves physics problems using a computational mesh. To provide this mesh, you can either (a) create a geometric model within the STAR-CCM+ environment and then use that model to create the mesh within STAR-CCM+, or (b) import an existing mesh. It is strongly recommended to use method (a), which corresponds to continuing with the `duct.sim` file that you saved from *Part 1*.

It is recommended that you save the simulation periodically during the set-up process. Use Save As to save progress under other file names to allow getting back to a previous stage should something not work as expected.

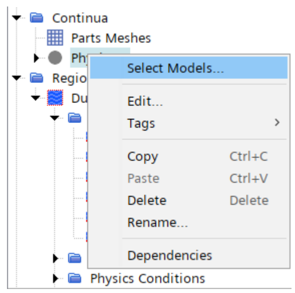
2.1 Starting from the Previous Simulation

1. Launch STAR-CCM+.
2. Open the previously created `duct.sim` file.
3. Save the simulation as duct-lam.sim in the file directory you created in *Part 1*. Close the `duct.sim` simulation tab.

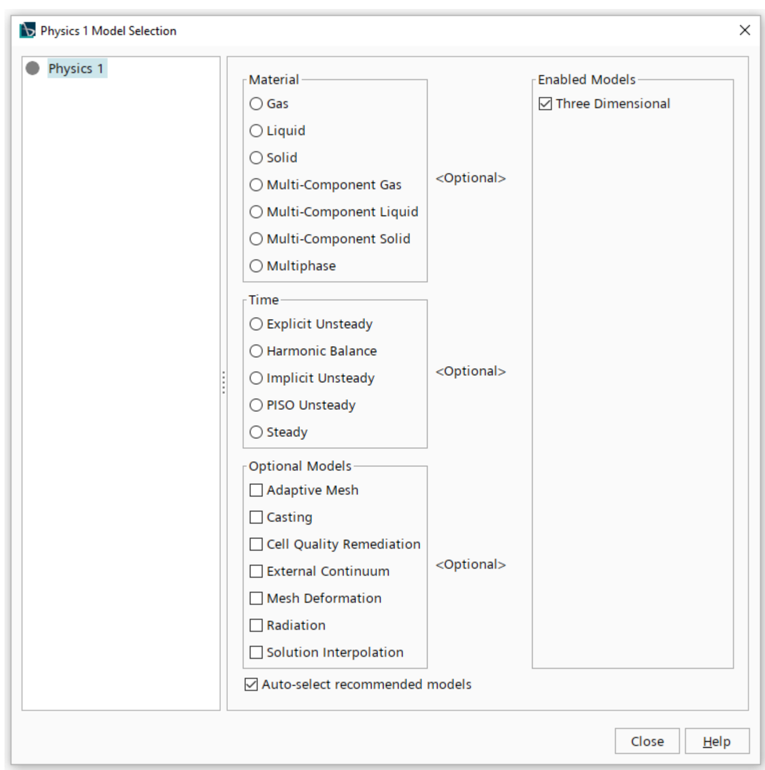
2.2 Selecting the Physics Models

A physics continuum defines a set of physics models and their associated material. A physics model is associated with one or more regions. With a mesh defined, a region now exists to which a physics model can be applied.

1. Expand the Continua node and right-click on Physics 1 (this is the physics continuum automatically created and assigned to the Duct Body region).



2. Click on Select Models.... The *Physics Model Selection* dialog should appear:



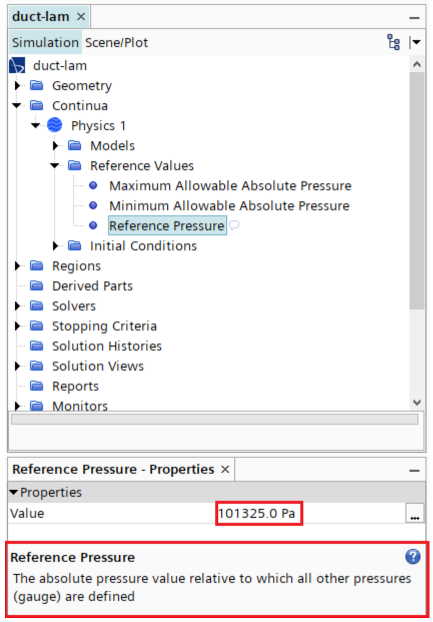
3. Select the following physics models, in order:

Group Box	*Model*
Enabled Models	**Three Dimensional** (pre-selected)
Material	**Liquid**
Flow	**Segregated Flow** **Gradients** (selected automatically with above)
Equation of State	**Constant Density**
Time	**Steady**
Viscous Regime	**Laminar**

Then press Close to exit the dialog box, as no optional models are required for this simulation. Notice the following:

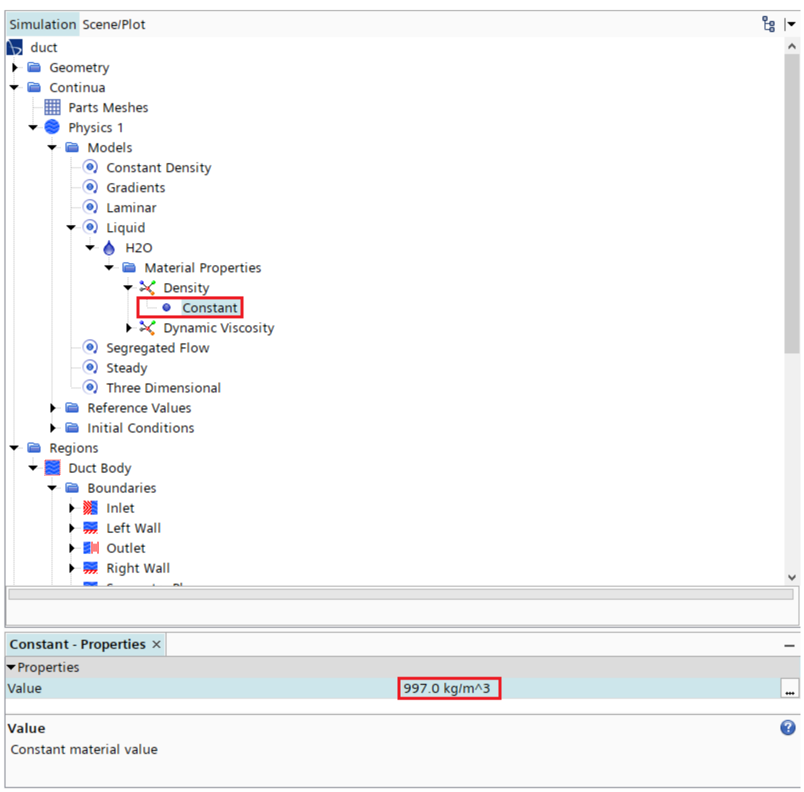
- Since **Auto-select recommended models** is engaged, the *Physics Model Selection* dialog guides you through the model selection process by prompting you for required models based on previous selections, and selecting certain default models automatically as you make choices. For example, when a continuum has contains a liquid or a gas, it also needs a flow model. Once it has a flow model, it needs a viscous model. If turbulence is activated in a fluid continuum, it will need a turbulence model.
- Water is the default substance defined for a liquid continuum. This can be seen by expanding Continua → Physics 1 → Models → Liquid → H20.
- To reverse part or all of the model selection processes, simply clear the checkboxes of the models you want to deactivate. Active models that are grayed out require other models to be deactivated first.
- The color of the Physics 1 node has turned from gray to blue, indicating that required models have been activated.

4. Open the Continua → Physics 1 → Reference Values node and select Reference Pressure. Ensure that the Value property is set to to 101325.0 Pa.
   - Flow simulations require the specification of a reference pressure. The reference pressure is simply a device that is used to reduce the numerical roundoff error in the numerical calculations involving pressure. This is necessary since the differences in pressure are important, and these differences can be small relative to the absolute value of the pressure. Other reference values may be required in more complex flow simulations.
   - A short explanation is given in the *Properties Window* when a Property is selected. As well, (Help) or F1 can be used to find the entry in the *STAR-CCM+ User Manual* to learn more about most objects.



