CFD Analysis of Thermal Control System in NX Thermal & Flow

C. R. Fortier
M. F. Harris
(NASA KSC)

Presented By
Craig R. Fortier

Thermal & Fluids Analysis Workshop
TFAWS 2014
August 4 - 8, 2014
NASA Glenn Research Center
Cleveland, OH

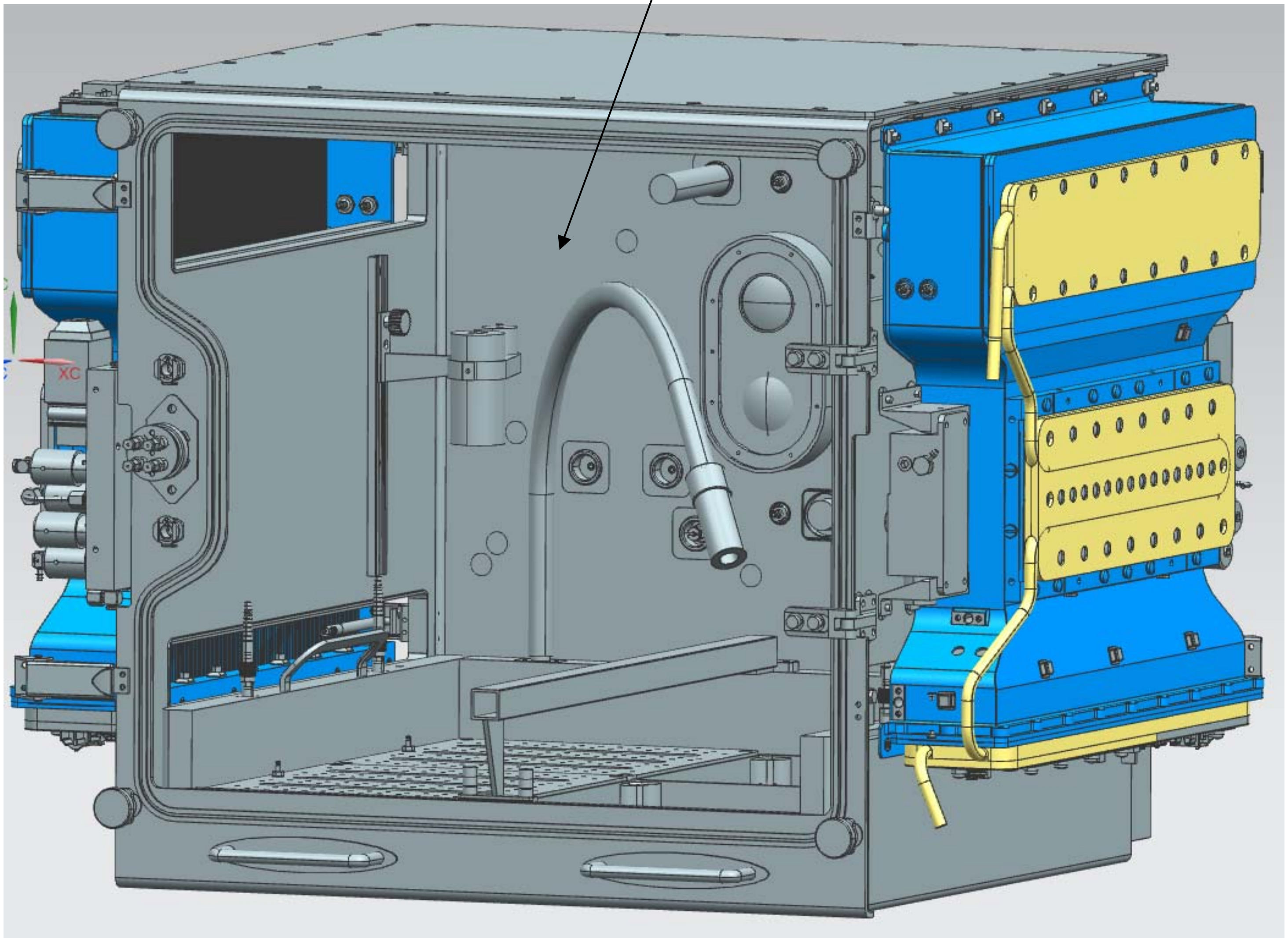
Background

- Thermal Control System (TCS) provides thermal control, mainly cooling, to several Advanced Plant Habitat (APH) subsystems. One of these systems is the Environmental Control System (ECS), which provides temperature and humidity control inside the growth chamber to optimize plant growth.

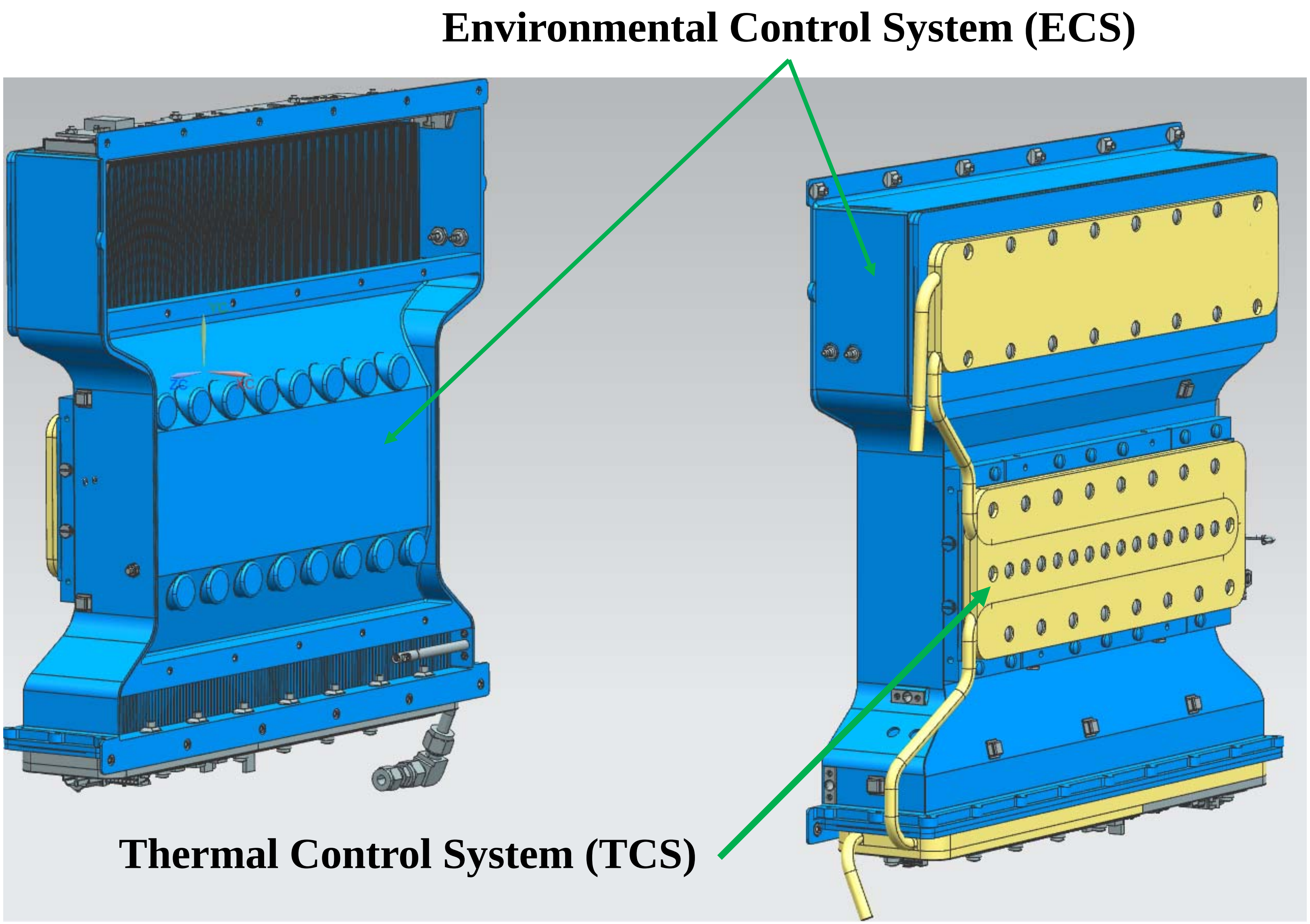
- TCS consists of three cold plates that use Thermoelectric Coolers (TECs) to heat or cool a water loop as needed to meet ECS temperature requirements.

