

Convection from a Fin in a Duct

ME 448/548: StarCCM+ Tutorial

Gerald Recktenwald*

27 February 2019†

Motivation

The goal of this tutorial is to introduce the following concepts in StarCCM+ simulation:

- Conjugate heat transfer: conduction in a solid coupled with convection in a fluid;
- Turbulent flow with heat transfer;
- Use of symmetry to reduce the scale of the problem being analyzed.

Figure 1 shows a one-half model of a plate-fin heat sink in a duct. The vertical plane through the heat sink is a symmetry plane. By invoking symmetry, we only need to simulate half of the duct/heat sink combination, which allows us to solve a model at a higher mesh resolution for the same computational effort, i.e., RAM usage and execution time.

The geometry of the model is created in Solidworks and imported into StarCCM+. It would also be possible to create the geometry in the 3D CAD tool built into StarCCM+.

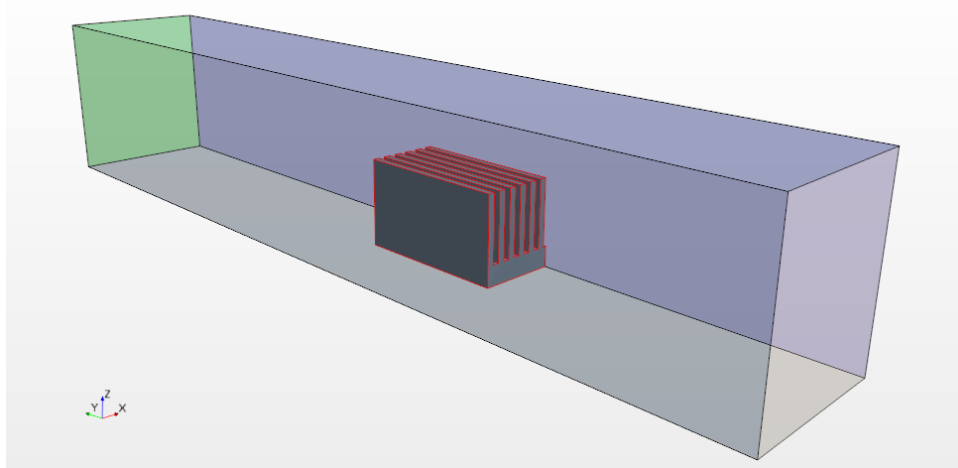


Figure 1 Half-model of a heat sink in a duct. The purple (far side) vertical wall is symmetry plane.

* Mechanical and Materials Engineering Department, Portland State University, gerry@pdx.edu

† Updated 2 October 2020 to clarify steps in renaming surfaces (pp. 3–4).

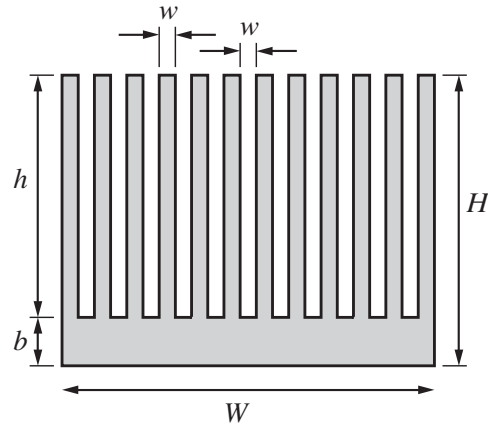


Figure 2 Sketch of the heat sink in the duct, viewed from the upstream direction.

Geometry

Figure 2 defines the geometric parameters of the upstream face of the heat sink. Figure 3 is a side view that shows the location of the heat sink relative to the inlet and outlet faces of the duct. Table 1 lists the numerical values of the dimensions. The small heat sink is typical of what might be used to cool an electronic component. The dimensions of the heat sink could be scaled up without affecting the procedure – other than some mesh size parameters – used in this tutorial.

Table 1 Dimensions of features in the solid model of the fin in Figure 1 and duct in Figure 2. The heat sink is centered in the duct cross section.

Symbol	Value	Description
W	34.5 mm	Width of the heat sink
w	1.5 mm	fin thickness and fin gap
H	19.5 cm	overall fin height $H = h + b$
h	15 mm	fin length
b	4.5 mm	thickness of fin base
L	34.5 mm	Length of the fin into the page
W_h	15 mm	Width of the heater surface
L_h	15 mm	Length of the heater surface
W_d	69 mm	Full width of the duct
H_d	39 mm	Height of the duct
L_d	138 mm	Length of the duct
S	69 mm	Location of leading edge of heat sink

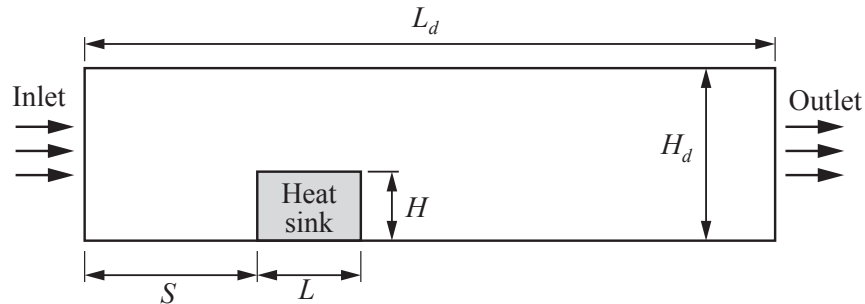


Figure 2 Side view of duct geometry.

Import Parasolid Models

The CFD model is assembled from two “solid” models created in Solidworks and exported as Parasolid files. One model is for the heat sink and one model is for the air around the heat sink.

- *File* → *Import* → *Import Surface Mesh*. Select one of the .x_t files.
- Accept defaults and click *OK*.
- Repeat the import for the other .x_t file.

The parts should be visible in a new *Geometry Scene*, but the duct will obscure the fin. You can click on the part names in the simulation tree to highlight each part in the *Geometry Scene* as shown in Figure 3, to the right.

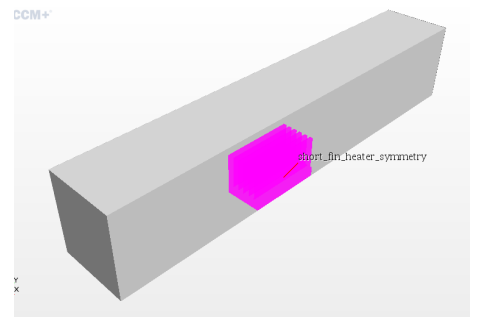


Figure 3 Geometry Scene after import of two Parasolid surface meshes.

Rename surfaces on the geometric parts

Separate the surfaces of each part using *Split by Patch*. This process effectively gives each surface a name. Choose descriptive names to aid in many subsequent model manipulation tasks.

- Identify and label the following parts on the duct
 - » Inlet on end closest to the fin
 - » Outlet on the far end
 - » Bottom wall (does not include the fin, which is not part of the air volume)
 - » Side wall
 - » Top wall
 - » Symmetry plane
- Close the *Split by Patch* tool, leaving the remaining surfaces un-named.
- In the simulation tree, rename the remaining surfaces, “air contact with fin”. This step is important because these surfaces will form the interface between the air in the duct and the surfaces of the fin in contact with the air.
- Identify and label the following parts on the fin. Figure 4 shows a view of the heat sink.

