

1 Overview

Figure 1 depicts a two-dimensional fluid domain bounded by a cylinder of diameter D . Inside the domain is a smaller cylinder of diameter d . The smaller cylinder is offset an x -direction distance e from the center of the larger cylinder. The inner cylinder is held at temperature T_{in} , while the outer cylinder is held at $T_{\text{out}} < T_{\text{in}}$. The gap between the cylinder is filled with air, and effect of buoyancy, due to the temperature difference between the cylinders, causes the air to move.

This document provides instructions for using **StarCCM+** to simulate the flow and heat transfer in the gap between the two cylinders. Two features of **StarCCM+** are demonstrated: simulation of flow on two-dimensional meshes and simulation of flows with buoyancy. One could analyze the flow in a three-dimensional domain, but it is much more computational efficient to convert the three-dimensional mesh to a two-dimensional mesh, and perform the analysis in a two-dimensional domain. It is not possible to create a two-dimensional mesh directly in the current version of **StarCCM+**. One way to set up a two-dimensional model is to follow the procedure used in this tutorial: create a three-dimensional mesh, and then extract the two-dimensional mesh coincident with the $z = 0$ plane.

The geometry, boundary conditions, and mesh for the eccentric cylinder problem are all created in **StarCCM+**. A three-dimensional model is created with the built-in CAD tool in **StarCCM+**. The planar boundaries parallel to the plane of Figure 1 are symmetry planes in the three-dimensional model.

The thermal and geometric parameters chosen for this tutorial are:

$$T_{\text{in}} = 40\text{ }^\circ\text{C}, \quad T_{\text{out}} = 20\text{ }^\circ\text{C}, \quad D = 16\text{ cm}, \quad d = 4\text{ cm}, \quad e = 4\text{ cm}.$$

The fluid is air. At the mean temperature ($30\text{ }^\circ\text{C} = 303.15\text{ K}$) of the cylinders

$$\rho = 1.16 \frac{\text{kg}}{\text{m}^3}, \quad \mu = 1.86 \times 10^{-5} \frac{\text{kg}}{\text{m} \cdot \text{s}}$$

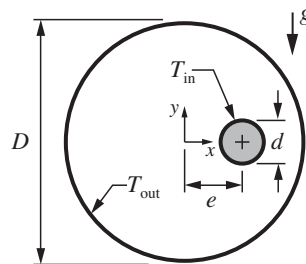


Figure 1: Two-dimensional gap between parallel cylinders that are not coaxial.

For these physical parameters, the Grashof and Rayleigh numbers are

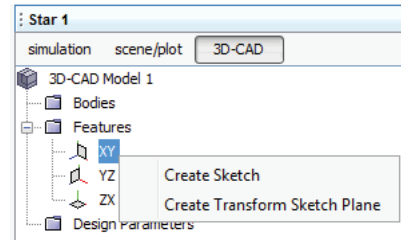
$$\text{Gr} = \frac{g\beta(T_{\text{in}} - T_{\text{out}})d^3}{\nu^2} = \frac{(9.8 \frac{\text{m}}{\text{s}^2}) \left(\frac{1}{303.15 \text{ K}} \right) (20 \text{ K})(0.02 \text{ m})^3}{\left(\frac{1.86 \times 10^{-5} \frac{\text{kg}}{\text{m}\cdot\text{s}}}{1.16 \frac{\text{kg}}{\text{m}^3}} \right)^2} = 3.1 \times 10^4$$

$$\text{Ra} = \text{GrPr} = (3.1 \times 10^4)(0.7) = 2.2 \times 10^4.$$

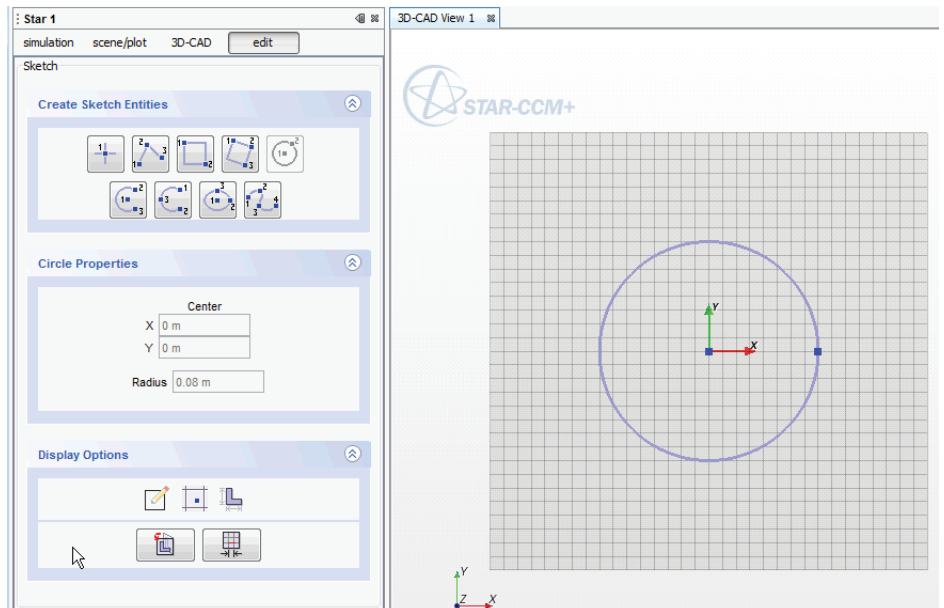
2 Model Geometry

The solid model is created with the 3D CAD tool built into StarCCM+. Start by creating a new StarCCM+ simulation. The model geometry is created as the 3D-CAD model in StarCCM+.

1. Open the CAD tool by expanding the Geometry node, right-clicking the 3D-CAD Models node and selecting **New**
2. Create a sketch on the (x, y) plane. Later, when the mesh is converted from 3D to a 2D, one of the mesh planes must be at $z = 0$. Creating the sketch on the (x, y) plane takes care of that requirement.