Advanced Plant Habitat

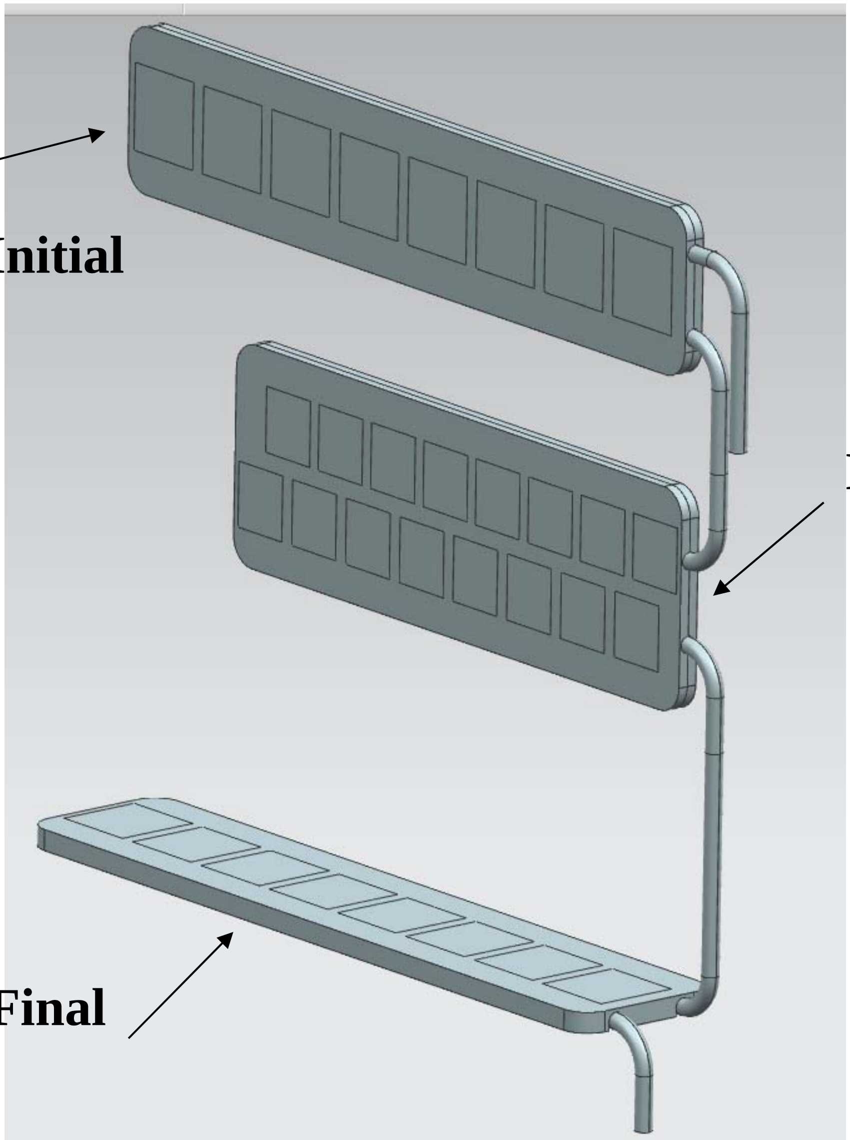
Growth Chamber



Environmental Control System



Thermal Control System (TCS)



Thermal Control Unit Initial (TCUI)

Humidity Control Unit (HCU)

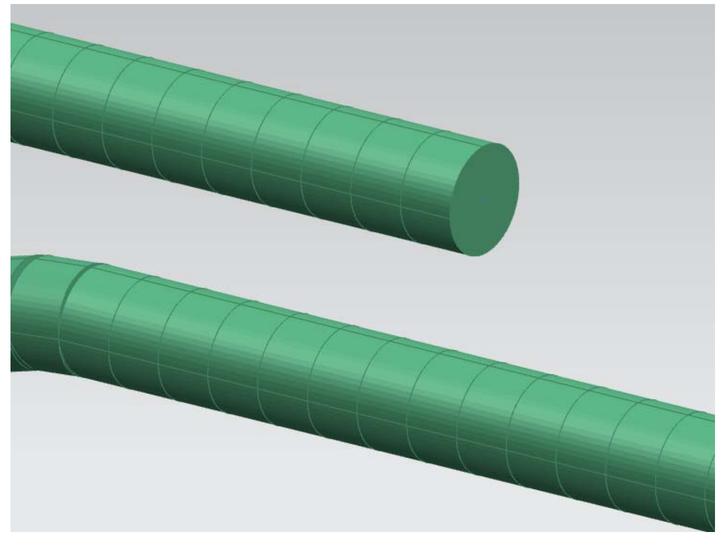
Thermal Control Unit Final (TCUF)

