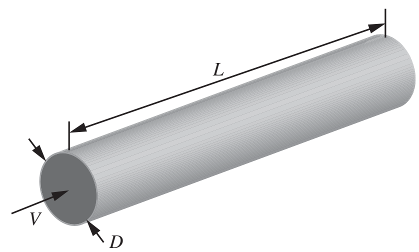
Pipe Flow Tutorial for STAR-CCM+

ME 448/548 Gerald Recktenwald
January 24, 2022 gerry@pdx.edu

1 Overview



This tutorial provides a recipe for simulating laminar flow in a pipe with STAR-CCM+. The focus in on the procedure for creating a model and visualizing the results. The flow field itself is trivial. The schematic to the right depicts flow through a pipe of diameter D and length L, with an average inlet velocity of V.

The flow is two-dimensional and axisymmetric. We will solve this problem with a three-dimensional model even though it is computational wasteful. The goal of this tutorial is to create a simple end-to-end recipe for laminar flows in three-dimensional geometries. In other tutorials we will explore how to exploit symmetry and how to reduce a three-dimensional mesh to a two-dimensional mesh for even greater savings in computational effort.

1.1 Modeling Procedure

The main steps in obtaining a solution are

1. Build a geometric model of the fluid domain.
2. Assign fluid properties to the material in the domain.
3. Assign boundary conditions.
4. Generate a mesh.
5. Set parameters that control the solution.
6. Run the solver.
7. View the results.

1.2 Model Parameters and Boundary conditions

The pipe flow model is specified by defining the geometry D and L and the inlet velocity V. The fluid is air, and STAR-CCM+ provides the density ρ and viscosity μ from its internal database of thermophysical properties. The location of the inlet and outlet boundaries *and* a choice of models at those boundaries must be specified.

The inlet boundary condition can be specified as uniform velocity or prescribed mass flow rate[1]. We will choose a Reynolds number, Re_D, so that the flow is laminar. We then choose a pipe diameter that is convenient, and from Re_D and pipe diameter compute the average velocity.

[1]A velocity profile could also be specified at the inlet, but the procedure is a bit more involved than the two simple boundary condition treatments discusses in this tutorial.

Let the tube have a diameter of 3 cm. The density and viscosity of air at 15 °C are

$$\rho = 1.23\ \frac{\mathrm{kg}}{\mathrm{m}^3}, \qquad \mu = 1.79 \times 10^{-5}\ \frac{\mathrm{N}\cdot\mathrm{s}}{\mathrm{m}^2} = 1.79 \times 10^{-5}\ \frac{\mathrm{kg}}{\mathrm{m}\cdot\mathrm{s}}$$

For laminar flow, choose $\mathrm{Re}_D = 500$

$$\mathrm{Re}_D = \frac{\rho V D}{\mu} \quad \Longrightarrow \quad V = \frac{\mu}{\rho D}\mathrm{Re}_D = \frac{1.79 \times 10^{-5}\ \frac{\mathrm{kg}}{\mathrm{m}\cdot\mathrm{s}}}{\left(1.23\ \frac{\mathrm{kg}}{\mathrm{m}^3}\right)(0.03\,\mathrm{m})}\,500 = 0.24\ \frac{\mathrm{m}}{\mathrm{s}}$$

The mass flow rate is

$$\dot{m} = \rho V A = \left(1.23\ \frac{\mathrm{kg}}{\mathrm{m}^3}\right)\left(0.2425\ \frac{\mathrm{m}}{\mathrm{s}}\right)\left(\frac{\pi}{4}(0.03\ \mathrm{m})^2\right) = 2.11 \times 10^{-4}\ \frac{\mathrm{kg}}{\mathrm{s}}.$$

The outlet boundary condition can be specified as a *pressure outlet* or a *mass flow split* condition. For this tutorial, we will use a pressure outlet. The choice of outlet boundary condition has little influence on the simulation results for the simple pipe flow problem.

Although the type of outlet boundary condition is not important for this problem, we need to consider *where* to locate the outlet boundary. In other words, how long should the pipe be? White[2] gives the following estimate for the entrance length for laminar fully-developed flow in a pipe

$$\frac{L_e}{D} \approx 0.06\mathrm{Re}_D$$

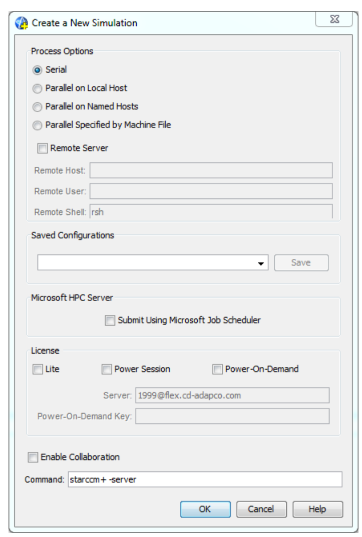
For $\mathrm{Re}_D = 500$ and $D = 3\,\mathrm{cm}$, White's formula gives $L_3 \sim 15D = 45\,\mathrm{cm}$. The purpose of this demonstration is not to simulate the developing flow. However, using L_e as a reference value allows us to choose a realistic length for the domain. We will choose $L = 45$ cm.

2 Start a New Simulation

Create a new *Simulation Model*

1. Launch STAR-CCM+
2. From the File menu, select New Simulation.
3. In the dialog box, make sure the Serial button (in upper left corner) is selected.
4. Check the Power-On-Demand box, enter the *Power-On-Demand Key*, and click OK.

The result is a new, blank simulation window, as shown in Figure 1.