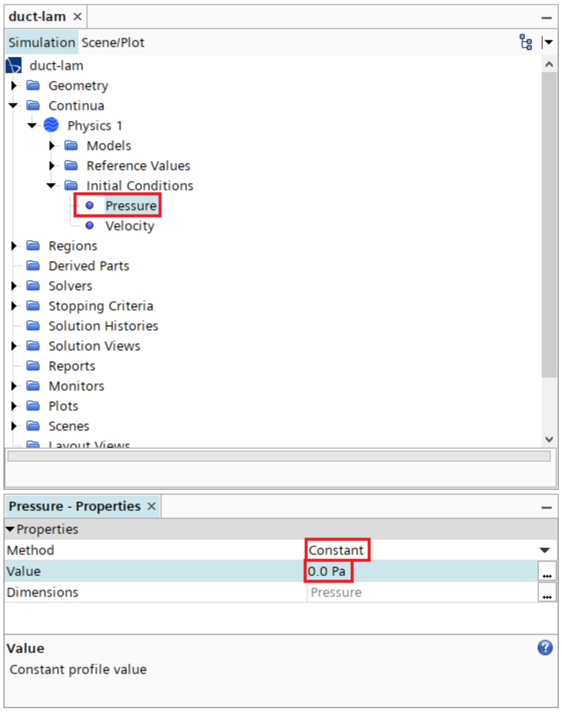
2.3 Setting the Fluid Properties and Initial Conditions

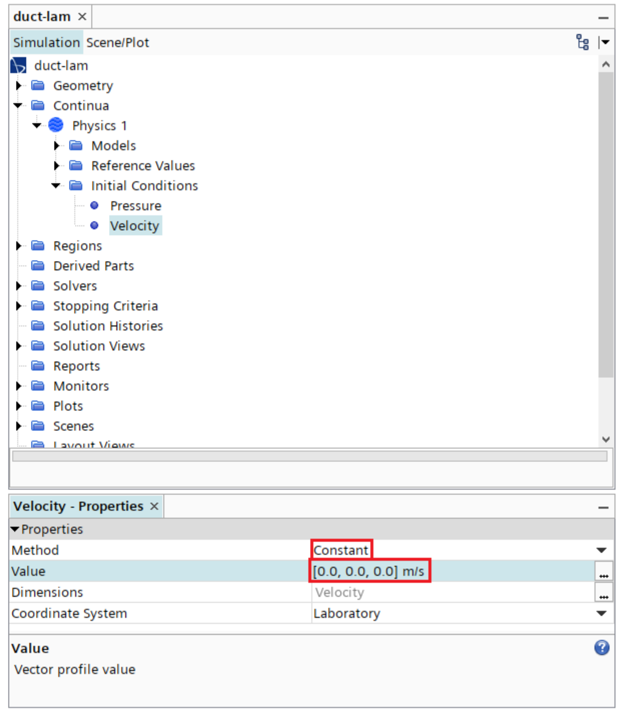
1. Expand the Continua → Physics 1 → Models → Liquid → H20 → Material Properties Density node and and select the Constant Node.
2. Set the Value property to that given for density in Table 1 (enter 997.0).



3. Similarly, set the Value property for Dynamic Viscosity to that given in Table 1 (enter 8.889E-04).

4. Expand the Physics 1 → Initial Conditions node and select the Pressure node. Verify that the Method is Constant and the Value is 0.0 Pa. This defines the initial pressure field of the fluid at the start of the simulation.
5. Similarly, select the Velocity node and verify that the Method is constant and that the value is [0.0, 0.0, 0.0] m/s. This defines the initial velocity field of the fluid as stationary in the Laboratory coordinate system, which is the default reference frame of any continuum.





2.4 Specify the Boundary Conditions

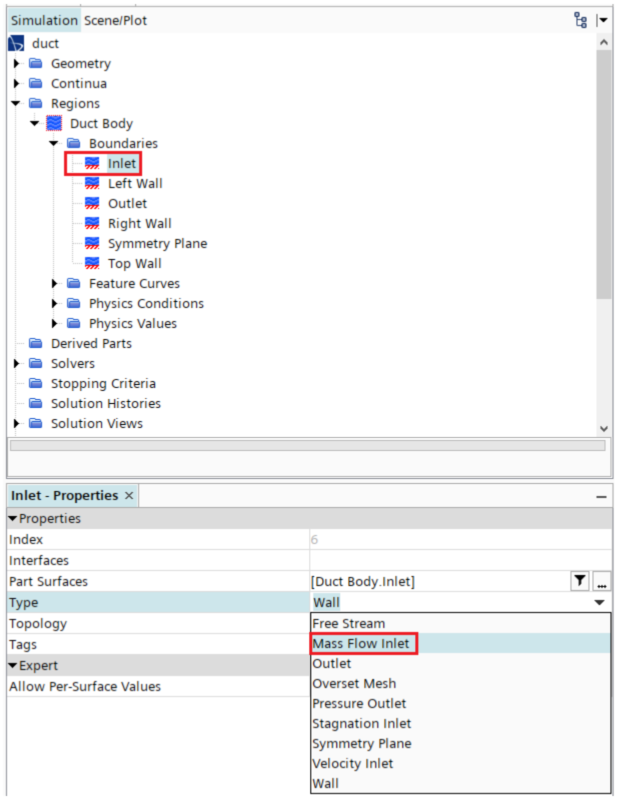
Boundary conditions are applied under the Regions → Boundaries node. In *Part 1*, you Assigned Parts to Regions when creating the mesh. This step created Boundaries from previously defined part surfaces. The following boundary conditions are applied to those boundaries:

Boundary	*Condition*
Inlet	**Mass Flow Inlet**
Left Wall	**Wall**
Outlet	**Pressure Outlet**
Right Wall	**Wall**
Symmetry Plane	**Symmetry Plane**
Top Wall	**Wall**

1. Check that the boundaries are properly defined: with Mesh Scene 1 open in the *Graphics Window*, open the Regions → Duct Body → Boundaries node and click on each boundary in sequence. The

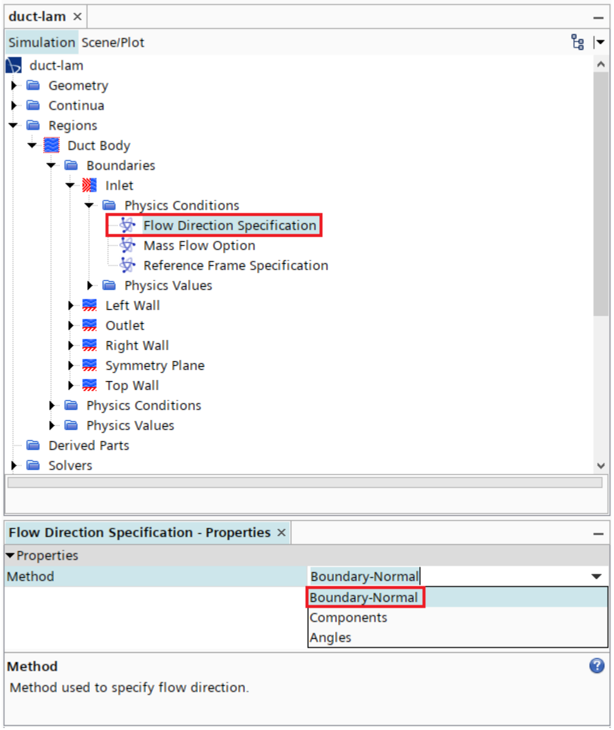
selected boundary should be highlighted in pink in the graphics window. As well, ensure all boundary conditions are set in the default Laboratory (or Lab) coordinate system.