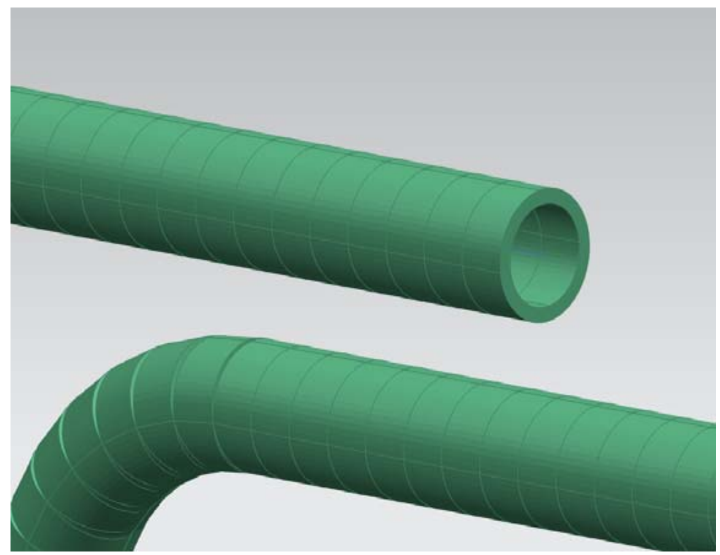
Problem Statement

- Pressure drop and heat transfer analysis was performed to optimize the TCS design.
- Computational analysis was performed in Siemens NX Thermal/Flow.
  - Method 1: Uses NX Thermal only. Includes 3D mesh thermal solver and 1D duct network solver for fluid flow in pipes or tubes. This may include wall thickness (if desired).
  - Method 2: Uses NX Thermal/Flow, which includes 3D mesh thermal solver and 3D computational fluid dynamics (CFD) flow solver.
  - Both methods were used and the results will be compared in this presentation.

Method 1: 1D duct flow network analysis

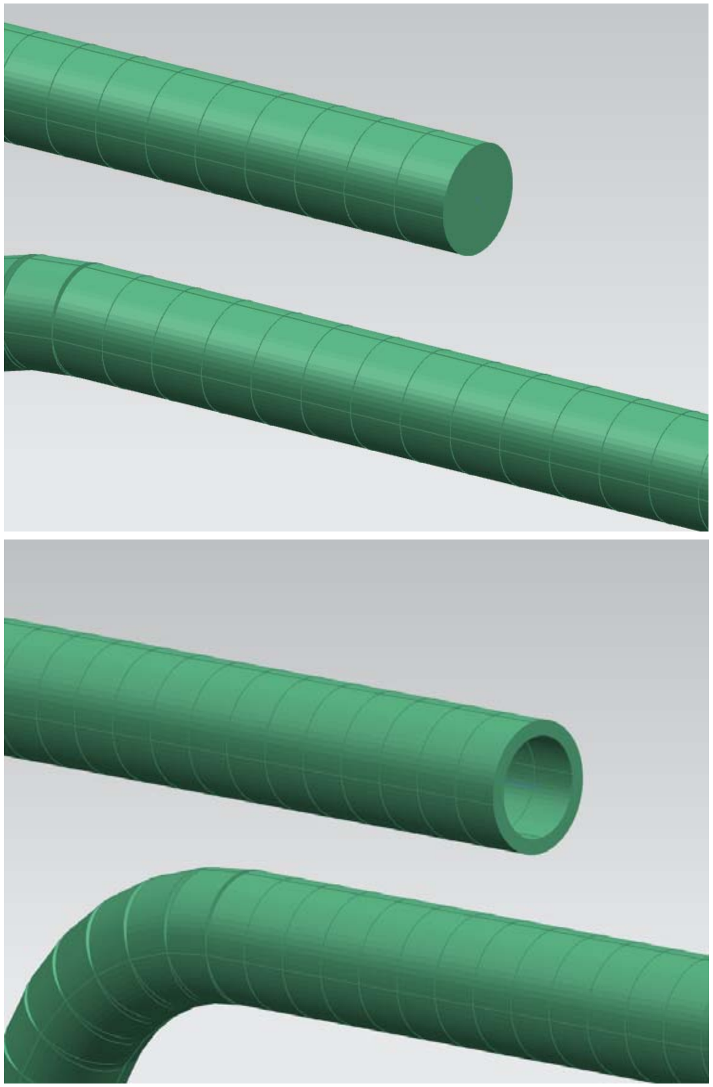
- Models fluid region as one-dimensional fluid duct.
  - Pressure
  - Velocity
  - Mass flow rate
  - Temperature
- 1D mesh
  - Fluid only
  - Fluid and tube wall
- Heat transfer (Thermal coupling)
  - Conduction, convection, radiation
  - Free, forced and user defined convection



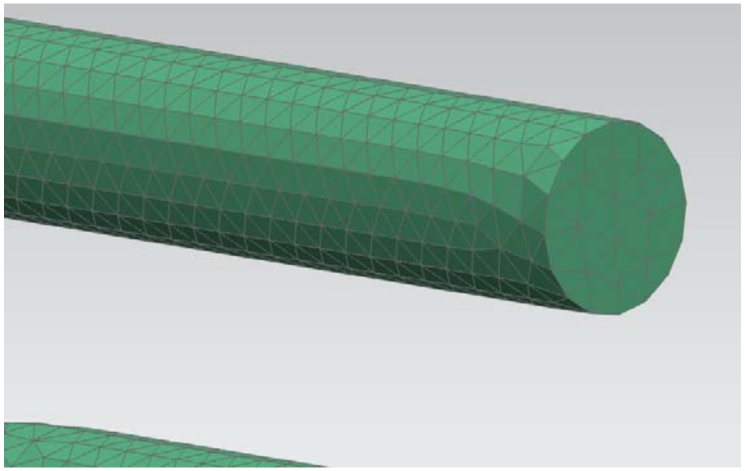


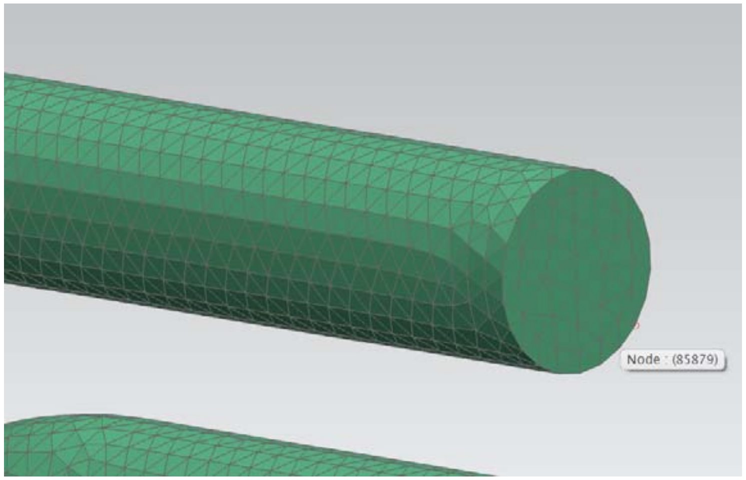
Method 1: 1D duct flow network analysis

- Does not use NX Flow
  - Only NX Thermal is used
- Duct boundary conditions
  - Fan/pump
  - Opening
  - Pressure
  - Flow properties
  - Duct to 3D flow connection
    - Connects 1D duct to 3D CFD model