[2] Frank M. White, *Fluid Mechanics*, sixth ed., 2008, McGraw-Hill, p. 347

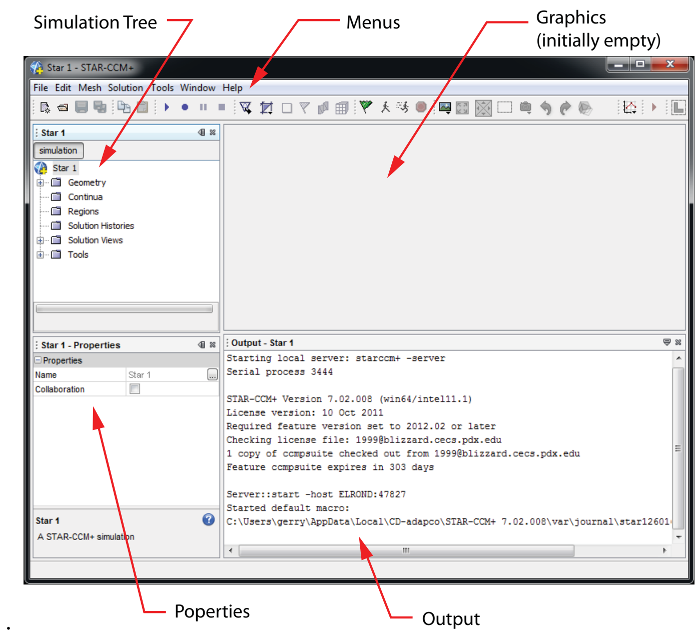


Figure 1: STAR-CCM+ user interface appearance after creating a new simulation.

2.1 Orientation to the User Interface

STAR-CCM+ has a complex user interface that is more similar to a CAD tool than a typical business application. Figure 1 shows the state of the interface at the start of a new simulation.

The STAR-CCM+ interface is divided into four primary panes[3]. The upper left pane contains the *Simulation Tree*, which appears as a list of icons that look like folders in a file browser. Each of the folder icons is a node in a hierarchy of model features and parameters. The Simulation Tree is manipulated by expanding nodes with **+** sign to the left of the node, or right-clicking on the node names to reveal a pop-up menu. During model creation, additional nodes appear when features that are selected by the user require additional parameters to be specified.

Below the Simulation Tree, in the lower left corner of the user interface, is the *Properties Pane*. When you select a node in the Simulation Tree, parameter choices or numerical values associated with that node are revealed in the Properties Pane. Thus, an important mode of setting simulation parameters is to select a node in the Simulation Tree, and then choose the value of a corresponding parameter in the Properties Pane.

The upper right corner of the user interface is the *Graphics Pane*, which is empty at the start of a simulation. In creating a model and solving for the flow field, you will create *Scenes*, which appear as tabbed sub-panes of the Graphics Pane. For example, a *Geometry Scene* is used to display and manipulate manipulate three-dimensional images of the domain geometry and mesh during the problem specification phase of the simulation. Other Scenes in

[3] Refer to the *Using the Workspace* section of the User Guide for more information.

the Graphics Pane are used to monitor convergence of the solution, and display field variables (velocity, pressure, turbulence quantities, etc.) after a solution is obtained.

The *Output Pane* in the lower right quadrant is used by STAR-CCM+ to display messages for the user. During meshing and during iterations toward convergence, large amounts of text messages will scroll by in the Output Pane. Although this information seems superfluous at first, the Output Pane can reveal important aspects of the simulation status or numerical values important for interpreting the results of a simulation.

For more information on the STAR-CCM+ user interface, refer to *Using the Star-CCM+ Workspace*, in the on-line help: Select Help from the menus at the top of the window.

3 Build the geometry of the model

The geometry of simple flow models can be created with a 3D-CAD tool that is built into STAR-CCM+. We will use the 3D-CAD tool to create a cylindrical object, and then incorporate that object as the fluid domain for the pipe flow simulation.

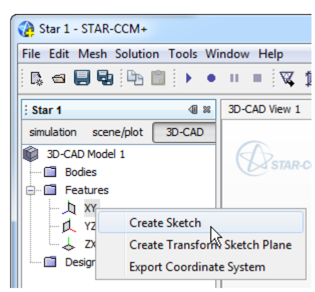
CFD simulations require specifying the geometry and boundary conditions on a fluid volume. Thus, unlike a typical CAD tool for drawing physical parts, the STAR-CCM+ interface is designed to ultimately create volumes occupied by fluid, not volumes occupied by solid parts. Although we are creating a model of flow in a pipe, we will not be drawing the pipe. Rather, we will be drawing a cylindrical plug that corresponds to the fluid *inside* the pipe.

3.1 Open the CAD tool and set the grid scale

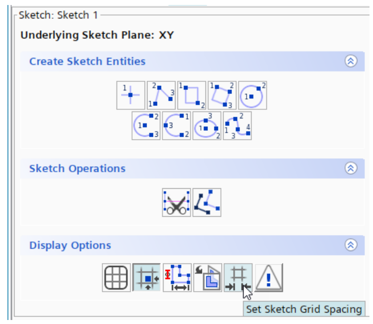
1. Right click on the plus sign (+) next to the Geometry node in the Simulation Tree.

2. Right click on the 3D-CAD Models node and select New from the pop-up menu. The Simulation Tree pane is replaced by a CAD Tree pane.

