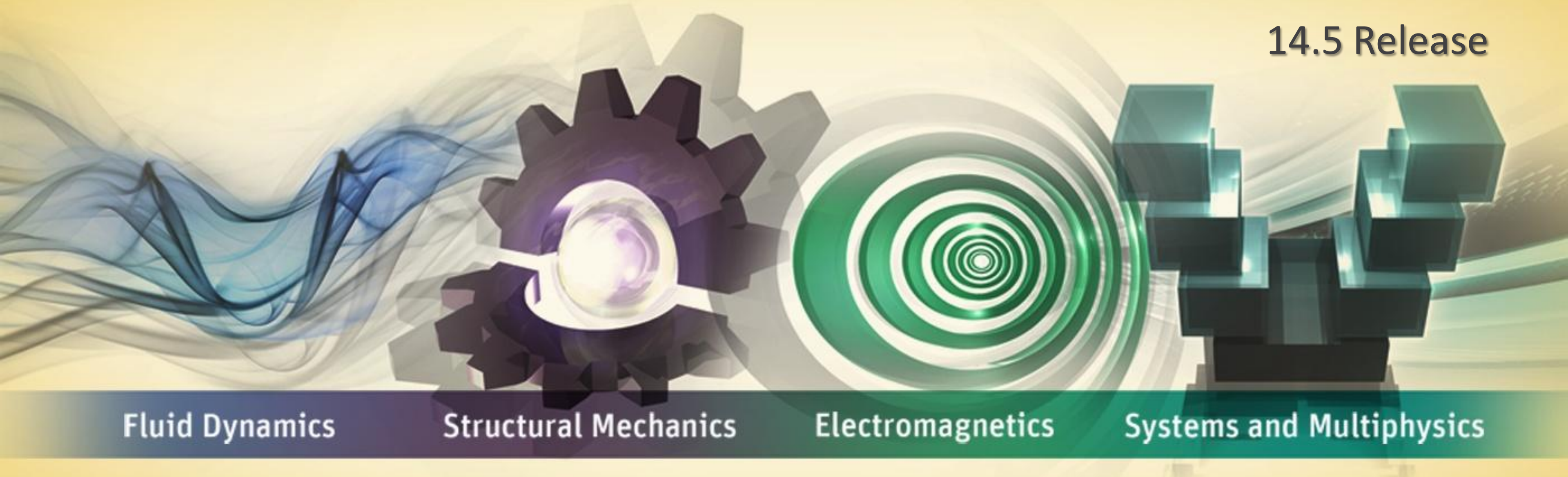
ANSYS

Realize Your Product Promise™

Lecture 7 – Heat Exchangers

14.5 Release



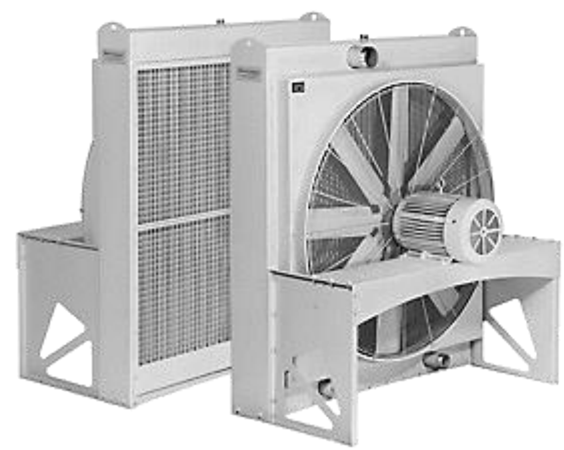
Heat Transfer Modeling using ANSYS FLUENT

Outline

- Introduction
- Simulation of Heat Exchangers
- Heat Exchanger Models in ANSYS Fluent 14.5
- Summary

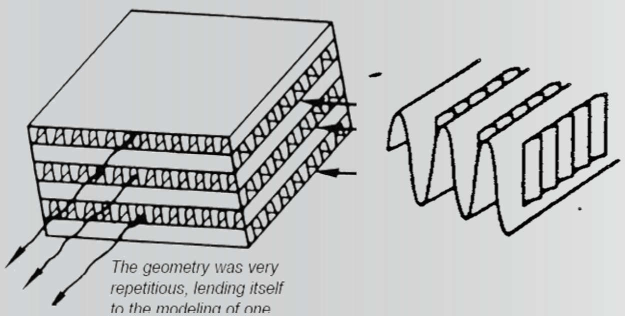
Introduction

- **Heat Exchangers are widely used in industry and need to be taken into account in many CFD calculations**
  - Boiler
  - Condenser
  - Radiators
  - Coolers

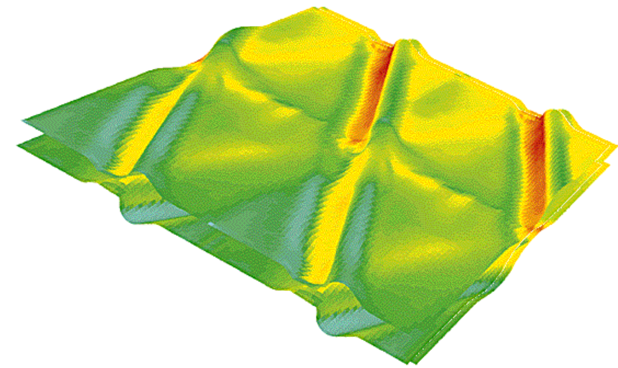


Introduction

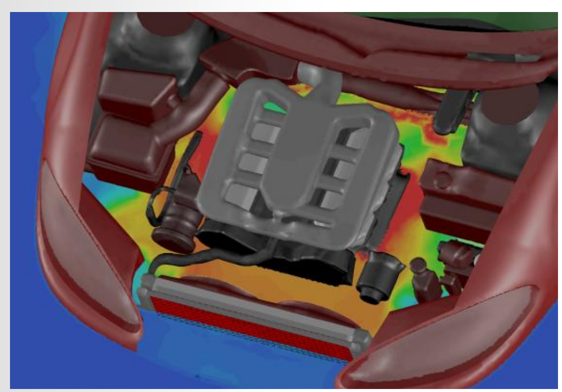
- **CFD can be used to calculate**
  - Local heat transfer near the heat exchanger walls



Local HTC Prediction on a Corrugated Plate Heat Exchanger



  - Global influence of the heat exchanger on its environment



Introduction

- **Heat exchanger geometries are generally complex and cannot be included in the CFD domain due to widely varying spatial length scales.**
- **Many difficulties will be alleviated if models were available to compute:**
  - Pressure loss generated by the heat exchanger for the primary flow
  - Heat transfer between the primary and auxiliary flows

Outline

- Introduction
- **Simulation of Heat Exchangers**
- Heat Exchanger Models in FLUENT 14.5
- Summary

Simulation of Heat Exchangers

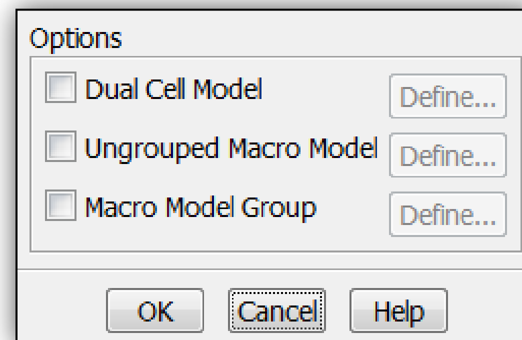
- **Multiple models are available to simulate heat exchangers.**
- **Models range from very simple to very complex.**
  - Radiator model
    - Surface zone
    - Specific condition built-in
    - Global pressure loss and heat transfer calculation
    - User input – pressure loss coefficient, heat transfer coefficient, radiator temperature or heat flux
  - Porous Media + Energy Source for fluid zone
    - Volume zone (fluid)
    - Non specific to heat exchanger
    - UDF can be used to defined velocity, position or time dependent profile
    - Refer to lecture on Heat Transfer in Porous Media

Outline

- Introduction
- Simulation of Heat Exchangers
- **Heat Exchanger Models in FLUENT 14.5**
- Summary

Simulation of Heat Exchangers

- **Macro Models (ungrouped and grouped)**
  - ANSYS FLUENT allows you to chose between two heat transfer models
    - Simple effectiveness: The coolant can be single phase or two-phase
    - Number of Transfer Units (NTU)
  - A 1D flow is assumed for the auxiliary or coolant flow
- **Dual-Cell-Based Heat Exchanger Models**
  - Uses the NTU method for heat transfer calculation
  - Two volume zones defined on top of one another
    - Primary flow
    - Auxiliary flow
  - Allows more flexibility as far as the shape of the heat exchanger is concerned

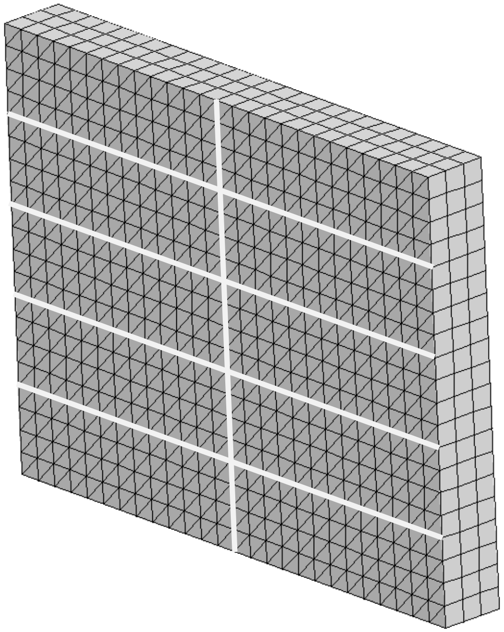


Macro Heat Exchanger Models Overview

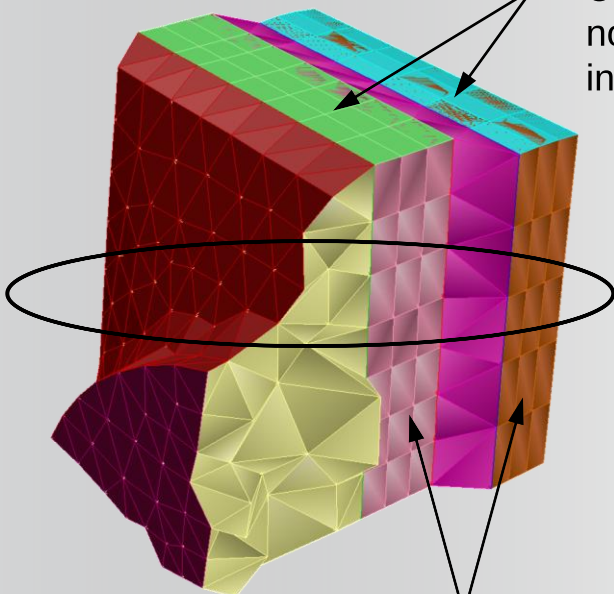
- **Auxiliary fluid temperature in heat exchanger is stratified.**
  - The “coolant” is referred to as the “auxiliary fluid.”
- **Heat rejection not constant over core**
- **Heat exchanger split into macroscopic cells or “macros” to account for non-constant heat rejection**
- **The primary flow is unidirectional and must be aligned with one of the three orthogonal axes defined by the rectangular core**
- **Auxiliary fluid flow rate is assumed to be one-dimensional.**

