CFD ANALYSIS OF THERMAL CONTROL SYSTEM USING NX THERMAL & FLOW

C. R. Fortier[1], M. F. Harris[1], S. McConnell[2]
[1]Engineering Design Analysis Branch
[2]Environmental Control & Life Support Systems Branch
NASA Kennedy Space Center, FL

ABSTRACT

The Thermal Control Subsystem (TCS) is a key part of the Advanced Plant Habitat (APH) for the International Space Station (ISS). The purpose of this subsystem is to provide thermal control, mainly cooling, to the other APH subsystems. One of these subsystems, the Environmental Control Subsystem (ECS), controls the temperature and humidity of the growth chamber (GC) air to optimize the growth of plants in the habitat. The TCS provides thermal control to the ECS with three cold plates, which use Thermoelectric Coolers (TECs) to heat or cool water as needed to control the air temperature in the ECS system. In order to optimize the TCS design, pressure drop and heat transfer analyses were needed.

The analysis for this system was performed in Siemens NX Thermal/Flow software (Version 8.5). NX Thermal/Flow has the ability to perform 1D or 3D flow solutions. The 1D flow solver can be used to represent simple geometries, such as pipes and tubes. The 1D flow method also has the ability to simulate either fluid only or fluid and wall regions. The 3D flow solver is similar to other Computational Fluid Dynamic (CFD) software.

TCS performance was analyzed using both the 1D and 3D solvers. Each method produced different results, which will be evaluated and discussed.

INTRODUCTION

Advanced Plant Habitat is a large, closed loop plant growth chamber that is designed to be used in an EXPRESS rack for the International Space Station (ISS). Environment within the growth chamber (GC) will be controlled, such as temperature and humidity, using the Environmental Control System (ECS). Finally thermal control is provided to the ECS using the Thermal Control Subsystem (TCS), which is a series of cold plates and Thermoelectric Coolers (TECs) working in unison with a pressurized water loop.

The TCS is broken into three different components. The first is the thermal control unit initial (TCUI). This is made from 6061-T6 aluminum and has eight thermoelectrics attached to one side of the device. Each TECs is set to generate 18 watts of heat, used to cool the air and heat the fluid.

The second component is the humidity control unit (HCU). This is made from 6061-T6 aluminum and has 16 TECs. The TECs are attached to one side of the device, each generating 4.9 watts of heat, used to cool the air and heat the fluid. The final component of the TCS is the thermal control unit final (TCUF). This is also made from 6061-T6 aluminum and has eight TECs attached to the top of the unit. Each TEC on this unit is set to generate 6.1 watts of heat, used to heat the air and cool the fluid (opposite of the TCUI and HCU). A combined image of the TCS system, which includes all three components is displayed in Figure 1.