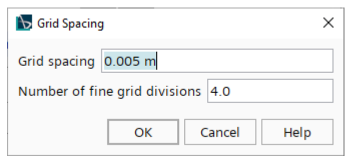
3. Right-click on the XY plane icon under the Features node, and select Create Sketch from the pop-up menu. The CAD Tree pane is replaced by a pane with two sub-panels, each one highlighted by a light blue box. The upper sub-panel labeled *Create Sketch Entities* contains icons for primitive sketching operations. The lower sub-panel labeled *Display Options* contains icons for grid spacing, snap-to-grid and other options. Note that the icons in both panels contain tool-tips that are displayed when the mouse pointer hovers above the icon.



4. Before drawing any features, set the grid scale for the CAD tool. Click on the icon in the lower right corner of the *Display Options* panel.



5. Set the grid spacing to 0.005 m



Note that all physical quantities are specified with dimensions. Thus, the input to the grid spacing dialog box could be `0.005 m`, `0.5 cm`, or `5 mm`. Only SI and metric units are accepted. If the units are not specified, the value is assumed to be in the m-kg-s system.

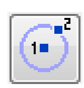
3.2 Sketch a circle and extrude it to create a cylinder

.

During the following steps, it may be helpful to zoom in or rotate the geometric model in the Graphics pane. Refer to the instructions in the box to the right for an introduction to the mouse movements that manipulate models in the graphics pane.

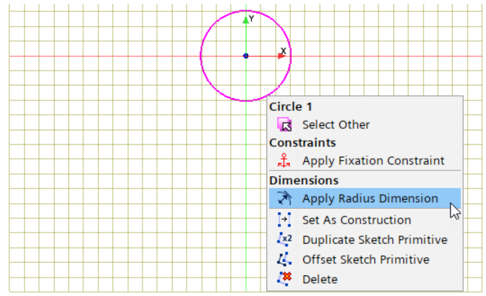
Mouse input to the CAD Tool

- Drag and rotate an object with the left mouse button
- Zoom in with the middle mouse button or scroll wheel
- Pan with the right mouse button

1. Select the *Create circle* from center tool. 

2. Left-click at the origin of the XY plane and then right-click and drag to enlarge the circle until its radius is 0.015 m. *Hint*: dragging mouse pointer along a grid line takes advantage of the snap-to-grid feature, making it easy to specify the radius.

3. Right-click on the edge of the circle and select Apply Radius Dimension.
   Enter 1.5 cm in the dialog box for the cylinder radius.



4. Click OK in the lower left corner of the CAD panel. *Don't skip this step!* The result is a new entity, Sketch 1, in the CAD tree.
5. Right-click on Sketch 1 node in the CAD tree, and select Create Extrude from the pop-up menu. A dialog box appears as shown in the left half of Figure 2.
6. Enter 0.45m for the Distance parameter, and click the OK button in the bottom left corner of the CAD pane.

The preceding steps create a three-dimensional cylinder in the Graphics Pane. You will likely need to manipulate the image with the mouse to make the cylinder visible. The default viewing position is so close to the origin that the cylinder does not fit into the viewport of the Graphics pane. For example, to make the cylinder visible, try zooming away by using the middle mouse button or the scroll wheel, and rotating the image with the left mouse button.

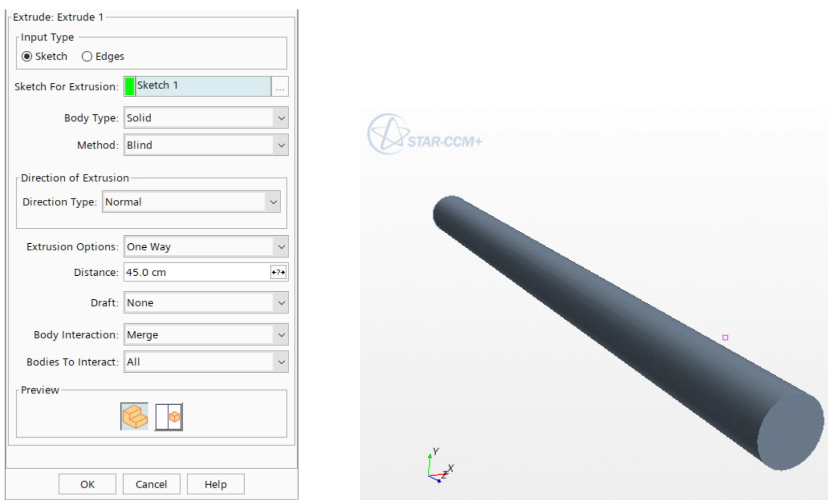


Figure 2: Dialog box for setting the extrusion distance (left). Extruded cylinder (right).

3.3 Save the Model

Select Save from the file menu, and save the `.sim` file to the hard drive of your computer. It's a good idea at this point to create a *directory* for the simulation results as well as a meaningful file name. As you build the model and run different cases, you will accumulate alternate `.sim` files for the same physical problem. You will also generate external graphics files to be incorporated into reports. Therefore, a little thought at this point will help you stay organized later.

3.4 Label the surfaces

Labeling surfaces in the 3D-CAD tool will make it easier to identify boundaries of the Regions later in the model setup. It is possible to label the surfaces later using tools in the simulation mode.

Note: The steps to label the surface geometry require that you are in working in the 3D-CAD mode of STAR-CCM+. If you are in the Simulation mode, you will see the Simulation Tree for the model as in the screen shot below right. Select the 3D-CAD mode by clicking the `3D-CAD` button as shown in the screen shot below left.