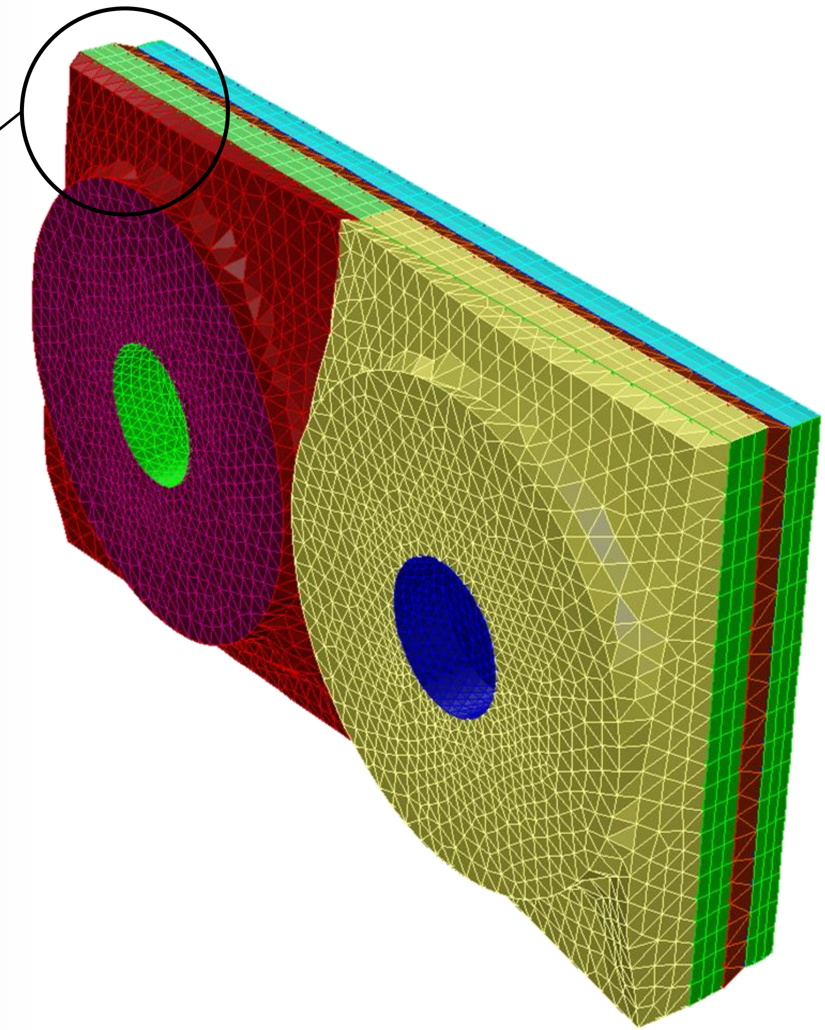
Mesh Considerations

- **The core must be approximately rectangular in shape.**
- **Evenly distributed Hex/Wedge cells must be used.**
  - Number of cells in the three coordinate directions must be based on macro discretization.
  - Equal number of cells in each macro
    - Quad or wedge elements are recommended (no pyramids)
- **Non-conformal interfaces can be used to connect sides with neighboring tetrahedral mesh.**



Mesh Considerations





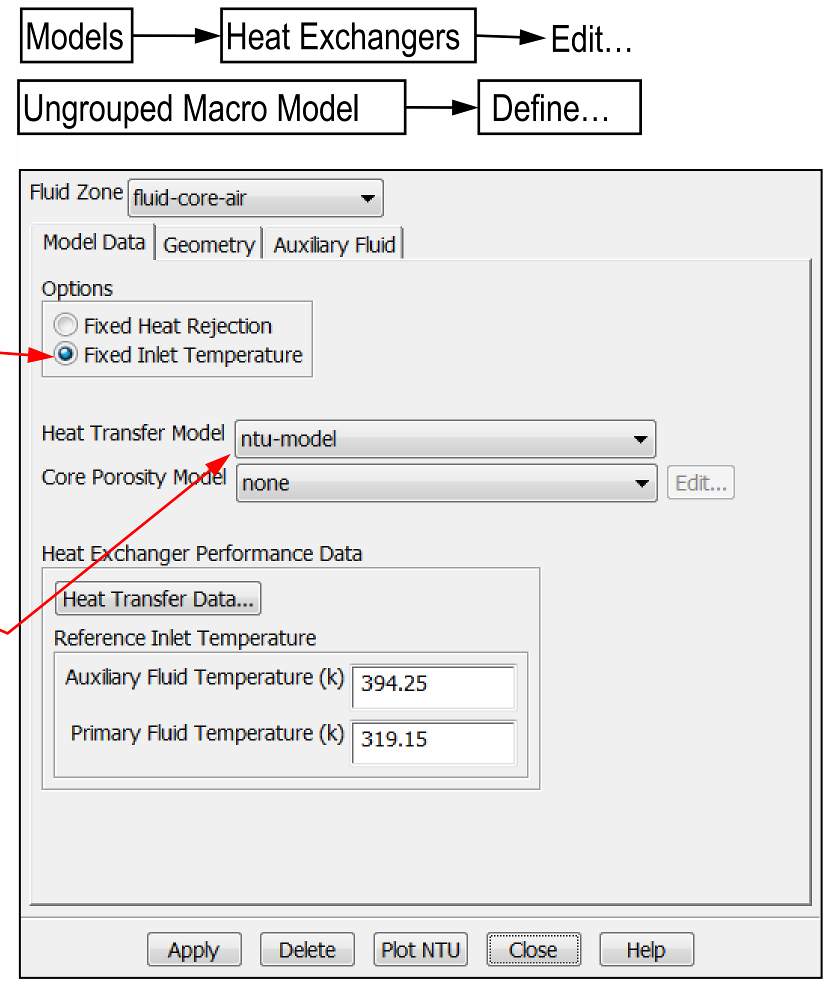
A tutorial is available on how to prepare the mesh

Models and Options

- **Heat exchanger conditions saved in**

  `file/write-bc`

- **Heat transfer options**
  - Fixed heat rejection
  - Fixed auxiliary inlet temperature
  - When one is fixed, the other is computed
- **Selection of the heat exchanger model**
  - Simple effectiveness
  - NTU-based
  - Both available in parallel



Simple Effectiveness Model

- **Can be used to model heat transfer from the auxiliary fluid to the gas**
  - The primary fluid capacity rate must be lower than the auxiliary fluid capacity rate

$$\left(\dot{m}C_p\right)_{\text{auxiliary}} > \left(\dot{m}C_p\right)_{\text{primary}}$$

  - Auxiliary fluid must be hotter than the primary fluid (otherwise a UDF is required)
- **Interpolate the effectiveness from a curve of velocity vs. effectiveness (provided by the user)**
  - Alternatively, a global effectiveness can be provided.
- **Auxiliary fluid may be single-phase or two-phase.**
  - Properties can be a function of temperature and pressure.
  - Phase change can be modeled.

Simple Effectiveness Model

- **Rate of heat transfer**

$$q = \varepsilon C_{\text{min}} (T_{\text{in,hot}} - T_{\text{in,cold}})$$

$$C_{\text{min}} = (\dot{m} C_p)_{\text{primary}}$$

- **Global efficiency is applied at each cell**

$$q_{\text{cell}} = \varepsilon \left(\dot{m} C_p\right)_{\text{pri}} \left(T_{\text{in,aux}} - T_{\text{cell}}\right)$$

$$q_{\text{macro}} = \sum_{\substack{\text{all cells} \\ \text{in macro}}} q_{\text{cell}}$$

- **Inlet temperature of macro can be defined**
  - As boundary condition for first macro
  - Equal to outlet temperature of previous macro
    - Calculated using energy balance across macro

$$q_m = \left(\dot{m} C_p\right)_{\text{aux}} \left(\underline{T_{\text{out,aux}}} - T_{\text{in,aux}}\right)$$

NTU-Based Model

- **The number of transfer units (NTU) is a dimensionless parameter used in heat exchanger performance analysis.**

$$\mathrm{NTU} = \frac{U\,A}{C_{\min}}$$

- **Can be used to model heat transfer between primary and auxiliary fluids.**
  - Unlike in the simple effectiveness approach, the auxiliary fluid can be either hotter or cooler than primary fluid
- **NTU enables calculation of macro effectiveness**
- **Accounts for primary side reverse flow**
- **Can be used with variable density gases.**

NTU-based Model

- **Global effectiveness:**

$$\varepsilon_{\text{full}} = \frac{q}{C_{\substack{\min\\ \text{full}}}\left(T_{\text{in,aux}} - T_{\text{in,prim}}\right)}$$

- **Relation between ε and NTU (cross flow, both fluids unmixed)**

$$\varepsilon = 1 - \exp\left[-\frac{\text{NTU}^{0.22}}{C_r}\left(1 - e^{-C_r\,\text{NTU}^{0.78}}\right)\right] \qquad C_r = \frac{C_{\min}}{C_{\max}}$$

- **Scaling of NTU**

$$\text{NTU}_m = \text{NTU}_{\text{full}}\frac{V_m}{V_{\text{full}}}\frac{C_{\min,\text{full}}}{C_{\min,m}}$$

$$q_m = \varepsilon_m C_{\min}\left(T_{\text{in,m}} - T_{\text{in,prim}}\right) \qquad q_{\text{cell}} = q_m\frac{V_{\text{cell}}}{V_m}$$