- » Heater – central surface on the bottom of the fin
- » Base – remainder of the bottom of the fin
- » Symmetry plane
- Close the *Split by Patch* tool, leaving the remaining surfaces un-named.
- In the simulation tree, rename the remaining surfaces, “fin contact with air”. This step is important because these surfaces will form the interface between the air in the duct and the surfaces of the fin in contact with the air. In other words, the “air contact with fin” (created earlier) and the “fin contact with air” are a matching pair of surfaces in the two different continua.

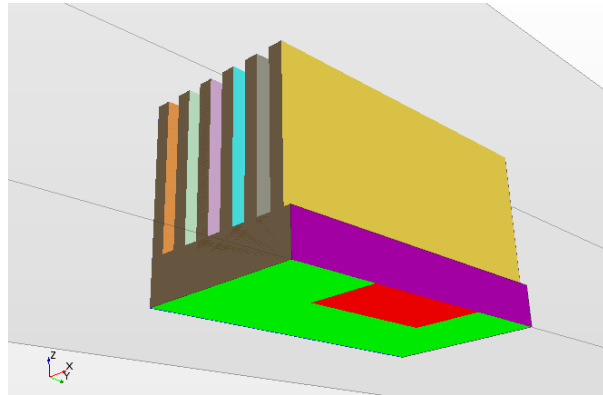


Figure 4 View of the heat sink part at the beginning of the *Split by Patch* operation. The colors of patches have not physical significance other than to identify distinct areas on the surface of the part.

Save the Model

It's a good idea to periodically save the model.

Create Physics and Mesh Continua

This model requires two physics continua – one for the fin and one for the air – and two mesh continua – one for the fin and one for the air..

Create the Physics Continuum for the Fin

- Right-click on *Continua* → *New* → *Physics Continuum*.
- Right click on *Physics 1* node and rename it to *fin*.
- Right click on *Models* → *Select Models*. In the *fin Model Selection* modal window, make the following selections:
 - » Three Dimensional
 - » Steady
 - » Solid
 - » Coupled Solid Energy
 - » Constant density
 - » Click *Close*

Figure 5 shows the final state of the fin physics continuum definition.

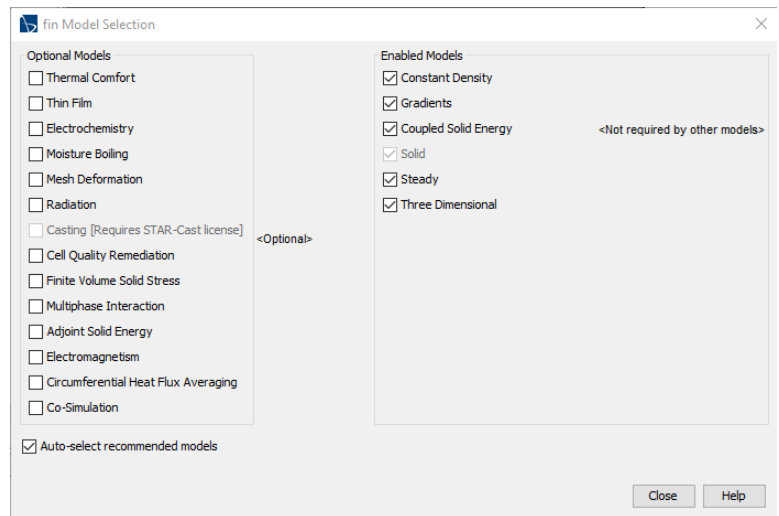


Figure 5. Selections for the fin physics continuum.

Verify the Fin Material Properties

- Expand the *Models* node under the (newly created) fin physics node
- Expand the *Solid* sub-node
 - » Observe that the default material is Aluminum. That is the appropriate material for the kind of heat sink we are simulating.

Create the Physics Continuum for the Air

- Right-click on *Continua* → *New* → *Physics Continuum*
- Right click on *Physics 1* node and rename it to *air*
- Right click on *Models* → *Select Models*. In the *air Model Selection* modal window, make the following selections, accepting any defaults.
 - » Three Dimensional
 - » Steady
 - » Gas
 - » Coupled Flow
 - » Constant Density
 - » Coupled Energy
 - » Turbulent
 - » K-Epsilon Turbulence
 - » Click *Close*

Figure 6 shows the final state of the fin physics continuum definition

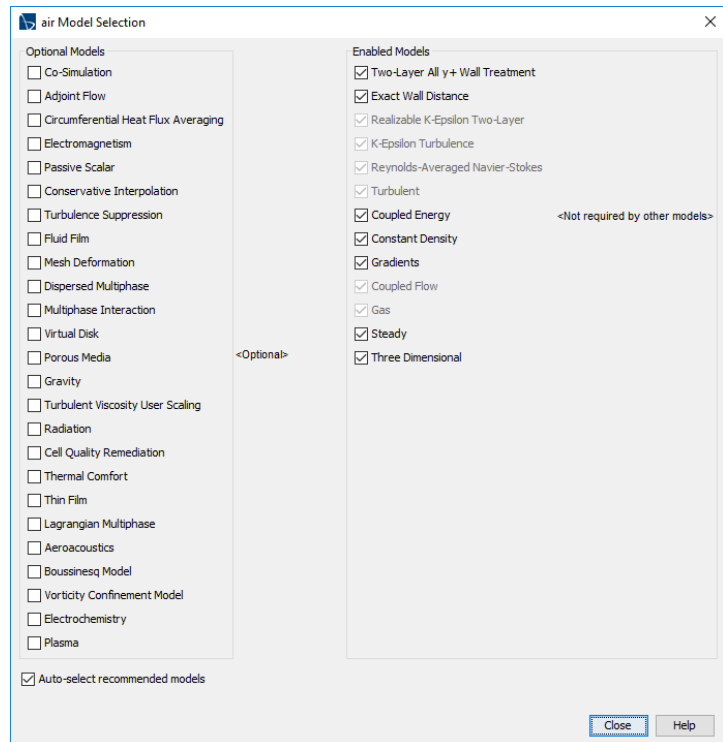


Figure 6 Selections for the air physics continuum.

Verify the Air Material Properties

- Expand the *Models* node under the (newly created) *air* node
- Expand the *Gas* node
 - » Observe that the default gas is air, which is what we want. No changes to the material properties are needed or desired.

Create the Mesh Continua for the Air

- Right-click on *Continua* → *New* → *Mesh Continuum*.
- Right click on *Mesh 1* and rename it to *air mesh*.
- Right click on *Models* → *Select Meshing Models*. In the *air mesh Model Selection* modal window, make the following selections. Figure 7 shows the result, with the Polyhedral meshing model selected.
 - » Select *Surface Remesher*
 - » Select *Trimmer mesh* (or *Polyhedral Mesher*)
 - » Select *Prism*.
 - » Click *Close*.