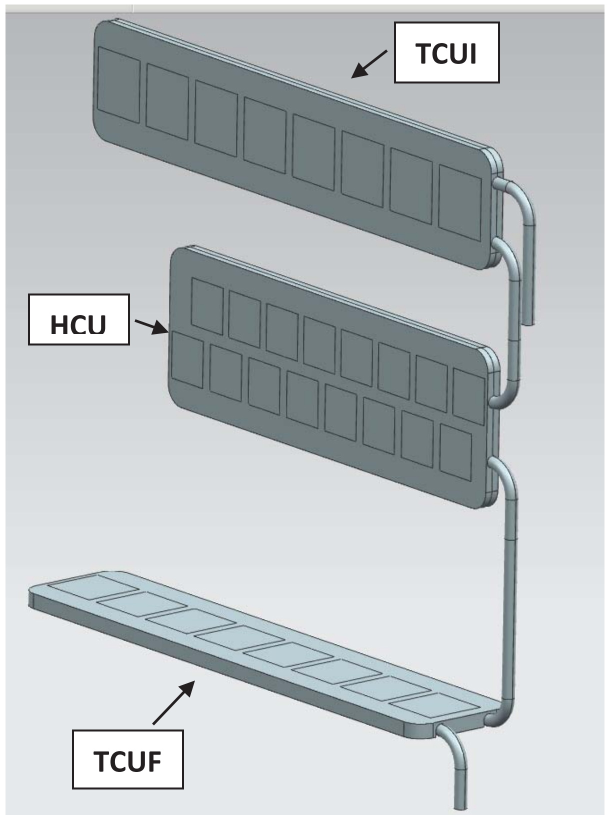


Figure 1. Image of Thermal Control Subsystem

DETAILED ANALYSIS

The analysis completed on the TCS was done using two different methods in Siemens NX Advanced Thermal/Flow (Version 8.5). The first method used only NX Advanced Thermal. This modeled the cold plates as 3D tetrahedral elements, while simplifying the fluid region and tube wall to a 1D mesh (1D flow solver). The second methods used NX Advanced Thermal & Flow and modeled the cold plates, tube wall, and fluid region using 3D tetrahedral elements (3D flow solver).

Method 1: 1D flow solver

Using this method the cold plates were meshed in 3D tetrahedral elements, while the fluid region and tube wall were meshed using a 1D mesh. The thermal and fluid regions were connected using convection coupling methods. This couple's convection between the polygon surfaces of the cold-plates, to the curves of the 1D fluid region, also called the duct flow model.

Additionally the boundary conditions and solver inputs to this method included the following:

- Constant mass flow rate for each flow case
- Serial processor solving
- Constant heat load for each TEC
- Rough walled tubing
- Turbulence begins at Reynolds > 2400 (NX default)
- Forced convection coupling
- Steady state solutions

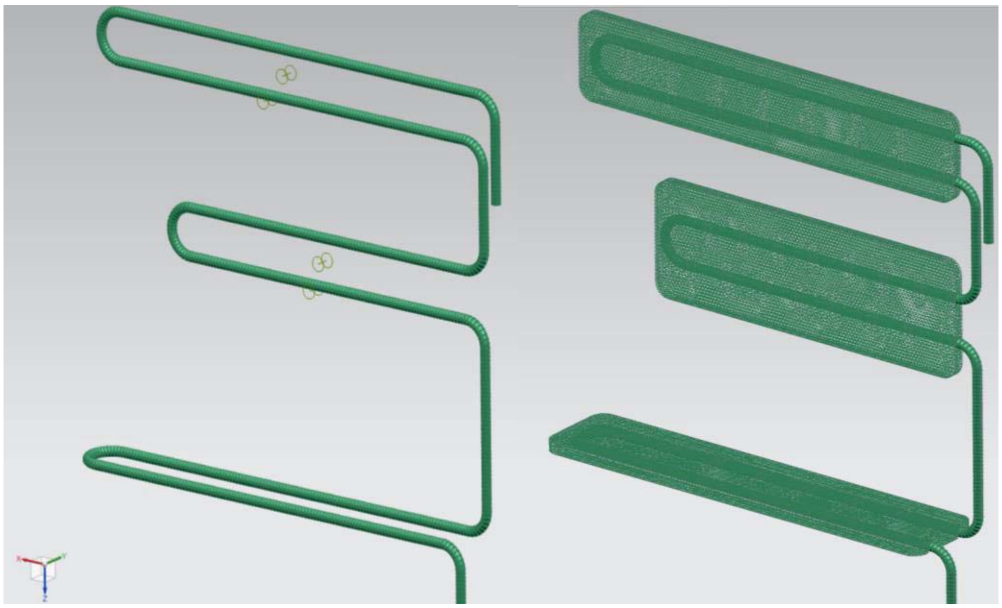


Figure 2. TCS Mesh using Method 1 (1D flow solver)

<u>Method 2: 3D flow solver</u>

In this method 3D tetrahedral elements were used to mesh the cold plates, tube wall, and fluid regions. For the fluid region a structured quadrilateral o-grid would typically be used to resolve the boundary layers. Instead an unstructured tetrahedral mesh was used in conjunction with wall functions. Using wall functions allows for the mesh to be larger in size, so the viscous sublayer does not need to be resolved. If wall functions were not used, a quadrilateral mesh would have been implemented and a $y+=1$ would have been desired. An image of the tube wall and fluid region mesh is shown in Figure 3.