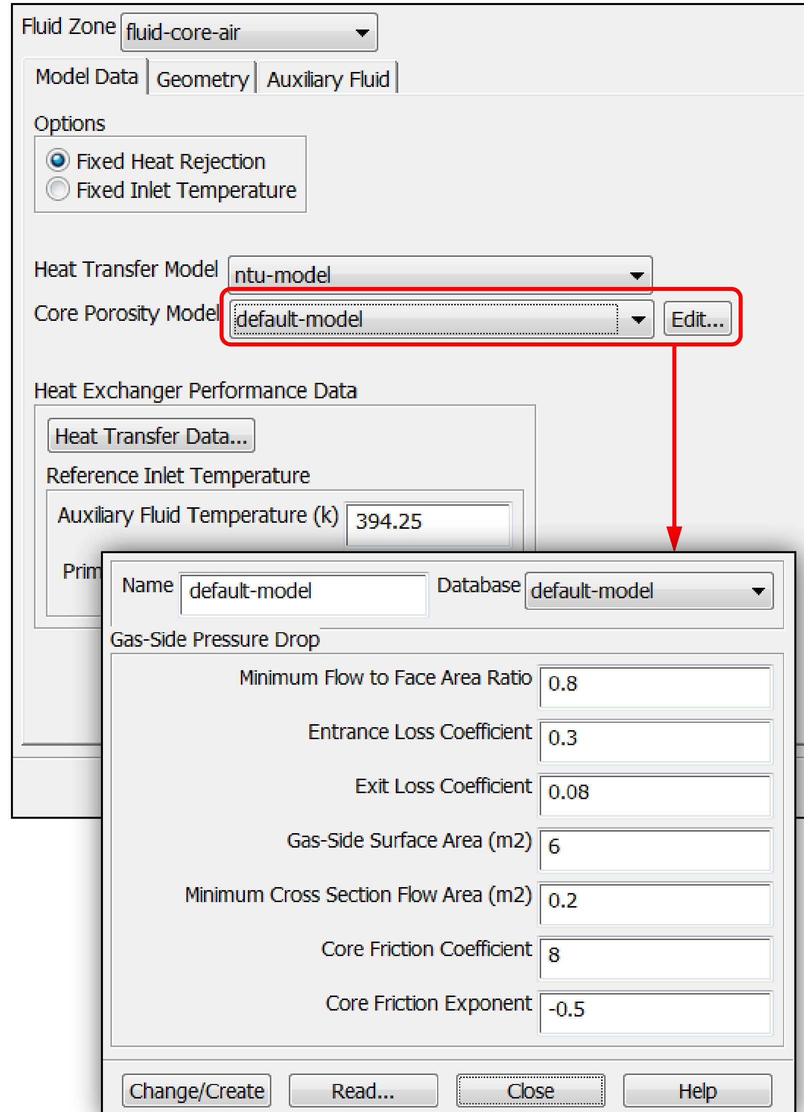
- **Energy balance across macro to determine temperature**

$$q_m = \left(\dot{m}c_p\right)_{\text{aux}}\left(T_{\text{out,aux}} - T_{\text{in,aux}}\right)$$

Core Porosity Model

- **Pressure drop coefficients can be defined**
  - Automatically using the Core Porosity of the exchanger
    - Pressure drop calculated using geometric description of the exchanger
  - By user-specified heat exchanger parameters (a default model is available).
  - From a data file.
    - Example:

```
("radiator" (0.8 0.3 0.08 6 0.2 8 -0.5))
```



Automatic Core Porosity Model

- **Pressure drop through the (porous) core can be expressed as**

Pressure loss coefficient → f

$$\Delta p = \frac{f \rho_m U_{max}^2}{2}$$

- **The pressure loss coefficient, f, is computed from**

$$f = \left(K_c + 1 - \sigma^2\right) - \left(1 - \sigma^2 - K_e\right)\left(\frac{\nu_e}{\nu_i}\right) + 2\left(\frac{\nu_e}{\nu_i} - 1\right) + f_c\left(\frac{A}{A_c}\right)\left(\frac{\nu_m}{\nu_i}\right)$$

σ = Minimum flow to face area ratio

K_c = Entrance loss coefficient

K_e = Exit loss coefficient

A = Gas side surface area

A_c = Minimum cross-sectional flow area

f_c = Core friction factor

v_e = Exit specific volume

v_i = Inlet specific volume

v_m = Mean specific volume

User-Defined Core Porosity Model

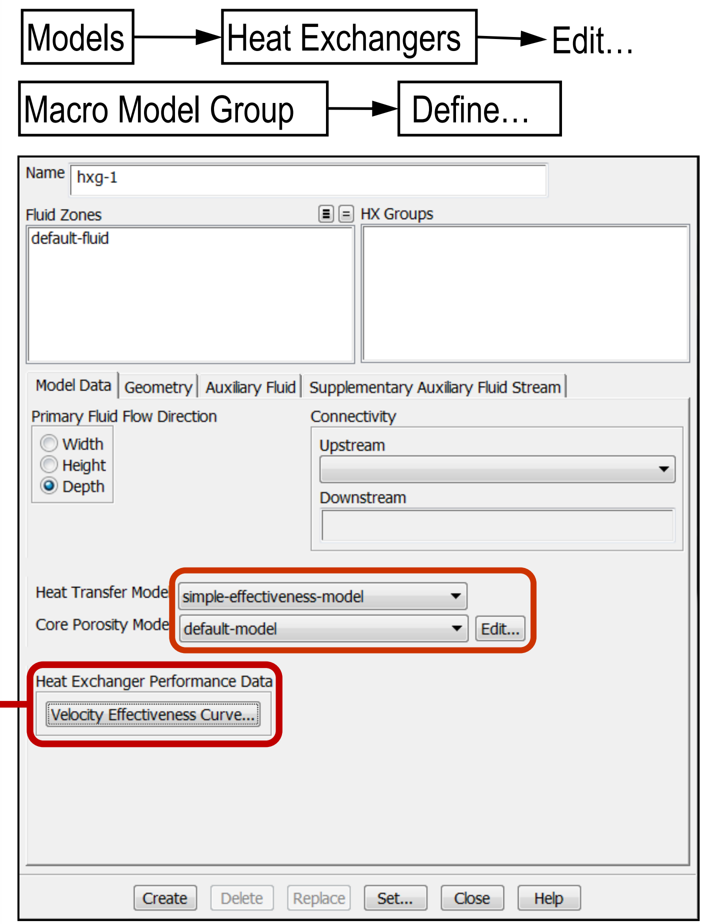
- **The core porosity model can be described manually using the Porous Media option for the fluid zone that represents the heat exchanger.**

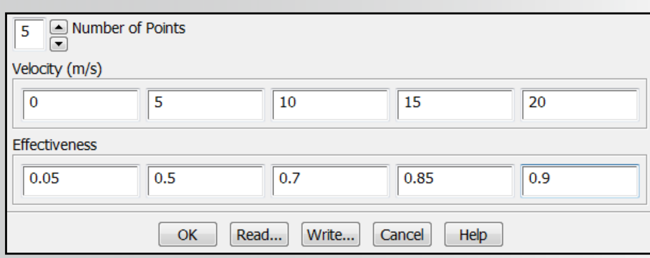
Cell Zone Conditions → Edit…

- Permeability and inertial resistance factor defined by the user.
  - Viscous and inertial resistances
  - Use two orders of magnitude higher in directions 2 and 3 for viscous and inertial resistance parameters (larger values may cause instability).
- The Plane tool can be used to define directions
  - Direction 1 defined by a red arrow in graphics window.

Specification of Exchanger Performance Data

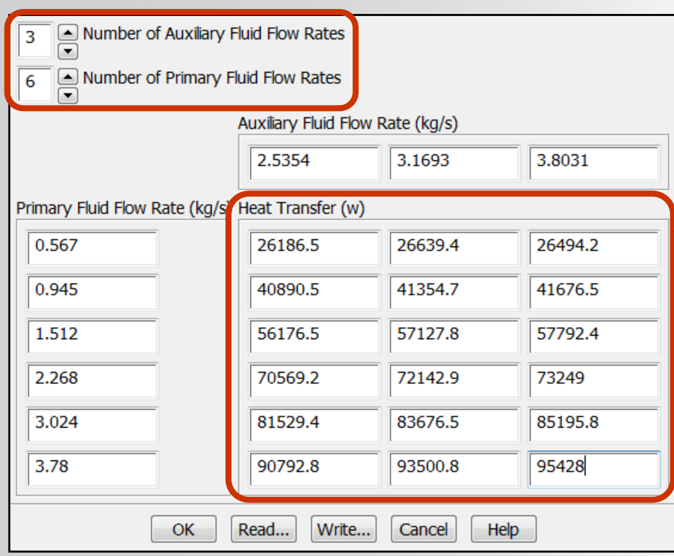
- **Simple effectiveness model**
  - Global effectiveness vs velocity curve defined by user
  - Interpolation to the operating point

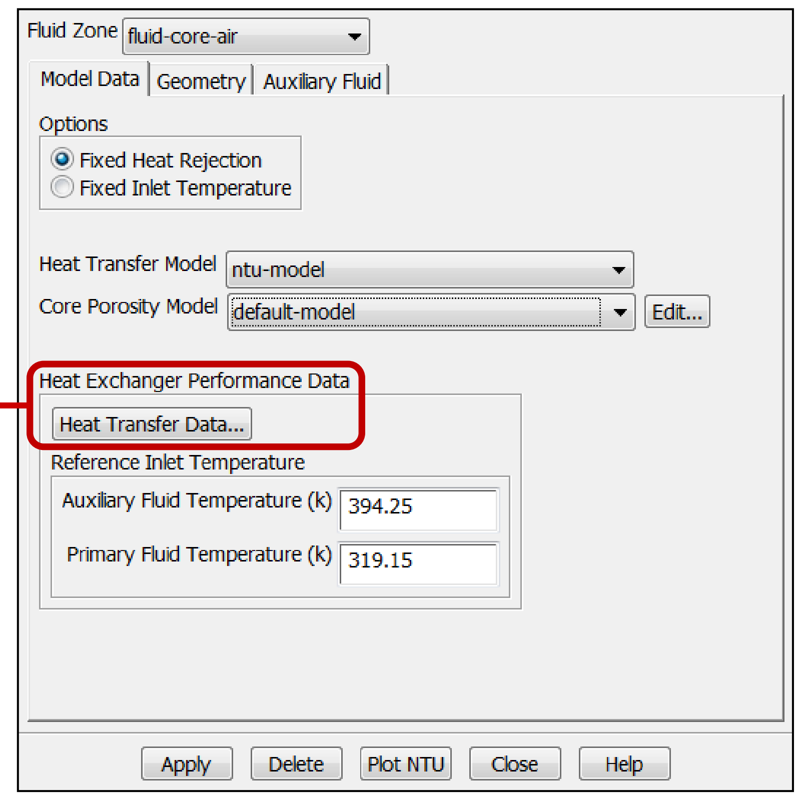




Specification of Exchanger Performance Data

- **NTU-based Model**
  - Raw heat exchanger performance data input
    - User enters mass flow rate vs. heat rejection.
    - FLUENT converts to mass flow rate vs NTU
  - Performance curves for multiple auxiliary fluid mass flow rates
    - Linear interpolation between curves