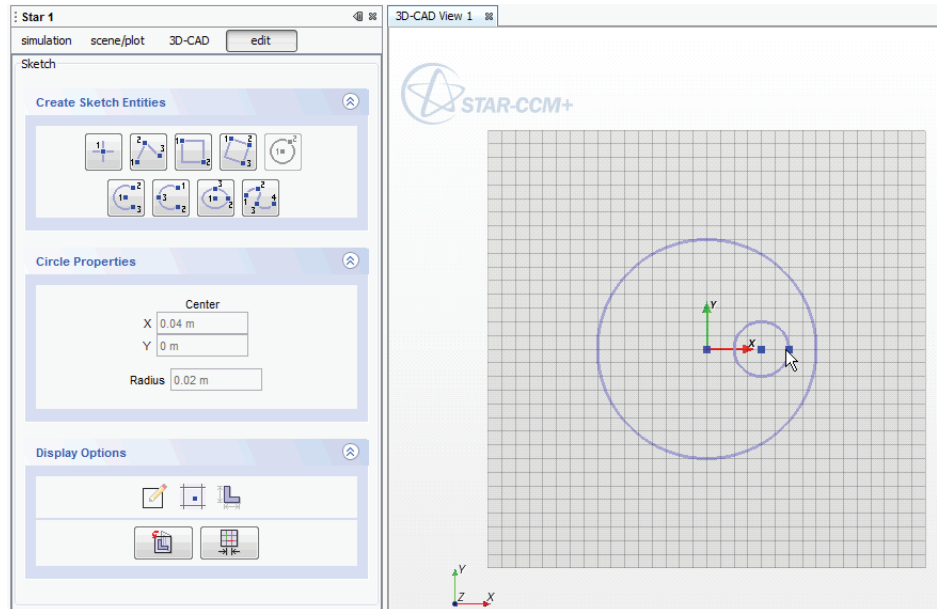
3. Sketch a circle with a radius 0.08 m centered at $(x, y) = (0, 0)$. Note that the origin and radius of the circle are displayed in the Circle Properties pane in the middle of the left side of the CAD window. You can sketch the circle without worrying about the precise dimensions, and then adjust those dimensions in the Circle Properties panel.



4. Sketch a second cylinder with radius 0.02 m centered at $(x, y) = (0.04, 0) m$.

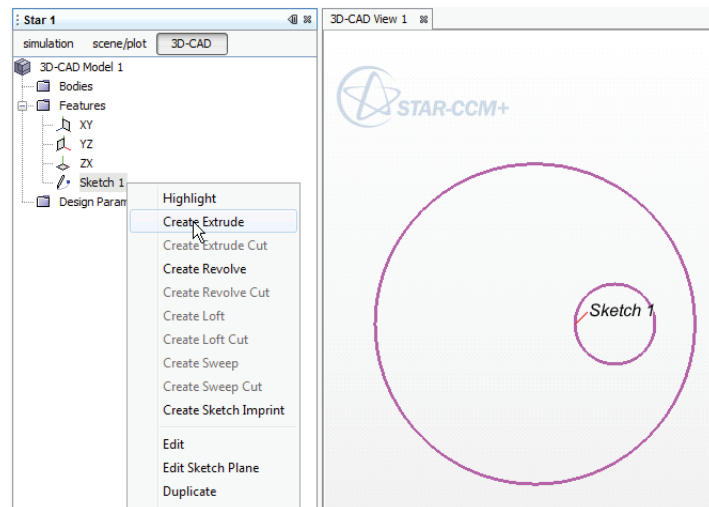
Click OK to close the sketch. You may need to adjust dimensions of the sketch components in the Circle Properties panel before clicking OK.

When you click OK, the sketch becomes a node called Sketch 1 under the Features node.



5. Right click on the (newly-created) Sketch 1 node and select **Create Extrude** from the pop-up menu.

In the Extrude properties panel, set the Distance parameter to 0.01 cm, and click OK. The size of the extrusion does not matter because the z direction dimension will be eliminated when the mesh is converted from 3D to 2D.



6. Click the Close 3D-CAD button to return to the StarCCM+ simulation mode.

3 Convert the Model Geometry to a Part with a 3D Mesh

3.1 Create the *Geometry Part*

The 3D-CAD model cannot be directly used in a StarCCM+ simulation. First it must be converted to a Geometry Part.

1. Right click on the 3D-CAD Model 1 node and select New Geometry Part. The new Geometry Part appears as a node under the Part Node.
Note: you may need to expand the 3D-CAD Models node under the Geometry node in order to see 3D-CAD Model 1.
2. For future reference, it is a good idea to rename the Geometry Part:
Right-click on Body 1 and select Rename. Give the part a name like *air volume*.
3. At this point it is also convenient to have a Geometry Scene to view the part.
Right-click on the Scenes node and select New Scene→Geometry.

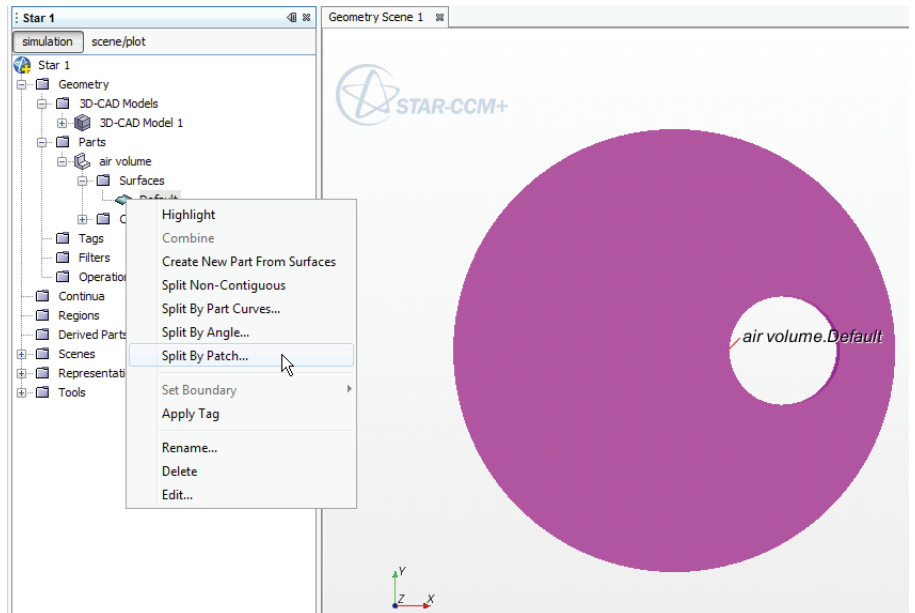
It's also a good idea to save the model at this state.