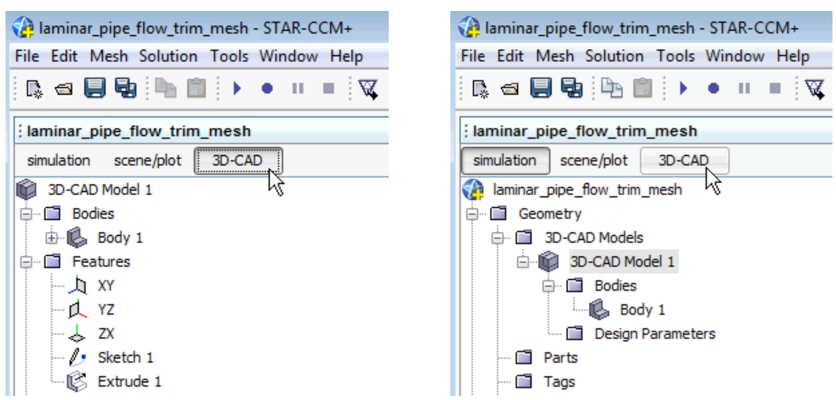


Figure 3: Toggling between the 3D CAD mode and the Simulation mode.

Label the inlet, outlet and pipe wall surfaces with the following steps. Make sure you are in the 3D-CAD mode as described in Figure 3.

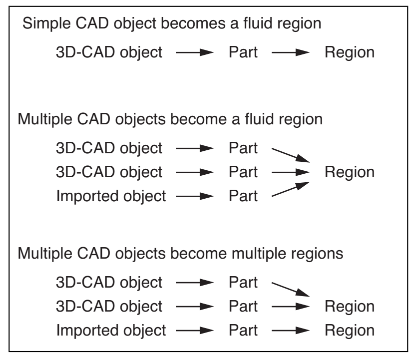
1. Rotate the model so that the inlet (or outlet) end is visible. We will define the inlet as the (x, y) plane containing $(x, y, z) = (0, 0, 0)$.
2. Right-click on the face, and select Rename from the pop-up menu.
3. Change the name of the surface to inlet (or outlet).
4. Repeat the preceding steps to label the outlet (or inlet) and the pipe walls.
5. Close the 3D-CAD model, and return to the Simulation Pane.

Notice that the view of the part shows up as a Geometry Scene.

It's possible that the geometry of the model will not be visible. In that case you will either need to open and select an existing Geometry Scene, or create a new Geometry Scene with the following steps.

1. Right click on Scenes at the top level of the simulation tree and select *New→Geometry Scene*.
2. Expand the Geometry Scene node and select the Parts corresponding to the CAD model you just created.

3.5 Create a Geometry Part from the CAD Model



The 3D-CAD model is part of the current STAR-CCM+ model file (the `.sim` file), but as a CAD model it is not directly used in a CFD simulation. All or part of a 3D-CAD model is a basis for a STAR-CCM+ *Part*, and the Part is used to create a Region. These steps are represented in the first row of the schematic to the right. This three-step procedure may seem like a lot of unnecessary work just to build a model of flow in a pipe. The complexity arises from the way that STAR-CCM+ manages definitions of the geometries used in flow simulations.

The complexity of the steps to define a fluid region for a simple model becomes an important asset when working with geometrically complex models imported from CAD tools. As depicted by the other examples in the schematic above and to the right, multiple CAD objects (which may be imported from other CAD tools) can be used to create separate *Parts*, which can be combined into a single *Region*. Alternatively, multiple CAD objects can be used to create separate *Parts*, which can be used to create multiple *Regions*.

3.5.1 Procedure to Create a Geometry Part

Use these steps to create a geometry *Part* from the 3D-CAD model. Make sure you are in the Simulation mode as described in Figure 3.

1. Expand the Bodies node.
2. Select Body 1 (or whatever you named the pipe object)
3. Right click and select New Geometry Part

You can rename the 3D CAD model (or not). I suggest `Pipe_fluid` or similar name for the part. Notice that a new Part is created under the Parts node. Also note that the names assigned in the CAD tool (`Inlet`, `Outlet` and `Walls`) have been propagated to the nodes of the Parts tree.

3.6 Save the Model

Select Save from the file menu.

4 Create a Region that contains the pipe object

The physical problem being modeled is specified in three distinct and independent categories.

- Regions
- Physics Continuum
- Mesh Pipeline

A *region* defines the topological relationship between material in the simulation, i.e., it defines how the geometric entities (volumes or areas and their boundaries) in the model are connected to each other. Boundary conditions are defined on the region and the boundary conditions are independent of the mesh, the kind of simulation (e.g., laminar or turbulent flow) or the thermophysical fluid properties.

Each region has a *Physics Continuum* that specifies the physical behaviors of the material in that region, e.g., fluid or solid, compressible or incompressible, liquid or gas. The Physics Continuum also specifies other factors that determine the physical behavior such as steady or transient as well as the solution methods used.

Mesh Pipeline that creates of a surface and volume mesh on each part. Once a model is specified and solved, modifying the mesh is an important part of ongoing model development. The mesh pipeline associates the mesh with individual parts that could be joined into a single region or maintained in different regions.

The three categories (Region, Physics and Mesh) influence each other. For example, wall boundaries and inlet boundaries influence the interior mesh differently. However, the three categories are sufficiently distinct that you can make major changes in on category, e.g., changing a boundary condition from constant wall temperature to adiabatic, without influencing the other categories. This flexibility comes at a cost of greater complexity in setting up the model.

From the Star-CCM+ User's Guide[4]:

> "Regions are volume domains (or areas in a two-dimensional case) in space that are completely surrounded by boundaries. They are not necessarily contiguous, and are discretized by a conformal mesh consisting of connected faces, cells and vertices."