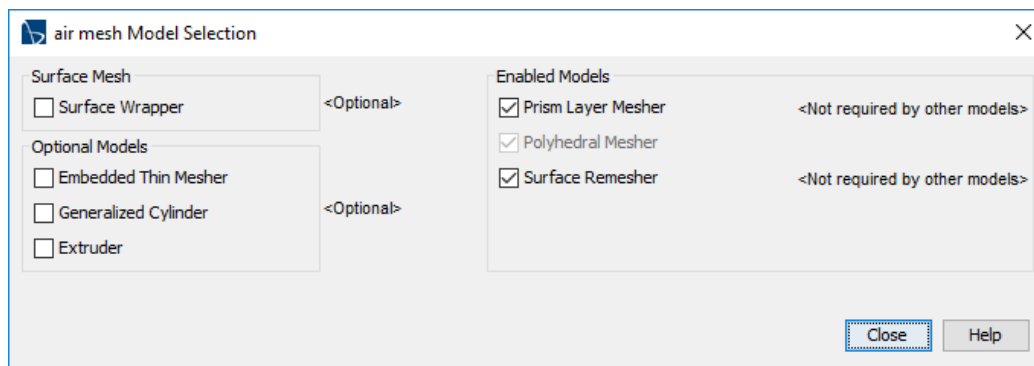


Figure 7 Selection of features for the air mesh continuum.

Create the Mesh Continua for the Fin

- Right-click on *Continua* → *New* → *Mesh Continuum*
- Right click on *Mesh 1* and rename it to *fin mesh*
- Right click on *Models* → *Select Meshing Models*. In the *fin mesh Model Selection* modal window, make the following selections
 - » Select *Surface Remesher*
 - » Select *Trimmer*
 - » Click *Close*.

Specify mesh parameters

Before a solution is possible, we need to create a mesh. However, in many flow simulations, the mesh behavior is influenced by the boundary conditions. For example, the meshes adjacent to an inflow or outflow boundary are different than meshes adjacent to a wall. Therefore, we should apply the boundary conditions *before* defining the mesh parameters. Nevertheless, the design of StarCCM+ allows meshes and boundary conditions to be manipulated independently. For example, changing boundary values does not require the mesh to be regenerated.

We'll return to the mesh definition after assigning the geometric parts to regions, and defining the boundary conditions of the region.

Assign the air and the fin to separate regions

Assign the air to a region

- Right-click on `duct_air_symmetry` sub-node of the *Parts* node, and select *Assign parts to regions*.
- IMPORTANT: As shown in Figure 8, set the middle pop-up menu on the left to *Create a boundary for each part surface*.
- Rename the region to air.
- Click *Apply* and then *Close*.

Assign the fin to a region

- Right-click on `short_fin_heater_symmetry` sub-node of the *Parts* node, and select *Assign parts to regions*.
- IMPORTANT: Set the middle pop-up menu on the left to *Create a boundary for each part surface*.
- Rename the region to fin.
- Click *Apply* and then *Close*.

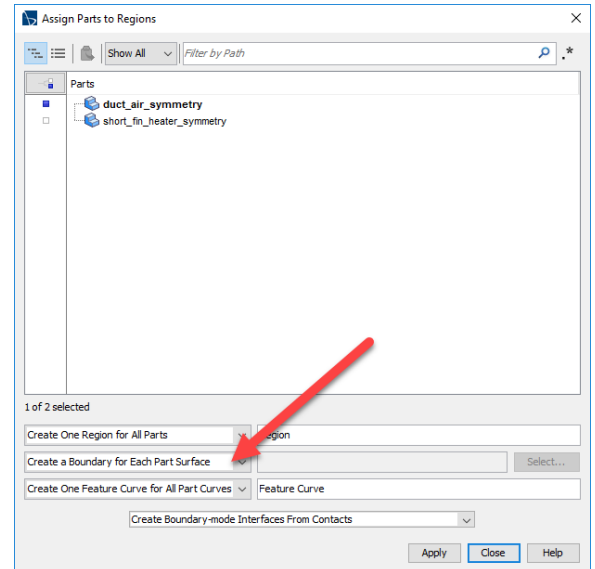


Figure 8 Assign the air duct part to a Region.

Verify the assignment of continua to Regions

The order of creating the mesh and physics continua has an effect on which of these continua are assigned to each region. When multiple continua are present in a model, it is important to check that each *Region* has the correct physics continuum and the correct mesh continuum.

- Click on the `air` Region in the simulation tree and inspect the *Properties* panel
 - » Verify that the *Mesh Continuum* is `air mesh`
 - » Verify that the *Physics Continuum* is `air`
- Click on the `fin` Region in the simulation tree and inspect the *Properties* panel
 - » Verify that the *Mesh Continuum* is `fin mesh`
 - » Verify that the *Physics Continuum* is `fin`

Figure 9 on the next page shows the results of checking the region assignments of physics and mesh continua. Use the pop-up menus in the *Properties* panel to change the continua as needed.