3.2 Create Separate Boundary Surfaces for the *Geometry Part*

The new Geometry Part has a single surface around the entire volume. We need to be able to refer to those surfaces separately. In preparation for creating a *Region* and assigning boundary conditions, isolate the surfaces of the *Geometry Part*.

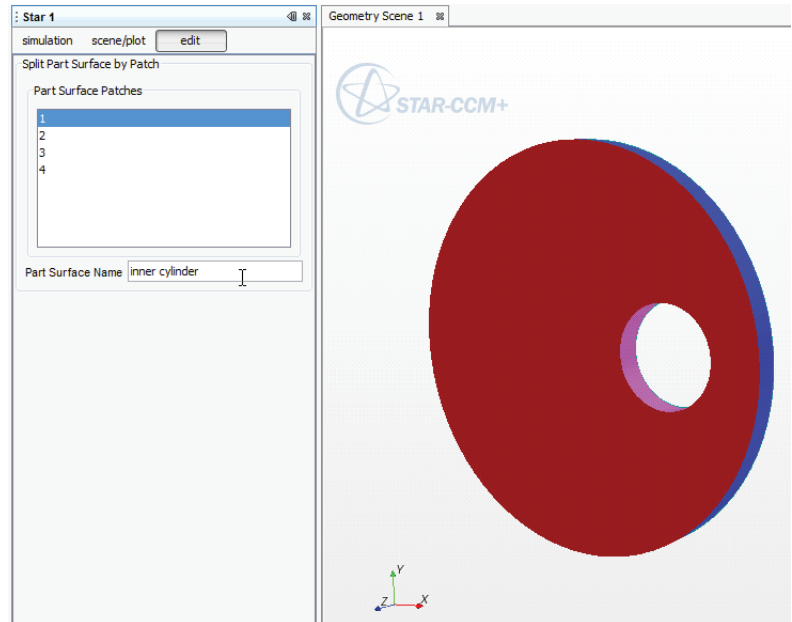
1. Expand the Surfaces node of the newly-created air volume part.
2. Right-click on the Default node and select Split by Patch...

This causes a Split Part Surface by Patch pane to appear.



- To make it easier to select patches with certainty, rotate the model in the geometry scene so that the sides of the air volume are visible.

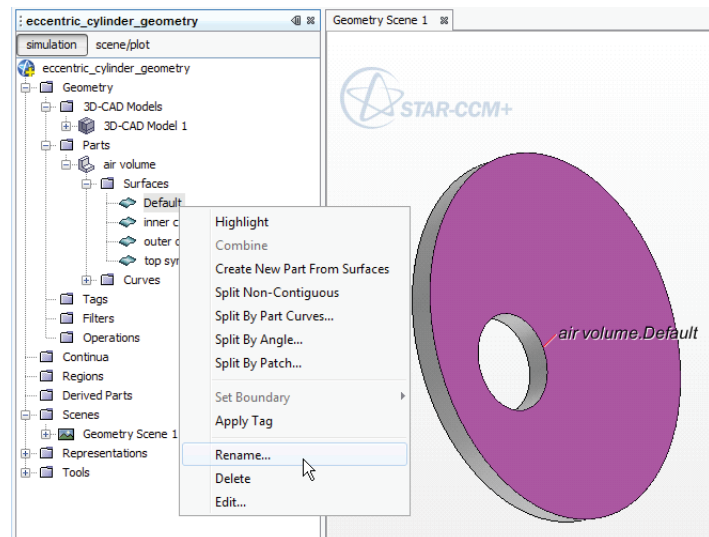
Click on the numbered surfaces in the **Split Part Surface by Patch** panel. When the inner surfaces is highlighted (in pink), change the name in the **Part Surface Name** box to *inner cylinder*, and click **Create** at the bottom of the **Split Part Surface by Patch** panel.



- Repeat the preceding steps to create *outer cylinder* and *top symmetry* patches. When there is only one numbered patch left in the **Part Surface Patches** panel, you can no longer create any more named patches. Click the **Close** button at the bottom of the **Split Part Surface by Patch** panel.
- Rename the last patch, which retains the name *Default*. You do this by selecting the **Default** surface under the **Surfaces** node of the **air volume** part.

Right-click on the **Default** node of the **Surfaces** node and select **Rename...** from the pop-up menu. Change the name of the last remaining surface (which should be a symmetry plane) to *bottom symmetry*.