Translation: A region is a volume of material with the same physical properties and the same meshing model. For the pipe flow tutorial, there is only one Region. The boundaries of the region are used to impose boundary conditions on the model

Regions are distinct. Information (e.g. mass flow or heat transfer) between regions is only shared when the regions are explicitly joined by an interface. For the pipe flow tutorial, there is only one region and it is the volume occupied by the fluid inside the pipe.

[4] See "*What are Regions?*" in the StarCCM+ User Guide

4.1 Create the Region

For this model there is only one region. The region is created from a part that was created with the built-in CAD tool

$$\text{CAD Model} \longrightarrow \text{Part} \longrightarrow \text{Region}$$

1. Right click on the Region node and select New
2. Select the Geometric Parts for the region
   a. In the Properties pane, click in the area to the right of Parts
   b. Expand the Parts node in the pop-up window
   c. Select the Pipe Fluid region (or Body 1 if you didn't name the Part)
   d. Click OK

Notice that the there is one boundary called "*Default*" in the list of Boundaries. Expand the Boundaries node if you do not see the Default boundary.

4.2 Assign the Types of Boundary Conditions

There is only one Boundary called "Default". We will create two new boundary surfaces – the inlet and the outlet – and we will rename the remaining part of the boundary.

4.2.1 Assign the Inlet Boundary

1. Right click on Boundaries and select New.
2. Right click on the newly created boundary and select Rename... (from the bottom of the menu).
3. Enter "inlet" and click OK.
4. Click on the [...] button in the *Part Surfaces* item in the *Properties Pane*.
5. Expand the nodes and click on the inlet – See Figure 4
6. Click OK
7. Set the boundary type to inlet with a prescribed velocity – See Figure 6
   a. Select the Type characteristic in the inlet property pane
   b. Select Velocity Inlet from the pop-up menu

The *value* of the inlet velocity cannot be assigned until the fluid continua is defined in a later step in this tutorial.

4.2.2 Assign the Outlet Boundary

Repeat the steps used to create the inlet boundary. Instruction steps are abbreviated.

1. Create a new boundary and rename it "outlet". New.
2. Assign the *outlet part* to the *outlet boundary* using the *Part Surfaces* item in the *Properties Pane*.
3. Set the Type to *Pressure* outlet.

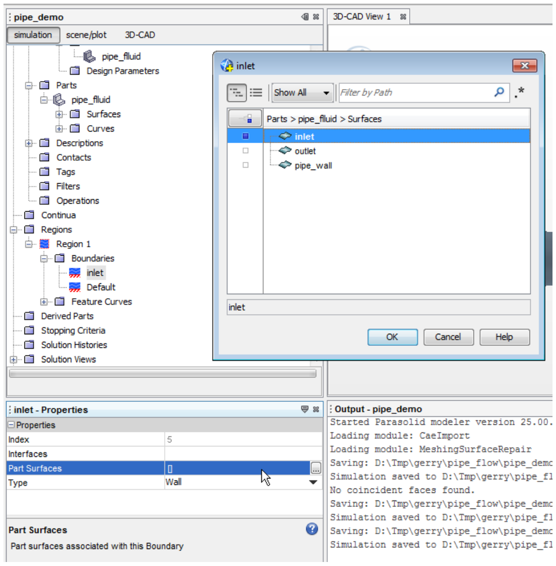


Figure 4: Assigning the inlet surface patch to the inlet boundary of the fluid region

4.2.3 Assign the Wall Boundary

The default condition is that all surfaces are solid, adiabatic walls. Therefore, no additional steps need to be taken. However, it is good practice to assign a meaningful name to each boundary.

1. Right click on the *Default* boundary in the list of boundaries.
2. Select *Rename...* and enter "`pipe_wall`".

4.3 Save the Model

Select Save from the file menu.

5 Create a Physics Continua

The Physics Continua defines the physical behavior of the material in the region. It is also used to specify the type of global solution algorithm can be used to solve the flow field. Different solution methods may be required as more (or less) complicated physics are included in the model. Refer to *Modeling Physics* section of the User Guide.

1. Right click on the *Continua* node and choose New→Physics Continuum.
2. Right click on the newly created *Physics 1* node and choose Select models...
3. In the Model Selection dialog box, make the following choices
   - Three Dimensional

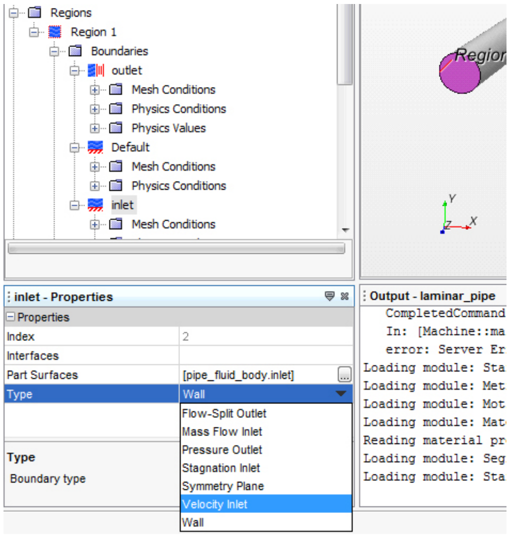


Figure 5: Specifying the type of inlet boundary condition.

- Gas
- Segregated flow
- Constant Density
- Steady
- Laminar

4. Click "Close"

6 Assign the Boundary Values

The type of boundary condition for each boundary surface was prescribed in § 4.2. After the fluid continua is selected, the values of boundary values can be specified. For the pipe flow problem, the boundary values are (1) no slip at the wall, (2) velocity at the inlet, and (3) pressure at the outlet.

The no-slip condition is the default condition for all surfaces. Therefore, there is no need to change the boundary value on the pipe wall.

