Steady Flow: Laminar and Turbulent in an S-Bend

This tutorial demonstrates the flow of an incompressible gas through an s-bend of constant diameter (2 cm), for both laminar and turbulent flow. The first part of the tutorial will cover the laminar flow, with a Reynolds number (Re) of 500, and the second part will cover the turbulent flow, Re = 50,000. The same pipe geometry will be used in both cases.

The geometry will be created using 3D-CAD, which is the parametric solid modeler available within STAR-CCM+. Following this, the necessary region will be created and a polyhedral mesh will be obtained using the generalized cylinder meshing model. This model extrudes polyhedral cells in the direction of the flow, which results in a mesh suited for pipe flows.

Prerequisites

To complete this tutorial, you will need to be familiar with the following techniques:

<table>
<thead>
<tr>
<th>Technique(s)</th>
<th>Associated Tutorial</th>
</tr>
</thead>
<tbody>
<tr>
<td>The STAR-CCM+ workflow</td>
<td>Introduction to STAR-CCM+</td>
</tr>
</tbody>
</table>
Creating the S-Bend Geometry

- **Start up STAR-CCM+** in a manner that is appropriate to your working environment and create a *New Simulation*.
- **Save the new simulation** to disk with the file name `sBend.sim`

The geometry will be created using 3D-CAD, which is the parametric solid modeler available within STAR-CCM+.
- To activate 3D-CAD, right-click on the **Geometry > 3D-CAD Models** node and select **New**.

This creates a new model and activates 3D-CAD. Start by renaming the 3D-CAD model.
- Select the **3D-CAD Model 1** node at the top of the object tree. **Rename** the model to **S-Bend**.

Creating the Geometry

The geometry of an s-bend pipe will be created. A circle representing the inlet face will be created first.
- Create a new sketch on the YZ plane by right-clicking on the **Features > YZ** node and selecting **Create Sketch**.
- Click on the (Set sketch grid spacing) button and change the **Grid Spacing** to **0.0025 m**.
- Click on the (View normal to sketch plane) button to bring the sketch plane into view.
- Use the (Create circle) tool to draw a circle with a radius of **0.01 m**.
and whose center is on the origin, position [0, 0].

- Click OK to exit the sketch.

The profile of the pipe along its length will now be created.
- Create a new sketch on the XY plane.
- Click on the (View normal to sketch plane) button.
- Use the (Create line) tool to draw a line of length 0.035 m starting at
the origin and extending in the positive x direction.

- Press <Esc> when done to exit the line tool.
- Use the (Center point circular arc) tool to draw an arc with a radius of 0.02 m. Three mouse clicks are required:
  a. The first click should be at position [0.035 m, 0.02 m]; this defines the center point.
  b. The second click should be at position [0.035 m, 0.0 m]; this defines the start point.
  c. The third click should be at position [0.055 m, 0.02 m]; this
defines the end point.

A second arc needs to be created to complete the “S” shape.

- Use the \texttt{Center point circular arc} tool to create the second arc. The coordinates for the three clicks are listed below:
  
  a. Center: \([0.075 \ m, 0.02 \ m]\).
  
  b. Start point: \([0.075 \ m, 0.04 \ m]\).
c. End point: [0.055 m, 0.02 m].

Now draw the final section of the pipe.
- Draw a line with start point [0.075 m, 0.04 m] that extends 0.065 m.
in the positive x direction.
The completed sketch is shown below.

- Click OK to exit the sketch.

The solid body of the pipe will be created using the sweep feature. This requires two sketches: one to act as the profile being swept; the other to act as the path for the sweep.

- Select both sketches from the 3D-CAD object tree, holding down the <Shift> key to select multiple items.
• Right-click on one of the highlighted nodes and select Create Sweep.

The **Sweep** dialog will appear.
- Accept the default settings and click OK.

A new body node, **Body 1**, is added below the **Bodies** node in the object tree.
- Select the **Bodies > Body 1** node and rename it to **Fluid**.
• Position the geometry as shown below.

Specifying Inlet and Outlet Faces

The final step in preparing the model geometry is to specify the inlet and outlet faces of the model by setting face names.

• Zoom in to the pipe end, right-click on the circular face and select
Rename, as shown below.

- Enter **Inlet** in the **Rename** dialog and click **OK**.