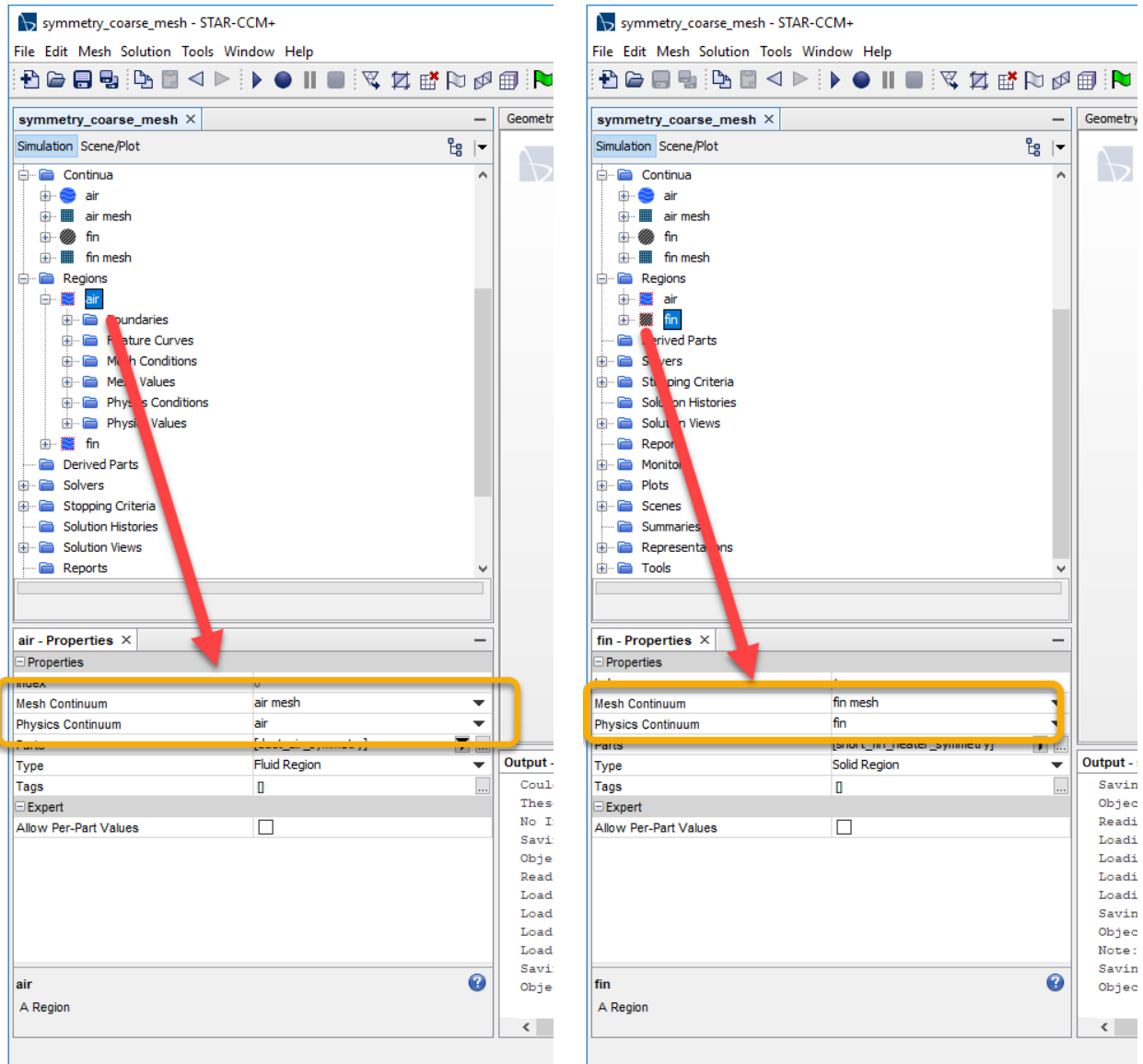


Figure 9 Verifying that the air and fin regions have the correct physics and mesh continua.

Boundary Conditions and Boundary Values

Set the boundary conditions for the air region

In the following instructions, we refer to the boundaries by their abbreviated names, “inlet”, “outlet”, “symmetry”, which were assigned in the *Split by Patch* operation. The full names appearing in the simulation tree also include the part name, e.g. “duct_air_symmetry.inlet”.

Expand the *Boundaries* sub-node of the air region.

- Mass flow inlet
 - » Click on the inlet sub-sub node.
 - » In the *Properties* pane, change *Type* from *Wall* to *Mass Flow Inlet*.
 - » Expand the inlet sub-sub node. Expand the *Physics Values* sub-sub-sub node
 - » Click on the *Mass Flow Rate* sub-sub-sub-sub node and enter a value of 0.015 kg/s in the *Properties* pane. A flow rate of 0.015 kg/s for the half-width gives $V_{in} = 9.25$ m/s and $Re_{Dh} = 30,000$
- Outlet
 - » Click on the outlet sub-sub node.
 - » In the *Properties* pane, change *Type* from *Wall* to *Mass Flow Outlet*. Note that a pressure outlet would also work.
- Symmetry plane
 - » Click on the symmetry plane sub-sub node.
 - » In the *Properties* pane, change *Type* from *Wall* to *Symmetry Plane*.

For all remaining surfaces, leave the default boundary condition settings: no-slip and adiabatic.

Set the boundary conditions for the fin region

Expand the *Boundaries* sub-node of the fin region.

- Heat input
 - » Click on the heater sub-sub node. Leave the *Type* as the default, *Wall*.
 - » Expand the *Physics Conditions* sub-sub-sub node.
 - » Click on *Thermal Specification*, and in the *Properties* pane, change the *Condition* from *Adiabatic* to *Heat Source*.
 - » Back in the simulation tree, expand the *Physics Values* sub-sub-sub node
 - » Click on the *Heat Source* sub-sub-sub-sub node.
 - » In the *Properties* pane, change the *Value* to 1 W. A heat input of 1 W for the half heater is a total heat input of 2 W, and a heat flux of $2W/(15 \times 10^{-3}m)^2 = 8890$ W/m².
- Symmetry plane
 - » Click on the symmetry plane sub-sub node.
 - » In the *Properties* pane, change *Type* from *Wall* to *Symmetry Plane*.

For all remaining surfaces, leave the default boundary condition settings: no-slip and adiabatic.

Create Contacts between the Fin and the Air

Although the air in the duct surrounds and is in contact with the fin, we must make sure that StarCCM+ knows that the fin can exchange heat with the air. We do this by establishing an *interface* object. The no-slip condition between the fin and the air will be imposed by the code through the same interface object. In the StarCCM+ model, we do not require any condition other than continuity of heat flux at the fluid-solid interface, as shown in Figure 10.

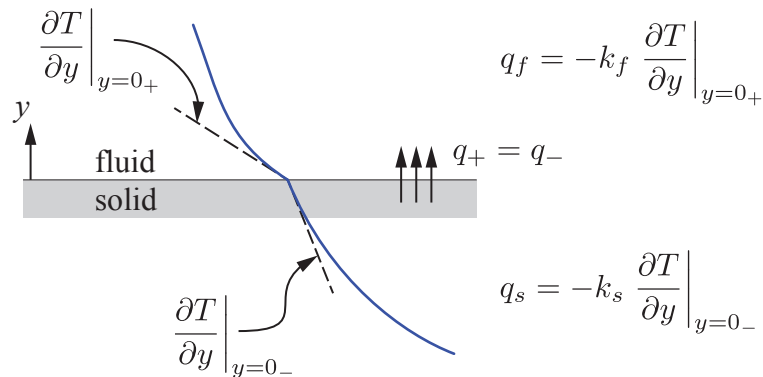


Figure 10 Continuity of heat flux at the fluid-solid interface. For simplicity of exposition, the interface is assumed to be located at $y = 0$.

Create the Interface

- Expand the *Boundaries* sub-nodes of both the air and fin Regions so that “air contact with fin” surface (in the air Region) and “fin contact with air” surface (in the fin Region) are visible. Note that the names of these surfaces were created during the *Split by Patch* operation.
 - » Click on air contact with fin and while holding the control key, click on fin contact with air surface.
 - » With both parts selected, right-click and choose *Create Interface* as shown in the left half of Figure 11 on the following page.
 - » After the interface is created, a new [Interface 1] sub-node appears under the *Boundaries* node of the air region. A matching [Interface 1] sub-node appears under the *Boundaries* node of the fin region.
- Explore the Properties of the air-side of the interface, but don't make any changes.
 - » Expand the *Physics Conditions* sub-sub node of the newly-created [Interface 1] sub-node of the air region
 - » Click on *Shear Stress Specification* and notice that in the *Properties* pane the *Method* is *No-Slip*. The alternative is a slip surface, which does not apply to the current model.
 - » Click on *Wall Heat Flux Coefficient Specification* and notice that in the *Properties* pane the *Method* is *None*. Leave this default value, but note that changing the *Method* to *Specified* allows input of a heat transfer coefficient on the air side of the interface.

- Explore the Properties of the fin-side of the interface, but don't make any changes.
 - » Expand the *Physics Conditions* sub-sub node of the newly-created [Interface 1] sub-node of the fin region
 - » Click on *User Wall Heat Flux Coefficient Specification* and notice that in the *Properties* pane the *Method* is *None*. Leave this default value, but note that changing the *Method* to *Specified* allows input of up to four coefficients to control heat flux on the fin side of the interface. These coefficients enable, for example, an additional heat source at the interface.