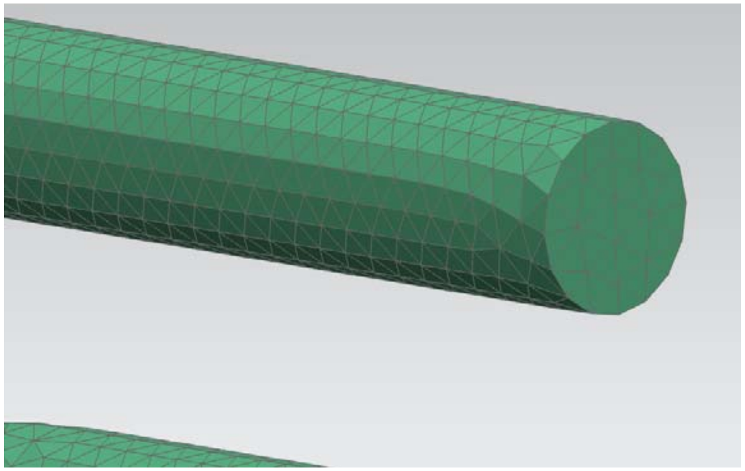
Method 2: 3D CFD Analysis



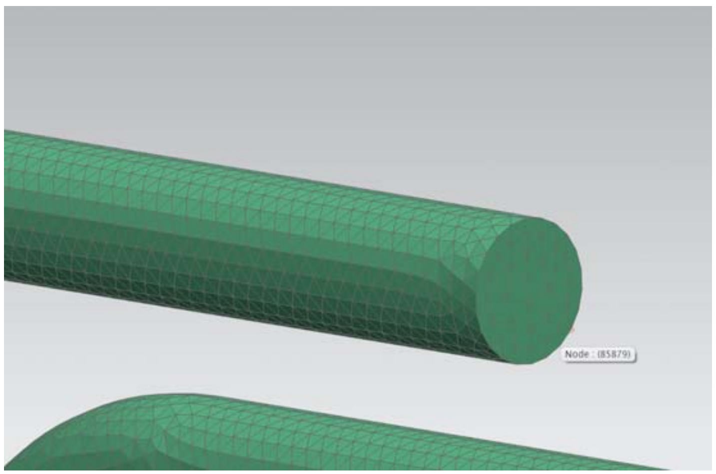


- 3D mesh
  - Structured hexahedral
    - Quad 2D face and sweep (manual)
  - Unstructured tetrahedral
    - Either automatic or manual meshing
- Turbulent model
  - *K-Omega*
  - K-Epsilon
  - *Mixing Length*
  - LES
- Both NX Flow and Thermal
- Serial & Parallel processing

Method 2: 3D CFD Analysis



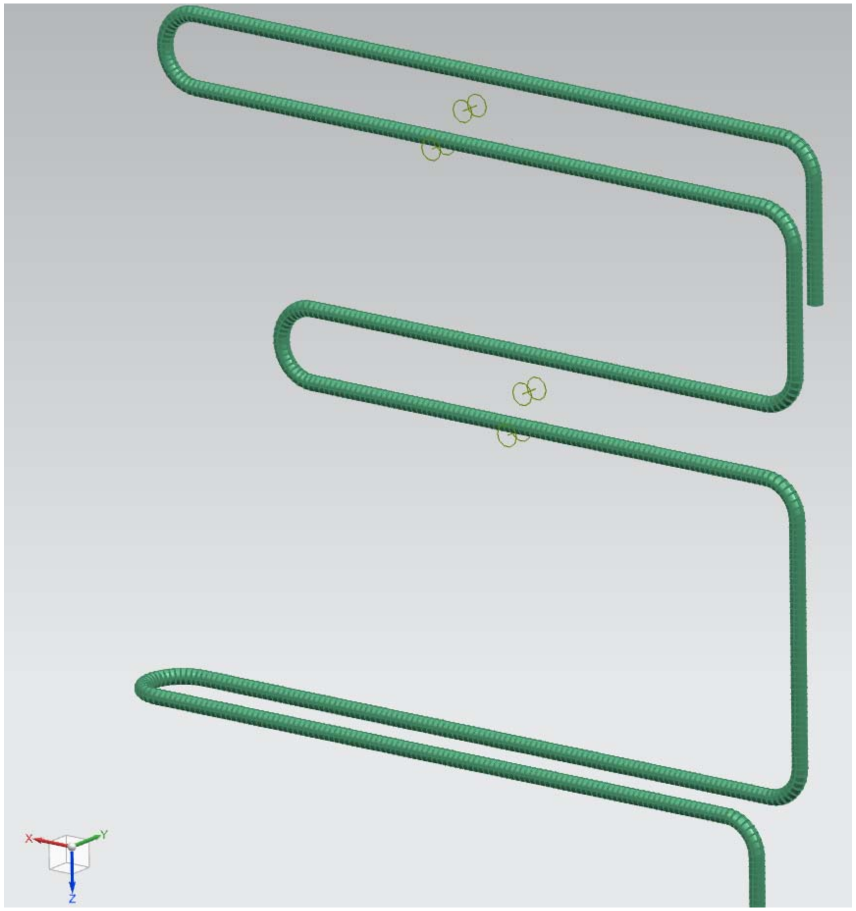
- Fully coupled pressure velocity scheme (parallel solver)
- All results converged for a steady state solution