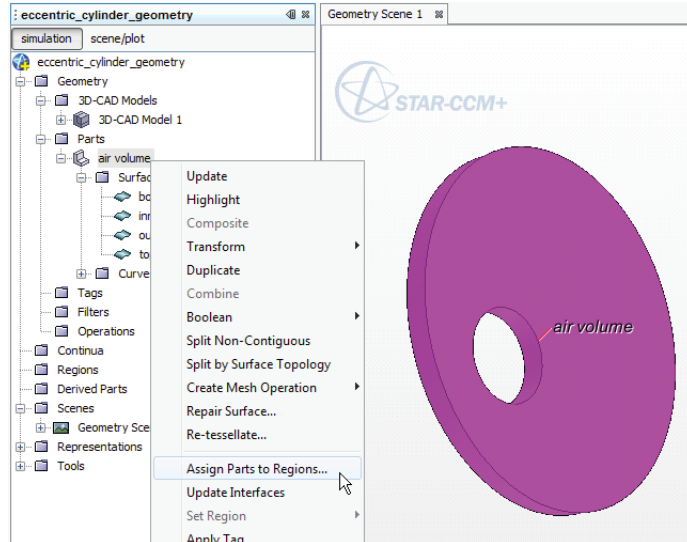
Save the model



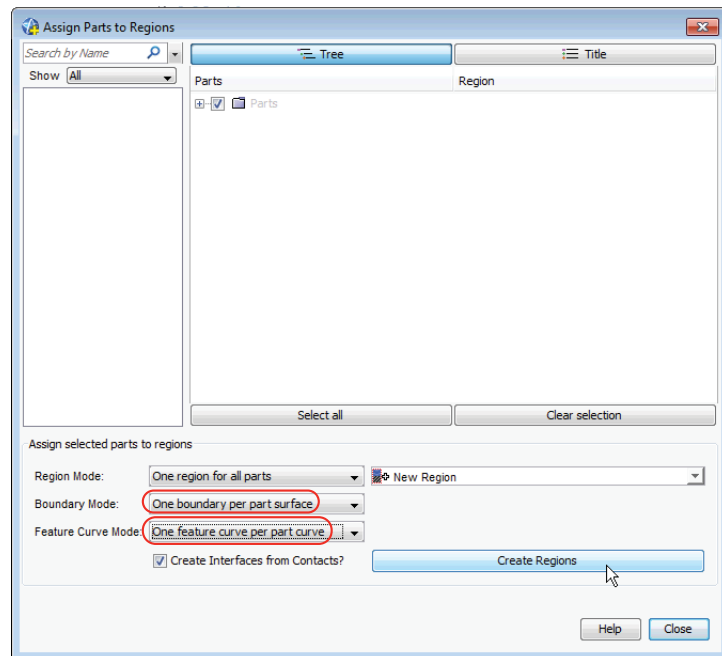
3.3 Create a Region from the Geometry Part

Now that the Geometry Part has distinct boundary surfaces, the next step is to create a Region from the Geometry Part.

1. Right-click on the air volume part and select Assign Parts to Regions from the pop-up menu.

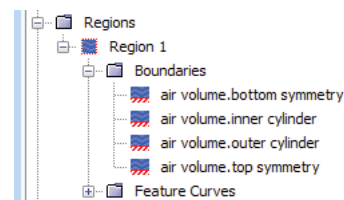


2. In the Assign Parts to Regions dialog box, change Boundary Mode to *One boundary per part surface* and change Feature Curve Mode to *One feature curve per part curve*, and then click Create Regions. Click the Close button.



3. Expand the Regions node to inspect the new region and the boundary surfaces assigned to it.

The names of the boundary surfaces could be shortened by removing the air volume. prefix from each boundary name.



3.4 Create a 3D Mesh

The analysis will be performed in 2D, but StarCCM+ (version 6) cannot create a 2D mesh on a 2D part. Instead, we need to create a 3D mesh and then extract a 2D mesh from the 3D mesh. That is why we have been working with a 3D part up to this point. Creation of the 3D mesh follows the standard procedure.

1. Right-click **Continua** and select **Mesh Continua**
2. Right-click on the newly-created **Mesh 1** node and select **Select Meshing Models**.

Select

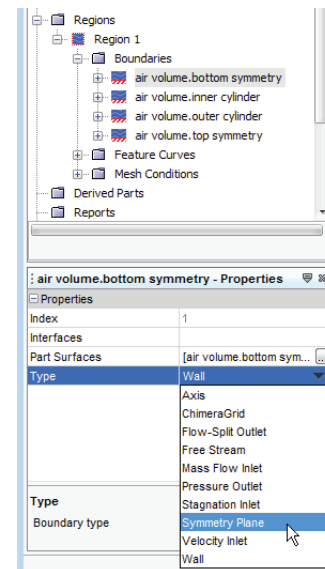
- Surface Remesher
- Polyhedral Mesher
- Prism Layer Mesher

Click the **Close** button.

3. Expand the **Reference Values** node under the **Mesh 1** node.
4. Select the **Base Size** leaf and in the **Base Size – Properties** pane, set the base size to 0.005 m.
5. Select the **Number of Prism Layers** leaf and in the **Number of Prism Layers – Properties** pane, set the **Number of Prism Layers** to 4.

Before constructing the 3D mesh, make sure the boundary conditions for the top and bottom symmetry planes are set to **Symmetry**.

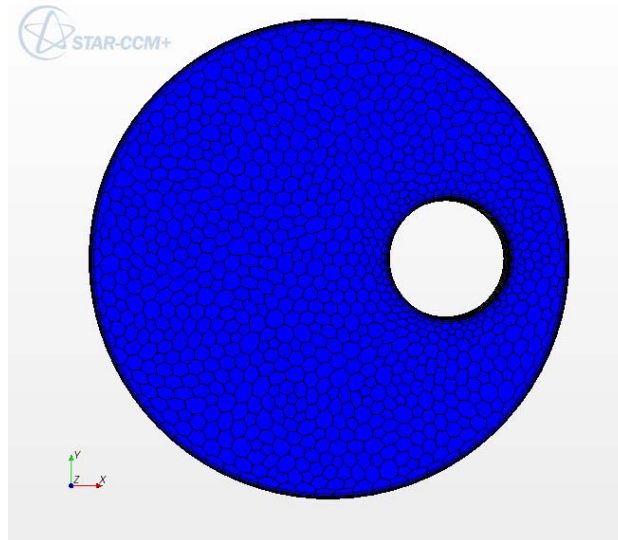
1. Select the **bottom symmetry** surface under the **Boundaries** node of **Region 1**. In the **Properties** pane, select **Symmetry Plane**.



2. Repeat the preceding step for the other boundary.

Generate the surface and volume meshes.

Select **Generate Volume Mesh** from the toolbar at the top of the window.



