Tutorial: Using Macro Heat Exchanger Model

Introduction

In a typical heat exchanger core, the auxillary fluid temperature is stratified in the direction of the auxillary fluid flow. As a result, heat rejection is not constant over the entire core. In ANSYS FLUENT the fluid zone representing the heat exchanger core is subdivided into macroscopic cells or macros along the auxillary fluid path. The auxillary fluid inlet temperature to each macro is computed and subsequently used to compute the heat rejection from each macro. This approach provided a realistic heat rejection distribution over the heat exchanger core.

Results obtained from the heat exchanger model are very important in the design of cooling modules. So it is important to make sure that the heat exchanger model is used properly.

This tutorial demonstrates the following:

- Set up the heat exchanger model.
- Run the case in ANSYS FLUENT.
- Use a simple example and change many input parameters to see if you get expected results.
- Inherent limitations in the macro model.

Prerequisites

This tutorial is written with the assumption that you have completed Tutorial 1 from the ANSYS FLUENT 13.0 Tutorial Guide, and that you are familiar with the ANSYS FLUENT navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

In this tutorialyou will use the macro heat exchanger model. For details about this model, see Section 6.1 The Macro Heat Exchanger Models in ANSYS FLUENT 13.0 Theroy Guide and Section 15.1 Overview and Restrictions of the Macro Heat Exchanger Models in ANSYS FLUENT 13.0 User's Guide.

Problem Description

A single pass heat exchanger is shown in Figure 1 (front and side view).

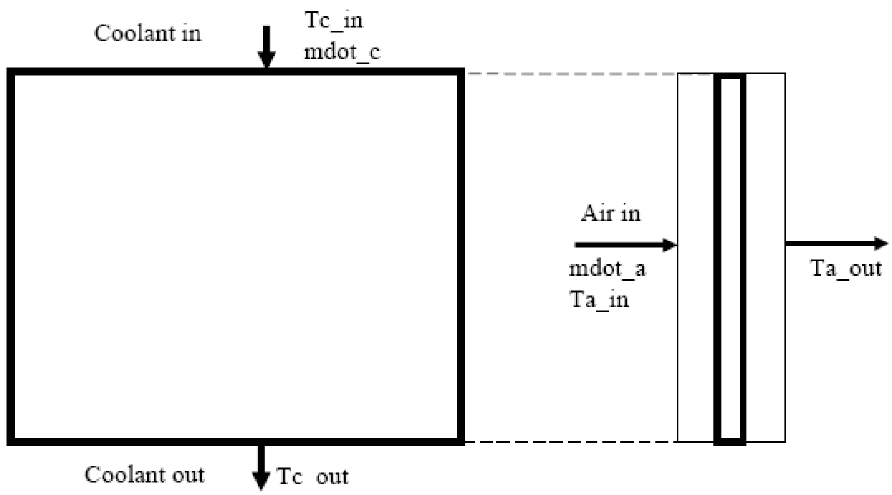


Figure 1: Single Pass Heat Exchanger

The following table shows the radiator performance data.

Air Inlet Temperature (Ta_in)	48.89 C
Coolant Inlet Temperature(Tc_in)	115.56 C
Air Mass Flow Rate (kg/s) (mdot_a)	1.140
Coolant Flow Rate (kg/s) (mdot_c)	2.870
Total Heat Rejection (Watts)	57345.960

Table 1: Radiator Performance Data

Since the operating conditions (mdot_c and mdot_a) are the same as one of the heat exchanger data, and since the operating inlet temperatures (Tc_in and Ta_in) are the same as the ones used to obtain the data (48.89 and 115.56 C respectively), we should get the same total heat rejection as the data, which is 57346 Watts. This will become more clear in the next tutorial.