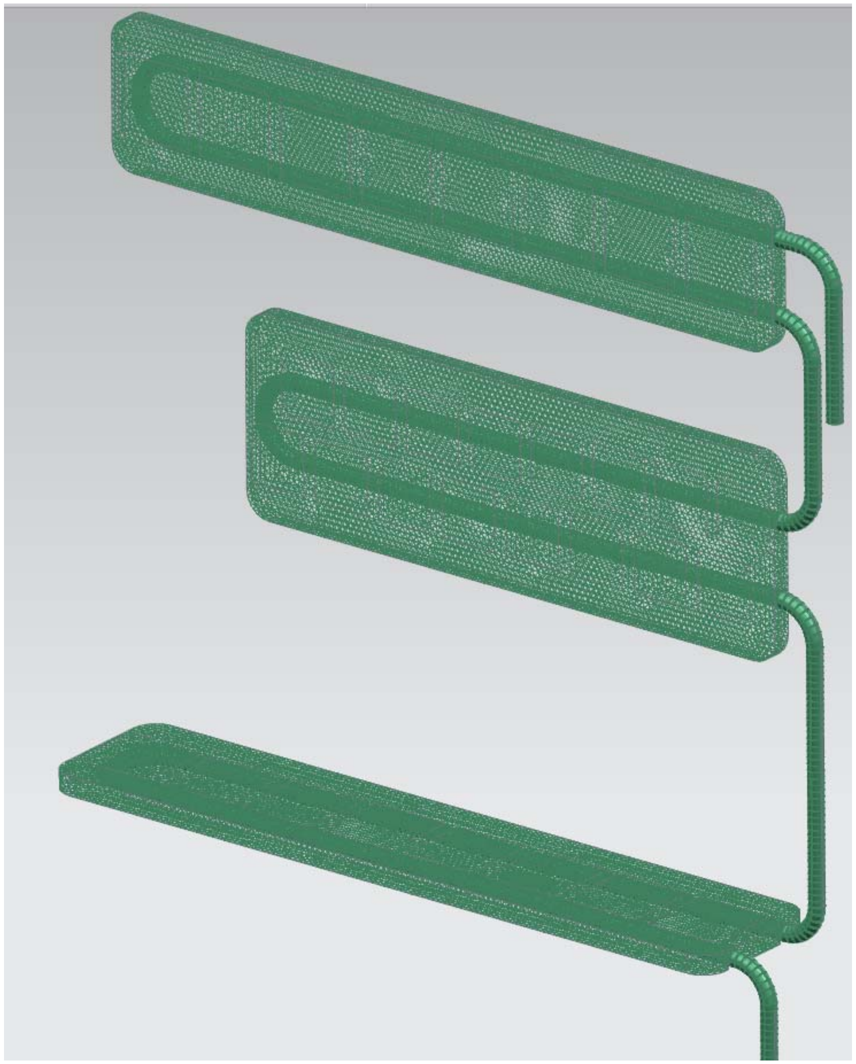


Method 1: Duct Mesh Sizing

- 1D duct network included both tube wall and fluid
  - Fluid and 3D thermal model for wall were tested
    - Tube used 0.075 inches element size
- 1D duct mesh (fluid)
  - 0.125 inches element size
- 1D duct mesh (tube wall)
  - 0.125 inches element size
- 3D tetrahedral cold plate mesh
  - 0.125 inches element size (0.395 inch recommended)
- Mesh sensitivity was tested at half and double of the element size and found that changing mesh size had no effect on the solution

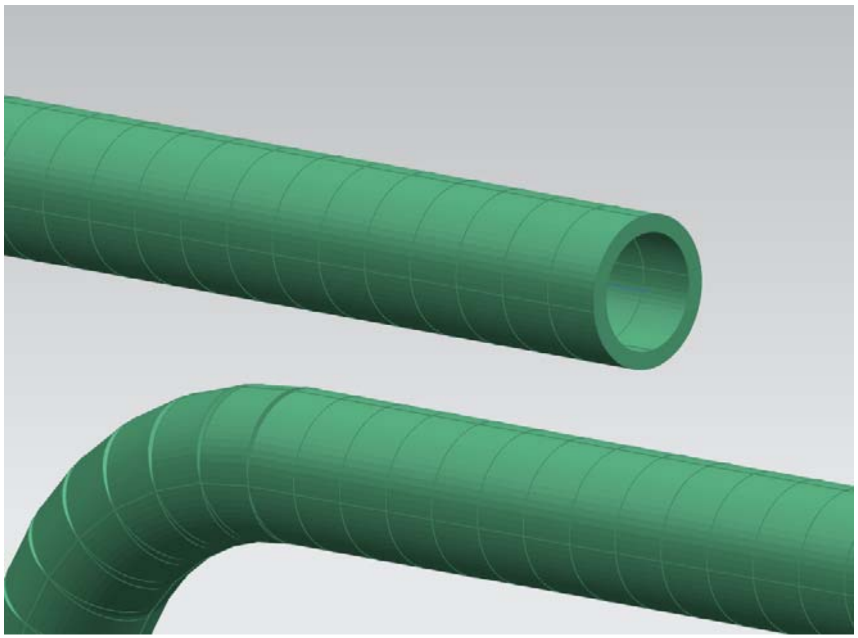
Method 1: Duct Mesh



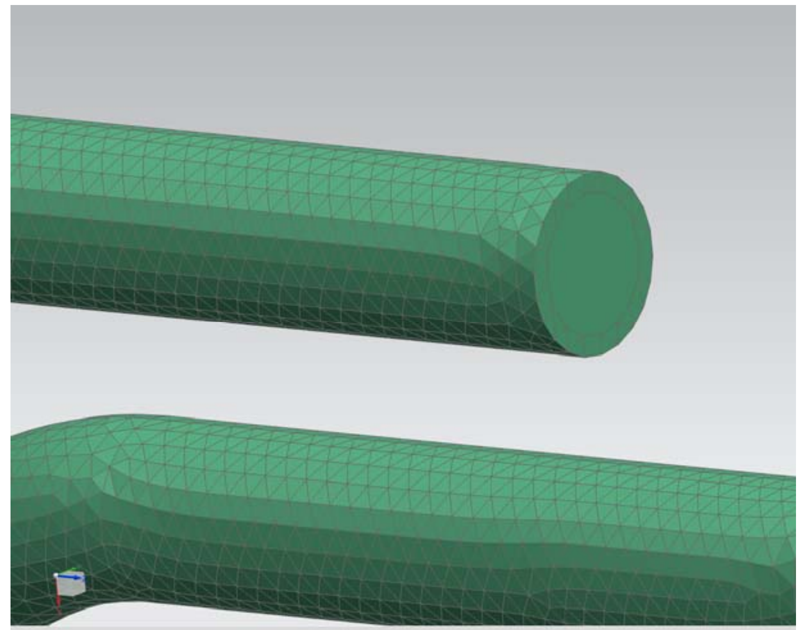


Method 1: Duct Mesh

1D duct with wall



1D duct with 3D wall



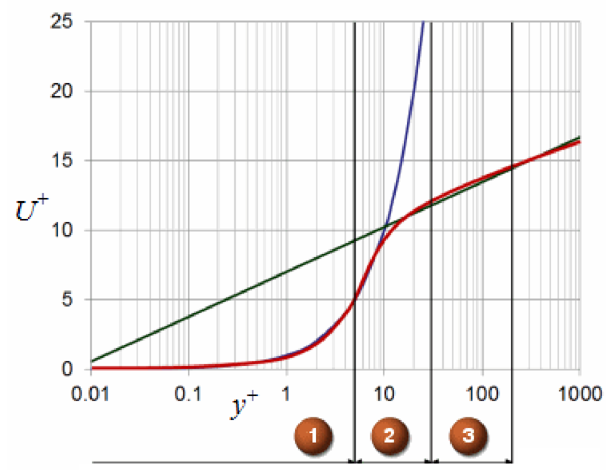
Method 2: CFD Mesh Sizing

- Fluid mesh (3D tetrahedral)
  – 0.33 inch element size
  – wall functions were used so viscous sublayer did not need to be resolved
- Tube wall mesh (3D tetrahedral)
  – 0.075 inch element size (0.164 inch recommended)
- Cold plate mesh (3D tetrahedral)
  – 0.125 inches element size (0.395 inch recommended)