The benefit of renaming the face in the 3D-CAD model is that it will retain its unique identity when the 3D-CAD model is converted to a geometry part in the next stage of the tutorial. The outlet will now be similarly renamed.

- Position the geometry so that the outlet face on the other end of the pipe is visible.

- Right-click on the circular face at the other end of the pipe, select **Rename**, and enter **Outlet** in the **Rename** dialog. Click **OK** to complete the action.

The geometry is now complete and we can exit 3D-CAD. Note that the **3D-CAD View** scene will be closed automatically upon exiting 3D-CAD.

- Click on the **Close 3D-CAD** button at the bottom of the object tree.

- Save the simulation.

**Creating a Geometry Part**

A new geometry part will be created using the 3D-CAD model.
• Right-click on the Geometry > 3D-CAD Models > S-Bend node and select New Geometry Part.

The Part Creation Options dialog appears.

• Click on OK to accept the default settings.
• Expand the Parts > Fluid > Surfaces node and note that the inlet and outlet faces specified in the previous section are defined as separate
Assigning a Part to a New Region

The geometry part will be assigned to a new region. As the inlet and outlet surfaces have been renamed in 3D-CAD, these will automatically be added to separate boundaries.
• Right-click on the **Parts > Fluid** node and select **Assign Parts to Region**.

![Assign Parts to Region](image)

• In the **Assign parts to region** dialog set:
  - **Region Mode** to **One region per part**
  - **Boundary Mode** to **One boundary per part surface**
• Click Create Regions and then Close.

A new region has been created with boundaries defining the inlet, outlet and main fluid volume.

• Create a new geometry scene. Use the mouse to rotate the model in the
Graphics window.

- Expand the Regions > Fluid > Boundaries node and select each of the boundary nodes to check that they have been specified correctly.

**Setting Up the Case**

In this first part of the tutorial, the case will be run using a steady state laminar flow (Re = 500).

By default all boundaries are set to be walls. We will modify the boundary type for the Inlet and Outlet.
• Select the Regions > Fluid > Boundaries > Inlet node.

• In the Properties window set the Type to Velocity Inlet.

The icon next to the Inlet boundary will change indicating that it is now a velocity inlet boundary.

• Select the Outlet boundary node and in the Properties window, set the Type property to Pressure Outlet.
The modified Boundaries node is shown below.

Generating a Surface Mesh

A polyhedral mesh will be generated using the generalized cylindrical mesher. This meshing model is particularly suited to pipe flow. A prism layer mesh will be defined along the wall boundary, to assure that near wall effects are adequately resolved.

• Right-click on the Continua node and select New > Mesh Continuum.
• Right-click on the Mesh1 > Models node and choose Select Meshing Models...

In the Meshing Model Selection dialog select:
• Surface Remesher from the Surface Mesh box
• Polyhedral Mesher from the Volume Mesh box
• Generalized Cylinder from the Optional Models box
• Prism Layer Mesher from the Optional Models box
• Click Close.

The mesh settings will now be defined.
• Select the Mesh 1 > Reference Values > Base Size node.

• Set the Value property to 0.0008 m.

• Continuing with the reference values section, select the Number of Prism Layers node and set its Number of Prism Layers property to 5.

• Select the Prism Layer Stretching node and set its Prism Layer Stretching property to 1.8.

• Select the Prism Layer Thickness > Relative Size node and set its Percentage of Base property to 45.

The surface mesh will be generated and checked before using the generalized cylinder mesher. A good quality surface mesh is needed before the volume mesh can be generated.

• Click on the (Generate Surface Mesh) button.
• Create a mesh scene and examine the surface mesh.

Generating a Volume Mesh

Before generating the volume mesh, we must define the boundaries that form the cylindrical geometry. This can be done automatically as follows:

• Right-click on the Mesh 1 > Models > Generalized Cylinder and select
Manage Cylinders.

The \textit{Select Cylinders} dialog appears. The default surface of the s-bend was found by the generalized cylinder mesher and is listed in the \textit{Available Cylinders} list.

- Click on the \textgreater\ button to move the \textit{Fluid: Default} cylinder into the \textit{Selected} list and click \textit{Apply} followed by \textit{Close}.
- Navigate to the Regions > Fluid > Boundaries > Default > Mesh Conditions >
Notice that the Extrusion law property was set to Constant in the Properties window.