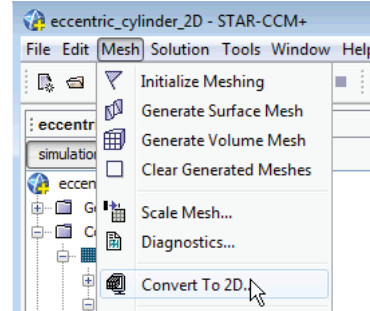
At this point, it would be good to save the model in a file that you set aside as a starting point for future geometry or mesh modifications. The next step is to convert the 3D mesh to a 2D mesh, and that step is *not reversible*.

1. Save the current model file as `eccentric_cylinder_3D.sim`.
2. Use **File**→**Save As...** to save the model file as `eccentric_cylinder_2D.sim`. We will continue to work with `eccentric_cylinder_2D.sim`.

4 Set up the 2D Model

4.1 Convert the 3D Mesh to a 2D Mesh

From the Menu bar select Mesh→Convert To 2D...



In the Convert Regions to 2D dialog box, check the Delete 3D regions after conversion option. Note, this destructive step makes it impossible to revise the mesh again without starting from the saved 3D model file.

Notice that the mesh can no longer be rotated in 3D, and that two continua have been created, Physics 1 and Physics 1 2D.

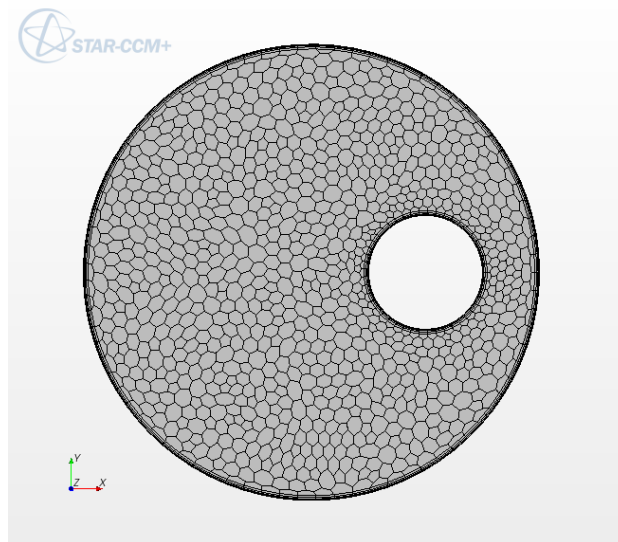
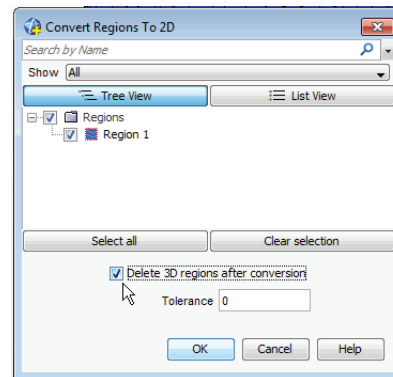


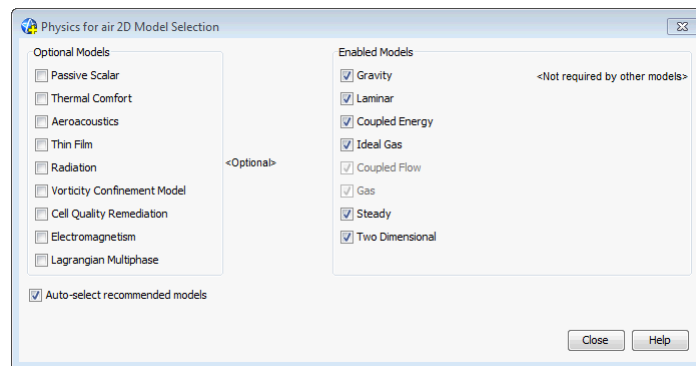
Figure 2: Two-dimensional mesh obtained with a base size of 5 mm.

4.2 Choose the Physical Models

1. Right click on the Physics 1 2D node and select **Rename...** from the pop-up menu. Choose a name like Physics for air 2D.
2. Right-mouse click on Physics 1 2D node and pick *Select Models...* from the pop-up menu. Make the following choices, by clicking on the check boxes:

Category	Choice
Time	Steady
Material	Gas
Flow	Coupled Flow
Equation of State	Ideal Gas
Viscous Regime	Laminar
Optional Models	Gravity

The finished model selection should look like this:



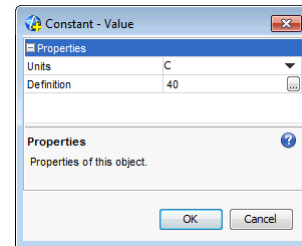
Click Close

3. Check the direction of gravity: Expand the Reference Values node under the Physics for air 2D continua node. Select the Gravity leaf and inspect the Value in the Gravity – Properties pane. The Value should be [0.0, -9.81, 0.0] m/s²

4.3 Apply the Boundary Conditions

Expand the Regions node. Notice that the old Region 1 is gone and there is a Region 1 2D node.

1. Select the Region 1 2D node, and expand the Boundaries node.
2. Expand the air volume.inner cylinder node and expand its Physics Conditions node.
3. Select the Thermal Specification leaf. In the Thermal Specifications – Properties pane, change the Method from Adiabatic to Temperature. When the change in the type of boundary condition is applied, a new node, Physical Values appears.
4. Expand the new Physical Values node and select the Constant leaf under the Static Temperature node. In the Constant – Properties pane, click on the [...] symbol. In the dialog box, change the Units to C (for convenience, only) and the Definition to 40.



5. Repeat steps 2 through 4 for the air volume.outer cylinder node. Set the outer cylinder temperature to 20 °C.

5 Solve the 2D Model

5.1 Create Scenes for Monitoring the Solution

1. Create a Vector Scene for the velocity field
2. Create a Scalar Scene for the temperature field

5.2 Set Up the Stopping Criterion

Add a Stopping Criteria for the continuity equation

Select Stopping Criteria → Create from Monitor → Continuity

Note that the default value for the Minimum Limit is 1.0×10^{-4} .