2. Set the Inlet boundary condition:
   2.1. Select the Inlet node and set the Type property to Mass Flow Inlet.

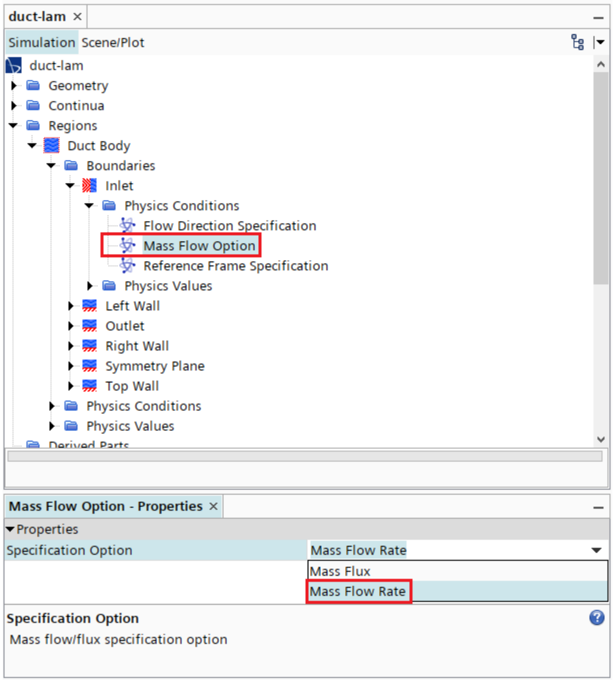


   Note here that the Part Surfaces property contains [Duct Body.Inlet]. The Part Surfaces property dictates the geometric entities (in this case a region) associated with this Boundary.

   2.2. Expand the Inlet → Physics Conditions node and select Flow Direction Specification. The Method should be set to Boundary-Normal, but note the other options available.



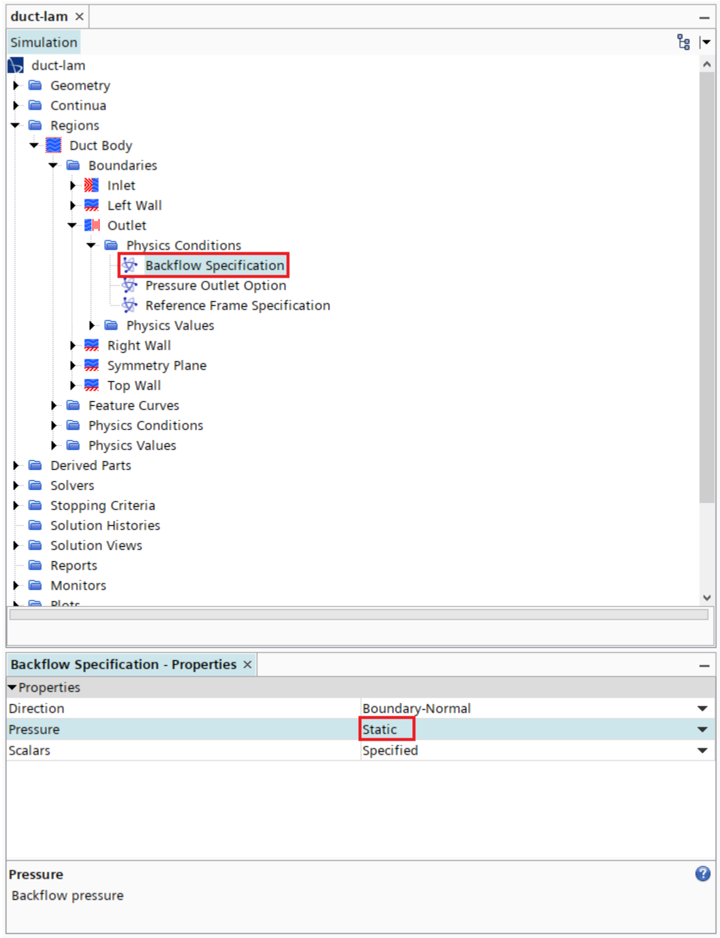
2.3. Select the Mass Flow Option node and verify that Specification Option is set to Mass Flow Rate. The alternative option is to define a mass flux at the boundary.



2.4. Expand the Inlet → Physics Values node and select Mass Flow Rate. Set the Value property to that given in Table 1 (enter 1.981E-02).

3. Set the Outlet boundary condition:
   3.1. Select the Outlet node and set the Type property to Pressure Outlet.
   3.2. Expand the Outlet → Physics Conditions node and select Backflow Specification Option.

Change the Pressure property from Environmental to Static. The Environmental condition subtracts the dynamic head at the pressure boundary, while the Static condition maintains the pressure at the user-defined pressure.



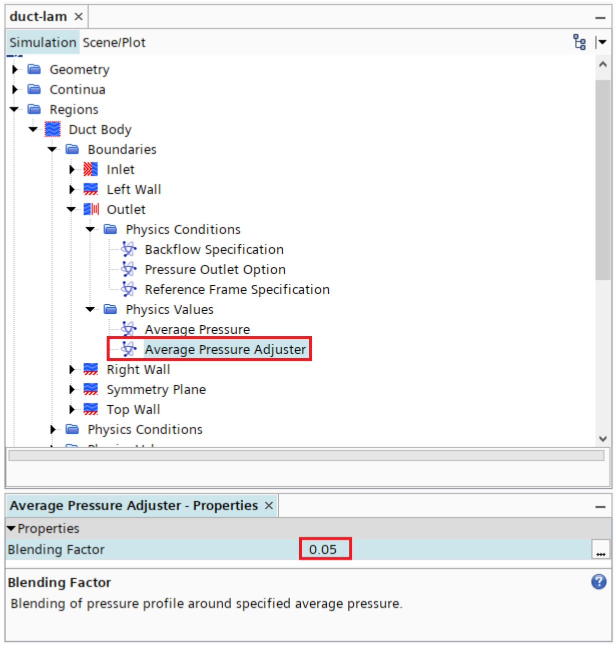
Leave the other conditions as default, noting:

- The Direction property controls the flow direction at the boundary, with options of:
  - Boundary Normal - as the name suggests, the flow is normal to the boundary.
  - Extrapolated - extrapolates the flow from the interior of the domain.
  - Components or Angles - these options specify the flow directly in 3D space through either a unit vector or Euler angles.
- The Scalars property controls how the scalar quantities (e.g., temperature) are defined at the boundary, with options of:
  - Specified - applies the scalar conditions that you specify if the flow reverses at the boundary, otherwise, extrapolates scalar conditions from the domain.
  - Extrapolated - extrapolates the conditions from the domain regardless of the flow direction.

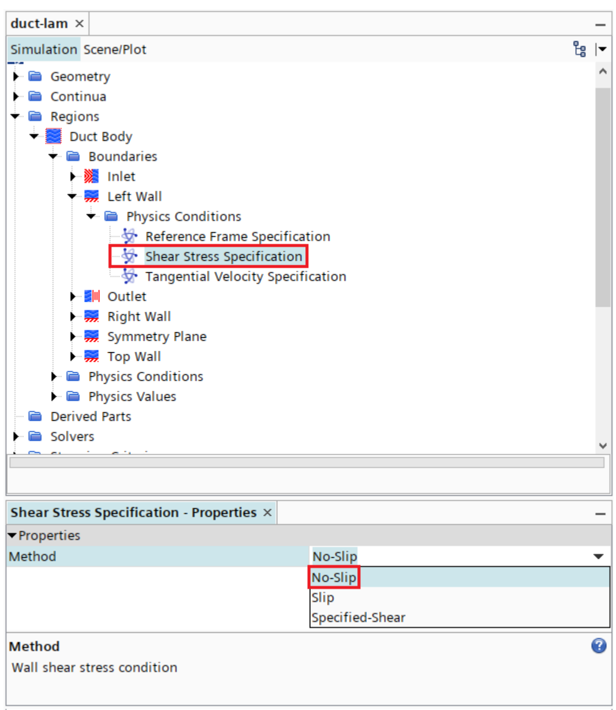
3.3. In the same node, select Pressure Outlet Option. Set the Option to Average Pressure.