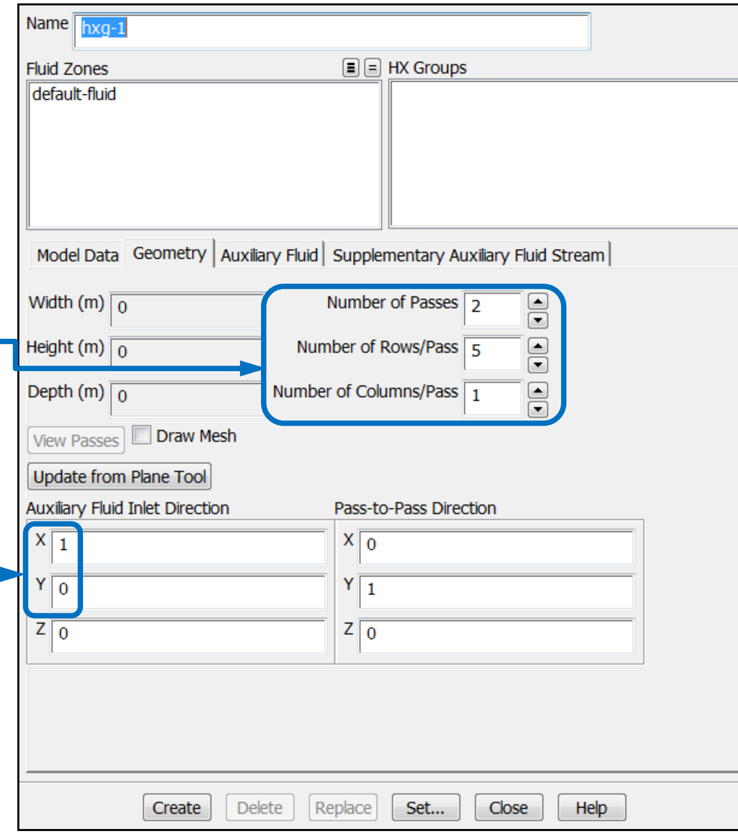




Defining the Macros

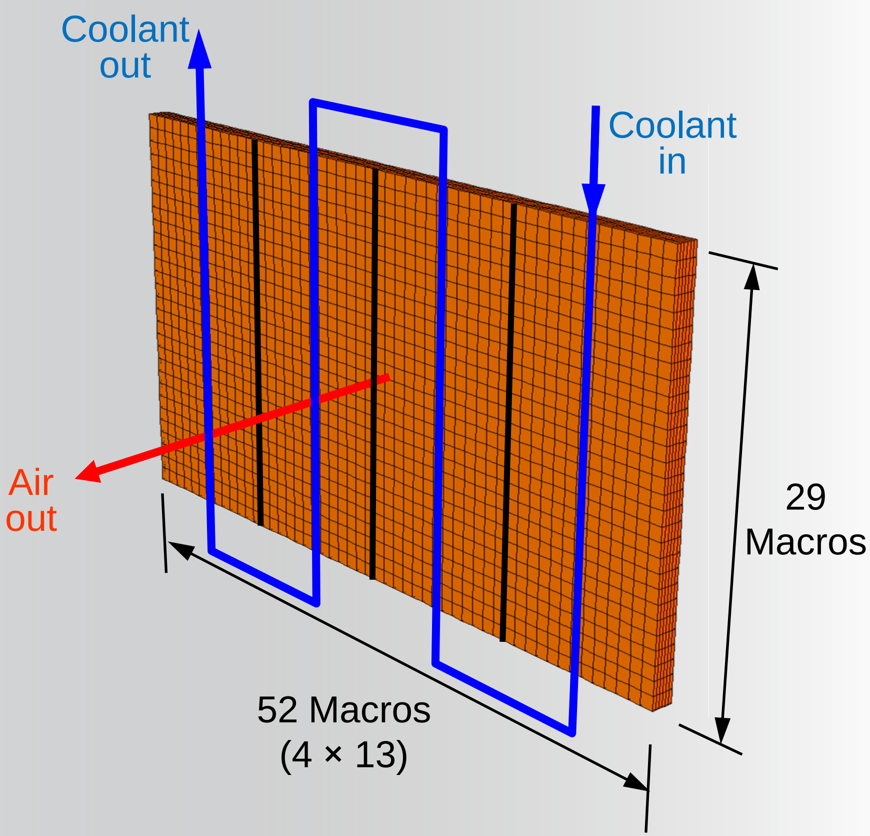
- **Heat exchanger core split into macros**
  - Macros are constructed based on
    - Number of passes
    - Number of macro rows per pass
      - Along auxiliary flow direction
    - Number of macro columns per pass
      - Along pass-to-pass direction
    - Auxiliary fluid inlet direction
    - Pass-to-pass direction

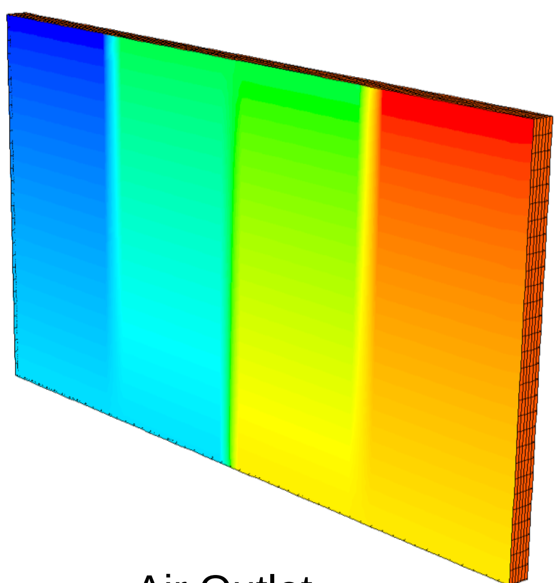
Note: Discretization in two directions only



Core Geometry

- **Successive passes are considered to be perpendicular to the primary fluid flow direction.**
  - Example – Four-pass exchanger with 29 rows and 13 columns.

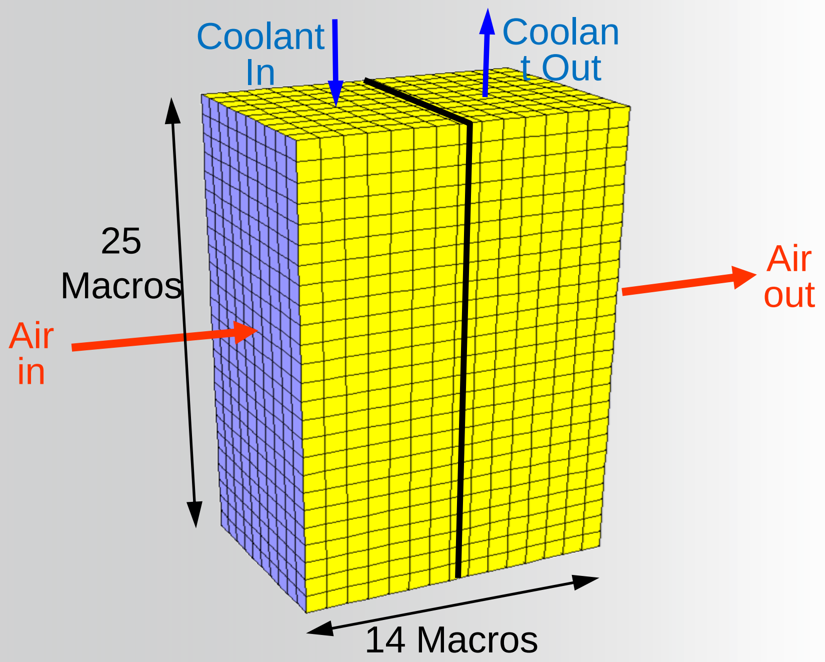


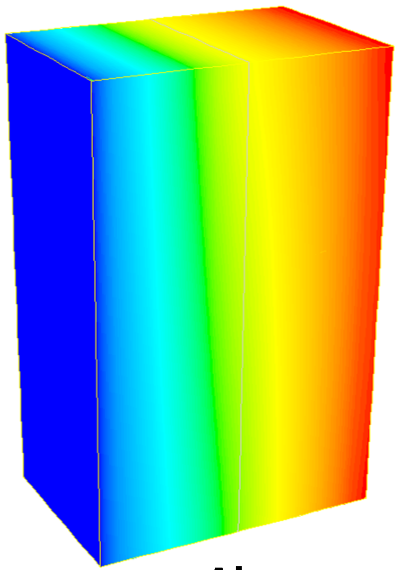


Air Outlet
Temperature

Core Geometry

- **Pass-to-pass is parallel to air flow direction**
  - Only one macro is used along the thickness of HXC
    - Less accurate if air flow distribution is non-uniform
  - Example: Two-pass exchanger with 25 rows and 7 columns