5.3 Run Solver

Click the Run icon from the Menu bar

6 Results

6.1 Medium Mesh Results

StarCCM+ solution was obtained on a mesh with a base size of 5 mm and four prism layers. The mesh is depicted in Figure 2.

Figure 3 shows the convergence of the coupled iterations. The stopping criterion of the continuity residual less than 1.0×10^{-4} was attained in 572 iterations.

The velocity field and temperature fields are shown in Figure 4. The dominant flow feature is a plume rising from the warmer inner cylinder. The plume impinges on the outer cylinder and continues in a counter-clockwise direction along the wall of the outer cylinder. Several weaker secondary recirculation zones are established.

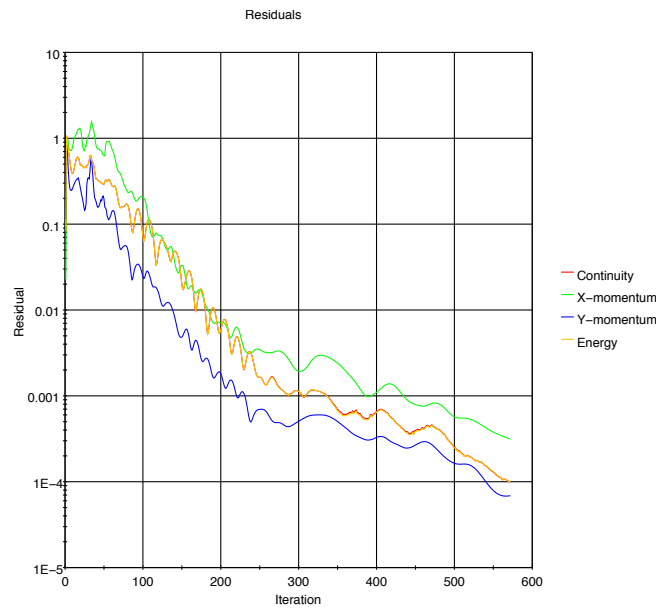


Figure 3: Residuals for the coupled solver on a medium mesh.

6.2 One Step of Mesh Refinement

A second model was created with a finer mesh by changing the base size parameter from 5 mm to 2 mm. Figure 5 shows two views of the new mesh, and Table 1 lists the number of cells in the preliminary (coarse) mesh and the refined mesh. The total number of elements in the finer mesh is approximately five times greater than in the original mesh.

The StarCCM+ solution did not converge for the finer mesh when the default initial guess of uniform (zero) velocity and uniform temperature was used as an initial condition. Convergence was obtained when the problem was treated as a transient and then switched to steady state. Figure 6 shows the residual history for the iterations after the switch to steady state. The steady solution was terminated after 4000 iterations with residuals between 10^{-3} and 10^{-4} .

The energy balance is another measure of convergence of the model. To expose the energy balance, create a report showing the heat transfer rates for regions of the model.

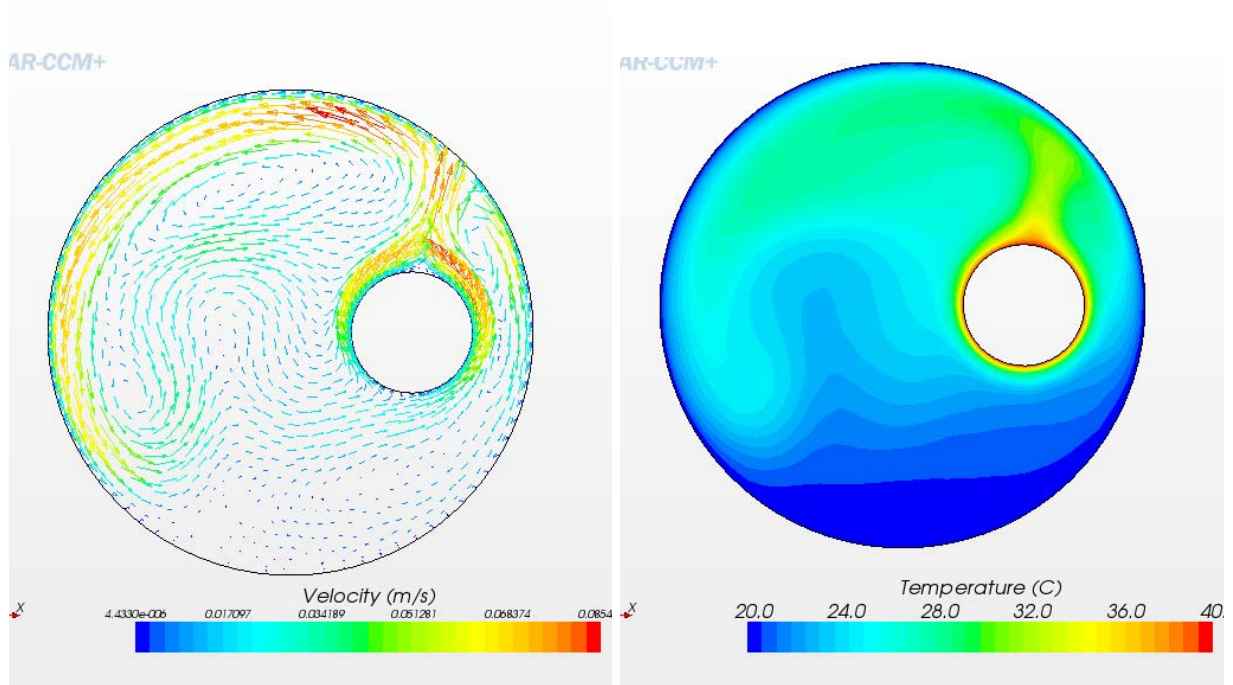


Figure 4: Velocity field (left) and temperature field (right) obtained on a mesh with base size of 5 mm.