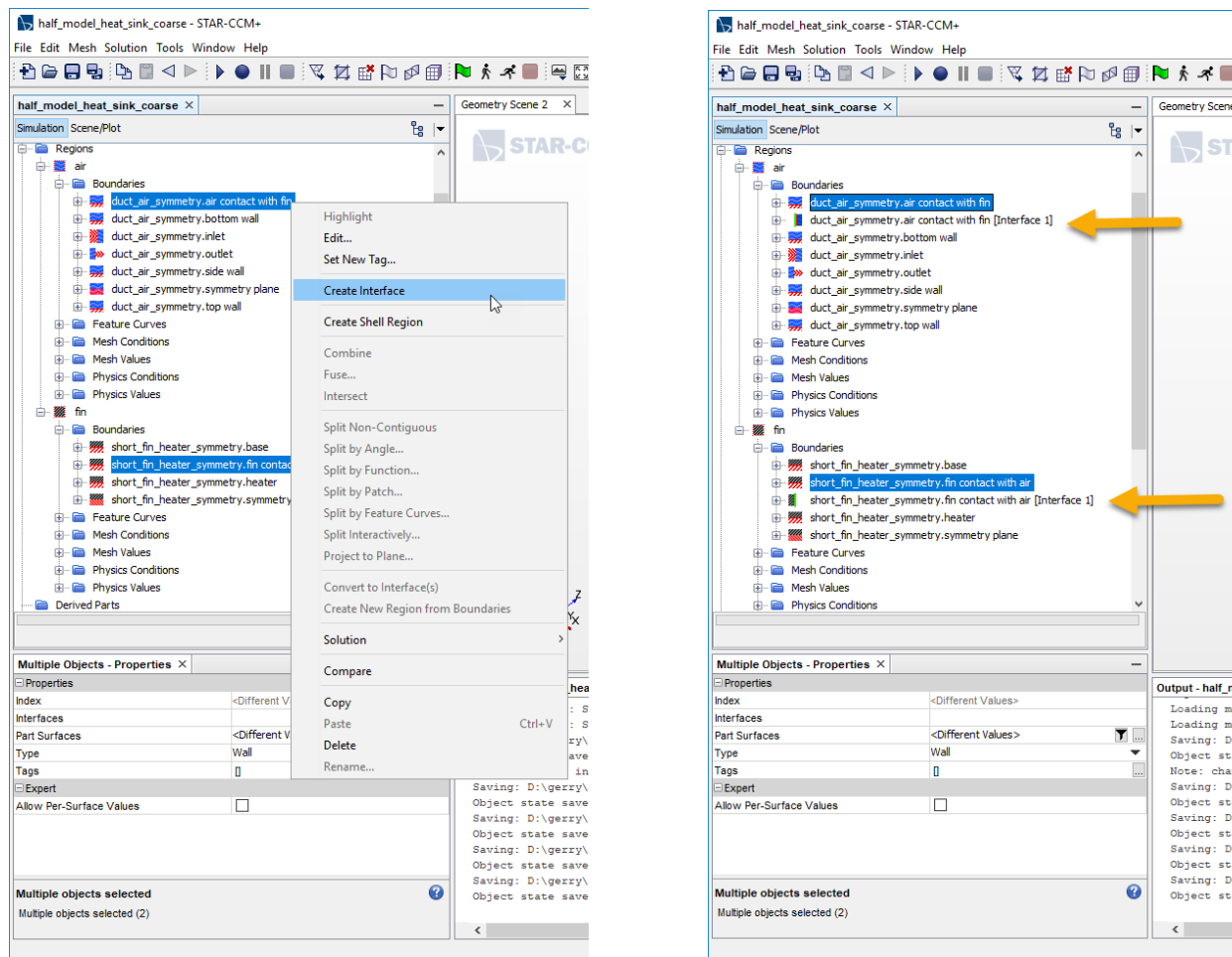


Figure 11 On the left: selecting the two surfaces to be connected with the *Create Interface* command. On the right: showing the two, matching interface objects under the *Boundaries* node of the air and fin Regions.

Generate Meshes

Start with coarse meshes to make sure that the model is correctly configured.

- For the air mesh, set the base size to 1 cm
- For the fin mesh, set the base size to 5 mm

Generate the volume mesh and view it in a *Mesh Scene*

Visualize the mesh

To aid in visualization, create a plane that is normal to the axis of the duct and intersects the fin

Create a new Mesh Scene

- Right-click on *Scenes* → *New Scene* → *Mesh*
- Expand the *Displayers* sub-node of the newly-created *Mesh Scene 1*
- Expand the *Mesh 1* sub-sub node, and the *Parts* sub-sub-sub node
 - » Click on the *side wall* node, and uncheck the *Visible* box in the *Properties* pane
 - » Click on the *top wall* node, and uncheck the *Visible* box in the *Properties* pane

Visualize the fluid mesh near the fin

- Create a plane as a Derived Part
 - » In the simulation tree, right-click *Derived Parts* → *New Part* → *Section* → *Plane...*
 - » To position the plane, use your mouse, or enter coordinates in the panel. The following table shows helpful values for the plane parameters. Note that the values of the *x* and *z* origin are not important, and can be left as their current values.

	Origin	Normal
x	0 cm	0 m
y	2.5 cm	1 m
z	1 cm	0 m

- » Click *Create*; Click *Close*.
- Rename the plane as an aid to post-processing (we will create other planes later)
 - » Expand the *Derived Parts* node
 - » Right-click → *Rename...* on the new *Plane Section* item and give the section a new name, like “mid fin vertical section”.
- Back in the *Mesh Scene*, click on the *Parts* sub-sub-sub node (under *Displayers/Mesh 1*)
 - » Click on the [...] in the *Properties* pane
 - » In the pop-up window, expand the *Derived Parts* node and select the *mid fin vertical section* part
 - » Click *OK*.

Figure 12 shows the *Mesh Scene* with the new plane section included in the *Parts* list, and with the top and side walls having their *Visible* box (in the *Properties* pane) unchecked. Note that

there are Prism Cells on the bottom, side and top wall of the duct, but not on the Symmetry plane. Also note that there are no Prism Cells in the fin. The mesh in the air volume is not pretty, but it will be sufficient to get the simulation started.