Preparation

1. Copy the files wedge.msh.gz and rad.tab to the working folder.

2. Use FLUENT Launcher to start the 3D version of ANSYS FLUENT.

3. Enable Double-Precision in the Display Options list.

   For more information about FLUENT Launcher *see Section* 1.1.2 Starting ANSYS FLUENT Using FLUENT Launcher *in the* ANSYS FLUENT *13.0 User's Guide.*

Setup and Solution

Step 1: Mesh

1. Read the mesh file, wedge.msh.gz.

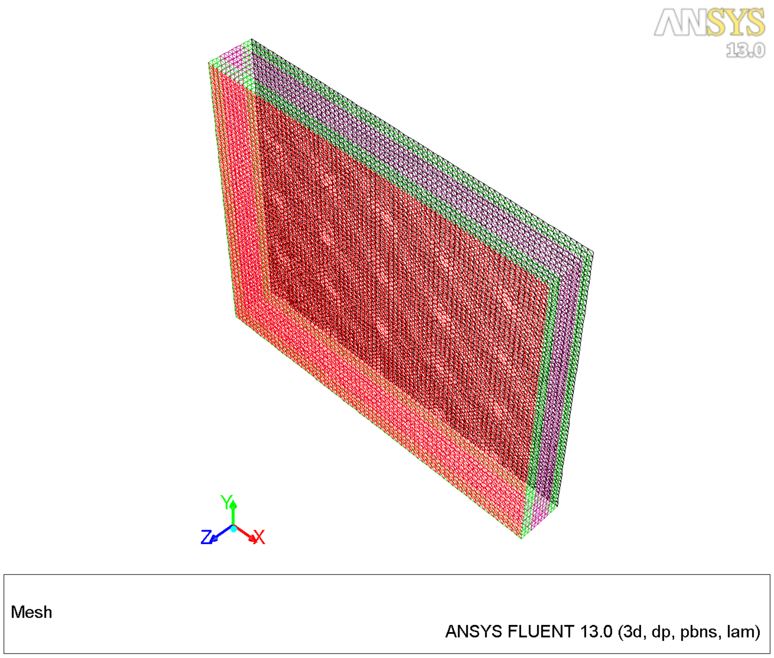


Figure 2: Mesh

Step 2: General Settings

1. Retain the default solver settings.

   General

2. Check the mesh.

   General ⟶ Check

3. Scale the grid.

   General ⟶ Scale...

   (a) Select mm from Mesh Was Created In drop-down list.

   (b) Click Scale and close the Scale Mesh dialog box.

Step 3: Models

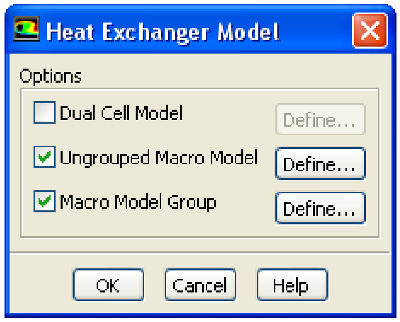
1. Enable the Energy Equation.

   Models ⟶ Energy ⟶ Edit...

   You will keep the flow laminar.

2. Enable the Heat Exchanger model.

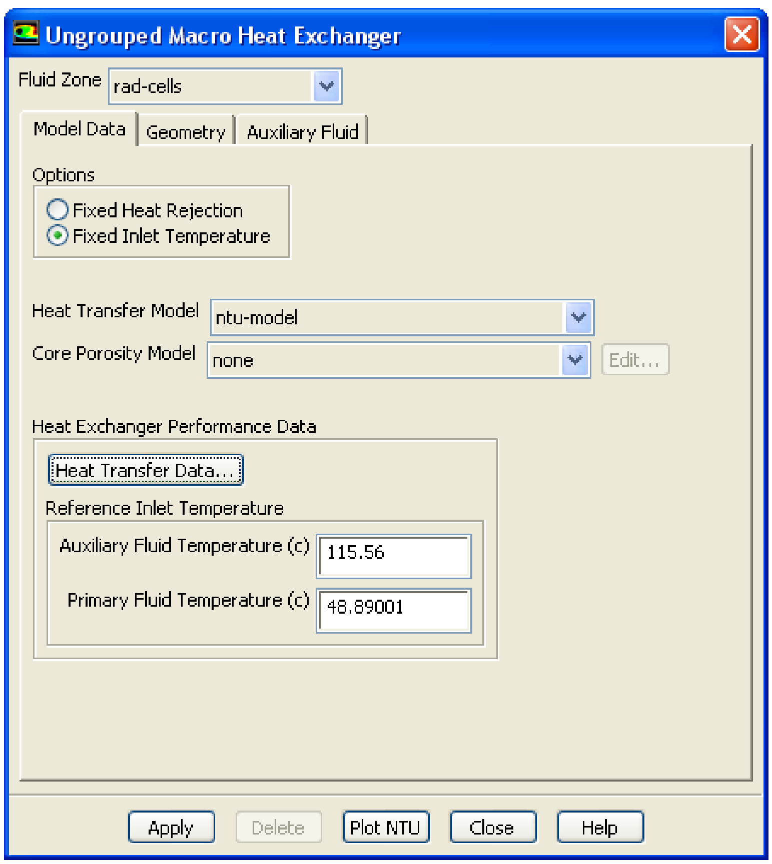
   Models → Heat Exchanger → Edit...