1. Expand the Physics Values node and select Constant
2. Set the Value to 0.24 m/s

7 Create a Mesh Pipeline

In this tutorial we will first use the polyhedral mesher with prism layers, which is the mesh type commonly used for complex geometries. Although the polyhedral mesher is easy to use, for this problem it creates a mesh that is topologically more complex than necessary for the simple pie flow. In follow-up exercises, we will explore different meshing models. Our immediate goal is to walk through

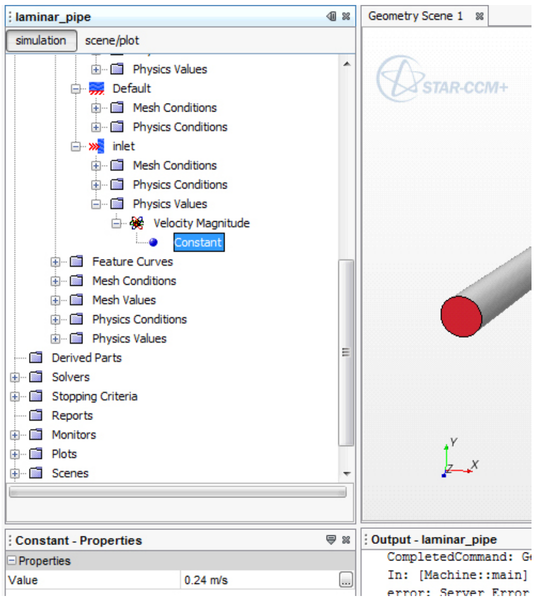


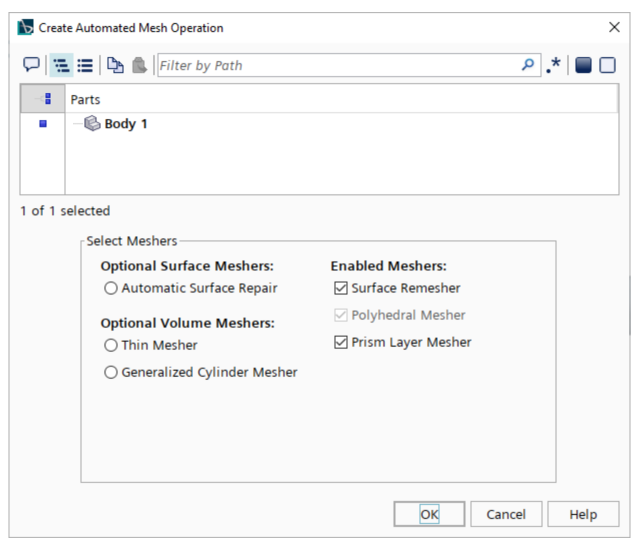
Figure 6: Specifying the magnitude of the inlet velocity.

the basic CFD analysis procedure from start to finish, not to use the optimal mesh.

It's a good idea to define the regions before defining the mesh because the type of boundary (not the boundary values) influences the allowable types of mesh adjacent to the boundary.

7.1 Create an Automated Mesh

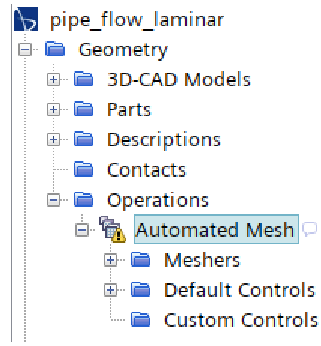
The mesh pipeline is associated with the geometry.



1. Expand the Geometry node at the top of the simulation tree
2. Right-click on Operations and from the pop-up menu select New→Mesh→Automated Mesh
3. In the dialog box that opens, select Body 1 in the Parts panel. Then in the lower half of the dialog box
    a. Select Surface Remesher
    b. Select Polyhedral Mesher
    c. Select Prism Layer Mesher

The completed dialog box appears to the right. Click OK

7.2 Specify the mesh parameters



Creating a good mesh is a very important part of a CFD simulation. We will pick a simple starting point for the mesh parameters and return to the mesh after obtaining the first simulation results.

Note that there is a yellow triangle with an exclamation point next to the Automated Mesh entry in the simulation tree. That indicates that the mesh pipeline has not been executed. We will first adjust the mesh parameters before executing the pipeline.

1. Expand the Default Controls node under the Automated Mesh node
2. Click on the Base Size node and enter 0.5 cm for the Base Size in the Properties pane in the lower left corner of the StarCCM+ window.
3. Click on the Number of Prism Layers node and change the default from 2 to 4 in the Properties pane.

7.3 Generate and view the mesh

1. Right-click on the Automated Mesh node under the Operations Node and select Execute
2. Scroll down the simulation tree, right click on the Scenes node and select New Scene→Mesh. A mesh scene should appear in the graphics pane.
3. Rotate the model in the mesh scene to inspect it. The view normal to the outlet should look like Figure 7.

8 Solve the flow field

Now that the physical domain and boundary conditions (region), fluid physics (physics continuum) and mesh are defined, we can tell StarCCM+ to solve the numerical model. For complex problems, the solution may not converge, or the solution may yield results that are not sufficiently resolved. For those reasons and others, we may try a solution and decide to modify the definitions of the regions, physics continuum or mesh continuum. In other words, for most problems of interest, we will need to iterate through a range of adjustments until the solution is acceptable. The pipe flow problem is very simple and as long as the preceding steps were completed correctly, the solution should be obtained relatively easily and quickly.