3.4. Expand the Outlet → Physics Values node and select Average Pressure. Verify that the Value property is 0.0 Pa. Recall that this is a gauge pressure to the Reference Pressure set in the Physics 1 continua. Therefore, the absolute average pressure at the outlet is 101325.0 Pa.

3.5. Next, select the Average Pressure Adjuster node and change the Blending Factor property to 0.05.



4. Set the Symmetry Plane boundary condition: select the Symmetry Plane node and set the Type property to Symmetry Plane.
5. Set the Wall boundary conditions:
   5.1. Expand the Left Wall node and verify that the Type property is set to wall.
   5.2. Expand the Left Wall → Physics Conditions node and select Shear Stress Specification. Ensure that the Method property is set to No-Slip.

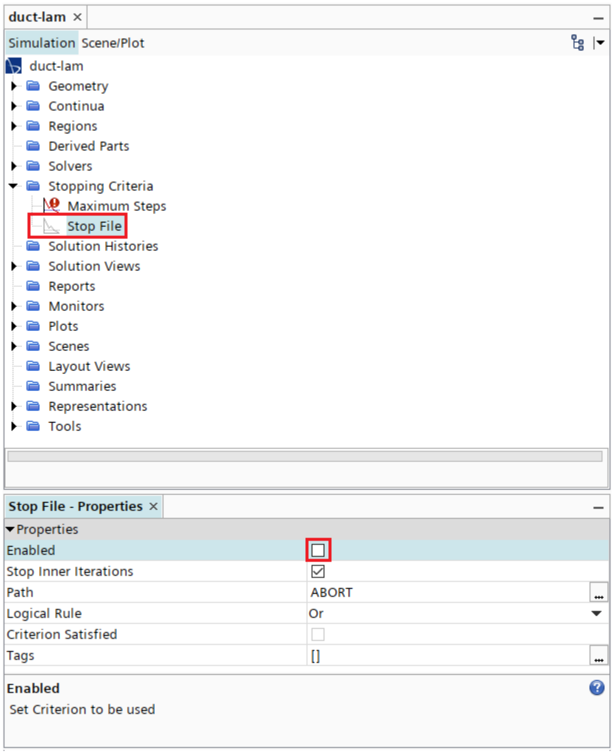


   5.3. Perform the last two steps for the both the Right Wall and Top Wall boundaries.

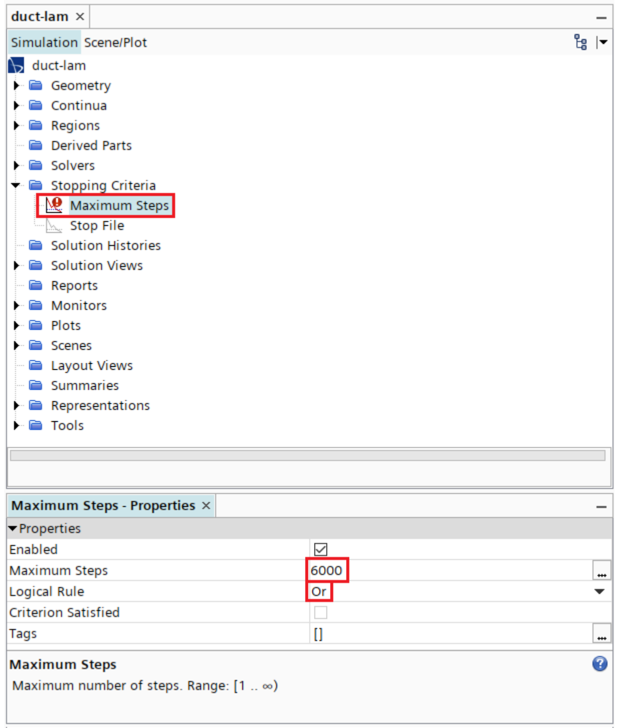
2.5 Set the Stopping Criteria

Stopping criteria for this simulation will consist of a maximum number of iterations and a minimum limit for the conservation equation residuals. Note that the criteria that are set below are nominal and work on most current desktop computers. There have been situations where the residual level is not met on some PC hardware. If this is the case when you run the simulation, you may need to adjust the criterion value.

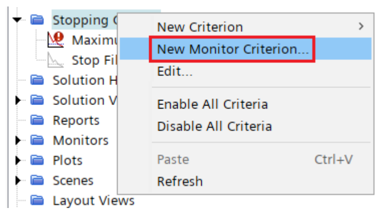
1. Expand the Stopping Criteria node and note that the two default criteria (Maximum Steps and Stop File) for steady simulations are present. Automatically generated stopping criteria cannot be deleted, but the Enabled Property can be activated or deactivated.
2. Select the Stop File node and uncheck the Enabled property to disable the criterion.



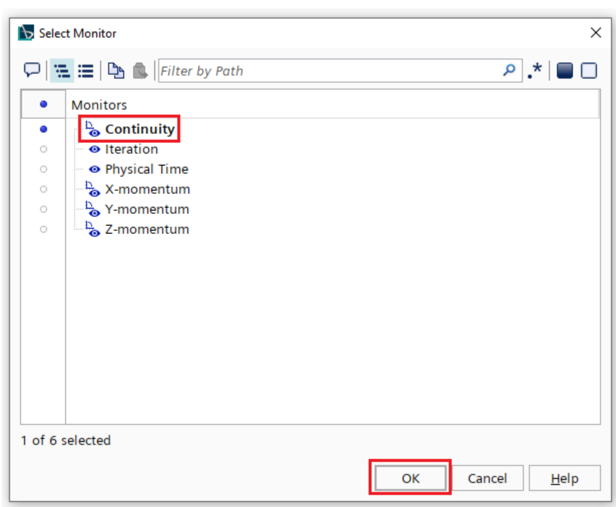
3. Select the Maximum Steps node and change the Maximum Steps property to 6000. Leave the Logical Rule property as Or.



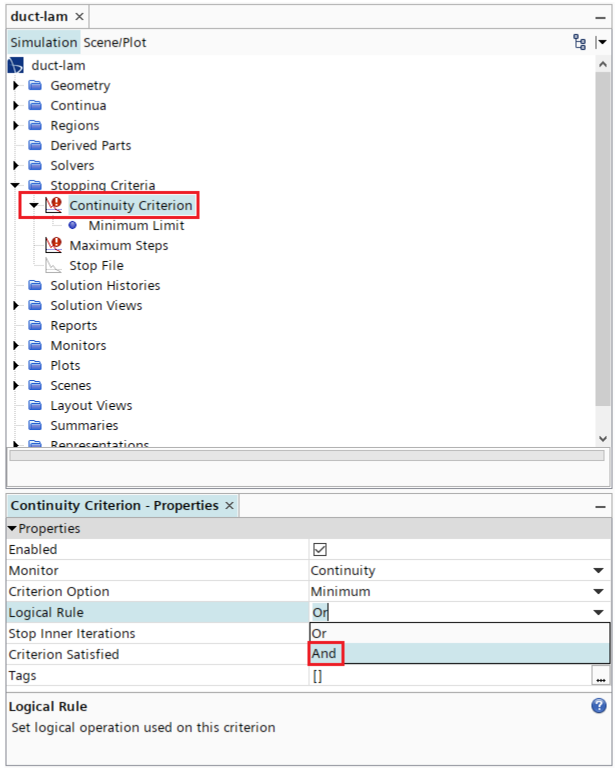
4. Create the stopping criterion for the continuity equation residual:
   4.1. Right-click the Stopping Criteria node and select New Monitor Criterion.



