Star-CCM+ Working with a CAD Model

I. Reference Files

A. S-bend_geometry_with_patches.sim
   Simplest model after importing from Solidworks and creating Surfaces

II. Import the model

A. Launch StarCCM+
   1. New Simulation

B. Import the model
   1. File Menu: Import Surface Mesh
   2. Select the file
   3. Choose - Create new part

C. Create separate surfaces for the inlet, walls, and outlet
   1. Select the Sbend Part in the Geometry tree
   2. Expand the Surfaces node
   3. Right-click on Faces, select: Split by Patch...
   4. Select the inlet and outlet patches, and name them (Click Create)
   5. Leave remaining patches untouched and click Close
   6. Select the remaining Faces node (under Surfaces) and Rename as Wall

D. Save the Model

III. Create Regions

A. Select Regions, right-click New...

B. Rename the Region S-bend

C. Define Parts that make up the S-bend
   1. Select S-bend Region
   2. Click the Parts box in the property pane
   3. Expand the pane to include

D. Right-click Boundaries, select New
   1. Rename as Inlet
   2. Select the Inlet Surface as the Part
      a. Click in the Parts row in the Properties Panel
      b. Select Inlet from the Parts tree
   3. Select Mass Flow Inlet as Type

E. Right-click Boundaries, select New
   1. Rename as Outlet
   2. Select the Inlet Surface as the Part
   3. Select Pressure outlet

IV. Create Mesh Continua

A. Right-click Continua, select New --> Mesh Continua

B. Right-click Models, select Select Meshing Models...
   1. Surface Remesher
   2. Polyhedral Mesher
   3. Prism Layer Mesher
   4. Close

C. Select Base Size

Star-CCM+ Working with a CAD Model

1. Measure the diameter of the pipe (d = 0.0635 m)
2. Expand the Reference Values node of the Mesh hierarchy
3. Select Base Size
4. Set the value of base size to 0.005 m

V. Create a Physics Continua

A. Right-click Continua, select New --> Physics Continua

B. Right-click Models node, select Select Models ...

1. Three Dimensional
2. Steady
3. Liquid
4. Segregated Solver
5. Constant Density
6. Laminar

VI. Generate Mesh

A. Generate Surface mesh

B. Create a new Scene for the Mesh

C. Generate volume mesh

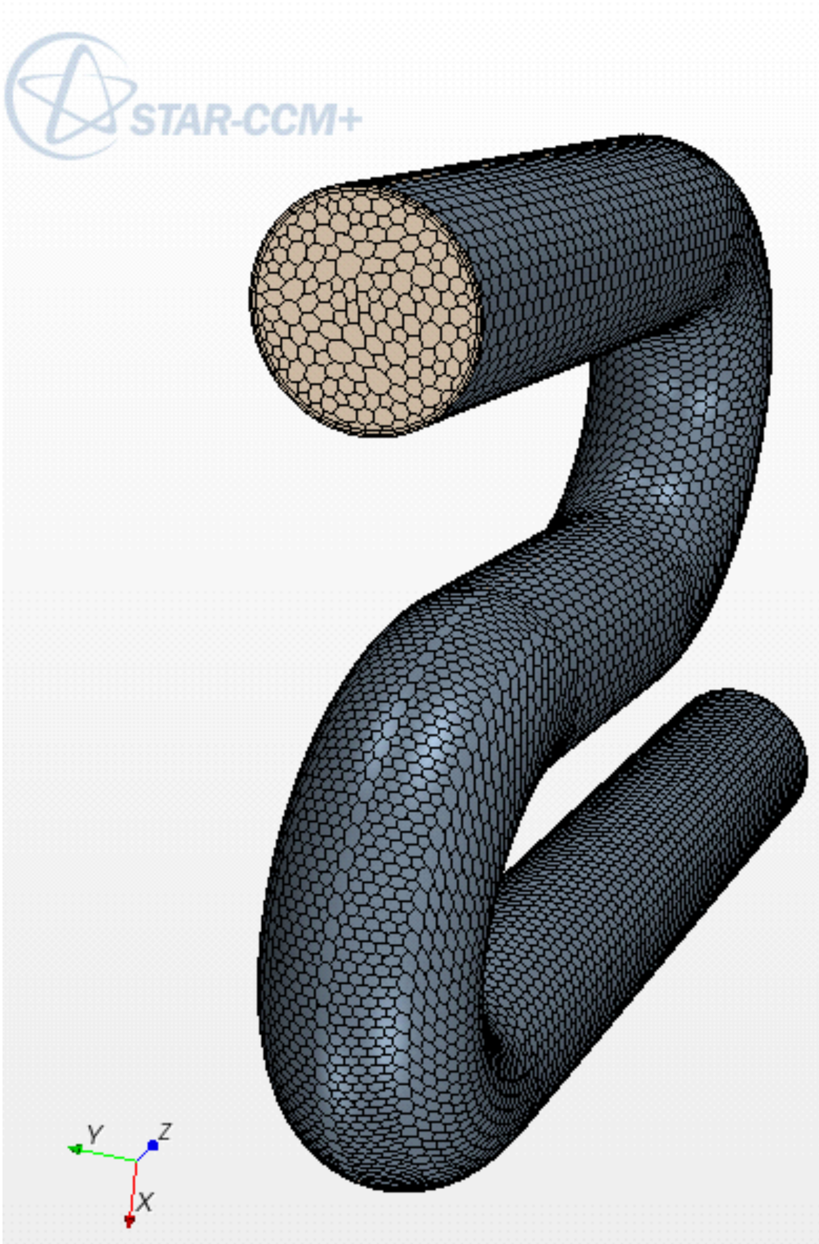
D. Create a section to view the midplane