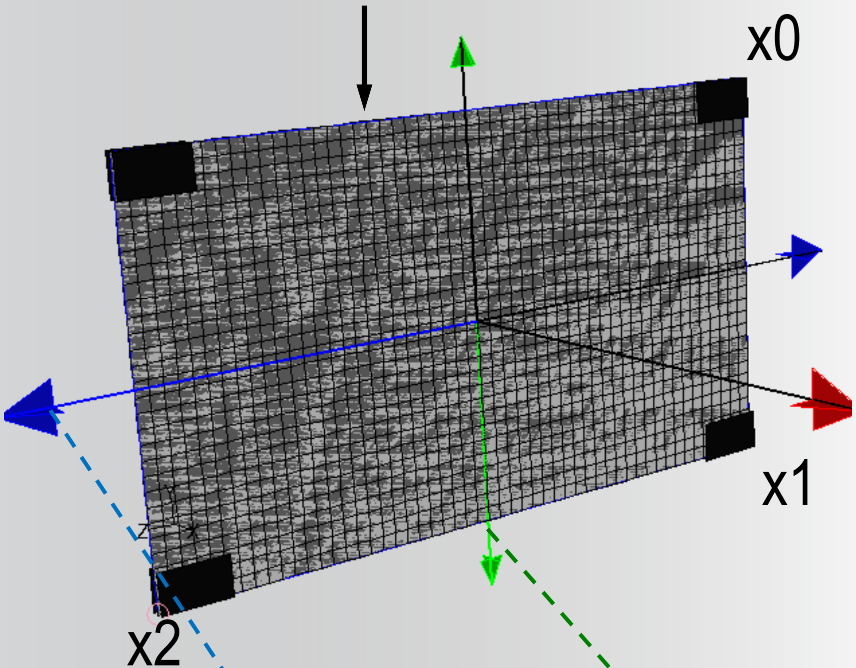
Air Temperature

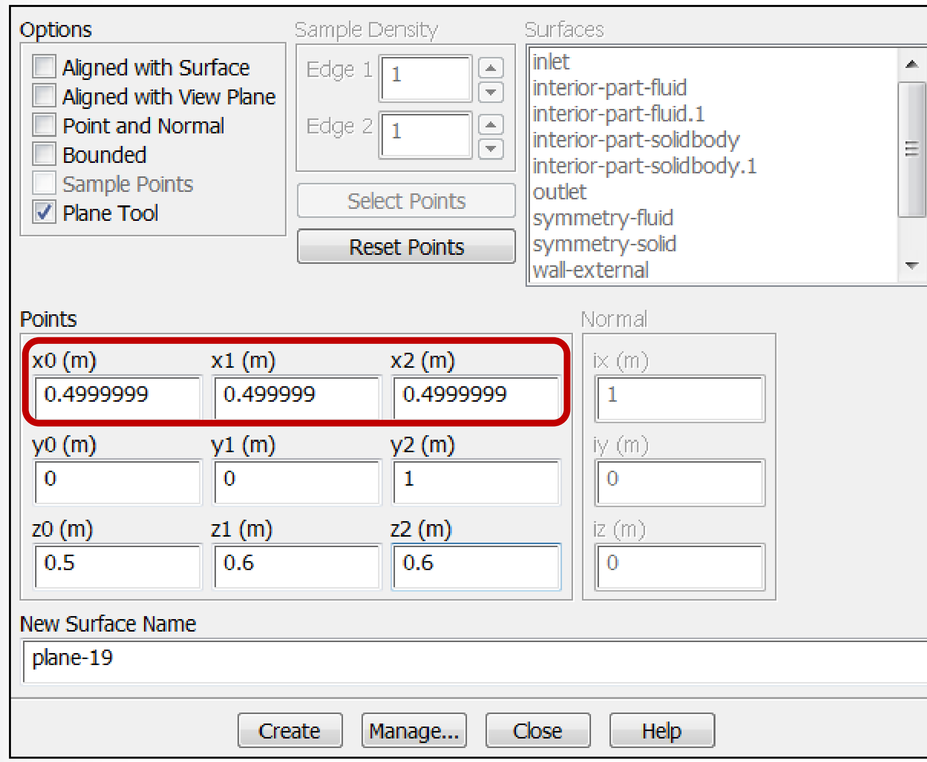
Core Geometry

- **Coolant Inlet Direction and Pass-to-Pass Direction can be defined using the Plane Tool (if the core is not aligned with coordinate directions).**
- **Plane tool setup**
  - Open Surface / Plane panel
  - Select three points to define the exchanger outlet plane
    - x0 to x1 : coolant inlet flow direction
      - Green arrow of the plane tool
    - x1 to x2 : pass-to-pass direction
      - Blue arrow of the plane tool
  - Click on Plane Tool option to define it.
- **Define core geometry by clicking Update from Plane Tool in Heat Exchanger panel**

Core Geometry

HXC Outlet surface





Pass-to-Pass Director

Coolant Inlet Direction

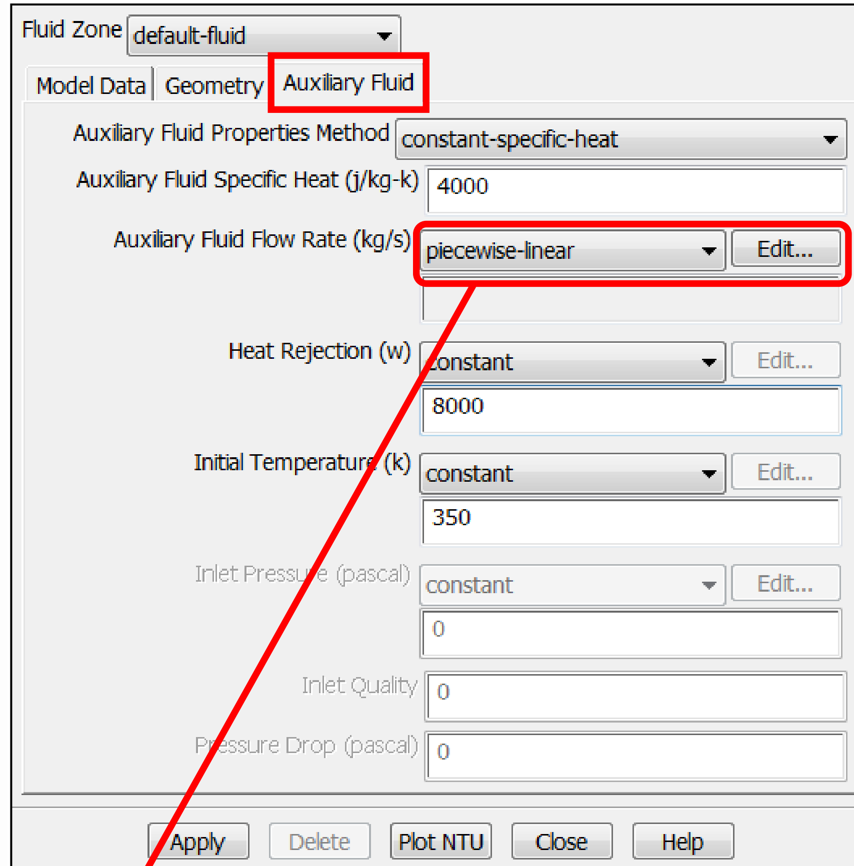
Core Geometry

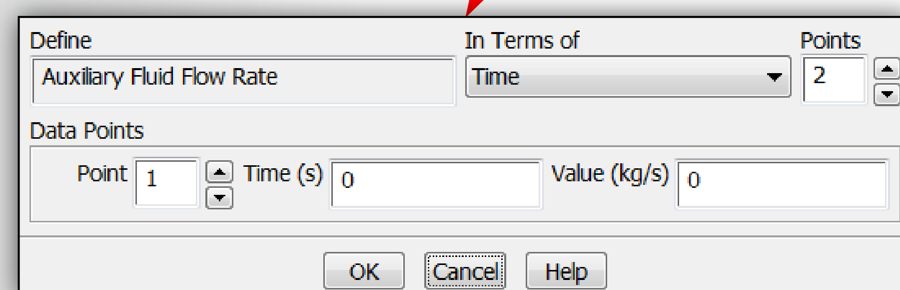
- **It is very important to use exact coordinates of the corner points x0, x1, and x2.**
  - Display / Mouse-Buttons
    - Select on for Probe
    - OK
  - Right-Click closest to the corner
    - Position gives the exact coordinates of the closest grid point
  - Copy/Paste the coordinates for x0, x1,and x2 into the Heat Exchanger panel

```
Probe position = (0.6020962 -0.001518894 0.49805915)
Surface group = hx_side1-2, id = 8, name = hx_side1-2

 Surface vertex 425
     Position = 0.6 0 0.5
  Current field value = 0

Thread 23, Cell 931
     Centroid = 0.59501 0.0250024 0.5025
       Volume = 2.4956e-006
    Partition = 0
```

Auxiliary Fluid

- **Boundary conditions for the auxiliary fluid**
  - Mass flow rate
  - Two thermal condition options
    - Heat Rejection (and Initial Temperature)
    - Inlet Temperature
  - Transient profile available
    - For mass flow rate as well as thermal conditions
    - Piecewise-linear or Polynomial

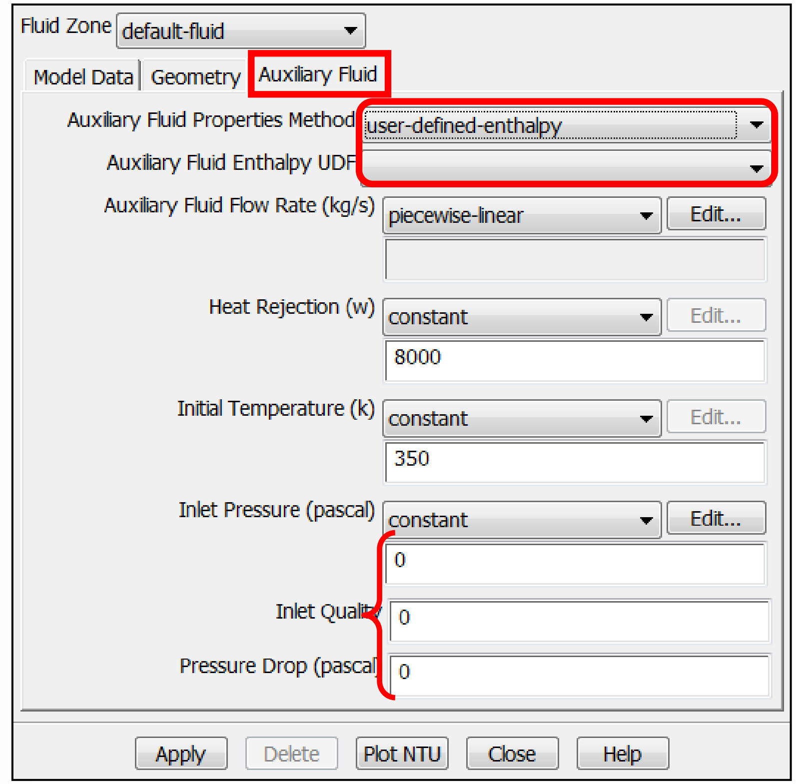


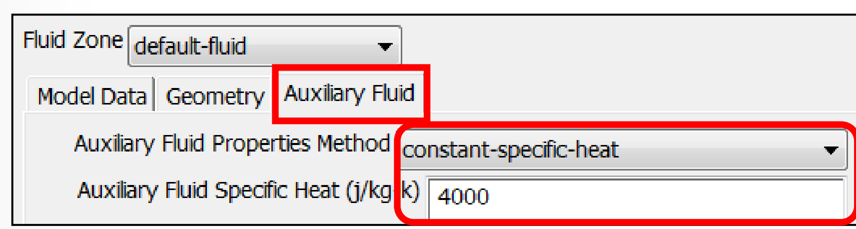


ANSYS

Auxiliary Fluid

- **Constant specific heat (enter the mean value)**
- **User-defined enthalpy**
  - UDF
  - H = f(p, t, x, ...)
  - Can be used when the auxiliary fluid specific heat is highly dependent on temperature.
    - Air/air heat exchanger