1. Report→New
2. Select heat transfer in the model property pane
3. Select the air region in the Parts to be analyzed in the report.

Table 2 shows the energy balances for the two models. The first two columns of data are for the coarse mesh with different stopping criterion. For the data in the first column, the model was terminated with the continuity residual was reduced to less than 10^{-4} , which occurred on iteration 572. The data in the second column show the energy balance when the coarse model was run for a total of 1000 iterations.

Figure 7 shows the velocity fields for the medium (base size 2 mm) and course (base size 5 mm) models. The finer mesh shows more detailed structure and a narrower plume of fluid rising from the small cylinder. The jet of air moving along the top and left wall also appears to be narrower. Overall, the two flow fields are very similar. This is especially encouraging because the finer mesh required a transient start-up, which could have resulted in a different stable flow pattern in the air.

Figure 8 shows the temperature fields for the medium (base size 2 mm) and course (base size 5 mm) models. Consistent with the velocity field, the plume rising from the inner cylinder is a bit narrower and more sharply defined. Other than the plume above the inner cylinder, there appears to be little difference in the temperature fields. Note that the alignment of the images is awkward because the images were not carefully planned before exporting a hard copy. That can be corrected without re-running the model.

Table 1: Cell counts for meshes used in the StarCCM+ analysis of free convection from eccentric cylinders.

Part	Element type	Coarse	Medium
		0.005 m	0.002 m
Air volume	cells	1477	7478
Inner cylinder surface	faces	46	46
Outer cylinder surface	faces	78	308
Total	elements	1601	7832

Table 2: Energy balance for StarCCM+ solutions to the free convection from eccentric cylinders.

Surface	Coarse mesh 572 iterations	Coarse mesh 1000 iterations	Medium mesh 4000 iterations
Inner cylinder	-10.9707 W	-11.0043 W	-11.4002 W
Outer cylinder	11.2110 W	11.0074 W	11.4049 W
Balance	0.24 W (2%)	0.0036 W (0.03%)	0.0047 W (0.04%)

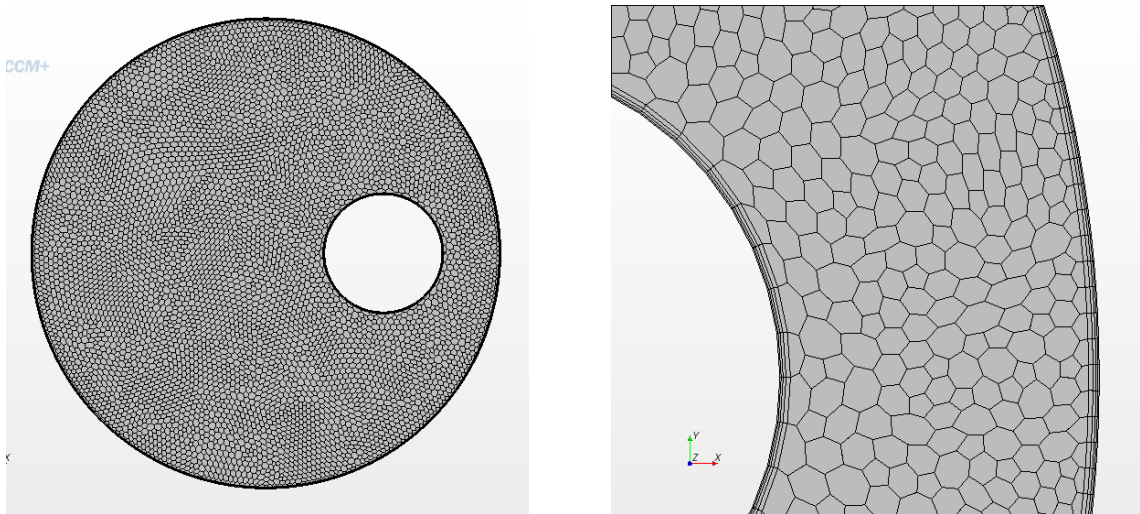


Figure 5: Computational mesh obtained with base size of 2 mm. Image on the right shows a zoomed-in view of the gap between the two cylinders.

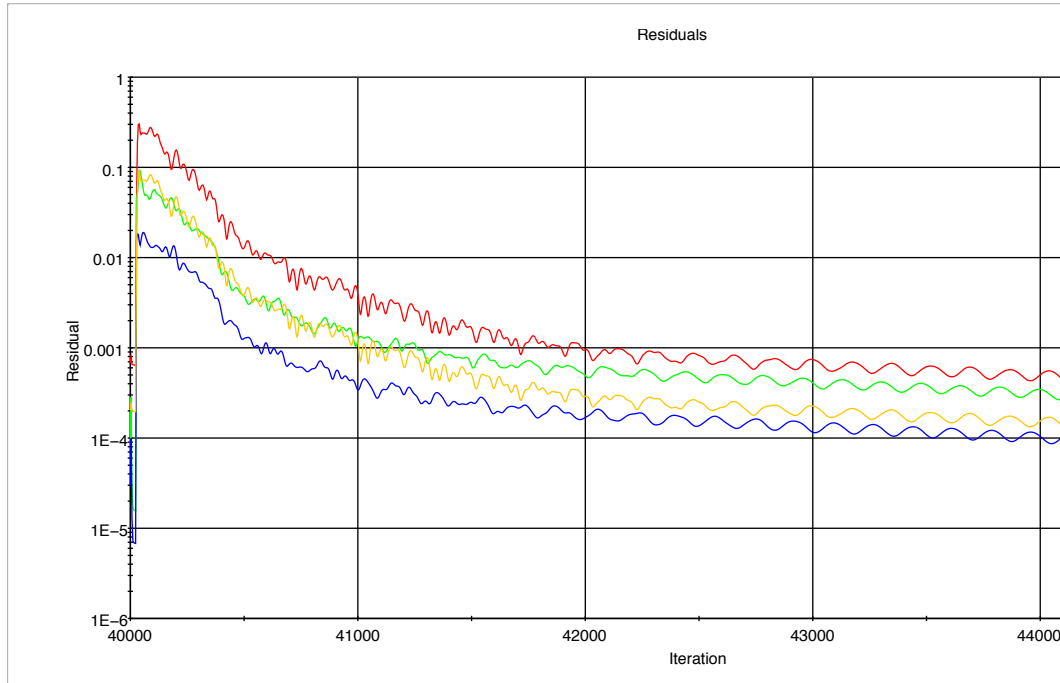


Figure 6: Residuals for the coupled solver on the mesh with a base size of 2 mm.