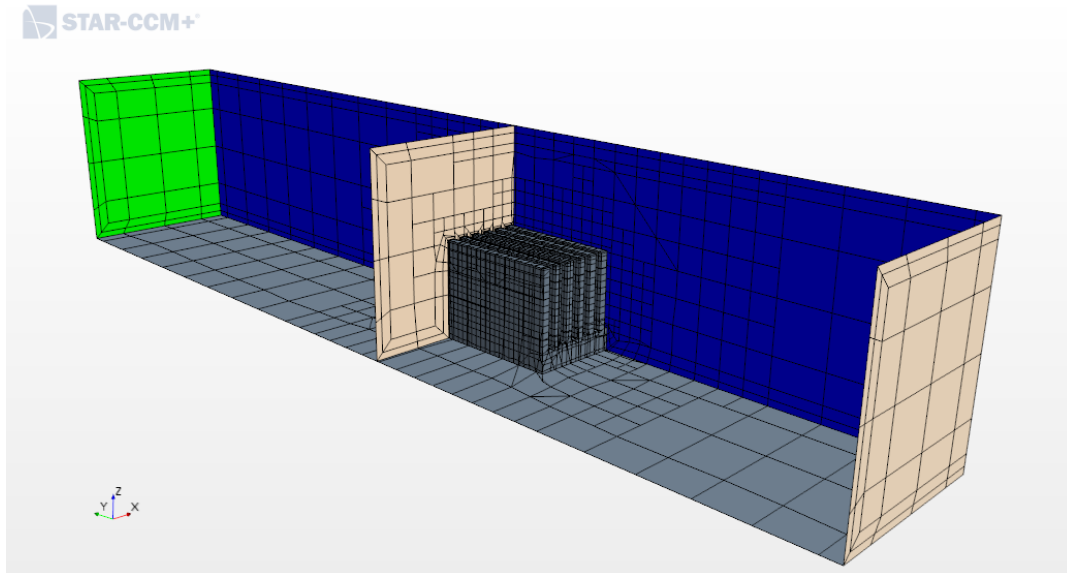


Figure 12 Initial coarse mesh created with trimmer meshes in the air and fin. Base size in the air is 1 cm and base size in the fin is 5 mm.

Run the Simulation

Set the Stopping Criteria

Before running the model, adjust the stopping criteria

- Expand the *Stopping Criteria* node in the simulation tree
- Set the *Maximum Steps* value to 200. This parameter is not critical for this problem since you can stop the simulation manually at any iteration.
- Right-click on *Stopping Criteria* → *Create new Criterion* → *From Monitor...*
 - » In the modal window, select *Continuity*
 - » Under the newly created *Continuity Criterion* sub-node, click on *Minimum Limit* and verify the (default) value of $1.0e-4$ in the *Properties* pane
 - » Set the *Logical Rule* to *And*. This will require the continuity criterion to be met even when other criteria (excluding the *Maximum Steps*) are met.
- Right-click on *Stopping Criteria* → *Create new Criterion* → *From Monitor...*
 - » In the modal window, select *Energy*
 - » Under the newly created *Continuity Criterion* sub-node, click on *Minimum Limit* and verify the (default) value of $1.0e-4$ in the *Properties* pane
 - » Set the *Logical Rule* to *And*. This will require the energy criterion to be met even when other criteria (excluding the *Maximum Steps*) are met.

- Run the simulation for 200 iterations (Maximum Steps)

Inspect the early solution

It makes sense to look at the solution before too much effort is expended. After letting the model run for 200 iterations – the initial value of *Maximum Steps* – check that the temperature and velocity fields make sense. Also create a heat transfer report that will be useful in mesh refinement studies.

1. Inspect the residuals

Figure 13 shows the residual plot after 200 iterations. There is a downward trend in the residuals, but more work needs to be done. Before doing more work, check that other aspects of the solution indicate that the model was set up correctly.

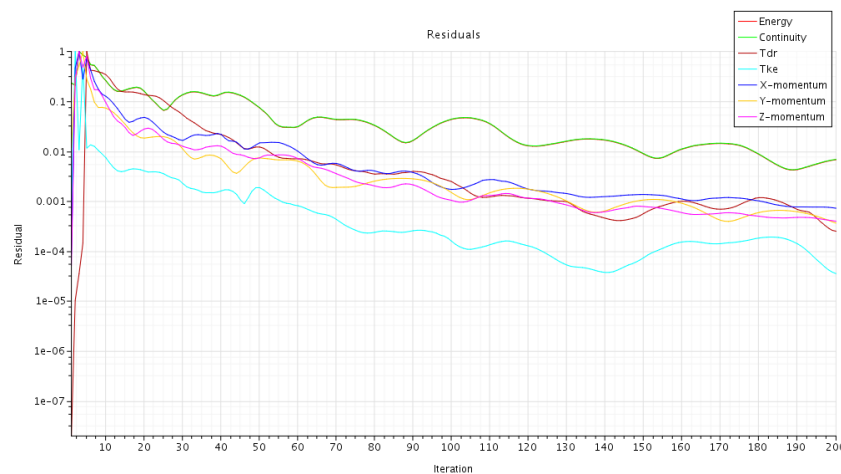


Figure 13 Residual plot after 200 iterations of the coarse mesh model.

2. Create a Scalar Scene to view the temperature

- Right-click on the *Scenes* node in the simulation tree: select *New Scene* → *Scalar*.
- Right-click on *Scalar Scene 1* → *Rename...* Choose a name like “Temperature”
- Expand the node for the newly created Temperature Scene
 - » Under the *Scalar 1* sub-sub node, Click on *Scalar Field*
 - » In the *Properties* pane, select *Temperature* as the *Function*
 - » In the *Properties* pane, change the *Units* from K to C.
 - » Under the *Scalar 1* sub-node, click on the *Parts* sub-sub node.
 - » In the *Properties* pane, click on the [...]
 - » In the pop-up window select the air and fin Regions, and click *OK*.
 - » Expand the air sub-node under *Parts*. Select the top wall sub-sub node and uncheck the *Visible* box in the *Properties* pane. Also select the side wall sub-sub node and uncheck the *Visible* box in the *Properties* pane.

After the preceding maneuvers, the Temperature scene should look like Figure 14

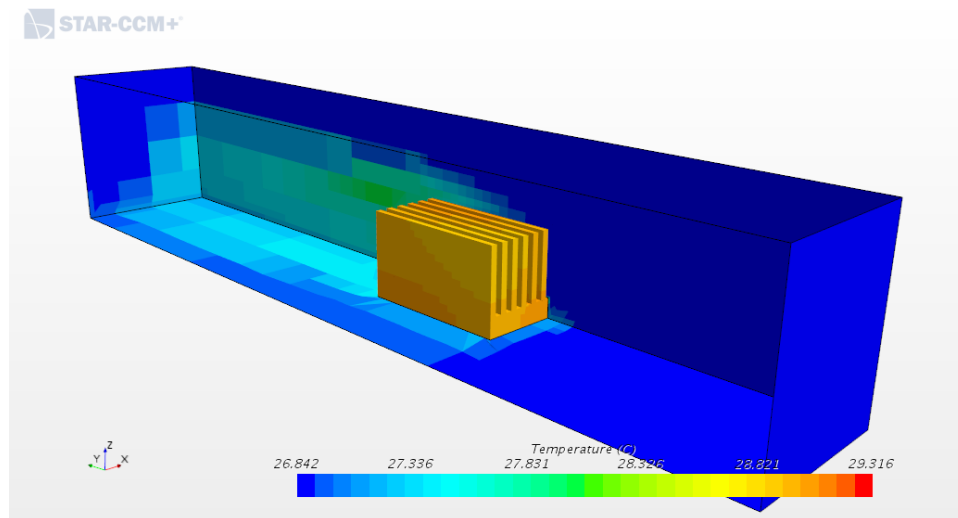


Figure 14 Temperature scene after 200 iterations

The temperature visualization can be enhanced by changing the display to show smoothed gradations of temperature, and by adding a vertical plane in the cross section to display temperature.