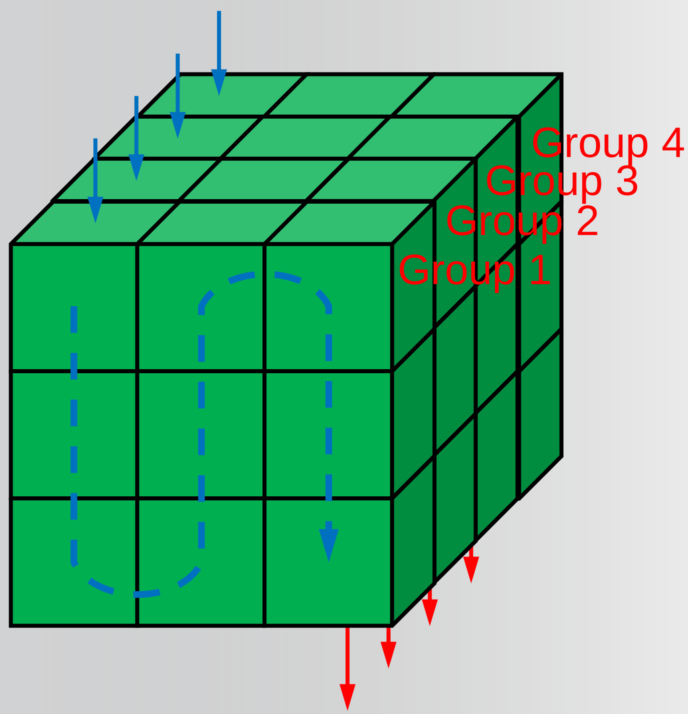


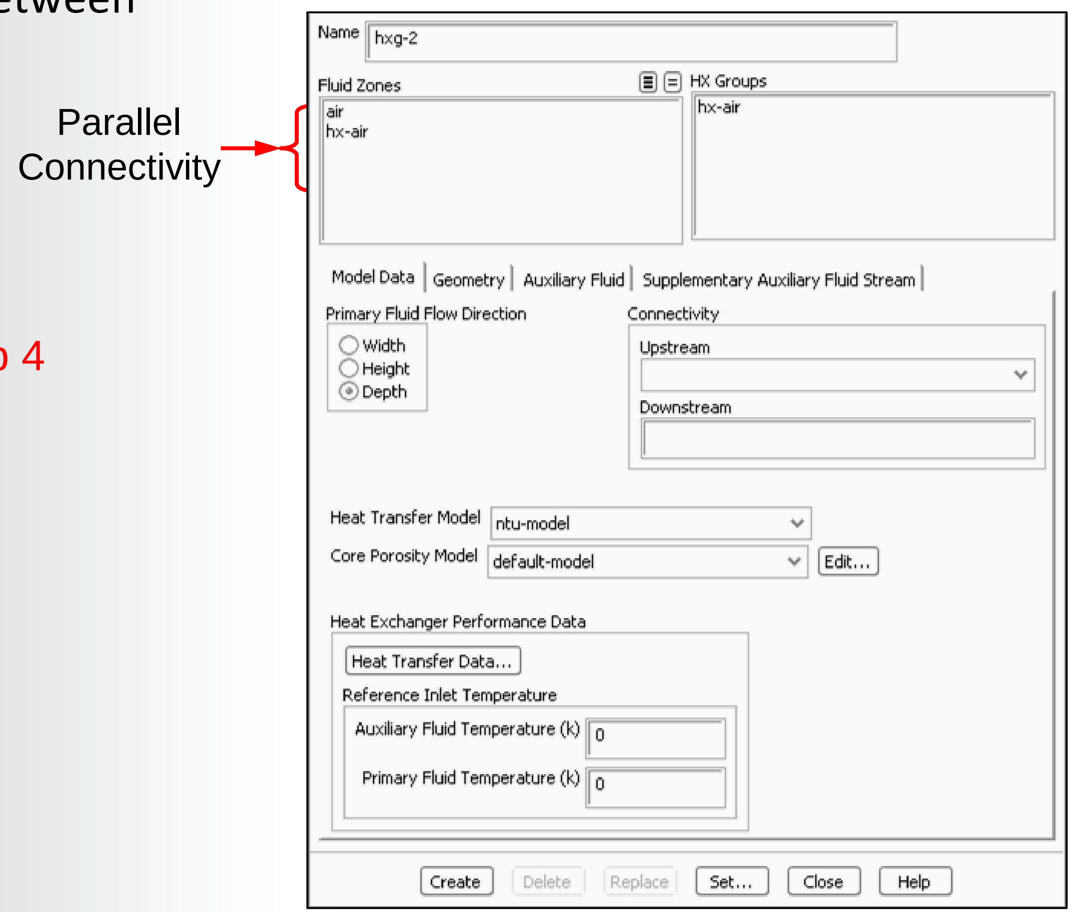
Heat Exchanger Group

- **Ability to group Heat Exchangers together**
  - In parallel
  - In series
- **Compatible with both simple-effectiveness and NTU-based models**
- **No need to define heat exchangers from fluid zones before defining heat exchanger groups**
  - The complete setup can be done in the same panel
- **When using groups, fixed inlet temperature is the only heat transfer option available.**

Heat Exchanger Group

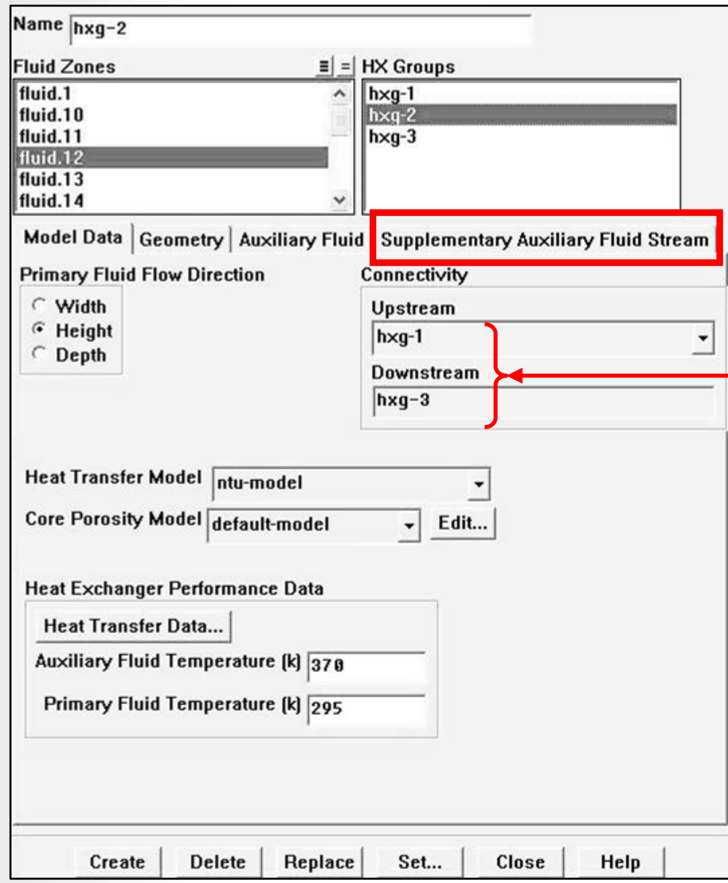
- **Parallel connectivity**
  - Auxiliary fluid flow split between fluid zones
  - Allow third direction discretization





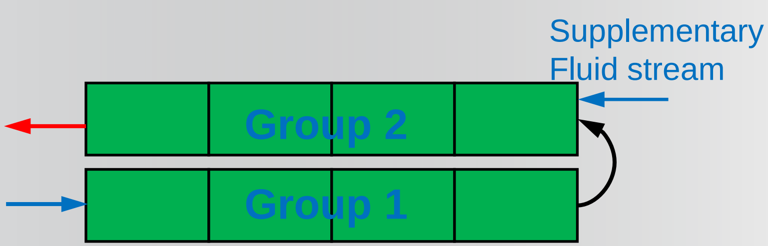
Heat Exchanger Group

- **Series connectivity**
  - The outlet of a particular group is the inlet of a downstream group
  - Each exchanger group can have a maximum of one upstream and one downstream group



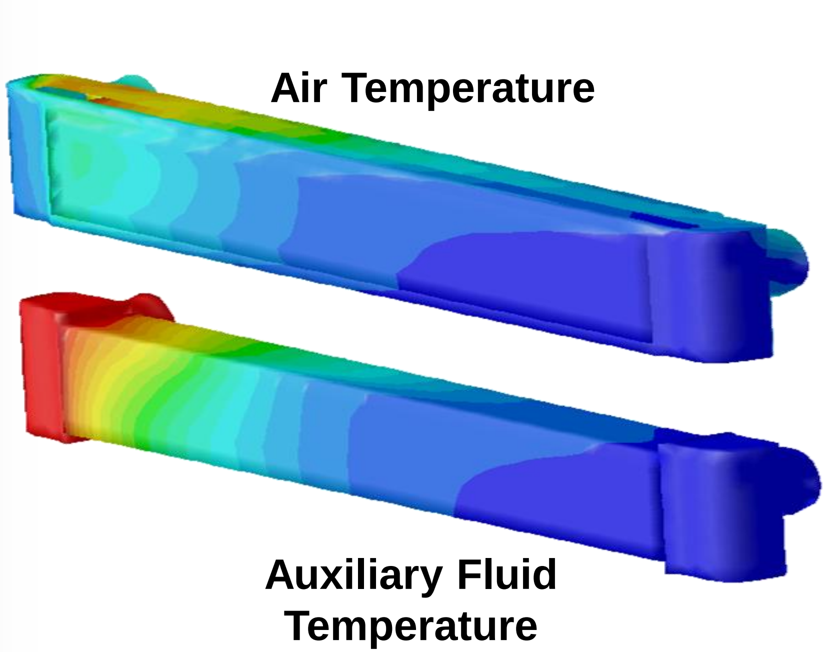
Supplementary auxiliary fluid stream can be added to a group

