Natural Convection: Concentric Cylinders

This tutorial demonstrates how to solve a natural convection problem in STAR-CCM+. An infinitely long cylinder of radius 1.78 cm, heated at a constant temperature of 306.3 K, is placed inside another larger cylinder of radius 4.628 cm at a lower fixed temperature of 293.7 K. These temperatures are chosen to give an even temperature distribution about a bulk temperature ($T_b$) of 300 K in the annulus. As the flow is expected to be symmetrical about the center-line, it is only necessary to work with half the geometry, as shown below.

The model will be set up for an ideal gas with the following material properties, evaluated at bulk temperature of 300 K and pressure 100,000 Pa.

- Density (kg/m$^2$) 1.1614
- Dynamic viscosity (Pa-s) 1.846E-5
- Kinematic viscosity (m$^2$/s) 1.589E-5
- Specific Heat (J/kg-K) 1007.0
- Thermal Coefficient of Volumetric Expansion (1/K) $1/T_b$ (Ideal Gas) = 0.00333
This case repeats the experiments of Kuehn and Goldstein [1]. The physics will be set up to correspond to a Rayleigh number of $2.66 \times 10^4$. The Rayleigh number ($Ra$) is a dimensionless number defined as

$$ Ra = \frac{g \beta \Delta T L^3}{\nu \alpha} \tag{1} $$

where $g$ is the gravity constant [m/s$^2$], $\beta$ is the thermal coefficient of volumetric expansion of an ideal gas [1/K], $\Delta T$ is the change in temperature [K], $L$ is the length of the convection ($R_o - R_i$) [m], $\nu$ is the kinematic viscosity [m$^2$/s] and $\alpha$ is the thermal diffusivity [m$^2$/s].

**Importing the Mesh and Naming the Simulation**

Start up STAR-CCM+ in a manner that is appropriate to your working environment and select the *New Simulation* option from the menu bar.

- Select File > Import > Import Volume Mesh... from the menu bar.
- In the Open dialog, navigate to doc/tutorials/heatTransfer/concylMesh.ccm.
- Click Open to start the import.
- Save the simulation under the name conCyl.sim.
A geometry scene will be displayed in the Graphics window. To view the internal mesh that has been imported, create a mesh scene.

![Mesh Scene](image)

### Setting up the Physics Models

We will next set up the physics for our model. As this problem deals with natural convection, we will be using an implicit coupled flow solver due to its increased robustness over the segregated solver. Further discussion on things to consider when deciding on an appropriate solver can be found in the section: How Do I Choose Between Coupled and Segregated?

A physics continuum, called Physics 1, is automatically created when the mesh is imported. By default, the continuum will include a Two Dimensional space model to account for the two dimensions of the imported mesh.

- Right-click on the Physics 1 node in the STAR-CCM+ tree and click on Select models...
- Select the following additional physics models:
  - Steady in the Time group box.
• Gas in the *Material* group box.
• Coupled Flow in the *Flow* group box.
• Ideal Gas in the *Equation of State* group box.
• Laminar in the *Viscous Regime* group box.
• Gravity in the *Optional Models* group box.
• Close the dialog.

The complete list of selected models under the *Models* node of **Physics 1** is shown below.

(2)

• Right-click on the **Continua > Physics 1 > Models > Gas** node and select **Edit**.

The *Edit* dialog will appear.

• Navigate to **Gas > Air > Material Properties**.
• Expand the Dynamic Viscosity, Specific Heat and Thermal Conductivity Constant nodes.
• Set **Dynamic Viscosity** to $1.846E-5$ Pa-s.
• Set **Specific Heat** to $1007.0$ J/kg-K.
• Set **Thermal Conductivity** to $0.0263$ W/m-K.
The completed dialog is shown below.

![Concentration dialog](image)

- Close the dialog.

- Navigate to the Contiuas > Physics 1 > Reference Values > Reference Density node.
In the Properties window, set the Value to $1.1614 \text{ kg/m}^3$.

Navigate to the Continua > Physics 1 > Reference Values > Reference Pressure node.

In the Properties window, set the Value to $100000.0 \text{ Pa}$.

Note that the initial static temperature is by default 300 K, which is the average of the fixed inner and outer wall temperatures. The initial static temperature is located under Continua > Physics 1 > Initial Conditions > Static Temperature. The material properties were evaluated at this bulk temperature.

**Setting Boundary Physics Conditions**

A fixed temperature difference across the annulus is achieved by setting the static temperatures of the inner and outer cylinder walls.

Select the Regions > ConvectionCylinders > Inner Wall > Physics Conditions > Thermal Specification node.
• In the Properties window, select Temperature for the Method property.

• Select the Regions > ConvectionCylinders > Inner Wall > Physics Values > Static Temperature > Constant node.

• In the Properties window, set the Value to 306.3 K.
• Repeat the process to set a static temperature for the Outer Wall node. Its value should be set to 293.7 K.