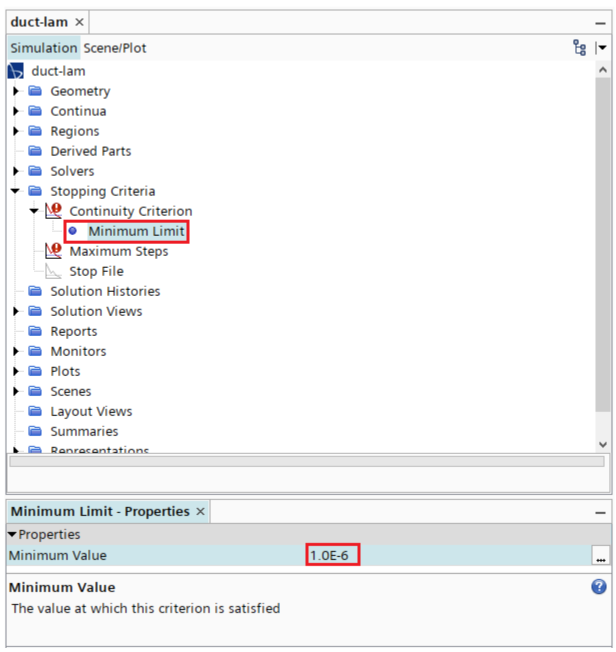
   4.2. In the *Select Monitor* dialog that appears, select the Continuity residual. Click OK to close the dialog box.



   4.3. Select the newly created Stopping Criteria > Continuity Criterion and change the Logical Rule property to And.



4.4. Select the Minimum Limit and set the Minimum Value to 1.0E-06.



5. Create stopping criteria for the X-, Y-, and Z-momentum residuals in a similar fashion, with the Logical Rule set to And and the Minimum Value set to 1.0E-6.

3 Running the Simulation

Note that if there are any monitoring values or plots that you wish to observe during the simulation, they should be set up before the simulation is run. For this case, post-processing will be demonstrated after the run.

1. In the toolbar, click the (Initialize Solution) or use the Solution → Initialize menu item to manually initialize the solution. If your had existing visualizations, you could use them to examine the initial conditions at this point.

2. Run the simulation by clicking (Run), pressing `<Ctrl><R>`, or use the Solution → Run menu item. You may also use the (Step) to step through the iterations one by one.

3. A residual plot will automatically be created and you may watch it as the solution progresses to convergence.

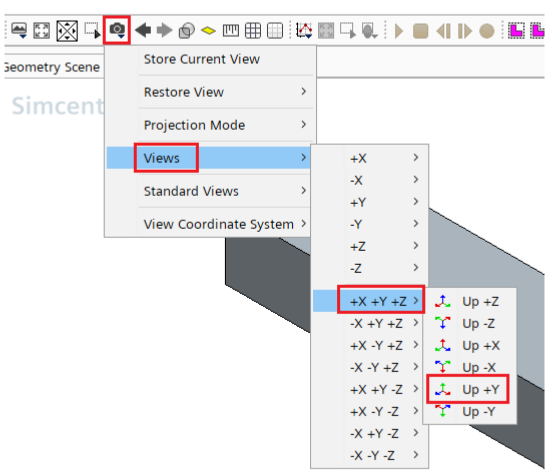
4. After the solution has converged, try clearing it using the Solution → Clear Solution... menu item. Then run the solution again, stopping it midway with the (Stop) button on the toolbar. Press (Run) again and notice that the solution continues iterating from where it was stopped.

4 Post-Processing the Simulation in STAR-CCM+

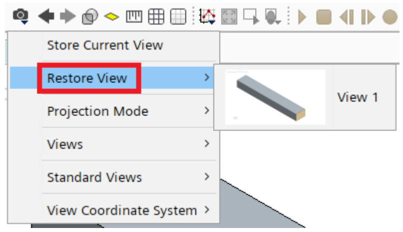
If existing simulation surfaces are not sufficient to use in plotting, visualizing, or reporting data, Derived Parts are created. Here, we will create a line probe, which is used to sample the velocity field at cells along the Outlet boundary. A visual for the velocity field along the Symmetry Plane will also be created.

1. Create the line probe derived part:

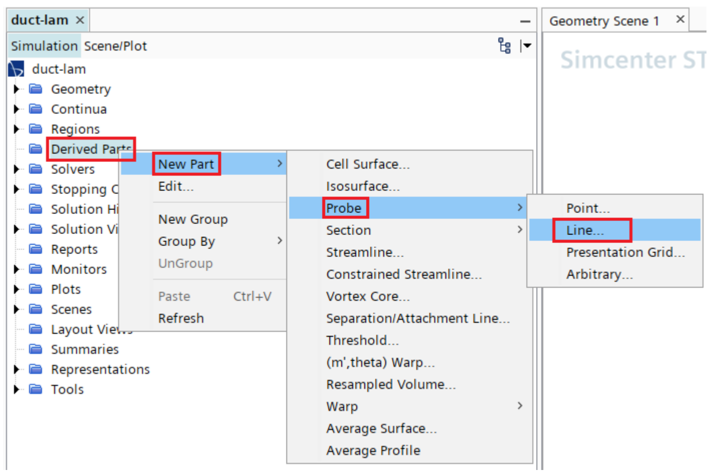
   1.1. Open the Geometry Scene 1. We must change the view to properly visualize the line probe – we will use this opportunity to find out how to achieve specific views within the *Graphics Window* and save them for later use. Select the (Save-Restore-Select Views) button in the toolbar, and navigate to Views → +X +Y +Z → Up +Y.



1.2. Save the view by using the Store Current View option under . This view can now be accessed at any time through → Restore View → View 1.

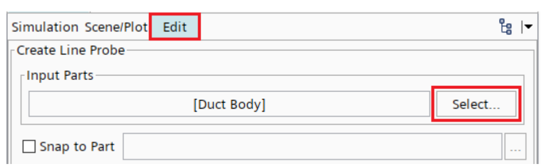


1.3. Right-click on the Derived Parts node and select New Part → Probe → Line...



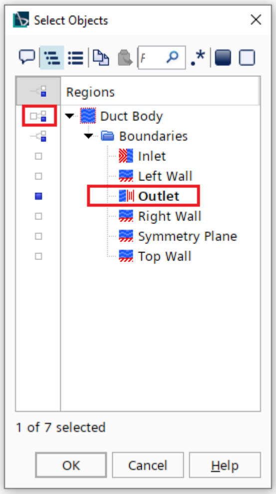
This should open an Edit window automatically. If it does not, with the mesh or geometry scene open in the *Graphics Window*, right-click the Outlet Velocity node and select Edit Part in Current Scene.

