1 Overview

This tutorial provides a recipe for simulating laminar flow in a pipe with STAR-CCM+. The focus is 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 $L$. 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 $\rho$ and viscosity $\mu$ 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. The value of inlet velocity or inlet mass flow rate are chosen so that the flow in the duct is laminar. We will choose a Reynolds number, $Re_D$, and pipe diameter, and then compute the average velocity consistent with $Re_D$.

---

1A 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
\[ \rho = 1.23 \text{ kg/m}^3, \quad \mu = 1.79 \times 10^{-5} \text{ kg/m/s} = 1.79 \times 10^{-5} \text{ kg/m/s} = 1.79 \times 10^{-5} \text{ N·s/m}^2 \]

For laminar flow, choose \( \text{Re}_D = 500 \)
\[ \text{Re}_D = \frac{\rho V D}{\mu} \quad \Rightarrow \quad V = \frac{\mu}{\rho D} \text{Re}_D = \left( \frac{1.79 \times 10^{-5}}{1.23 \text{ kg/m}^3} \right) \left( 0.03 \text{ m} \right) \left( 0.24 \text{ m/s} \right) = 0.24 \text{ m/s} \]

The mass flow rate is
\[ \dot{m} = \rho V A = \left( 1.23 \text{ kg/m}^3 \right) \left( 0.2425 \text{ m/s} \right) \left( \frac{\pi}{4} \left( 0.03 \text{ m} \right)^2 \right) = 2.11 \times 10^{-4} \text{ kg/s} \]

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
\[ \frac{L_e}{D} \approx 0.06 \text{Re}_D \]

For \( \text{Re}_D = 500 \) and \( D = 3 \text{ cm} \), White’s formula gives \( L_e \sim 15D = 45 \text{ cm} \). 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 \( L = 45 \text{ cm} \).

## 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, and click OK.

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

---

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. 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 clicking on nodes to reveal subnodes. One also uses a right-click to expand a pop-up menu for a node. 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

---

The STAR-CCM+ user guide refers to these panes as windows. Refer to the What is the Client Workspace? in the Using STAR-CCM+ section of the User Guide.
BUILD THE GEOMETRY OF THE MODEL

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.

![CAD tool interface](image)

5. Set the grid spacing to 0.005 m

![Grid spacing dialog box](image)

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

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

![Create circle tool](image)

2. Click at the origin of the XY plane 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. 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.

4. 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.

5. Enter 0.45m for the *Distance* parameter, and click the OK button in the bottom left corner of the CAD pane.

![Mouse input to the CAD Tool](image)

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. Refer to the short list of hints in the box above and to the right. 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.

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.
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.
3 BUILD THE GEOMETRY OF THE MODEL

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.

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 create a new Geometry Scene.

1. Right click on Scenes at the top level of the simulation tree and select New $\rightarrow$ 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.

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.
4 CREATE A REGION THAT CONTAINS THE PIPE OBJECT

Alternatively, multiple CAD objects can be used to create separate Parts, which can be used to create multiple Regions.

3.5.1 Procedure

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 may be specified in three distinct and independent categories.

- Regions
- Mesh Continuum
- Physics Continuum

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 Mesh Continuum that consists of a surface and volume mesh. Each region also has a Physics Continuum that specifies the physical behaviors of the material, 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.

The three categories (Region, Mesh and Physics) 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 one 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

4See “What are Regions?” in the StarCCM+ User Guide
“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 this problem, there is only one region and it is the volume occupied by the fluid inside the pipe.

Before defining the fluid properties or the mesh, we will create the region and name the surfaces on the boundaries. Think of regions as defining the mathematical properties of the solution – both the physical geometry and the boundary conditions – and not the artifacts of the solution such as the mesh or turbulence model.

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

\[
\text{CAD Model} \rightarrow \text{Part} \rightarrow \text{Region}
\]

1. Right click on the Region node and select \textit{New}

2. Select the Geometric Parts for the region
   a. In the \textit{Properties pane}, click in the area to the right of \textit{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 \textit{OK}

Notice that there is one boundary called “\textit{Default}” in the list of Boundaries. Expand the Boundaries node if you do not see the Default boundary.

4.2 Assign the Boundaries of the region

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 \textit{New}.

2. Right click on the newly created boundary and select \textit{Rename}… (from the bottom of the menu).

3. Enter “inlet” and click \textit{OK}.
4 CREATE A REGION THAT CONTAINS THE PIPE OBJECT

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

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.

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.
4 CREATE A REGION THAT CONTAINS THE PIPE OBJECT

4.3 Save the Model

Select Save from the file menu.
Figure 6: Specifying the magnitude of the inlet velocity.
5 Create a Mesh Continuum

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 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.