- Within the same Default boundary, select the Default > Mesh Values > Generalized Cylinder Parameters node.

Notice that the Number of Layers property was set to 82 in the Properties window. This value was calculated by STAR-CCM+ based on the geometry model and sets the number of layers that will be generated along the pipe.

The volume mesh can now be generated.

- Click on the (Generate Volume Mesh) button.
- Right-click on a blank space in Mesh Scene 1 and select Apply Representation > Volume Mesh.
• Examine the final volume mesh.

• Save the simulation

Selecting Physics Models

A physics continuum was automatically created during the mesh generation process. We will now select the physics models for the first part of the tutorial.

• Right-click on the Continua > Physics 1 > Models node and choose Select models...

The Physics Model Selection dialog will guide you through the model selection process. In the Physics Model Selection dialog, select the following models:

• Steady from the Time box.
• Gas from the Material box.
• Segregated Flow from the Flow box.
• Constant Density from the Equation of State box.
- Laminar from the Viscous Regime box.
- Click Close.

The expanded Models node is shown below.

Modifying Material Properties

The Dynamic Viscosity and the Density of air will be modified.
- Within the Physics 1 continuum, select the Models > Gas > Air > Material Properties > Density > Constant node. In the Properties window, set the Value property to 1 Kg/m\(^3\)
- Select the Dynamic Viscosity > Constant node. In the Properties window, set the Value property to 1.716E-5 Pa-s

Setting Initial Conditions and Boundary Settings

To achieve a Reynolds number of 500 for this gas and pipe diameter, we require a mean velocity of 0.429 m/s. To assist solution convergence, the initial condition for velocity will be defined.
- Select the Physics 1 > Initial Conditions > Velocity > Constant node. In the Properties window, set the Value property to [0.429, 0.0, 0.0] m/s

Set the same velocity on the Inlet boundary.
• Select the Regions > Fluid > Boundaries > Inlet > Physics Values > Velocity Magnitude > Constant node. In the Properties window, set the Value property to 0.429 m/s

Preparing a Scalar Scene

A scalar scene will be created to show the velocity magnitude on a section plane through the center of the s-bend. This will be used to visualize the solution while the simulation is running.

• Create a new scalar scene.
• Right-click on the Derived Parts node and select New Part > Section > Plane...
• In the Create Section panel, do the following:
  • Ensure that Fluid is selected in the Input Parts box
  • Set the normal vector to [0, 0, 1] m
  • Under Display, select the Existing Displayer radio button and choose Scalar 1 from the drop-down menu.
The completed dialog is shown below.

- Click Create followed by Close.
Scalar Scene 1 is shown below.

- Click on the scene/plot button above the object tree and select the Displayers > Scalar 1 > Scalar Field node.
- In the Properties window set the Function to Velocity > Magnitude.
- Select the Displayers > Scalar 1 node and set the Contour Style to Smooth Blended.
- Click on the Simulation button to return to the STAR-CCM+ simulation object tree.

A vector plot displaying the velocity magnitude will be created.
- Create a new vector plot.
- Using the drag-and-drop method, add the section plane from the
Setting Up Stopping Criteria

The number of iterations for the solver will be limited to 500. This should be sufficient for the solution to converge

- Select the Stopping Criteria > Maximum Steps node. In the Properties window, set the Maximum Steps property to 500
- Save the simulation

Running the Simulation

The simulation is now ready to be run.

- Click on the Run button.

While the simulation is running you can click on the tabs at the top of the Graphics window to view each of the scalar scenes. The Residuals display will be created automatically and shows the progress of the solvers.
During the run, it is also possible to stop the process by clicking the (Stop) button on the toolbar. If you do halt the simulation, it can be continued again later by clicking the (Run) button. If left alone, the simulation will continue until 500 iterations are complete.

- While the simulation is running, select the Scalar Scene 1 tab at the top of the Graphics window to visualize the solution.
- When the simulation has finished running, click on the (Save) button.

**Visualizing the Results**

The scalar scene is shown below.
The vector scene is shown below.

Changing to a Turbulent Flow

The case will now be edited to have a turbulent flow (Reynolds number = 50,000).

- Save the simulation as sBendTurbulent.sim

Modifying the Physics Continuum

We will edit the physics continuum, disable the laminar model and enable the turbulent and K-epsilon models.