1.4. Click the Select... button to select Input Parts for the line probe.

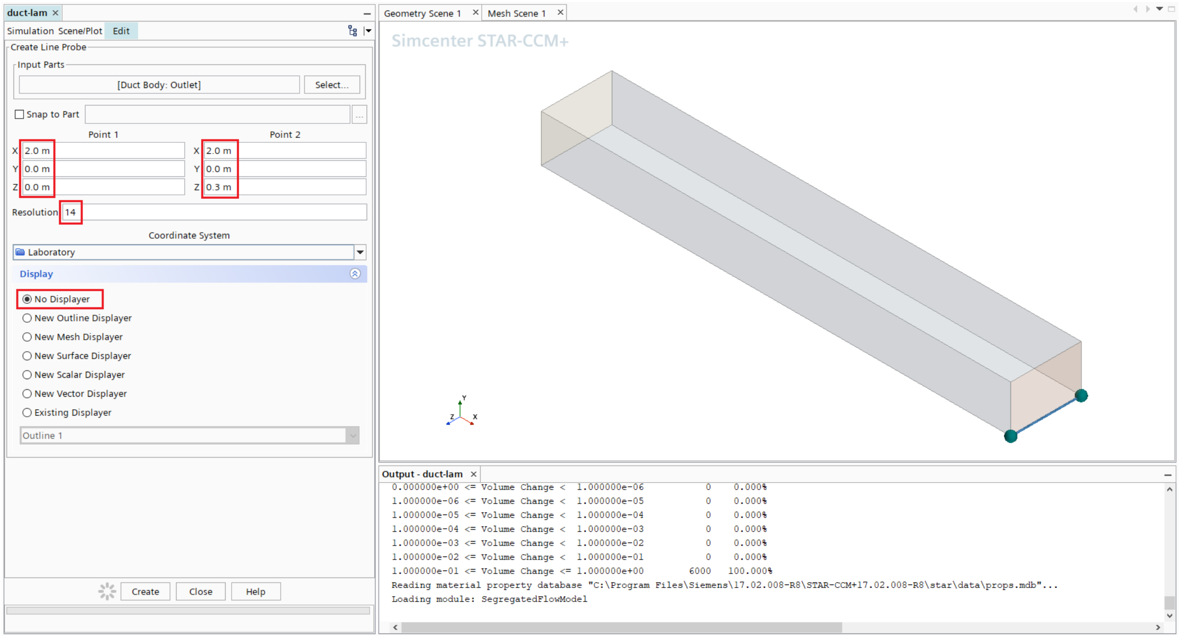


This will open the Select Objects dialog box.

1.5. De-select the Duct Body region by clicking twice on the side of the item-tree. Then, select just the Outlet boundary.



1.6. Enter (2.0, 0.0, 0.0) m and (2.0, 0.0, 0.3) m respectively for Point 1 and Point 2. Set the Resolution to 14 – this is the number of points the line probe will sample. Since there are only 14 cells in the *z-direction*, having greater than 14 points makes little sense. Set the Display to the No Displayer radio button. The line you are describing should be visible in the *Graphics Window* as a barbell.



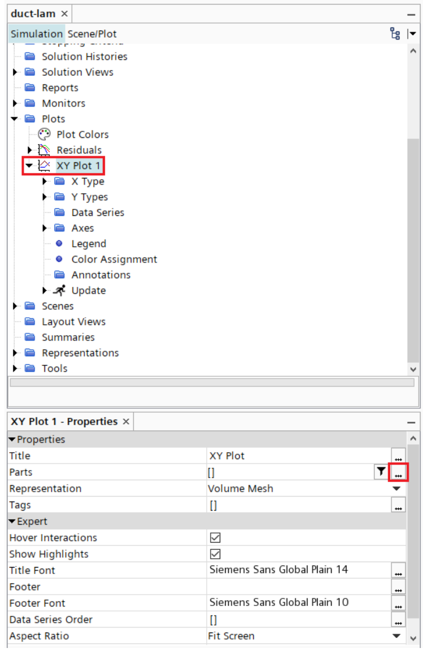
1.7. Click Create at the bottom of the Edit window, and then click Close.

1.8. Rename the newly created Line Probe derived part by right-clicking and selecting Rename, or selecting it and pressing <F2>. Rename the part to 'Outlet Line'.

2. Create an XY Plot to visualize the velocity field at the outlet:

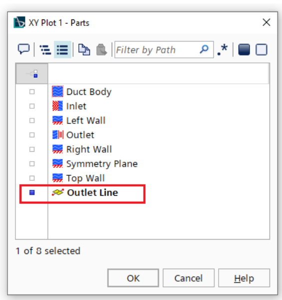
2.1. Right-click the Plots node and select New Plot → XY Plot The newly created XY Plot 1 node should open and expand automatically when it is created, and a new window should appear in the *Graphics Window* titled 'XY Plot 1'.

2.2. With the XY Plot 1 node selected, open the Parts property selection by clicking the [...] (Property Customizer) button.

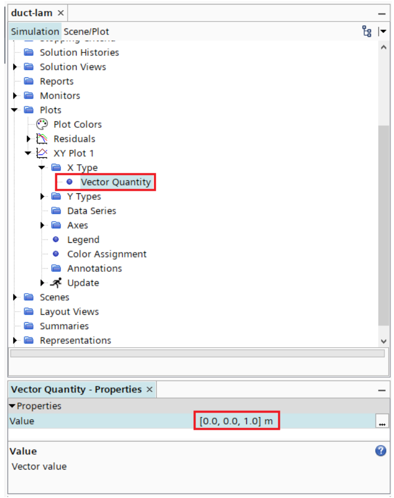


The *XY Plot 1 - Parts* dialog box will open.

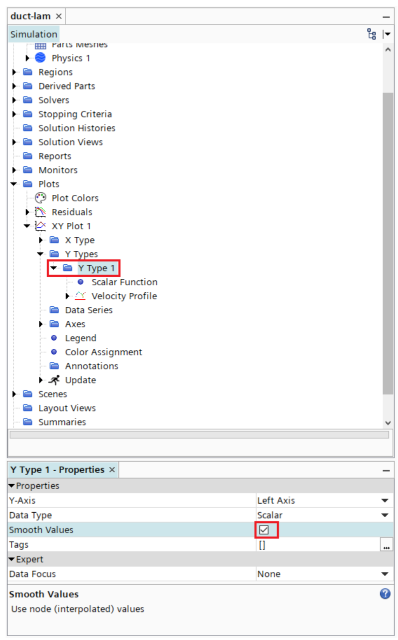
2.3. Select the Outlet Line part and click Close to exit the dialog box.



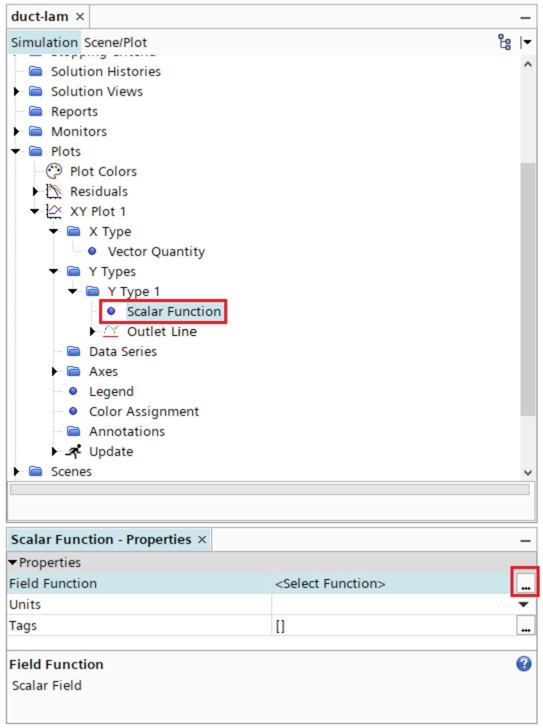
2.4. Expand the XY Plot 1 → X Type node and select the Vector Quantity node. Change the Value property to [0,0,1] m (this is the direction of the line probe).



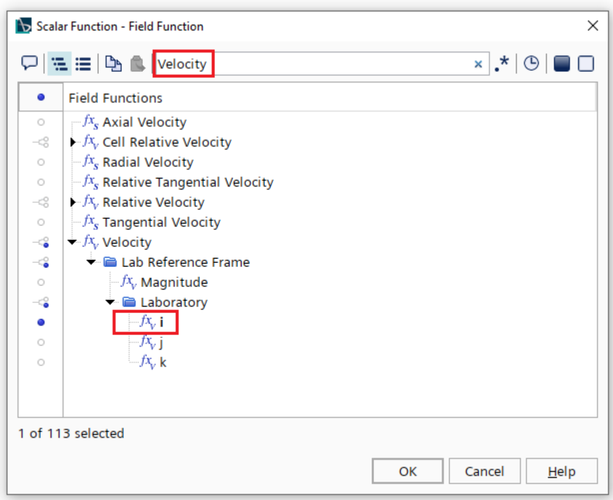
2.5. Select the Y Types → Y Type 1 node and engage the Smooth Values option. This activates solution interpolation, and is an extremely important step to generally include when using XY Plots.