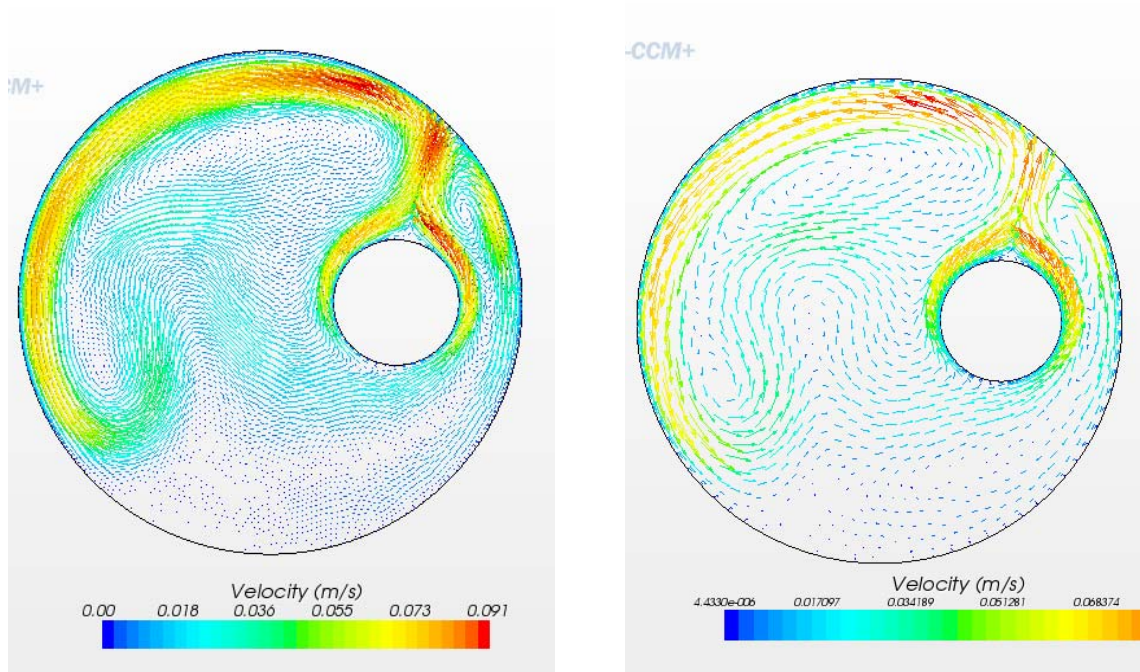


Figure 7: Velocity field on the mesh with a base size of 2 mm (left) compared to the velocity field on a mesh with a base size of 5 mm (right).

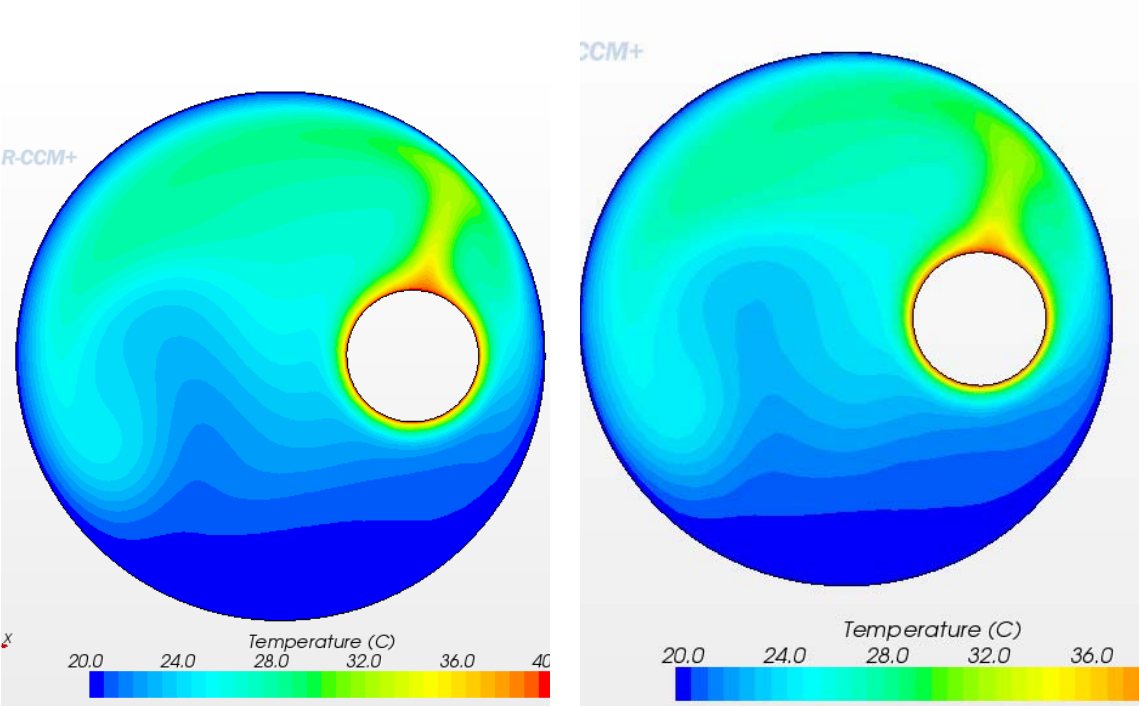


Figure 8: Temperature field on the mesh with a base size of 2 mm (left) compared to the temperature field on a mesh with a base size of 5 mm (right).