- Right-click on the Continua > Physics 1 > Models node and choose Select models...

In the Physics Model Selection dialog, do the following:

- Clear the checkbox next to the Laminar model in the Enabled Models box.
- Select Turbulent from the Viscous Regime box.
The Reynold-Averaged Navier-Stokes model is automatically selected.

- Select K-Epsilon Turbulence from the Reynolds-Averaged Turbulence box; The Realizable K-Epsilon Two-Layer and Two-Layer All y+ Wall Treatment models will be automatically selected.
- Click Close.

The updated Models node is shown below.

![Diagram of Models node]

**Modifying Initial Conditions and Boundary Settings**

To achieve a turbulent flow ($Re = 50,000$) in the existing pipe, the velocity of the air needs to be increased to 42.9 m/s. This value needs to be changed at the Initial Conditions node and at the Inlet boundary condition. The Turbulence Specification will also be changed so that the turbulence intensity and length scale can be defined.

- Select the Continuum > Physics 1 > Initial Conditions > Velocity > Constant node. In the Properties window, set the Value property to $\begin{bmatrix} 42.9, & 0.0, & 0.0 \end{bmatrix}$ m/s
- Select the Turbulence Specification node and set its Method property to Intensity + Length Scale.

The Turbulent Length Scale node is added to the object tree.
• Select the Turbulent Length Scale > Constant node and set its Value property to 0.0014 m. The turbulent length scale used in this case is approximately 7% of the hydraulic diameter, which for circular pipes is the same as the pipe diameter.

• Select the Turbulent Velocity Scale > Constant node and set its Value property to 4.29 m/s. The turbulent velocity scale used in this case is 10% of the free-stream velocity.

• Select the Turbulence Intensity > Constant node and set its Value property to 0.12. The turbulence intensity in for this case is approximately 12% and was calculated using the following equation:

$$I = \frac{\frac{3}{2} v_r^2}{U^2}$$

where $I$ is the turbulence intensity, $v_r$ is the turbulent velocity scale, and $U$ is the free-stream velocity.

The same values will be used at the Inlet boundary. We will start by setting the Turbulence Specification to use Intensity + Length Scale.

• Select the Regions > Fluid > Boundaries > Inlet > Physics Conditions > Turbulence Specification node. In the Properties window, set the Method property to Intensity + Length Scale.

• Select the Physics Values > Turbulent Length Scale > Constant node and set its Value property to 0.0014 m.

• Select the Regions > Fluid > Boundaries > Inlet > Physics Values > Velocity Magnitude > Constant node. In the Properties window, set the Value property to 42.9 m/s.

• Continue by selecting the Turbulence Intensity > Constant node in the same Physics Values node, and set its Value property to 0.12.

**Extending the Stopping Criteria**

The turbulent flow requires a longer time to converge. Increasing the number of iteration to 1000 should give the solver sufficient time to converge.

• Select the Stopping Criteria > Maximum Steps node. In the Properties window, set the Maximum Steps property to 1000.

• Save the simulation

**Clearing the Solution and Running the Simulation**

The solution from the laminar flow needs to be cleared before running the solver.
• Select Solution > Clear Solution... from the menu.

The Clear Solution dialog will appear.
• Accept the default settings and click OK.

The simulation is now ready to be run.
• Click on the (Run) button.
• While the simulation is running, select the Scalar Scene 1 tab at the top of the Graphics window to visualize the solution.
• When the simulation has finished running, click on the (Save) button.

Visualizing the Results for the Turbulent Flow

The scalar scene is shown below.
The vector scene is shown below.

Suggestions for Further Practice

To see how the laminar and turbulent models differ, you could try running the Re = 50,000 case with the laminar physics model and see what will happen.

- Open the sBend.sim file in a STAR-CCM+ session.
- Increase the flow velocity to 42.9 m/s in the Initial Conditions and at the Inlet boundary.
- Clear the solution and run the case.

Notice how the residuals do not converge, but instead start to oscillate. This indicates that the model setup is incorrect.

Summary

This tutorial covered the following:
• Creating a geometry model in 3D-CAD
• Using the generalized cylinder mesher
• Solving a laminar flow through the pipe
• Solving a turbulent flow through the pipe
• Plotting results of each case