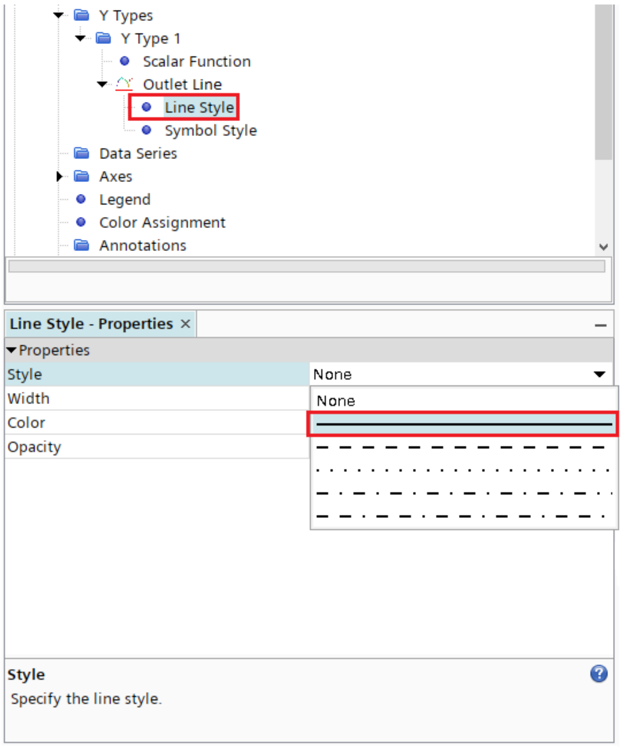
2.6. Expand the Y Types → Y Type 1 node and select the Scalar Function node. In the *Field Function* property, click [...] to open the Scalar Function - Field Function dialog box. This provides all of the scalar field functions available to be plotted.

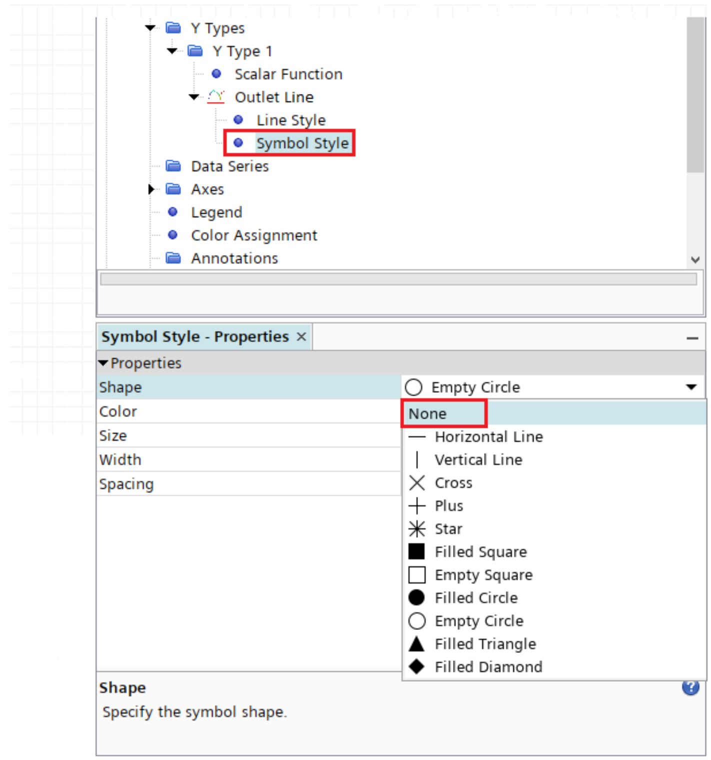


2.7. In the *Scalar Function - Field Function* dialog box, select Velocity → Lab Reference Frame → Laboratory → i. This selection is made easier by searching for 'Velocity' in the search bar.



2.8. Expand the Y Type 1 → Velocity Outlet node and select Line Style. Change the Style property from None to Solid. Similarly, select Symbol Style and change the Style property from Empty Circle to None.





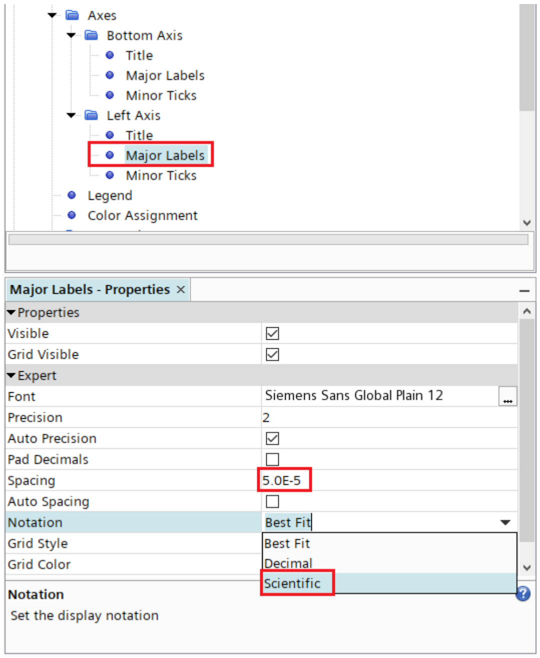
2.9. Select the Axes → Bottom Axis node. Change the Minimum property to 0.0 and the Maximum property to 0.3, to reflect the z-extent of the domain.

2.10. Expand the Bottom Axis node and select the Title node. Set the Title property to 'Z [m]'.

2.11. Select the Axes → Bottom Axis → Major Labels node and change the Spacing to 0.05.

2.12. Expand the Axes → Left Axis node. Change the Title property to 'U Velocity [m/s]' in the Title node.

2.13. Open the Left Axis → Major Labels node and change the Spacing property to 5.0E-5. Also change the Notation property to scientific.



The plot should appear as below:

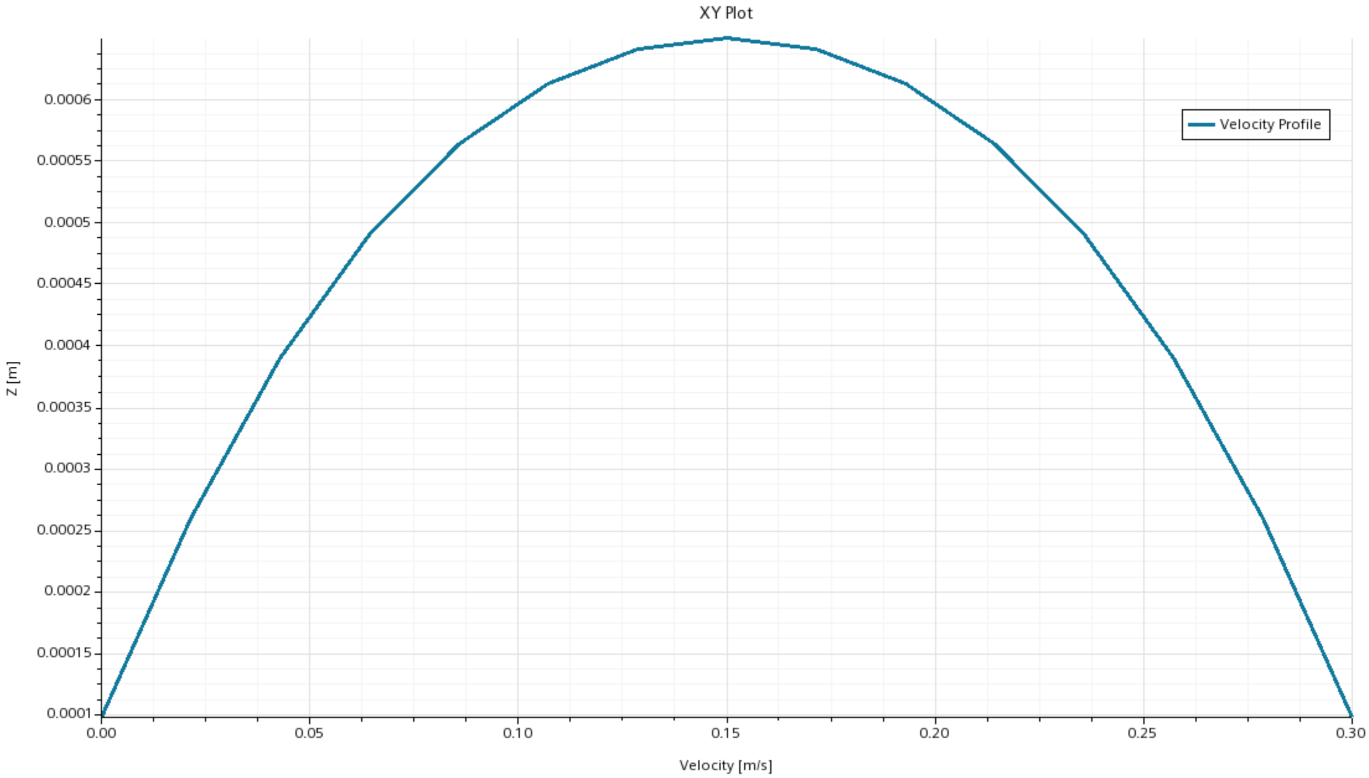


Figure 2: U-velocity profile at the outlet.

2.14. Export the U-velocity profile data as a `*.csv` for post-processing in another program (e.g. Gnuplot): right click anywhere in the XY Plot 1 scene, and select Export.... Enter an appropriate name and location in the *Save* dialog.

3. Create a vector scene of the velocity field on the symmetry plane:
   3.1. Right-click on Scenes and select New Scene → Vector. A vector scene should appear in the *Graphics Window*.
   3.2. Expand the newly created Vector Scene 1 → Vector 1 node and select Parts using [...].
   3.3. In the *Parts* dialog box, select the Symmetry Plane region and press OK.
   3.4. Select the Vector 1 → Vector Field node and change the Function property to Velocity.
   3.5. Select the Vector 1 → Color Bar node and change the Color Map to purple-red basic (large difference) to better visualize the velocity.
   3.6. Select the Vector 1 → Glyph > Relative Length node and change the Glyph Length (%) to 2.5 to better visualize the velocity.
   The scene should appear as below:

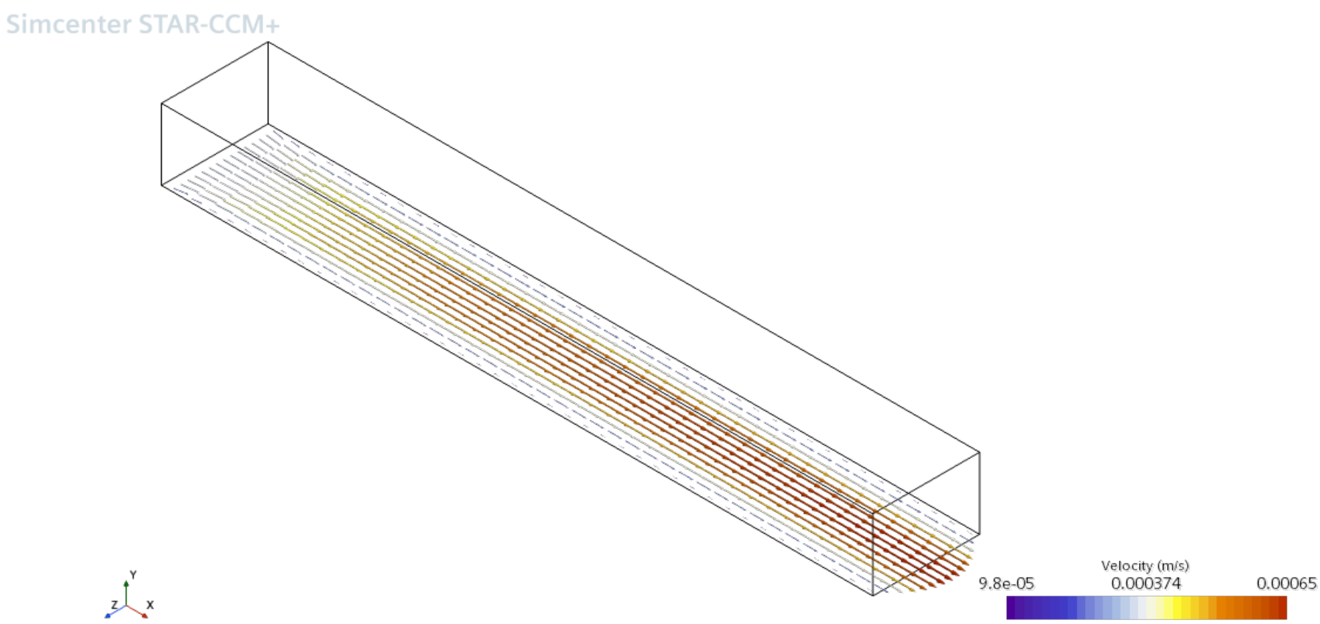


Figure 3: Velocity field at the symmetry plane.

   3.7. Save the vector scene as an image: right-click anywhere in the Vector Scene 1, and select Hardcopy... Enter an appropriate name and location in the *Save* dialog that appears. Note that a variety of image file types are available, and the resolution of the image may be adjusted.

4. Clear the solution, then initialize and re-run or step through it with Outlet Velocity Profile or Vector Scene 1 open – watch the velocity field change as the solution converges.

5 Further Exploration

In order to gain more experience using STAR-CCM+, you can complete the following additional tasks.

1. Restart the flow calculation and converge to a tighter tolerance.
   1.1. Load the `duct-lam.sim` file.
   1.2. Export the outlet velocity profile for post-processing in an external software by right-clicking on XY Plot 1 and selecting Export... and then save as `duct-lam-1E-06.csv`.
   1.3. Open the stopping criteria and change:
      - Maximum iterations to 500
      - Minimum limit for all residuals to 1.0E-8.
   1.4. Create a new dataset by expanding the Tools node and right-clicking on Tables, then selecting New Table... → File Table and open the `duct-lam-1E-06.csv` file.
   1.5. Open the XY Plot 1 node and right-click on Data Series, then select Add Data. In the *Add Data Providers to Plot* dialog box, select the duct-lam-1E-06 dataset and press OK. Open XY Plot 1 in the *Graphics Window* to view the results.

2. Add the energy equation calculation and thermal boundary conditions.
   2.1. Load the `duct-lam.sim` file.
   2.2. Right-click on the Continua → Physics 1 node and press Select Models.... Select the Segregated Fluid Temperature model from the list of Optional Models.
   2.3. Open the Continua → Physics 1 > Models → Liquid → H20 → Material Properties and note that Specific Heat and Thermal Conductivity were added. Leave the default values.
   2.4. Also notice that Static Temperature has been added to the Initial Conditions and that Maximum and Minimum Allowable Temperature have been added to the Reference Values. Leave the default values.
   2.5. Expand the Inlet boundary node and then open the Physics Values → Total Temperature node. Verify the Value property is set to 300 K.
   2.6. For each of the Left, Top, and Right Wall boundaries, change the Physics Values → Thermal Specification from Adiabatic to Heat Flux. Set the Physics Values → Heat Flux to a constant value of 666.67 W/m^2 for each wall.
   2.7. Add a new Stopping Criterion for the energy conservation equation as done for the other conservation equations.
   2.8. Clear and re-run the simulation.
   2.9. Create a contour plot of the temperature field at the outlet face: Create a new Scalar Scene. Set the Parts for both the Outline 1 and Scalar 1 displayers to [Duct Body: Outlet]. Set the Scalar Field to the Temperature field function.
   2.10. Create a new plot of the temperature profile at the outlet using the existing line probe (created for the velocity profile).
   2.11. Create a new line probe that goes down the center of the duct, and then create a chart that plots temperature along this line.