Setting Solver Criteria

We will increase the default value of the Courant number of the coupled implicit solver to achieve faster convergence. We generally want a Courant number that balances the total number of iterations against the time per iteration. The section on the coupled implicit solver provides further guidelines for setting an appropriate Courant number.

• Select the Solvers > Coupled Implicit node.
• In the Properties window, set the Courant Number to 100.0.

Because we are using a relatively large Courant number on a simple convection problem, we can expect fast convergence. We can therefore decrease the number of iterations for which to run the solver.
• Set the number of maximum steps to 300.

Preparing Scalar and Vector Scenes

A scalar and a vector scene will be created to display the temperature and velocity vectors in the region bounded by the concentric cylinders during the simulation.

• Create a new Scalar Scene.
• Click on the scene/plot button.
• Navigate to Scalar Scene 1 > Displayers > Scalar 1 > Scalar Field.
• Select the Temperature scalar field function in the Scalar Field - Properties
• Create a new Vector Scene. Note that by default, the vector field is set to the Velocity function.

Preparing Report to Monitor Convergence

To judge convergence for this natural convection problem, we will monitor the heat balance in our model, which should sum to zero for a converged case.

• Right-click on the Reports node and select New Report > Heat Transfer.
• Rename the Heat Transfer 1 node to Heat Transfer Inner.
• In the Properties window of the Reports > Heat Transfer Inner node, set the
Parts to Regions > ConvectionCylinders > Inner Wall.

- Repeat the above procedure to create another report, called Heat Transfer Outer, for the heat transfer on the Outer Wall region.

We will next create an expression report that will calculate the heat balance using the results of the newly created reports. A corresponding Field Function is generated for each report, allowing us to use them in an expression.

- Right-click on the Reports node and select New Report > Expression.
- Rename the Expression 1 node to Heat Balance.
- In the Properties window of the Reports > Heat Balance node, set the Definition to $HeatTransferOuterReport +
Running the Simulation

We are now ready to run the simulation. Click the Run button in the toolbar.

The Residuals display will be created automatically and will show the solver’s progress.

Save the simulation after the solver has finished.

Visualizing the Results

The scalar and vector scenes are shown below. As expected, the temperature decreases between the wall of the inner cylinder and that of the outer cylinder. This corresponds to the difference in the fixed wall temperatures of the two cylinders. The warmer air rises as it moves from the wall of the inner cylinder towards the outer cylinder, and sinks to the bottom of the annulus as it cools. The air recirculates within the annulus due
to natural convection, with the largest velocities seen near the walls of the cylinders, towards the top.
The heat balance quickly drops to about zero, showing that the problem converged correctly.

Comparing Results to Experimental Data

Kuehn and Goldstein reported their experimental and numerical results in the form of an equivalent conductivity. This equivalent conductivity is defined as the ratio of the actual heat transfer measured (or predicted) across the annulus to the heat transfer that would be obtained by pure conduction alone. That is,

\[ k_{eq} = \frac{q_{act}}{q_{cond}} \]  

(3)

For concentric cylinders, \( q_{cond} \) is defined as

\[ q_{cond} = \frac{2\pi k\Delta T}{\ln(R_o/R_i)} \]  

(4)

For the case being considered in this tutorial, Kuehn and Goldstein obtained an equivalent conductivity of 2.58.
To evaluate conductivity, we first calculate the conductive heat transfer based on Eqn. (4) using values provided earlier in our setup. This comes out to \( q_{\text{cond}} = 2.177 \, \text{W/m} \). The value of \( q_{\text{act}} \) for one half of the cylinders can be retrieved by double clicking on the Reports > Heat Transfer Inner or Heat Transfer Outer node. The heat transfer will be displayed in the Output window.

![Output - conCyl](image)

The total \( q_{\text{act}} \) for the complete annulus is therefore 5.56 W. When inserting into Eqn. (3) above, we calculate an equivalent conductivity of

\[
 k_{eq} = \frac{5.56}{2.177} = 2.55
\]  

(5)

There is only about a 1% difference between this result and the value given by Kuehn and Goldstein, showing that the numerical model has predicted the heat transfer consistent to what has been reported by others. Note that because we are working with a 2D representation of the cylinders, our prediction of heat transfer is per meter length, that is, W/m.

**Summary**

This tutorial demonstrated how to solve a natural convection problem in two concentric cylinders. The steps covered were:

- Importing a volume mesh.
- Defining physics models for laminar flow with natural convection.
- Defining the material properties for the selected models.
- Defining fixed physics conditions at the regions level.
- Setting a convergence condition for the coupled implicit solver.
- Preparing scalar and vector scenes.
- Preparing a heat balance report.
- Running the simulation.
- Examining the temperature, velocity field and heat balance solutions.
• Comparing solution to experimental results.

References