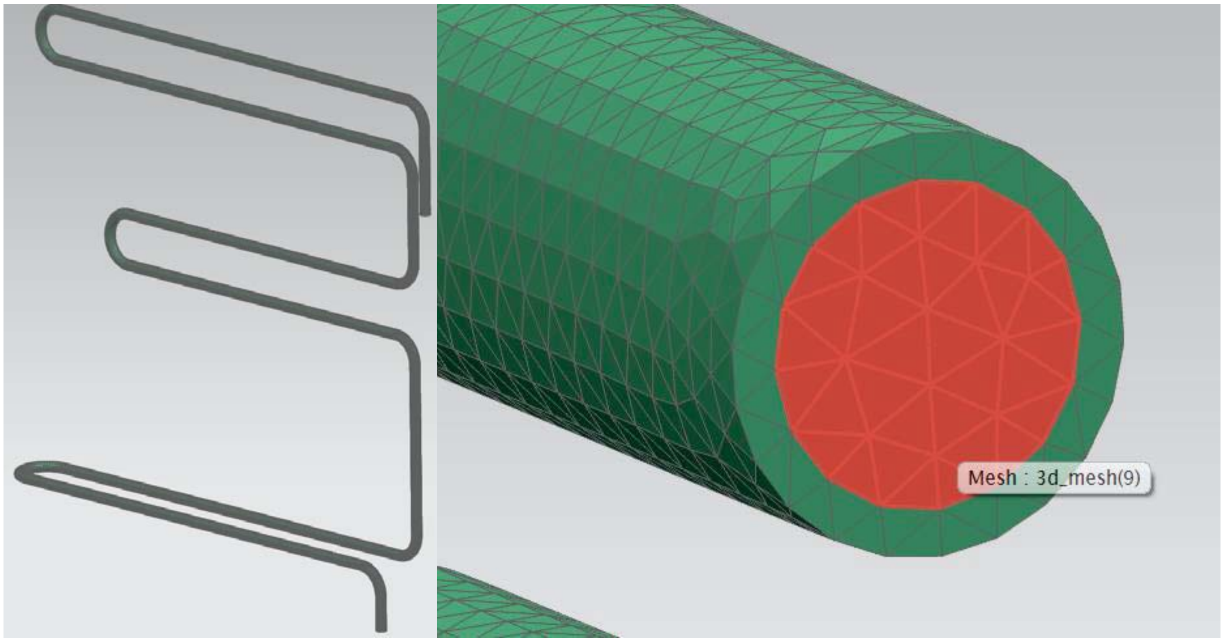


Figure 3. TCS Fluid Region Mesh using Method 2 (3D flow solver)

The $y+$ was calculated using [1] estimated for each turbulent flow case, which was assumed to be when the Reynolds number was greater than 2000. The equation typically used to determine this may be seen in Equation 1.

$$y^+ = \frac{y \cdot u^*}{\nu} \qquad \textbf{Equation 1}$$

From [2] *pages 358-359*

The purpose of calculating the $y+$ values for these flow cases was to determine the mesh size that would be required. An image of the turbulent boundary layer from [3] is shown below in Figure 4. In this figure we are concerned with the inner layer shown by the #4. This is compose of the viscous sublayer #1 $(y^+ < 5)$, the buffer layer #2 $(5 \leq y^+ < 30)$, and the log-law region #3 $(30 \leq y^+ < 200)$. When using wall functions you must be sure that the y^+ produced by the model is greater than 5 and in the buffer layer or higher. The main goal of using wall functions is to be outside the viscous sublayer. For this analysis hand calculations were initially performed to try and keep the mesh size greater than a y^+ of 30 using the average velocity. After calculations were performed, it was determined that the y^+ value of the model ranged between 7.8 to 13.5, which was lower than desired, but still acceptable. The y^+ values determined at the model inlet are shown in Figure 5.