Series connectivity



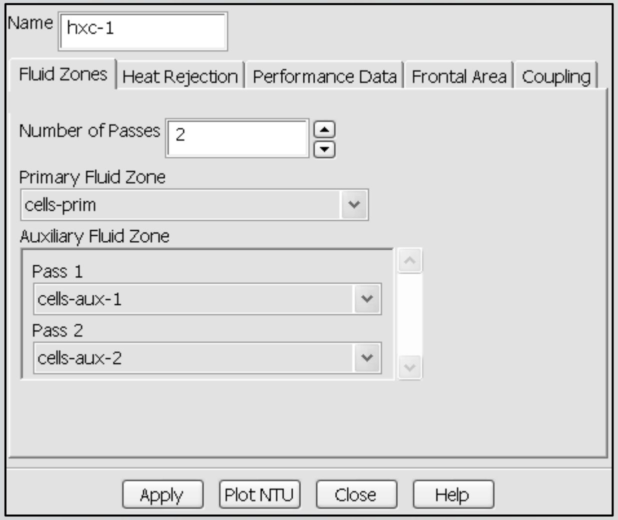
Dual Cell Heat Exchanger Model

- **Can predict the auxiliary flow field (inside of tank and pipes) as well as primary flow field (through the core)**
  - Core represented by two identical structured superimposed meshes that are coupled only through heat transfer
  - Should be used when auxiliary flow distribution inside the core is highly non-uniform
  - Coolant density can vary
  - Multiple pass heat exchangers cannot be modeled, this will require hooking a UDF

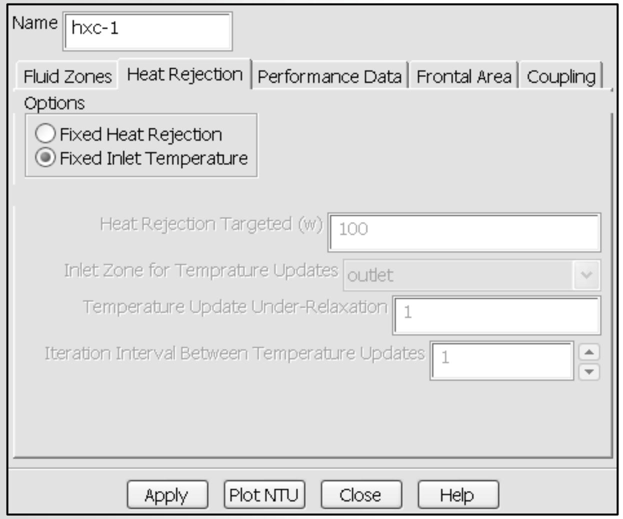


Dual Cell Heat Exchanger Option

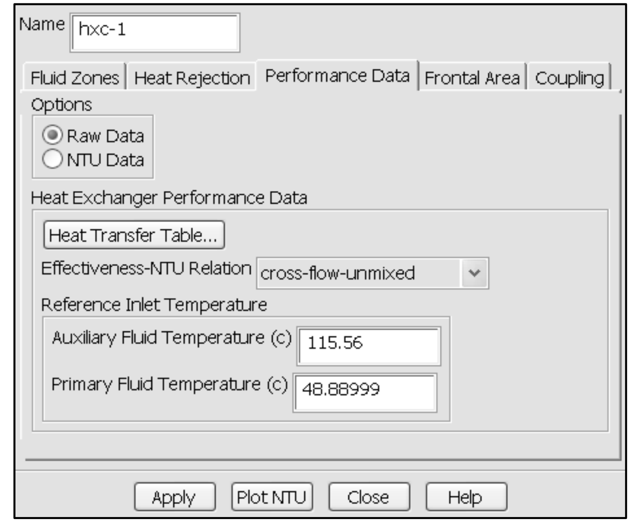
Fluid zones Input



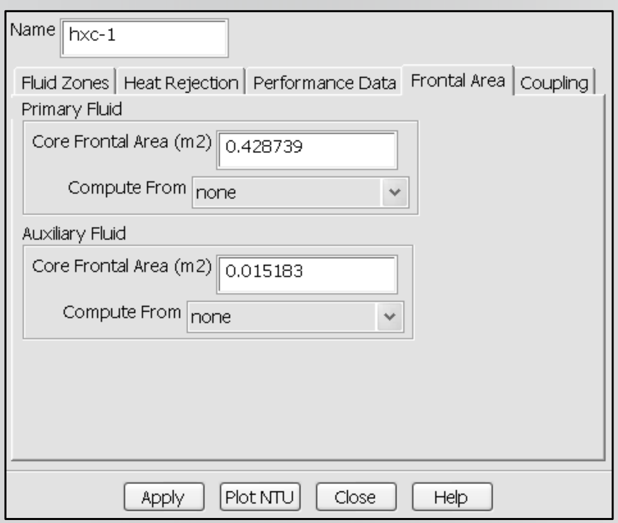
Heat Rejection Method



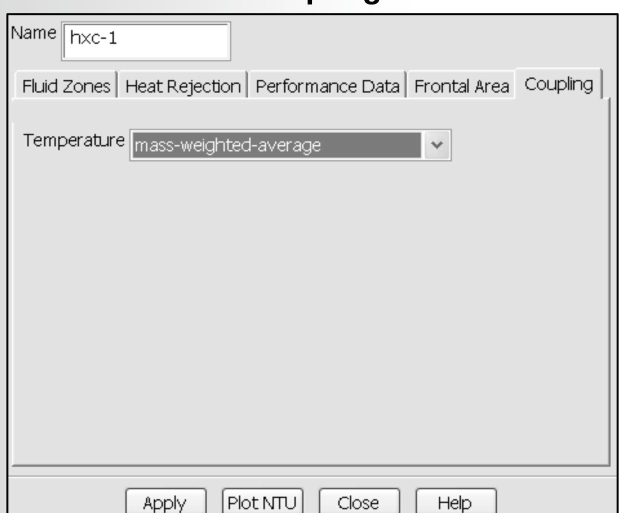
Setting a Performance Data



Frontal Area



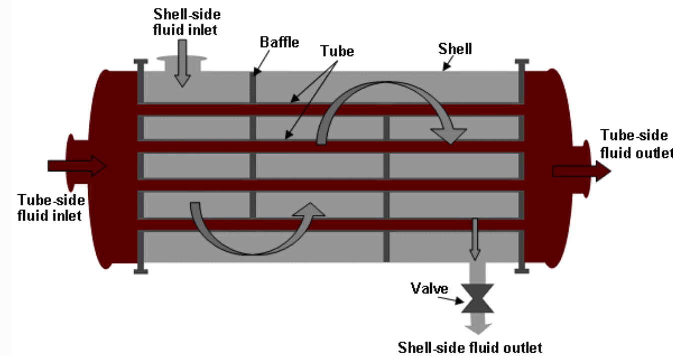
Coupling



Raw or NTU data can be entered manually or read through file

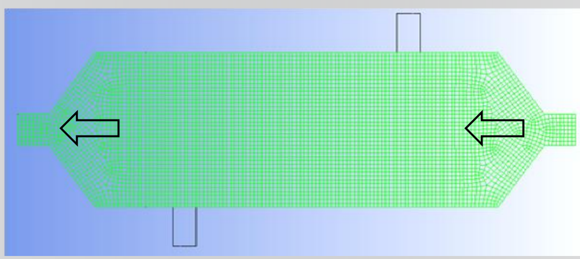
Dual Cell Heat Exchanger Model – 2D Example