   (a) Enable Ungrouped Macro Model.

   (b) Enable the Macro Model Group.

   (c) Click Define... next to Ungrouped Macro Model.



   i. Enable Fixed Inlet Temperature from the Options group.

   ii. Enter `115.56` for Auxiliary Fluid Temperature.

   iii. Enter `48.89001` for Primary Fluid Temperature.

   iv. Click on the Heat Transfer Data.

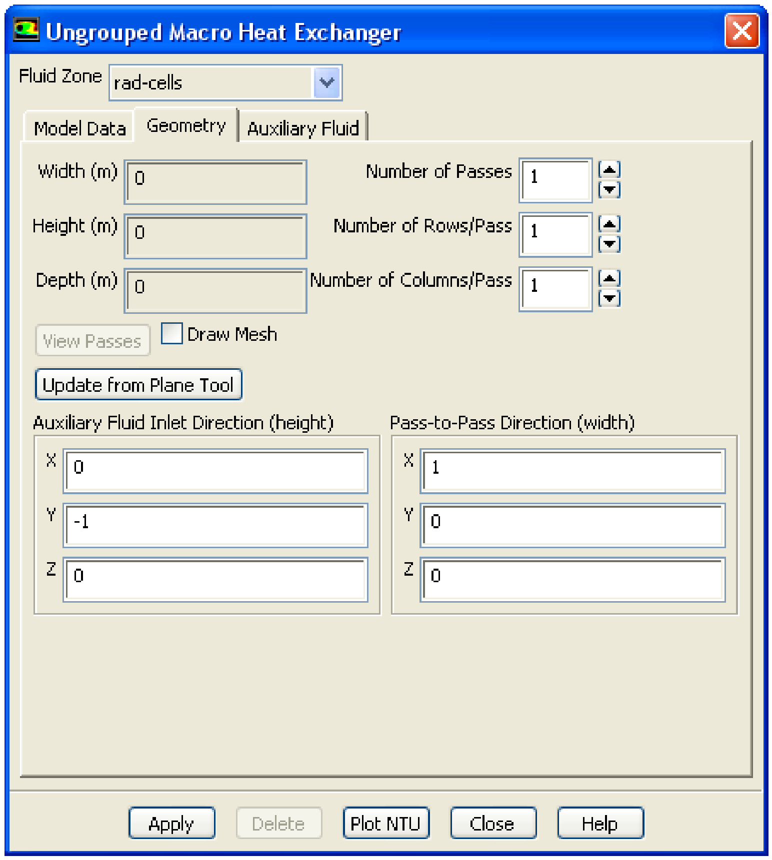
A. Click on Read....

B. Select file rad.tab and click OK.

C. The Heat Transfer Data Table dialog box is updated.

D. Click OK.

v. Click the Geometry tab.