- Adjust the temperature field rendering
 - » Click on the *Scalar 1* sub-sub node of the *Displayers* node of the Temperature scene
 - » In the *Properties* pane, change the *Contour Style* from *Automatic* to *Smooth Filled*
- Add a plane section downstream of the fin and normal to the duct axis. The easy way to do this is to make a copy of the existing vertical plane. Alternatively, you can create a new plane from scratch.
 - » In the simulation tree, click on the “mid fin vertical section” derived part.
 - » Select copy (right mouse button, or control-C).
 - » Click on the *Derived Parts* node and select paste (right mouse button or control-V).
 - » Select the newly created part and rename it “downstream fin vertical section”.
 - » Right-click on the newly renamed part and select *Edit Part in Current Scene*.
 - » Click *Cancel* in the pop-up window that asks about creating a new displayer.
 - » To position the plane, use your mouse, or enter coordinates in the panel. The following table shows helpful values for the plane parameters. Note that the values of the *x* and *y* origin are not important, and can be left as their current values.

	Origin	Normal
x	0 cm	0 m
y	5 cm	0 m
z	1 cm	1 m

- » Click *Apply*; Click *Close*.

- The Temperature Scene, with the smooth contours and the newly create vertical plane section should look like Figure 15

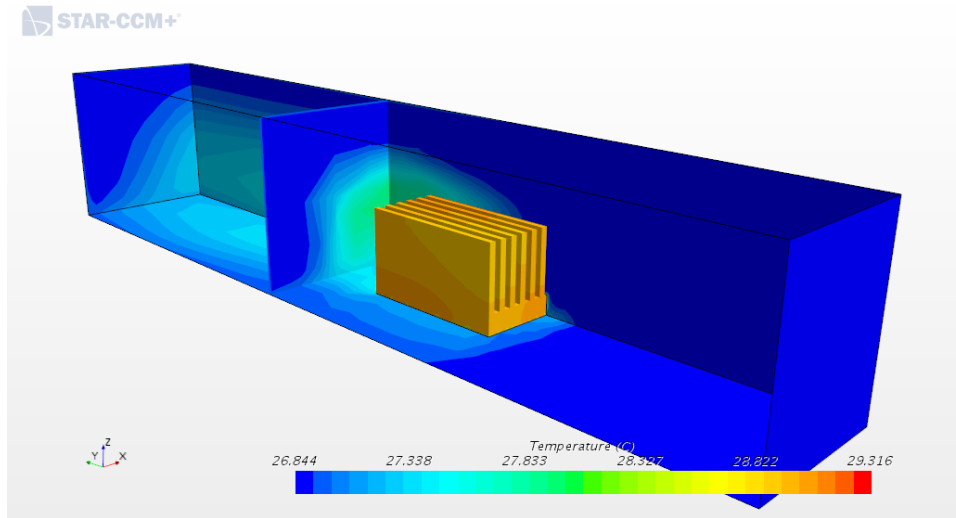


Figure 15 Temperature on the fin surface, symmetry plane, bottom wall and on a vertical plane downstream of the fin.

3. Create a Vector Scene to view the velocity

Plot the velocity vectors on the symmetry plane and on a horizontal plane midway up the fin.

- Create the Velocity Scene: right click *Scenes* → New Scene → Vector
 - » Optional: Rename the scene as “Velocity Field”
 - » Click on the *Outline 1* sub-node of the *Displayers* node of the *Velocity Field*
 - » In the *Properties* pane, check the *Feature Lines* box.
- Select the Parts
 - » Expand the *Displayers* sub-node, and expand the *Vector 1* sub-sub node
 - » Click on *Parts*, and in the *Properties* panel, click on the [...] box
 - » In the Parts pop-up window, expand the *Regions* node and the *air/Boundaries* sub-node. Select the symmetry plane surface
 - » Click *OK*.

The Velocity Field scene should look like Figure 16.

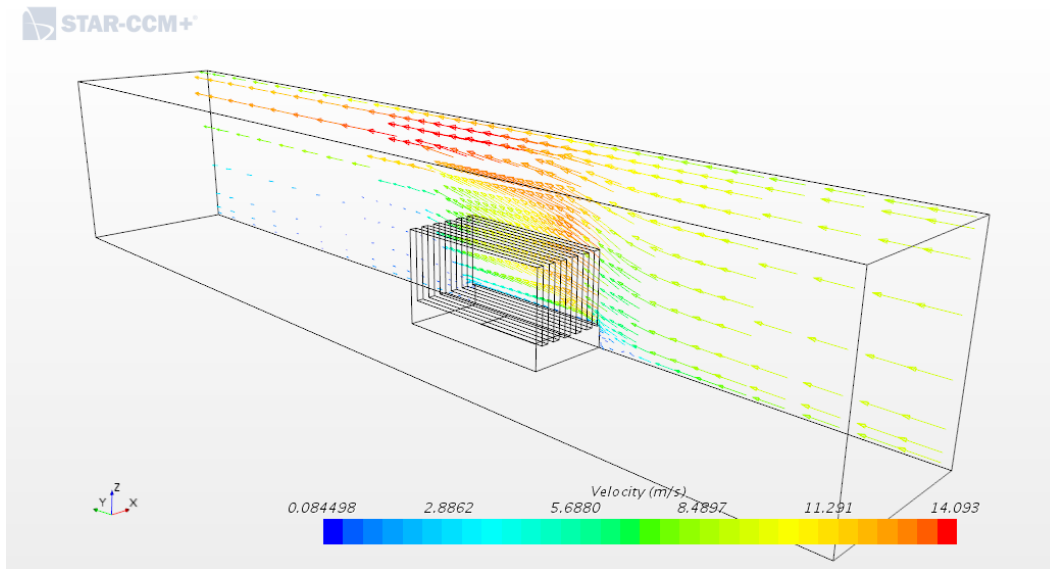


Figure 16 Velocity vectors on the symmetry plane after 200 iterations of the coarse mesh solution.

Enhance the velocity field scene by showing velocity vectors on a horizontal plane.

- Create a horizontal plane as a derived part
 - » In the simulation tree, right-click *Derived Parts* → *New Part* → *Section* → *Plane...*
 - » To position the plane, use your mouse, or enter coordinates in the panel. The following table shows helpful values for the plane parameters. Note that the values of the *x* and *y* origin are not important.

	Origin	Normal
x	0 cm	0 m
y	3 cm	0 m
z	1 cm	1 m

- » Click *Create*; Click *Close*.
- » Right click on the newly-created plane node under *Derived Parts*, and select *Rename*. Change the name to “horizontal plane”.
- Display velocity vectors on the newly-created plane
 - » In the *Velocity Field* scene, select the *Parts* sub-sub node of the *Vector 1* sub-node of the *Displayers* node.
 - » Click on the [...] box in the *Properties* pane.
 - » Select the horizontal plane under *Derived Parts*.
 - » Under the *Displayers* node of the *Velocity Field*, a new *Section Geometry 1* node appears. Select that node.
 - » In the *Properties* pane, uncheck *Surface* and check *Outline*.

The *Velocity Field* scene should now look like Figure 17.