Actual Heat Exchanger

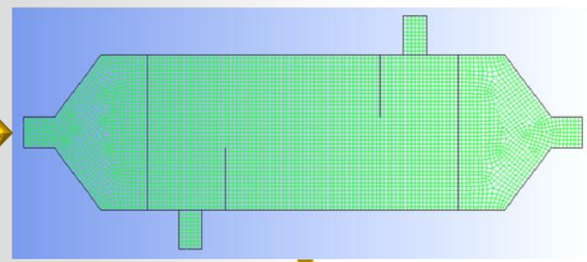


Schematic Representation of Heat Exchanger

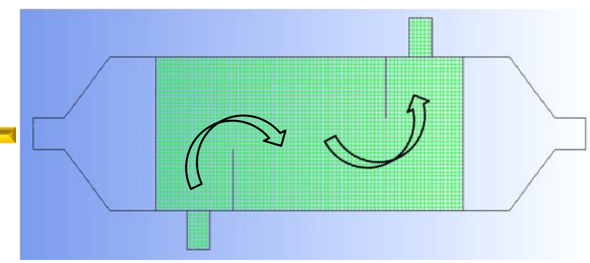
Tube Side Mesh



Overlapping Meshes

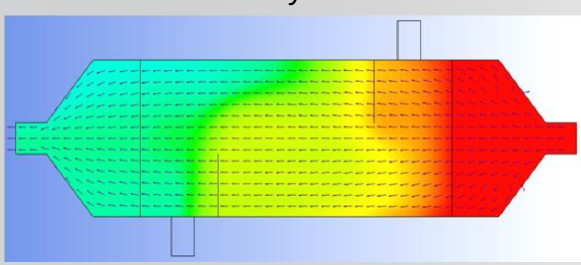


Shell Side Mesh

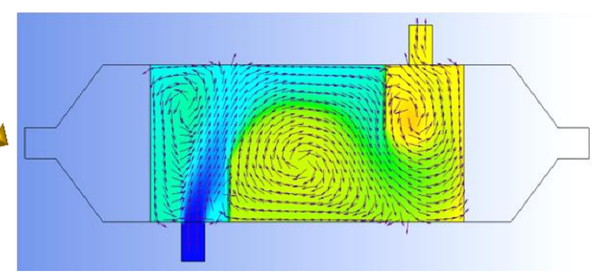


Dual Cell Heat Exchanger Model

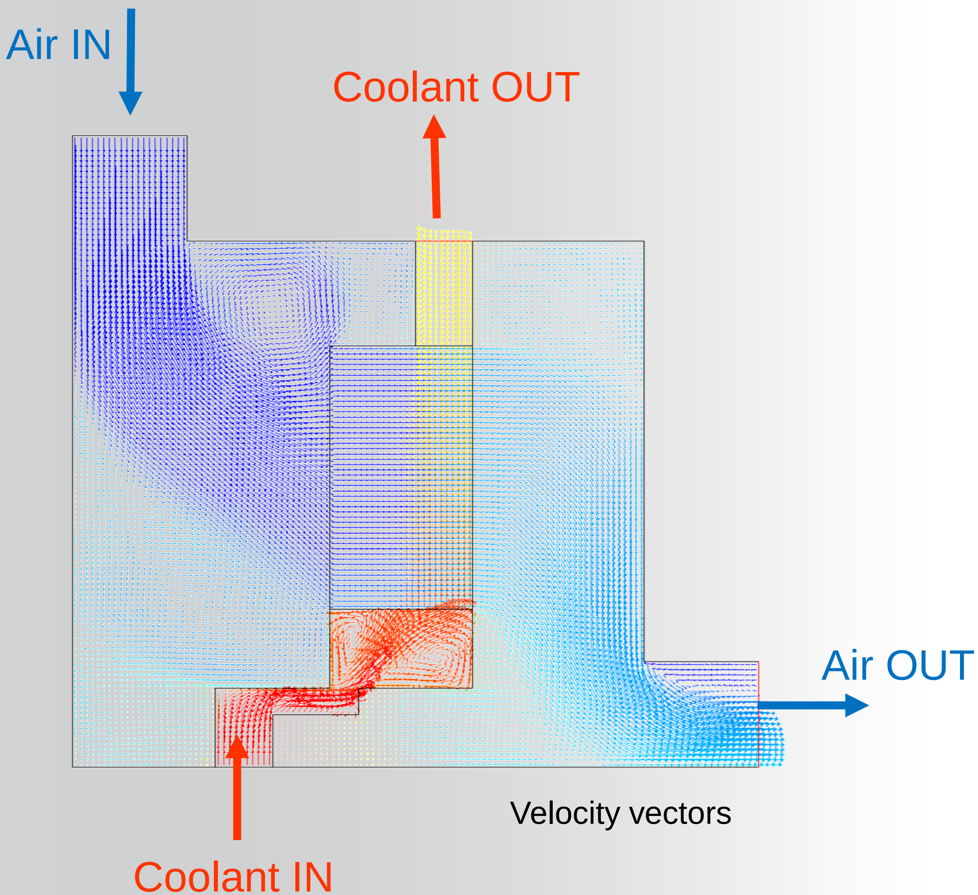
Contours of Temperature at Primary Side



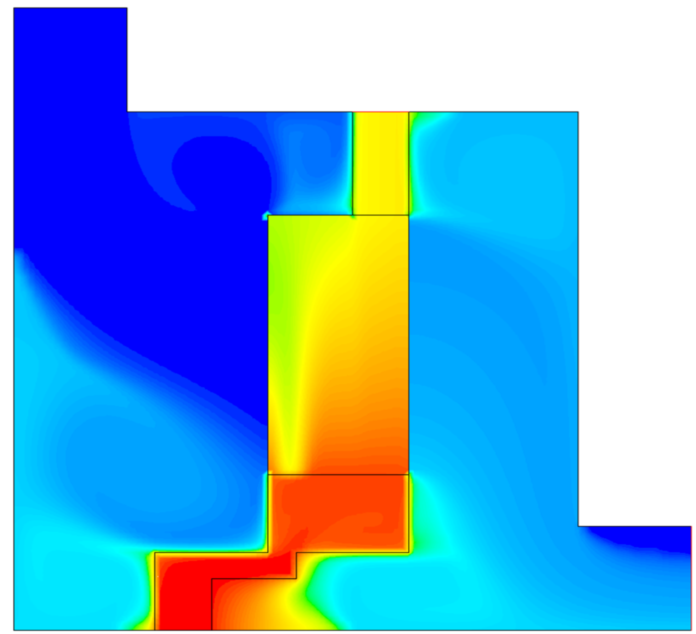
Contours of Temperature at Auxiliary Side



Dual Cell Heat Exchanger Model – 2D Example



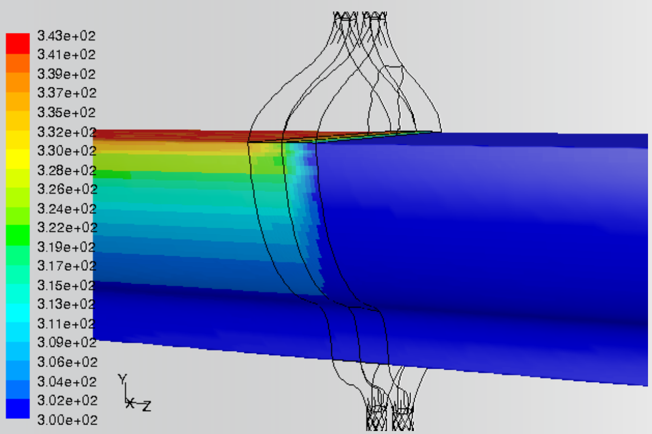
Velocity vectors



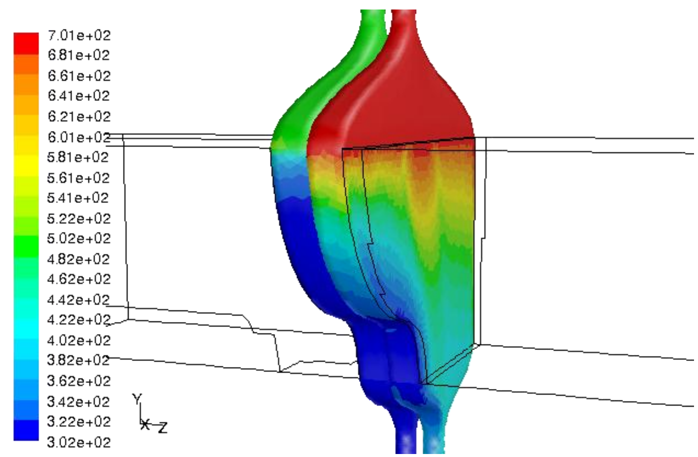
Contours of Temperature

Coincident Heat Exchanger

- **UDF and scheme interface panel done as consultancy**
  - Heat transfer mechanism provided by customer (proprietary)
  - Supports tetrahedral meshes, and meshes need not be identical.
  - Exchanger core does not have to be rectangular
  - Interior geometry of the core can be represented
  - Developed for parallel calculations



Temperature Contours In the Main Flow

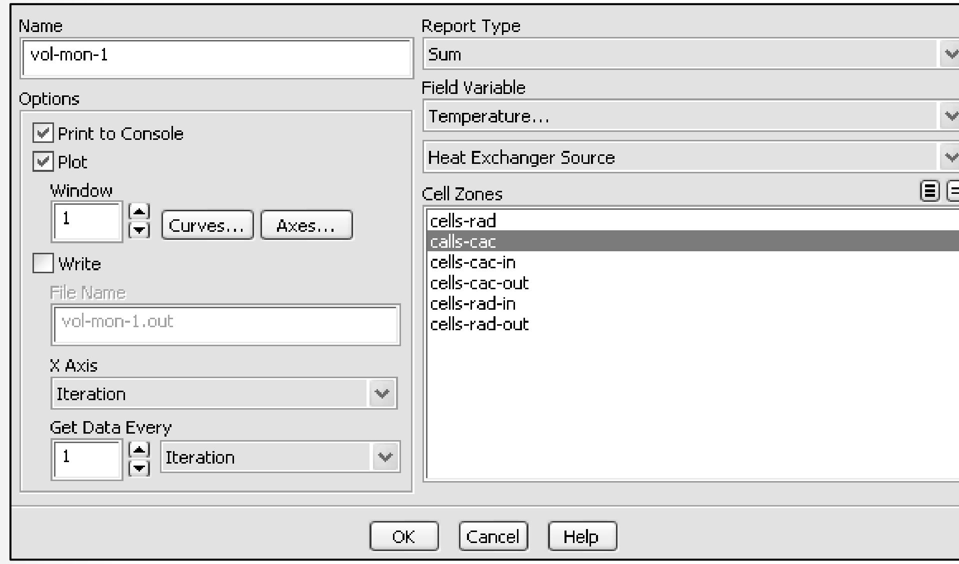


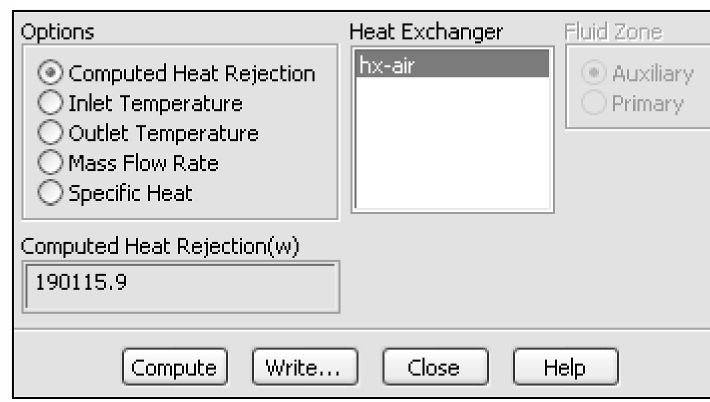
Temperature Contours in the Exchanger Core

Post Processing

- **Global heat transfer reports**
  - Can post process total heat rejection rate

  - Can report the following variables:
    - Computed heat rejection
    - Inlet Temperature
    - Outlet Temperature
    - Mass Flow Rate
    - Specific Heat





Outline

- Introduction
- Simulation of Heat Exchangers
- Heat Exchanger Models in FLUENT 14.5
- **Summary**

Summary

- **Start by closely reviewing your modelling goals. The modelling approach selected for the simulation will heavily depend on your objectives.**
- **The macro models, are modelling auxiliary flow passes as 1D flow.**
  - The macro model is quite suitable for a thin 3D heat exchanger core with a rectangular cross section, where the pass-to-pass plane is perpendicular to the primary flow direction, the auxiliary flow is uniform, and the mesh is uniform and structured.
- **The dual cell model allows the solution of the passes of the auxiliary flow on a separate mesh**
  - The dual cell model provides the greatest flexibility with regard to the shape of the heat exchanger core and the nature of the mesh, and allows the auxiliary fluid to be highly non-uniform as it enters the core.
- **ANSYS Fluent User's Guide – Chapter 14.1 provides further guidance on how to chose a heat exchanger model.**