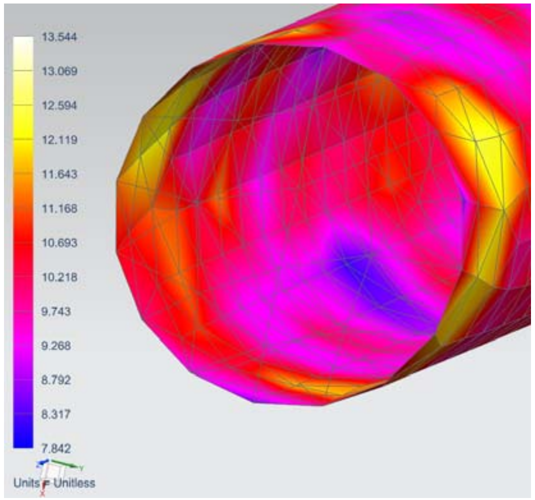
- Mesh sensitivity was tested by varying the mesh size and no effects to the solution was noticed

Method 2: CFD Mesh

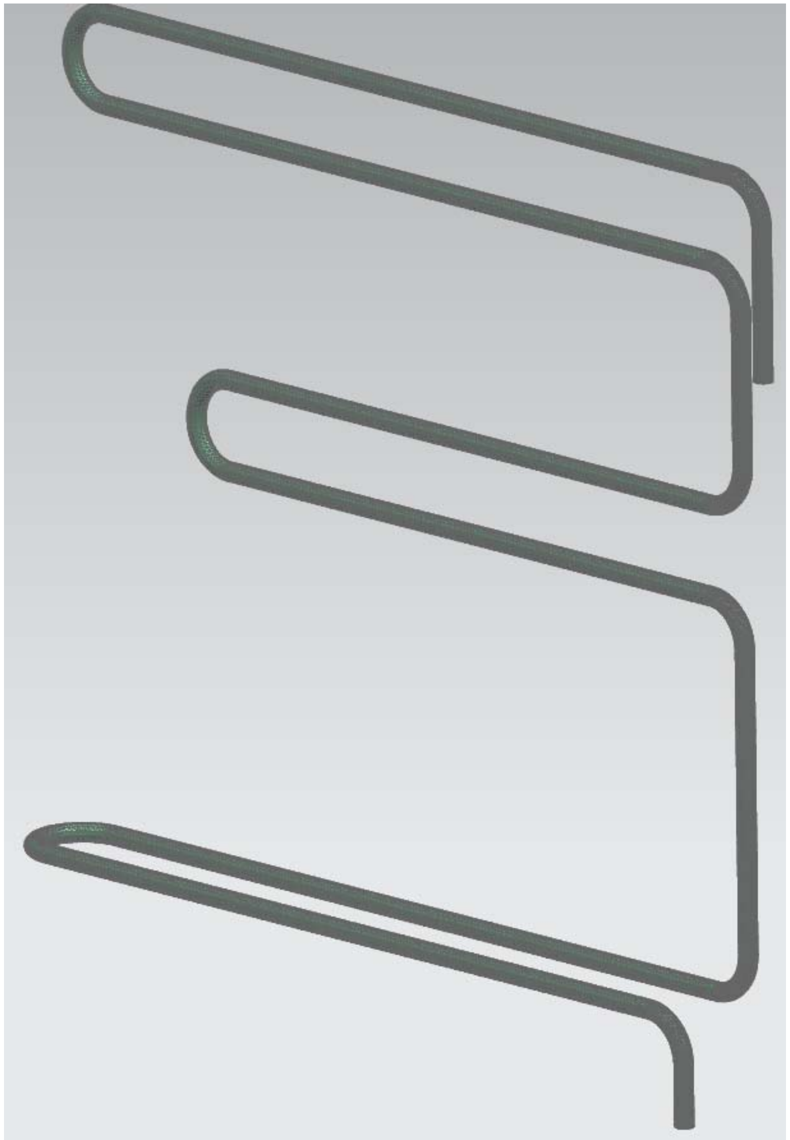
- Inner layer
  1. Viscous sublayer
  2. Buffer layer
  3. Log-law region