Additionally the boundary conditions and solver inputs to the model included the following:

- Constant mass flow rate for each flow case

- Parallel processor solving (6 processors)
- Mixing Length Turbulent model (Reynolds >2000)
- Constant heat load for each TEC
- Rough walled tubing
- Mesh mating conditions
- Steady state solutions

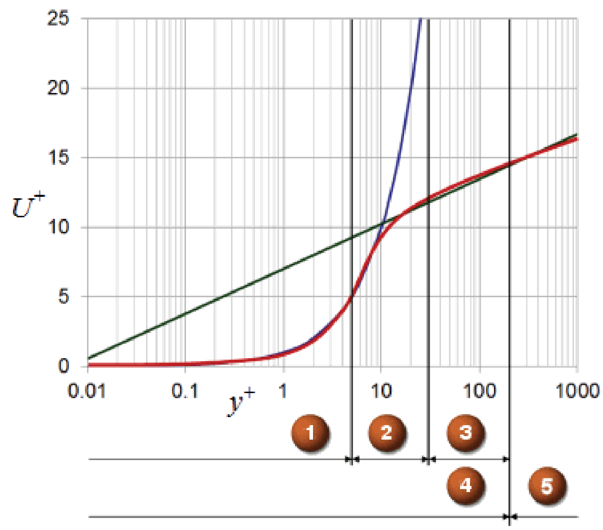


Figure 4. Turbulent Boundary Layer Depiction from [3]

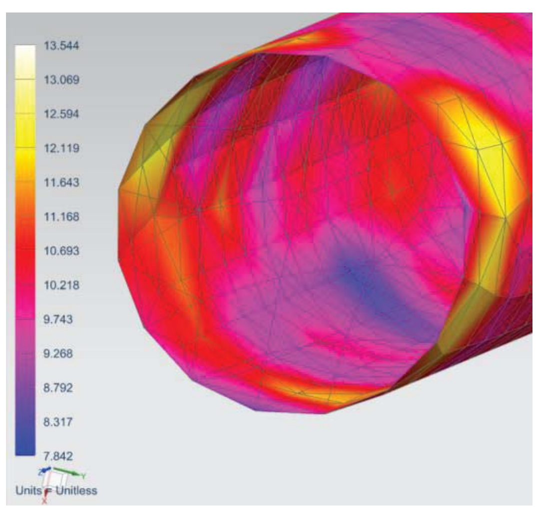


Figure 5. Y+ Values for inlet of TCUI for Method 2

RESULTS

Originally the two different methods were intended to be used for results verification. After initial results were completed, the results for cold plate temperature and fluid pressure drop were scene to have large discrepancies, while the fluid temperature exiting the TCS was not seen to change. The following sections will give an overview of the results determined from the analysis.

Method 1: 1D flow solver results

The exact results were extracted from contour profiles shown in the Siemens NX software. The average temperature on all three cold plates may be seen in Figure 6. This shows that the average temperatures of the TCUI and HCU start high and begin to reduce as flow rate increases, while the TCUF has the opposite response (due to TECs cooling). While the trending of the results looks reasonable, the magnitude of the cold plate temperature difference seemed large and will be compared to Method 2. The temperature contour profiles at 70 lbm/hr may be seen for all three components of the TCS in Figure 7.

In addition to the thermal surface temperatures of the cold plates, the fluid exit temperature and fluid pressure drop were also evaluated. The fluid temperature exiting each cold plate may be seen in Figure 8, while the fluid pressure drop through the TCS is shown in Figure 9. Both figures appear to have realistic trending results, however when the pressure drop numbers are compared to Method 2 a larger discrepancy exists. The total pressure drop across the TCS and several other subsystems must remain below 2.8 +/- 0.15 PSID, which is the system requirement. Pressure drop results from this method appear to be well within that range, showing a maximum pressure differential of 0.243 PSID. These results will be compared against those received from 3D flow model (CFD) used in Method 2.