1. Derived Part --> New --> Section --> Plane
2. Normal: [0, 1, 0]
3. Origin: [x, 0, z] where any x and z will work
4. Click Create
5. Click Close

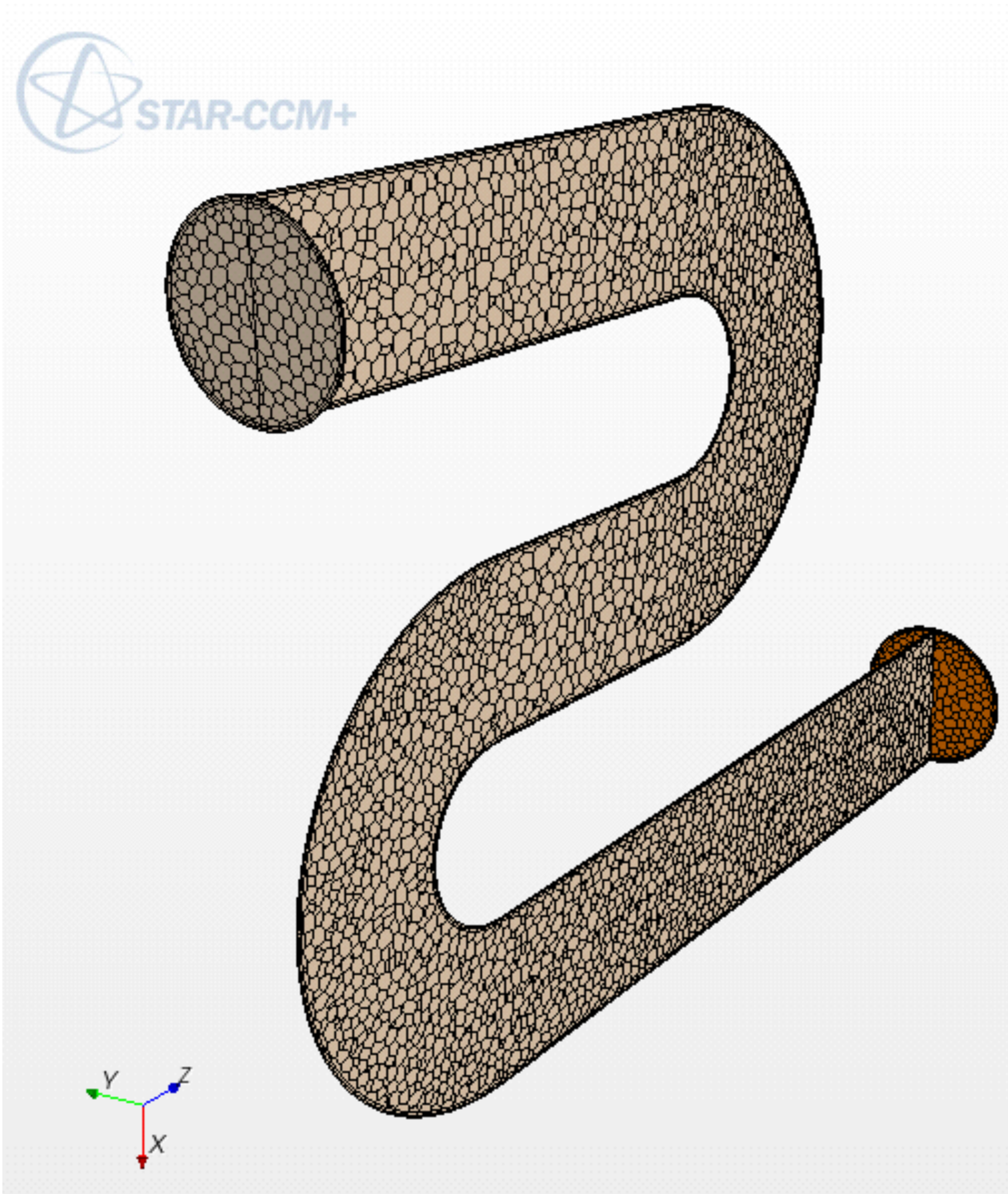
E. Expand the Mesh Scene node

Star-CCM+   Working with a CAD Model

F. Click representation, and select Volume Mesh



Star-CCM+ Working with a CAD Model



G. Select parts and turn off Wall region

VII. Prepare for solution and Run it

A. Create a Scene for displaying the Static Pressure
   1. Scenes --> New --> Scalar Scene
   2. Function: Static Pressure
   3. Parts: Wall off, plane section on

B. Limit the number of time steps
   1. Expand Stopping Criteria
   2. Click Maximum Steps and set to 200

Star-CCM+ Working with a CAD Model

VIII. Create a Streamline plot

A. Right click Derived Parts --> New Part --> Streamline
   1. Input Parts: S-bend
   2. Vector Field: Velocity
   3. Seed Mode: Line See
   4. Resolution: 10
   5. Click the "Display Tool" box
      a. Orient the line by dragging the two big balls
      b. Settle on the coordinates:
         i. Point 1: (x,y,z) = (-0.03, 0, 0)
         ii. Point 2: (x,y,z) = (0.03, 0, 0)
   6. Click Create -- Don't forget!