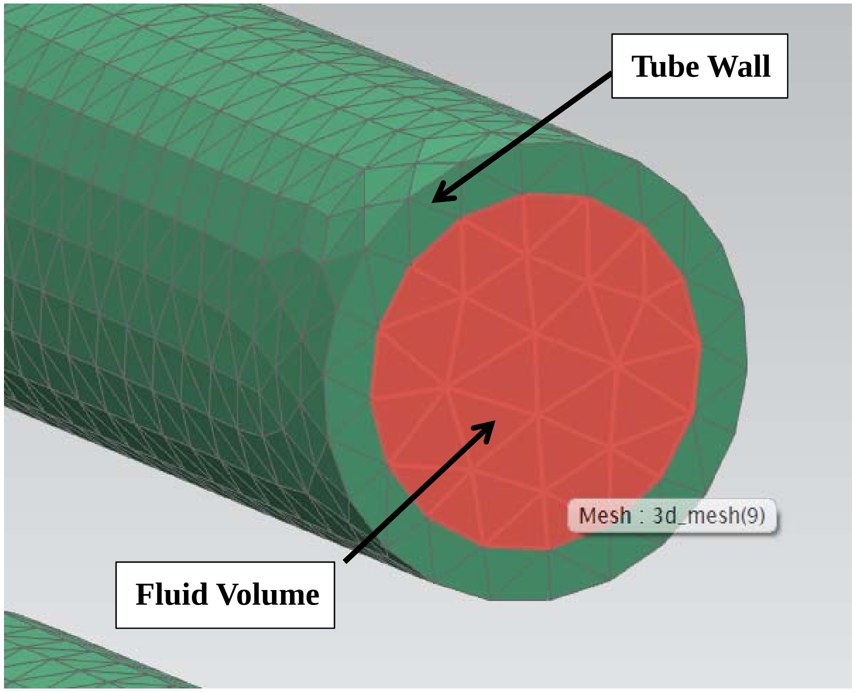


- Goal was to keep mesh large enough so y+>30
  – Actual ranged 7.8 to 13.5



Method 2: CFD Mesh



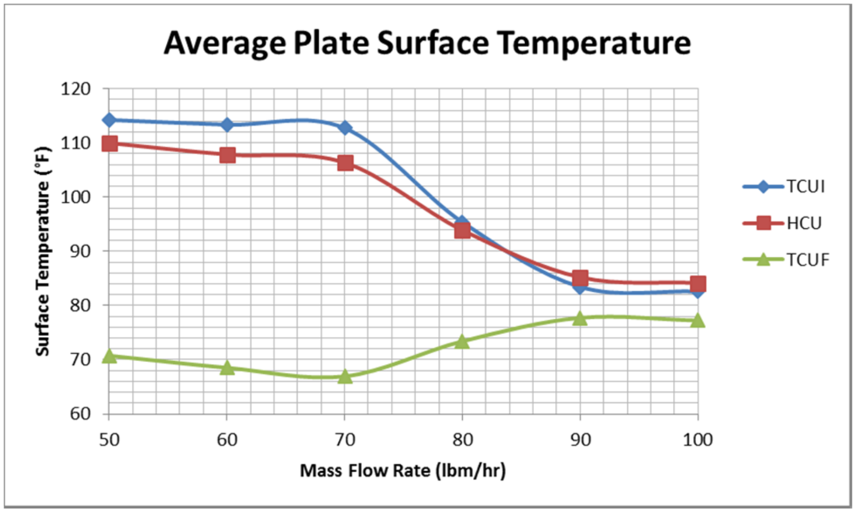


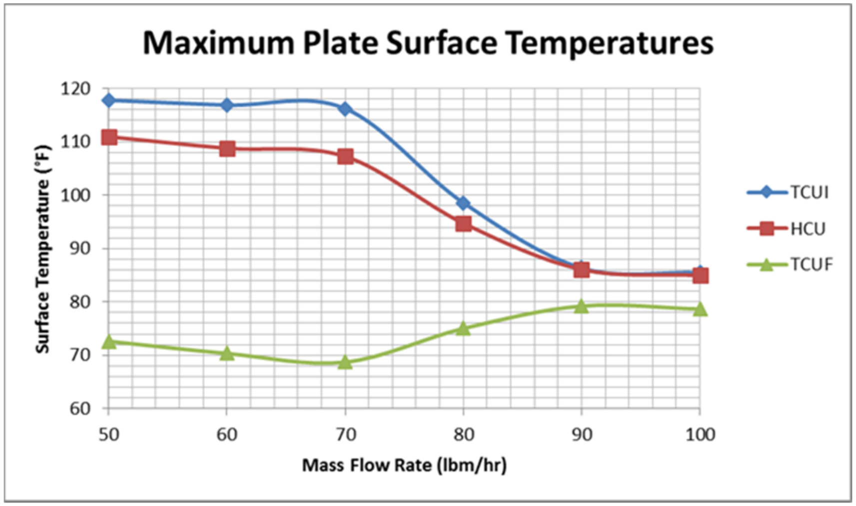
Case Summary

- Used constant mass flow rate at
  - 50, 60, 70, 80, 90 and 100 lbm/hr
- IATCS fluid from ISS
  - Water was used in model
  - Initial temperature of water was 73.4°F
- TCUI TECs
  - 18 Watts of heating per TEC
- HCU TECs
  - 4.9 Watts of heating per TEC
- TCUF TECs
  - -6.1 Watts of cooling per TEC