5.1 Specify the mesh type

1. Right click on the Continua node and choose New→Mesh Continuum.
2. Right click on the newly created Mesh 1 node and choose Select Meshing Models...
3. In the Model Selection dialog box, make the following choices
   - Surface Remesher
   - Polyhedral Mesher
   - Prism Layer Mesher
4. Click “Close”
5. Set the Base size
   a. Open the Reference Value node under the Mesh 1 node
   b. Select the Base Size node
   c. In the Properties Pane, set the base size value to 0.005 m

5.2 Generate the mesh

1. Set the Base size
   a. Open the Reference Value node under the Mesh 1 node
   b. Select the Base Size node
   c. In the Properties Pane, set the base size value to 0.005 m
2. Click on the Generate Solid Mesh icon
3. Open a Mesh Scene to view the mesh
   a. Right click on the Scenes node at the top level of the Simulation tree and select New Scene→Mesh
   b. Expand the newly created Mesh Scene 1 node to inspect properties
   c. Rotate and zoom in the new graphics pane for Mesh Scene 1
6  Create a Physics Continua

The Physics Continua defines the physical behavior of the material in the region. It also specifies which type of global solution algorithm is used to solve the flow field. 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
   - Steady
   - Gas
   - Segregated flow
   - Constant Density
   - Laminar
4. Click “Close”

7  Solve the flow field

7.1 Adjust the stopping criteria for the simulation
1. Open the Stopping Criteria Node
2. Change the Maximum Steps property to 100

7.2 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.

7.3 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.
7.4 Create a reports to monitor the pressure
1. Right-click Reports and select New Report→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.

7.5 Make the Monitor Plot visible
Select the Plots→Maximum Pressure Plot node in the Simulation Tree

7.6 Run the simulation
Click the
8 Inspect the solution

8.1 Residual Plot

Figure 7: Residual plot

8.2 Monitor Plot – Maximum Pressure

Figure 8: Maximum pressure value in the domain during iterations toward convergence
8.3 Scalar plot – pressure on pipe wall

1. Select Scalar Scene 1

2. Open the Displayers node and select the Scalar 1 node

3. Expand the Parts node
   a. Make the Pipe wall visible
   b. In the Parts property, click on the [ ] value
   c. Select wall and click the > icon to move the wall from the Select From list to the Selected list
   d. Click close

4. Select Scalar Field node in the Scalar 1 scene. The result should be a shaded surface plot of the pressure on the pipe wall.
8.4 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 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

![Diagram of velocity vectors](image)

**Figure 9:** Velocity Vectors

8.4.1 Modify vector length

1. Under the Geometry Scene 2 node
2. Open the Displayers
3. Select Section Vectors
4. Select Relative Length
5. In the properties pane, set the Glyph Length (%) to a new value, say 2
8.5 Plots to add

1. Velocity Profile plot
2. Pressure along pipe axis

8.6 Exporting plots and data for use in model documentation

Show how to export the profile data
9 Future Updates to the Tutorial

9.1 Modify the model and rerun it

Change the inlet BC from uniform velocity to a mass flow rate

10 Explore alternative mesh models

1. Use a polyhedral mesh without the prism layers
   a. Change the mesh models to de-select the prism layer
   b. Delete the current mesh and create a new volume mesh
   c. Re-run the analysis
2. Use a trimmed mesh
3. Use a generalized cylinder mesh

11 Engineering Analysis

The images created in post-processing with STAR-CCM+ usually need to be incorporated into a report or used in a presentation. It is important to know how to display the CFD simulation results in a way that contributes to engineering analysis. The three-dimensional velocity vector plots and surface contour plots are impressive and useful for obtaining a qualitative picture of the flow field. However, engineering analysis requires quantitative information such as pressure drop or total heat transfer rate, or the net aerodynamic force on an object.

11.1 To do

1. Add steps to the tutorial on how to extract engineering data
2. Write up a separate, short paper that shows these engineering results properly formatted and discussed
3. Interpret the engineering analysis write-up in terms of good practice for documentation and in terms of expectations for homework.

Plot velocity profile at axial locations.

1. Show how to plot velocity profiles in StarCCM+
2. Show how to export the data and create the plot in MATLAB

Plot pressure along the centerline.

1. Show how to plot pressure along the centerline in StarCCM+
2. Show how to export the data and create the plot in MATLAB

Discuss whether the flow is fully-developed.

1. Velocity profiles are not invariant along the length of the pipe
2. How does the velocity profile at the exit compare to the model of fully developed laminar flow (parabolic profile)?

3. Pressure drop becomes linear almost immediately after the entrance (?

Questions for further model development

1. How might mesh be refined to improve the result?

2. Does the domain need to be lengthened?

12 Homework

1. Refine the mesh

2. Add prism layers

3. Use cylindrical mesh elements

Does the inlet boundary condition affect the result? Compare the solution results for uniform inlet velocity versus total mass flow rate at the inlet.

Compare quantitative measures:

1. Pressure gradient at the inlet. Extract pressure gradient as a numerical value (line fit?)

2. pressure distribution over the inlet \( p = f(r) \)

Compare numerical friction factor to theoretical friction factor

1. Rerun the model for different inlet velocities

2. Tabulate and plot \( \frac{dp}{dx} \) versus \( \text{mdot} \)

3. Compute \( f_{\text{Re}} \) from \( \frac{dp}{dx} \) assuming that the flow is fully-developed

4. Repeat with a 2D model