A. Set Number of Passes to 1.

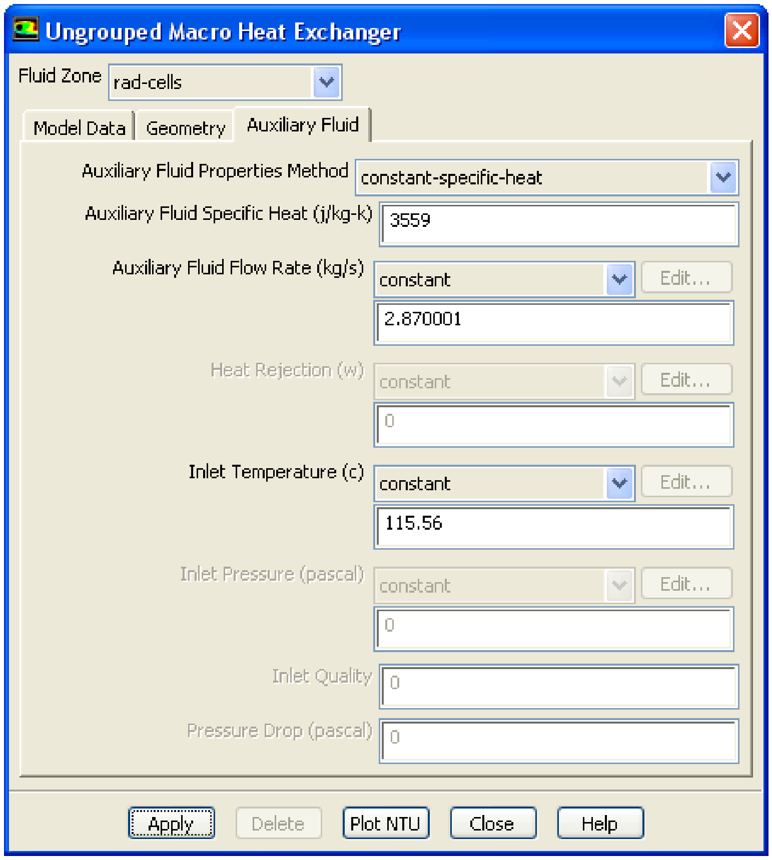
B. Set Number of Rows/Pass to 1.

C. Retain 1 for Number of Columns/Pass.

D. In the Auxiliary Fluid Inlet Direction (height) group set the X, Y, and Z values to `0, -1,` and `0` respectively.

E. Similarly, in the Pass-to-Pass Direction (width) group set the X, Y, and Z values to `1, 0,` and `0` respectively.

vi. Click on the Auxiliary Fluid tab.



A. Enter `3559` for Auxiliary Fluid Specific Heat (j/kg-k).

B. Enter `2.870001` for Auxiliary Fluid Flow Rate (kg/s) (mdot_c).

C. Enter `115.56` for Inlet Temperature (c) (Tc_in).

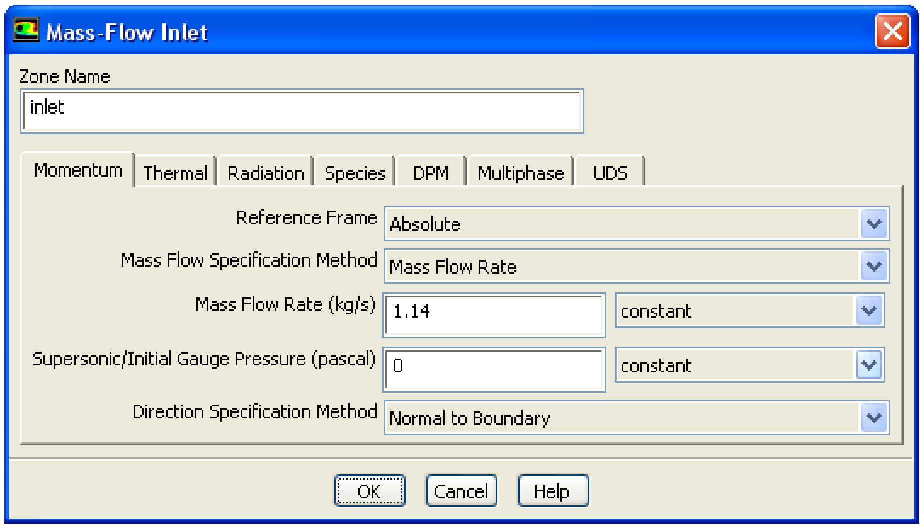
D. Click Apply and close the Ungrouped Macro Heat Exchanger dialog box

(d) Click OK to close the Heat Exchanger Model dialog box.