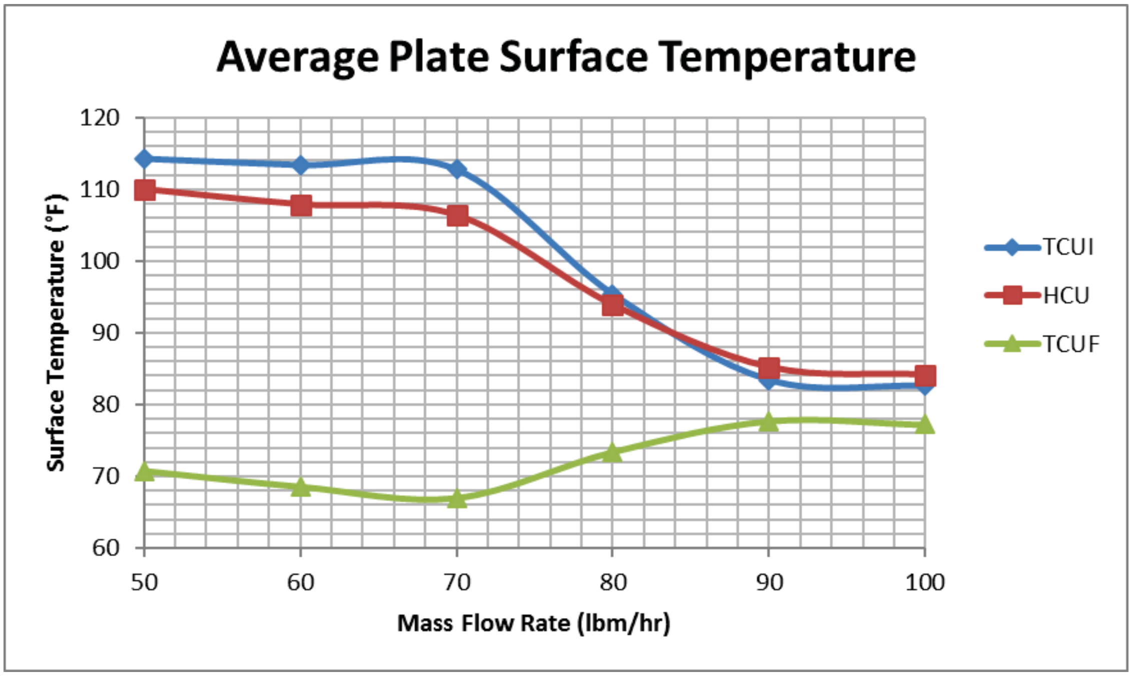


Figure 6. Average Surface Temperature of Cold Plates for Method 1

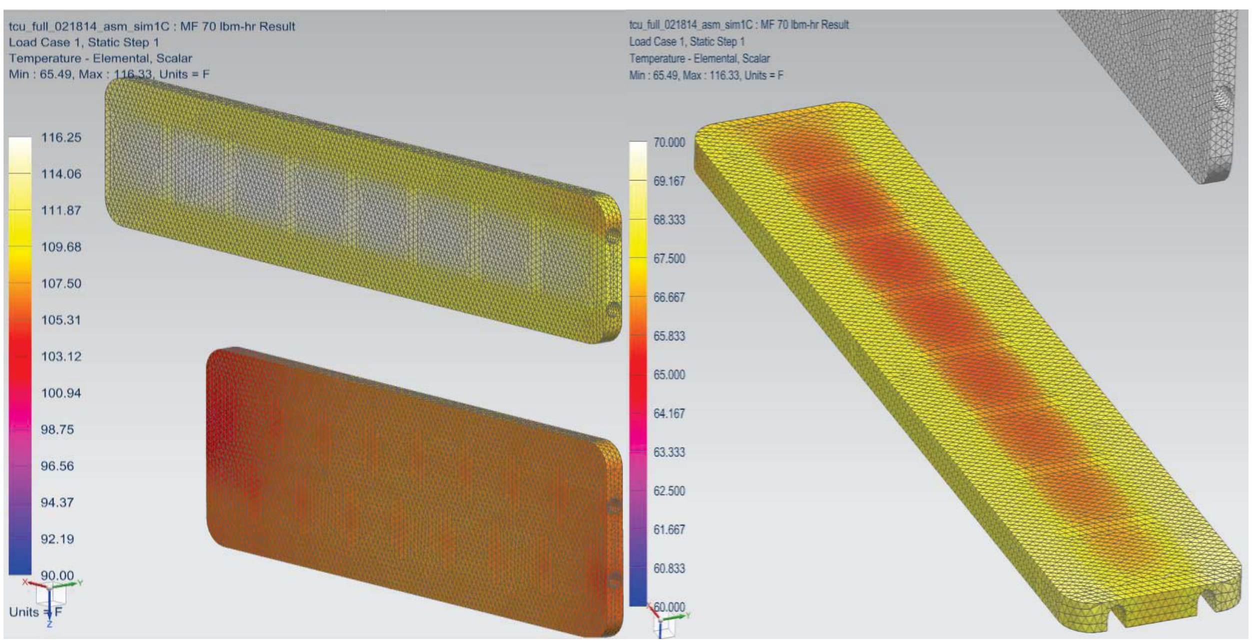


Figure 7. Temperature Contour of TCS Cold Plates for Method 1 @ 70 lbm/hr

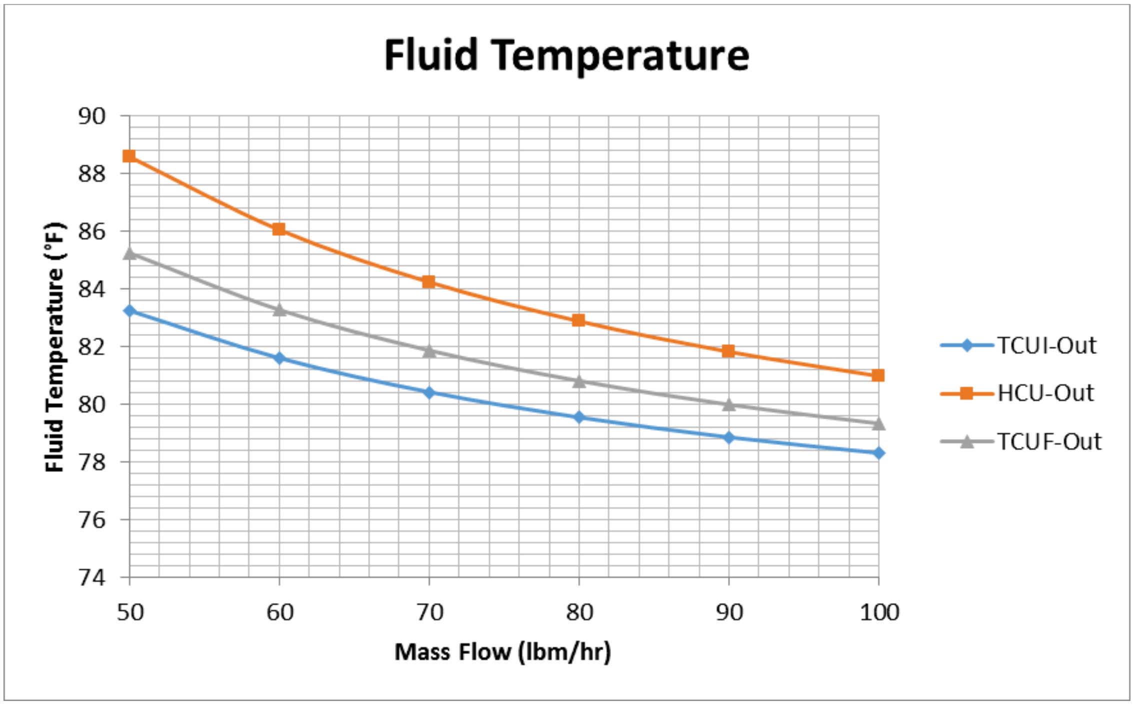


Figure 8. Fluid Temperature Exiting TCS System for Method 1

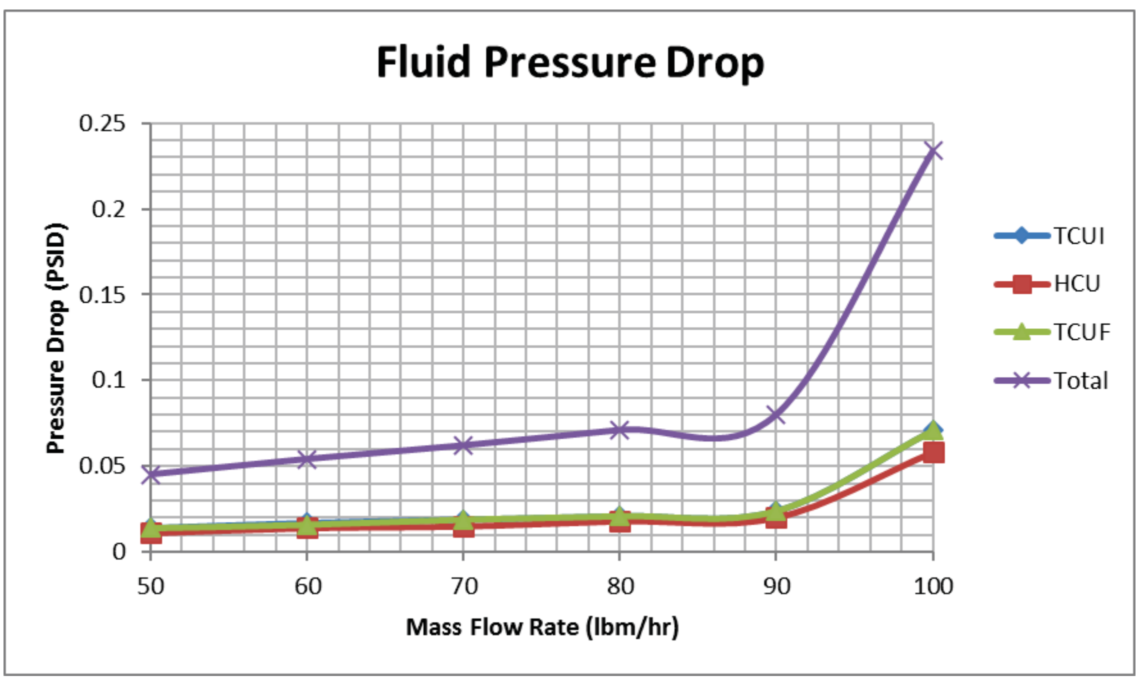


Figure 9. Fluid Pressure Drop in TCS for Method 1

Method 2: 3D flow solver results

The exact results were extracted from contour profiles shown in the Siemens NX software for both 3D thermal and fluid meshes. The average plate surface temperatures for all three cold plates have been plotted in Figure 10. When comparing these results with those shown previously in Figure 6 some discrepancies are noticed. Both Figure 6 (Method 1) and Figure 10 (Method 2) have surfaces temperatures decreasing as fluid travels from the TCUI to