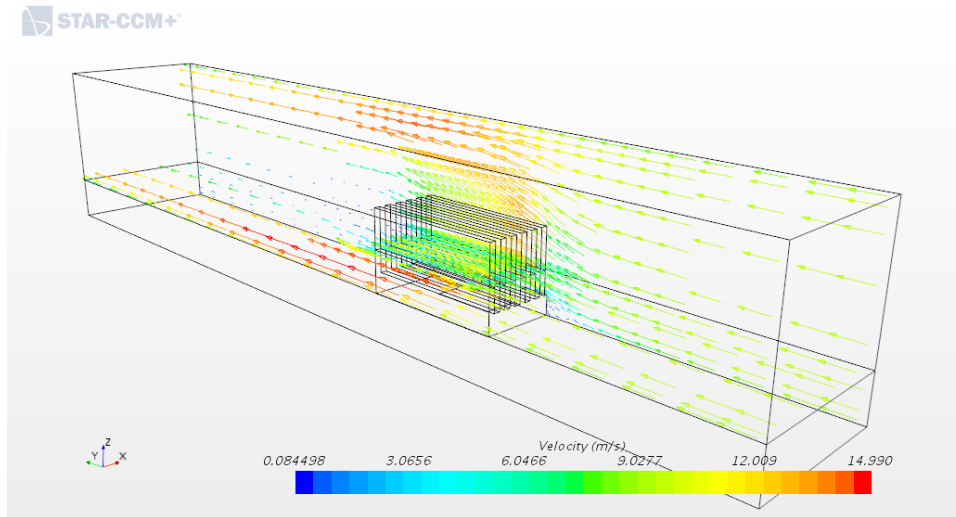


Figure 17 Velocity vectors on symmetry plane and horizontal plane approximately midway through the fin.

4. Create a Heat Transfer Report

A heat transfer report is useful for checking the energy balance and for extracting the heat transfer coefficient from the fin. Create the report with the preliminary solution to make sure the energy balance is closed. Later, we'll rerun the report on finer meshes as one way of assessing the grid independence of the solution.

- In the simulation tree, right-click Reports → New Report → Heat Transfer. Note that there may be an intermediate option [Element Count...X-Velocity Correction].
- Select the newly-created Heat Transfer 1 node
 - » In the *Properties* pane, click on the [...] box of the *Parts* properties
 - » In the pop-up window Select all of the Regions, and click OK
 - » Right-click on Heat Transfer 1 (under the Reports node) and select Run Report

The report output appears in Figure 18.

Heat Transfer of Boundary Heat Transfer on Volume Mesh		
Part	Value (W)	Errors
air: duct_air_symmetry.air contact with fin	0.000000e+00	no data
air: duct_air_symmetry.air contact with fin [Interface 1]	-9.988939e-01	
air: duct_air_symmetry.bottom wall	0.000000e+00	
air: duct_air_symmetry.inlet	-4.517792e+03	
air: duct_air_symmetry.outlet	4.518517e+03	
air: duct_air_symmetry.side wall	0.000000e+00	
air: duct_air_symmetry.symmetry plane	0.000000e+00	
air: duct_air_symmetry.top wall	0.000000e+00	
fin: short_fin_heater_symmetry.base	0.000000e+00	
fin: short_fin_heater_symmetry.fin contact with air	0.000000e+00	no data
fin: short_fin_heater_symmetry.fin contact with air [Interface 1]	9.988773e-01	
fin: short_fin_heater_symmetry.heater	-1.000000e+00	
fin: short_fin_heater_symmetry.symmetry plane	0.000000e+00	unable to compute
Total:	-2.750270e-01	incomplete computation

Figure 18 Heat transfer report after 200 iterations of the coarse mesh model.

Run the Model to Tighter Convergence

If we let the model run for up to 800 iterations, the energy and continuity criteria are met after about 400 iterations, as shown in Figure 19.

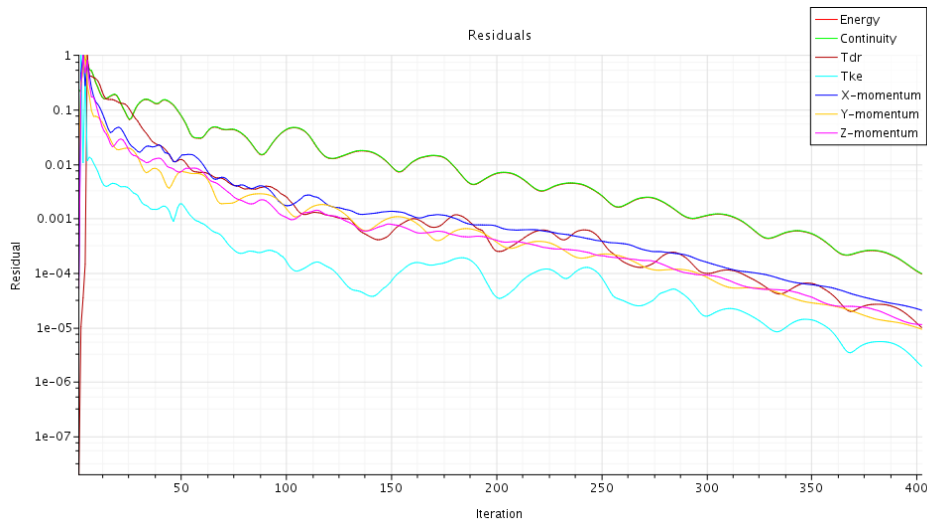


Figure 19 Coarse model convergence after 400 iterations.

Surface Average of Area: Magnitude on Volume Mesh

Part	Value (m ²)	Errors
fin: short_fin_heater_symmetry.base	4.764673e-06	
fin: short_fin_heater_symmetry.fin contact with air	0.000000e+00	no data
fin: short_fin_heater_symmetry.fin contact with air [Interface 1]	1.200161e-06	
fin: short_fin_heater_symmetry.heater	5.126977e-06	
fin: short_fin_heater_symmetry.symmetry plane	1.051476e-06	
Total:	1.463068e-06	

Figure 20 Area totals for surfaces of the fin.

Heat Transfer of Boundary Heat Transfer on Volume Mesh

Part	Value (W)	Errors
air: duct_air_symmetry.air contact with fin	0.000000e+00	no data
air: duct_air_symmetry.air contact with fin [Interface 1]	-9.996974e-01	
air: duct_air_symmetry.bottom wall	0.000000e+00	
air: duct_air_symmetry.inlet	-4.517523e+03	
air: duct_air_symmetry.outlet	4.518513e+03	
air: duct_air_symmetry.side wall	0.000000e+00	
air: duct_air_symmetry.symmetry plane	0.000000e+00	
air: duct_air_symmetry.top wall	0.000000e+00	
fin: short_fin_heater_symmetry.base	0.000000e+00	
fin: short_fin_heater_symmetry.fin contact with air	0.000000e+00	no data
fin: short_fin_heater_symmetry.fin contact with air [Interface 1]	9.998074e-01	
fin: short_fin_heater_symmetry.heater	-1.000000e+00	
fin: short_fin_heater_symmetry.symmetry plane	0.000000e+00	unable to compute
Total:	-1.040719e-02	incomplete computation

Figure 21 Heat transfer report after convergence of the coarse mesh model.