Step 4: Boundary Conditions

1. Set the boundary conditions for the inlet.

   Boundary Conditions ⟶ inlet ⟶ Edit...



   (a) Enter 1.14 for Mass Flow Rate (Mdot_a).

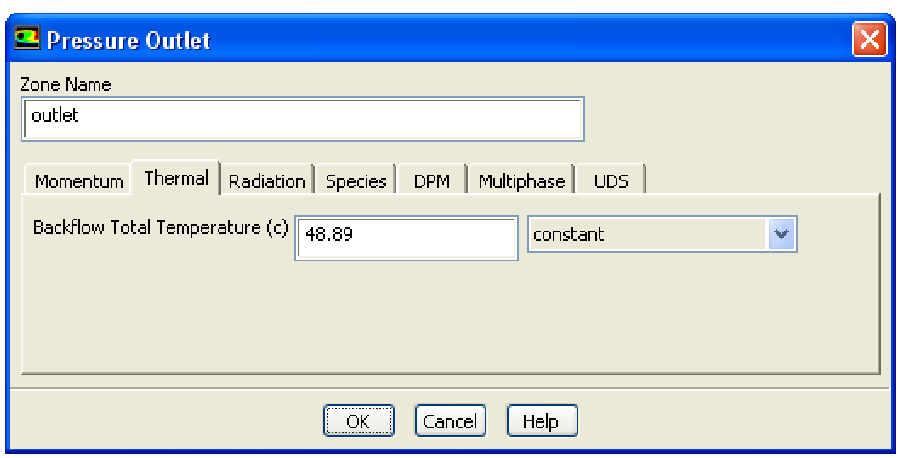
   (b) Select Normal to Boundary from Direction Specification Method drop-down list.

   (c) Click on the Thermal tab and enter 48.89 for Total Temperature (Ta_in).

   (d) Click OK to close the Mass-Flow Inlet dialog box.

2. Set the boundary conditions for the outlet.

   Boundary Conditions ⟶ outlet ⟶ Edit...



   (a) Click on the Thermal tab and enter 48.89 for Backflow Total Temperature.

   (b) Click OK to close the Pressure Outlet dialog box.

Step 5: Solution

1. Set the solution parameters.

   Solution Methods

   (a) Select Green-Gauss Cell Based from the Gradient drop-down list.

2. Select only Energy from the list of equations.

   Solution Controls ⟶ Equations...

3. Enable the plotting of residuals during the calculation.

   Monitors ⟶ Residuals ⟶ Edit...

   (a) Ensure that Print to Console and Plot are enabled.

   (b) Disable Check Convergence in the group of Equations.

   (c) Click OK to close the Residual Monitors dialog box.

4. Initialize the solution.

   Solution Initialization

   (a) Select inlet from the Compute from drop-down list.

   (b) Click Initialize

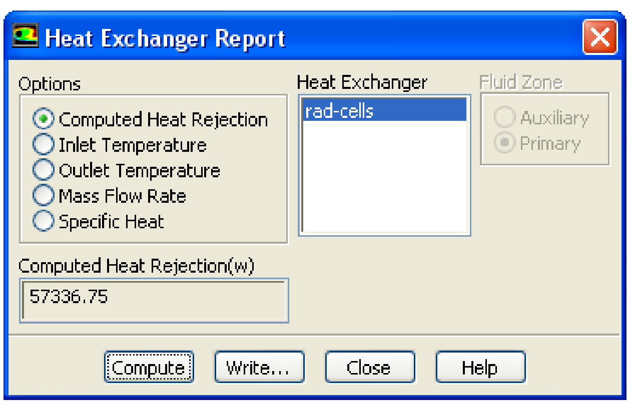
5. Calculate for 20 iterations.

   Run Calculation ⟶ Calculate

Step 6: Postprocessing

1. Compute the total heat rejection rate.

   Reports ⟶ Heat Exchanger ⟶ Set Up...



   (a) Select Computed Heat Rejection from the list of Options.