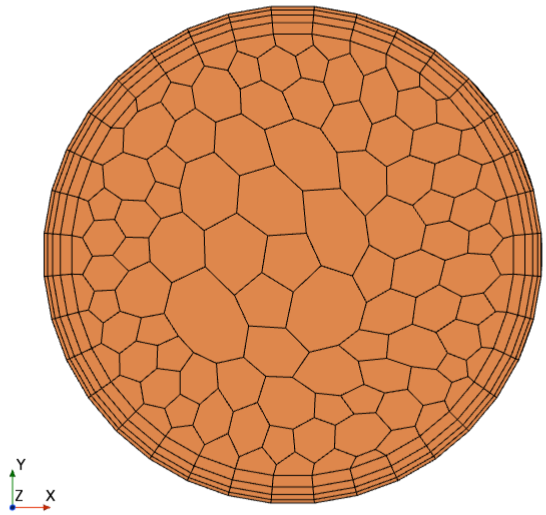


Figure 7: Mesh on outlet surface that is created with a base size of 0.5 cm and four prism layers. All other mesh parameters are set to the default values.

8.1 Adjust the stopping criteria for the simulation

The default setting is to have the solver complete 1000 iterations. For the pipe flow problem, that is more than necessary. Instead, we'll start with 100 iterations.

1. Open the Stopping Criteria Node
2. Change the Maximum Steps property to 100

The iterations should be allowed to continue until the residuals have been reduced below a tolerance. To automate the stopping, create a stopping criterion based on the residuals of the continuity equation.

1. Right-click on the Stopping Criteria node
2. Select New Monitor Criterion...
3. Click on the dot to the left of Continuity in the dialog box
4. Click OK
5. Select the Minimum Limit node under the newly created Continuity Criterion node
6. Notice that the Minimum Value in the Properties pane is `1.0E-4`. That is a good value for the stopping criterion. It means that the solution iterations will stop when the residuals of the continuity equation fall below 10^{-4}.

The default is that any of the Stopping Criteria can stop the iterations. You can inspect the relationship between the Stopping Criteria as follows.

1. Click on the Continuity Criterion node.

2. Notice that the Logical Rule property in the Properties pane is `Or`. That means the when the Continuity Criterion is satisfied it is combined with other Stopping Criterion with a logical "or" rule.

3. Click on the Maximum Steps node.

4. Notice that the Logical Rule property in the Properties pane is `Or`. That means the when the Maximum Steps is satisfied it is combined with other Stopping Criterion with a logical "or" rule.

Therefore, since both the Continuity Criterion and Maximum Steps are combined with a logical "or", the simulation stops when either one is satisfied.

8.2 Prepare to monitor the solution

It's possible to begin the solution immediately, but it is often helpful to have a way to visualize the solution as it is happening. We'll do that by creating a display of the fluid pressure.

8.3 Open a Scalar Scene

1. Right-click the Scene node and select New Scenes→Scalar
2. Open the Displayers
3. Select the *Scalar Field* node of the Scalar 1 node
4. In the Properties panel, click on the ¡Select Function¿ pop-up and (scroll down to) select Pressure.

8.4 Create a reports to monitor the pressure

In addition to plots, we may want to extract numerical quantities from the solution. Numerical results, like overall pressure drop, or total shear stress on a surface, are obtained by creating *reports*.

1. Right-click Reports and select New Report→User→Maximum
2. Click on the Maximum 1 node that was just created
3. Rename the report to *Maximum Pressure*
4. Click on the Scalar Property and select Pressure from the pop-up menu for scalar Properties
5. Click on the `[ ]` value for the Parts Property (or click the [...] icon.)
6. A dialog box opens

   a. Select the *Region 1*

   b. Click OK

7. Right-click on Reports and select New Report→Minimum

8. Repeat the steps for the maximum pressure report. This time the output of the report will be the minimum pressure.

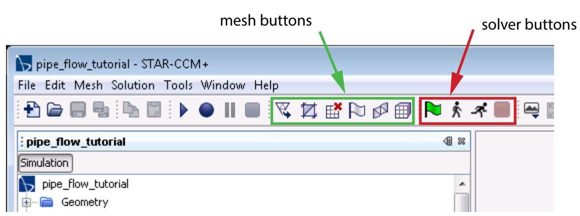
After a report is created, it must be executed by right-clicking on the report node and selecting `Run Report`. We will do that after the end of the simulation run. However, it is also helpful to create a plot that monitors the values from the report.

1. Select both the Minimum pressure and Maximum pressure nodes under the Reports node. (These are the reports that were just created in the preceding step.)

2. Right-click on the report selection (both reports) and select Create Monitor and Plot from Report. A dialog box pops up. Select Single Plot.

8.5 Run the simulation

Click the run button in the row of menu icons at the top of the screen. The run button is the second from the end of the solver buttons.



8.6 Residual Plot

While the simulation is running, inspect the residual plot. Figure 8 show the residual plot at the end of 200 iterations. As a solution progresses the residual plot is one of the best indicators of whether the solution converges. At convergence, the residuals are several orders of magnitude smaller than 1.