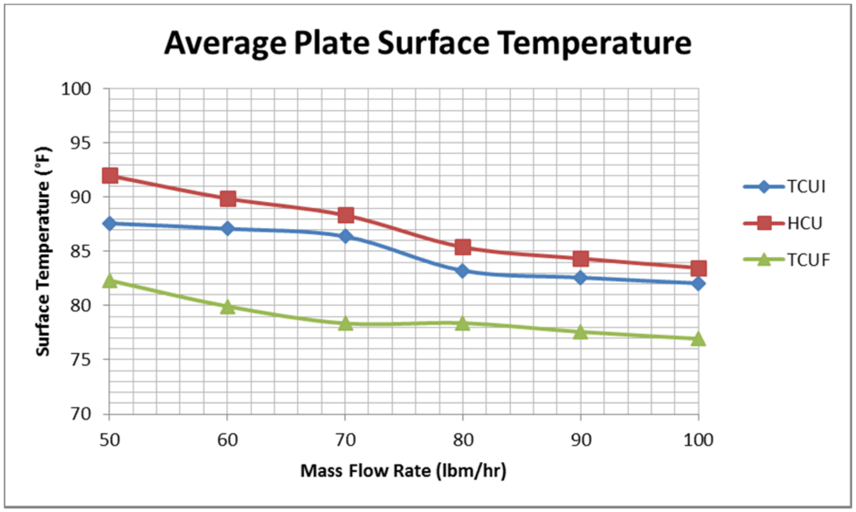
Cold Plate Surface Temperature Results

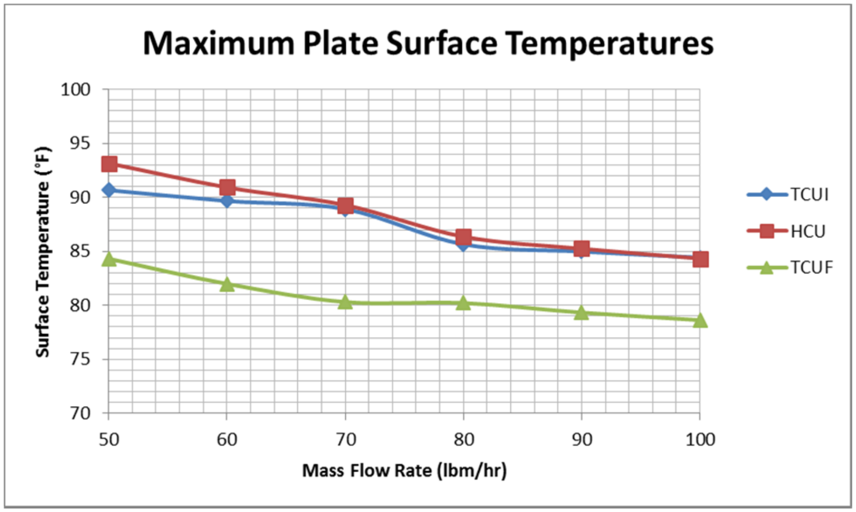
Method 1





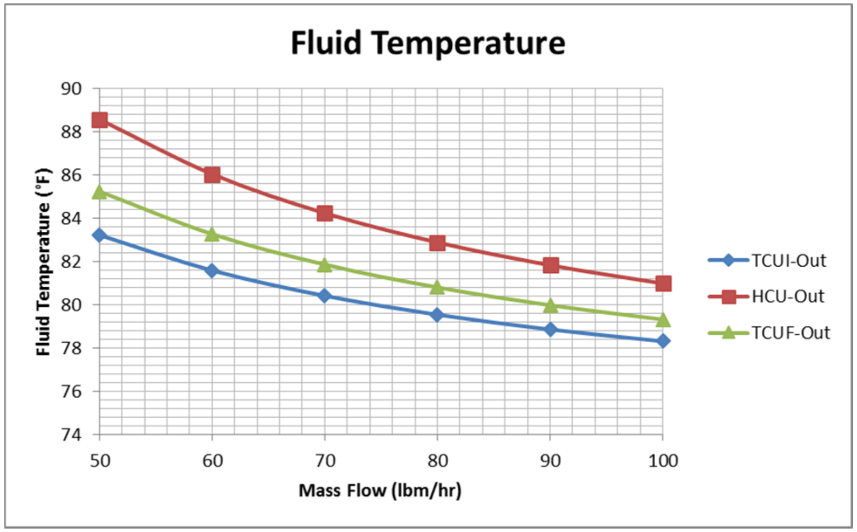
Method 2



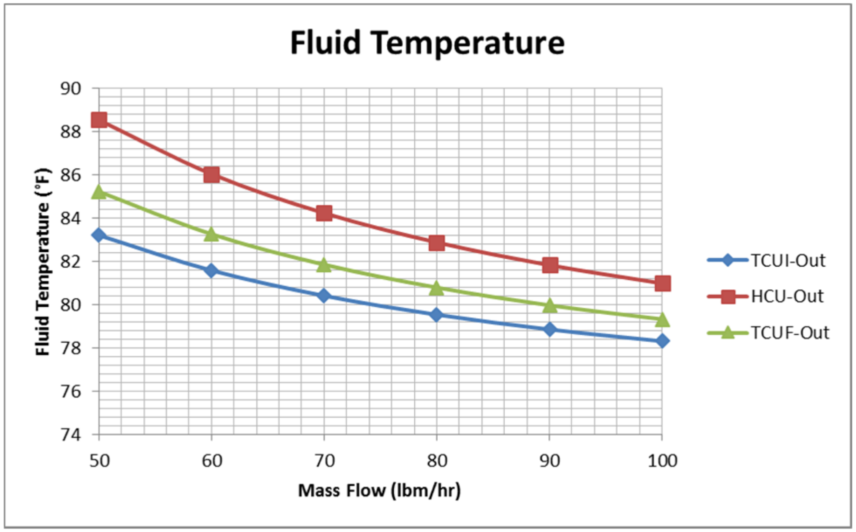


Fluid Temperature Results

Method 1

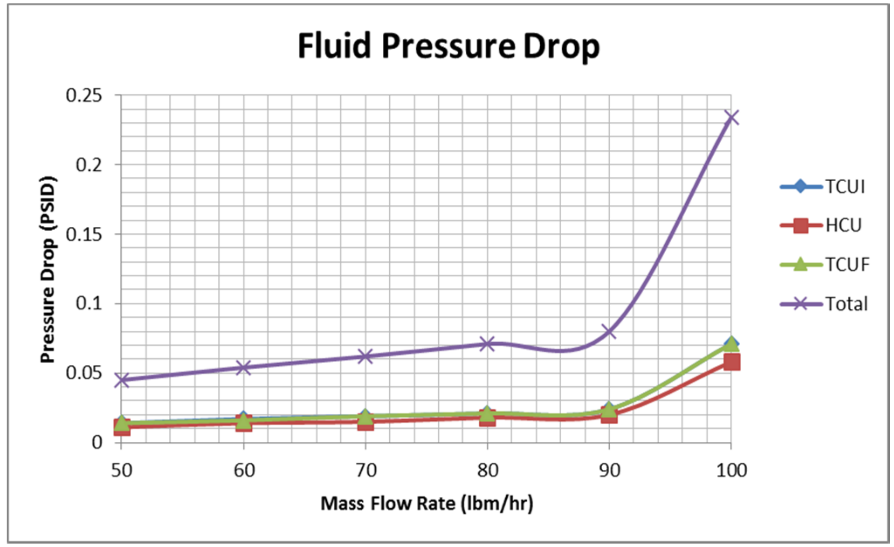


Method 2

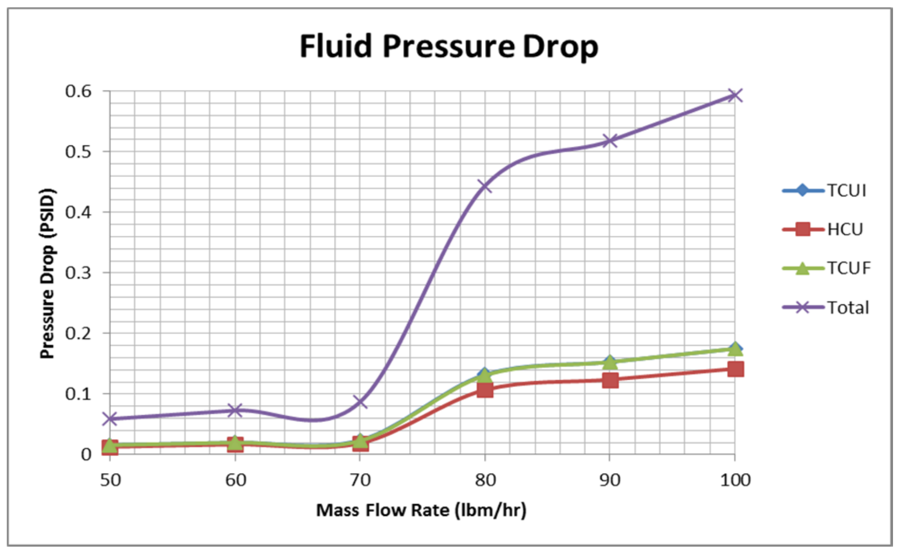


Fluid Pressure Drop Results

Method 1



Method 2



Conclusions

- Models produced similar fluid temperatures, but differed in pressure drop and surface temperature results.
- Testing sensitivity in Method 1 to convection coupling or Method 2 to turbulence model did not produce drastic changes
  - Grid size sensitivity also did not affect the solution
- Preliminary data from the integrated testing at Orbitec showed the CFD method was closer in actual pressure drop to prototype.
  - While CFD pressure drop analysis was closer to system pressure drop seen during testing, it was still 50% lower.
    - Analysis will need to be updated based on system changes during testing.
  - Currently no thermal data is available to correlate model to test data

Questions?