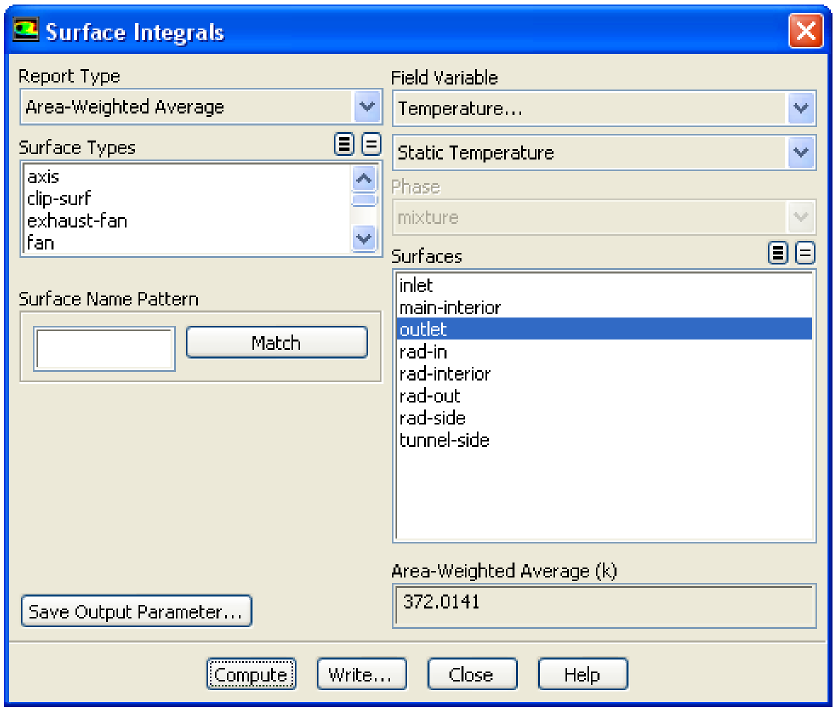
   (b) Click Compute.

(c) Close the Heat Exchanger Report dialog box.

The total heat rejection rate(Q) is 57336.75 *Watts. This is the same value as the heat exchanger data.*

2. Compute the air outlet temperature.

Reports ⟶ Surface Integrals ⟶ Set Up...



(a) Select Area-Weighted Average from the Report Type drop-down list.

(b) Select Temperature and Static Temperature from the Field Variable drop-down list.

(c) Select outlet from the list of Surfaces.

(d) Click Compute

The Area-Weighted Average *temperature is* 372.0141 k*(*Ta_out*).*

(e) Close the Surface Integrals dialog box.

3. Save the case and data files (`wedge1.cas.gz` and `wedge1.dat.gz`).

File ⟶ Write ⟶ Case & Data...

Further Improvements

1. Use more macros.

   (a) Read the case and data file (`wedge1.cas.gz` and `wedge1.dat.gz`).

   Models ⟶ Heat Exchanger ⟶ Edit...

   i. Click Define... next to Ungrouped Macro Model.

   ii. Click on Geometry tab.

   iii. Set Number of Rows/Pass to 60 and Number of Columns/Pass to 70.

   iv. Click Apply and close the Ungrouped Macro Heat Exchanger dialog box.

   (b) Close the Heat Exchanger Model dialog box.

   (c) Run calculation.

   Run Calculation ⟶ Calculate

   (d) Compute the total heat rejection rate.

   Reports ⟶ Heat Exchanger ⟶ Set Up...

   The total heat rejection rate(Q) is 56815.05 *Watts. the result is almost the same. There is only 0.9% under-prediction in total heat rejection. Since, you are scaling each small macro, there are numerical inefficiencies.*

   (e) Display the temperature contours.

   Graphics and Animations ⟶ Contours ⟶ Set Up...

   i. Enable Filled from the Options group box.

   ii. Select Temperature and Static Temperature from the Contours of drop-down list.

   Temperature contours at the outlet gives variation as expected.

2. Similarly read the case and data file again and set the Number of Rows/Pass to 30 and Number of Columns/Pass to 35.

   The results are still the same, but slight improvement due to reduced numerical inefficiencies.