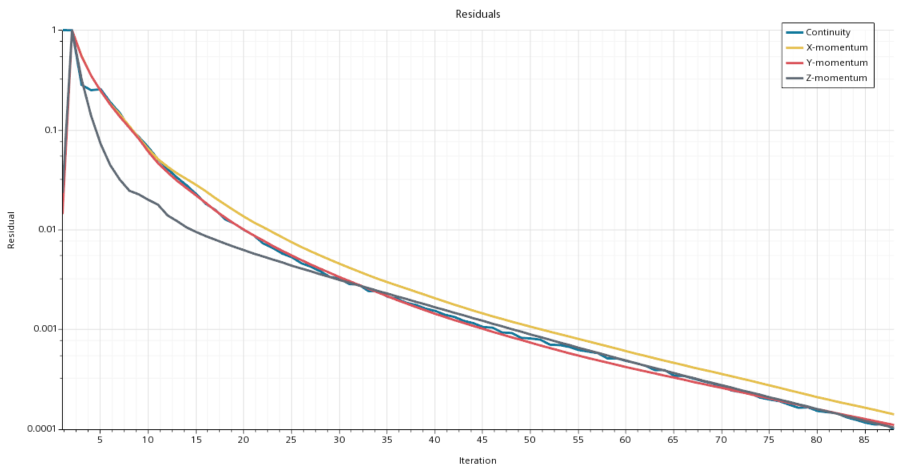


Figure 8: Residual plot

9 Inspect the solution

9.1 Monitor plot of maximum and minimum pressure

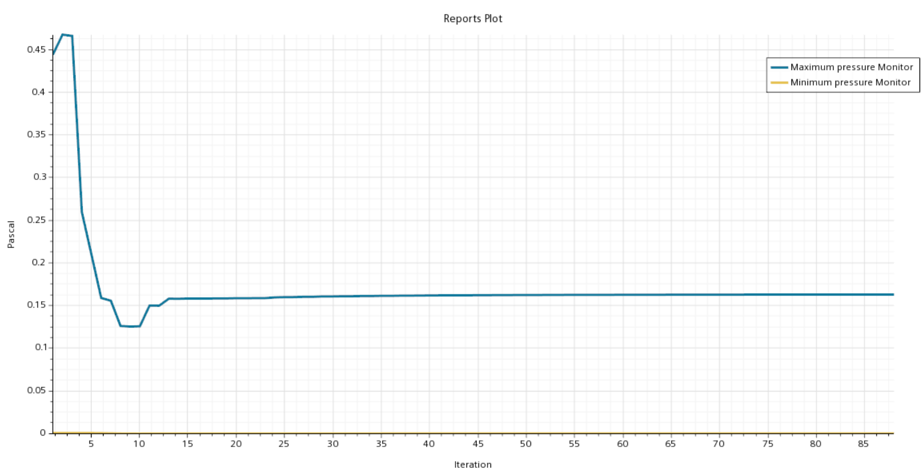


Figure 9: Maximum and minimum pressure values in the domain during iterations toward convergence

.

9.2 Scalar plot – pressure on pipe wall

9.3 Velocity Vector Plot

By default, scalars and vectors are only viewable on the surfaces of the fluid Regions. To display the velocity vectors inside the domain, we first need to

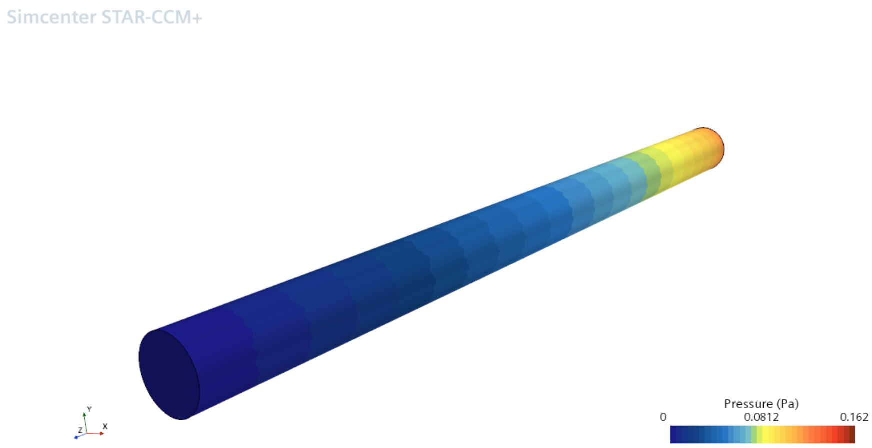


Figure 10: Pressure on the surface of the fluid inside the pipe.

create a surface on which the vectors are displayed. The simplest surface is a plane aligned with one of the coordinate axes.

1. Click on the Derived Parts node and Select New→Plane Section.
2. In the *New Section* dialog box, leave defaults for *Plane Parameters* (or adjust as necessary to put the plane through the center)
3. Scenes→New Scene→Geometry

With a new (derived) part ready to be selected, create the velocity field plot as a *Vector Displayer*

1. Right click Scenes node and select New→Vector Displayer
2. Click create

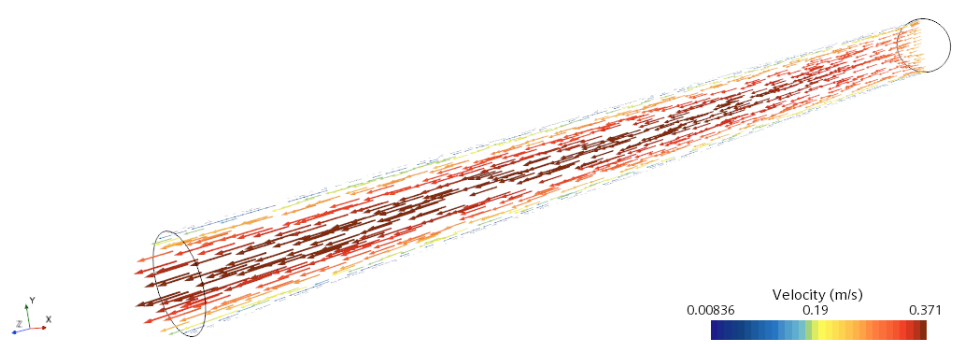


Figure 11: Velocity Vectors