3. Read the case and data file and set Number of Rows/Pass to 31 and Number of Columns/Pass to 36.

   There is about 0.94% under-prediction of total heat rejection, which is tolerable, but the contour of air temperature is not so good. In conclusion choose the Number of Rows/Pass *and* Number of Columns/Pass *(and uniform grid) so that each macro is the same size.*

   Note: *This limitation is inherent in the macro-based models.*

4. Read the case and data file and set Number of Rows/Pass to 120 and Number of Columns/Pass to 140.

Note: *Now you have more macro than available cells. So some macros will be empty. If you click* Apply *you will get a message for each empty macro. The end result is not realistic.*

Note: *This limitation is inherent in the macro-based models.*

5. Predict coolant inlet temperature for a given total heat rejection.

   Note: *The heat exchanger report gives a total heat rejection of* 5.73367e+004 *for coolant inlet temperature of* 115.56. *In this exercise you will predict coolant inlet temperature for a given total heat rejection. The way the algorithm works is that it will predict total heat rejection starting from an initial guess of coolant inlet temperature. If predicted total heat rejection is different from the targeted value, it will adjust the coolant inlet temperature accordingly for next iteration.*

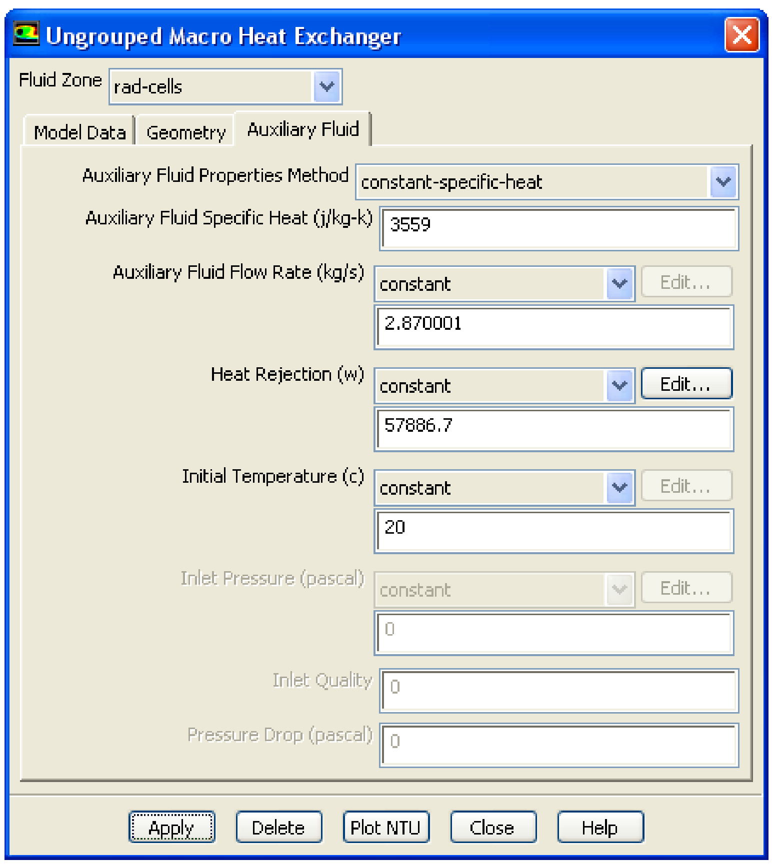
   (a) Read the case and data file.

   (b) Make changes to the heat exchanger model.

   Models ⟶ Heat Exchanger ⟶ Edit...

   (c) Click Define... next to Ungrouped Macro Model.

   i. Select Fixed Heat Rejection from the Options group box.



   ii. Click on Auxiliary Fluid tab and set Heat Rejection and Initial Temperature to `57886.7` and 20 respectively.

(d) Ensure that under-relaxation factor of energy is 1.

Solution Controls

(e) Initialize using value computes from the inlet.

Solution Initialization

i. Select inlet from the Compute from drop-down list.

(f) Run calculation.

Run Calculation ⟶ Calculate

It will take about six iterations for residuals to go below 1e-17. After converging it will predict Tc_in = 343.15 K*, which is way off.*

Summary

The tutorial shows how to set up the heat exchange model, run the case in ANSYS FLUENT and compare the results. Special care must be taken when using the macro-based heat exchanger models.