TCUF. Both methods use the same TECs heat input and cold plate mesh. The thermal and fluid models for Method 2 are connected with mesh mating conditions, instead of convection coupling. This could attribute to the different results. The temperature contour profiles at 70 lbm/hr may be seen for all three components of the TCS in Figure 11.

As with Method 1, the fluid exit temperature and fluid pressure drop through each cold plate was captured in Figure 12 and Figure 13 for Method 2. By comparing the fluid exit temperature in Figure 8 (Method 1) and Figure 12 (Method 2) shows very little difference between the two graphs. Results appear to be similar trending and have similar temperature values. Next the pressure drop differences were compared between Figure 9 (Method 1) and Figure 13 (Method 2). Results from Method 2 have a larger overall pressure drop then in Method 1. The increased pressure drop from Method 2 to 1 is about a 246% increase. Furthermore cases for Method 2 (specifically at 80, 90, and 100 lbm/hr) were turbulent, and turbulence modeling was used. The two models tested were the Mixing Length (single equation model) and the K-ε (two-equation model). The variation in pressure drop using these models was found to be less than 3% and the higher pressure drop results were displayed here (Mixing Length model).

While the fluid temperatures in both analyses are almost identical, other parameters differ greatly. Pressure drop in the fluid is higher with the 3D modeling, which is expected with turbulent energy is being modeled. Furthermore there is a very large difference in the predicted plate temperatures. These different methods show two very different temperature profile trends. This difference could be related to issues coupling fluid and thermal interactions and these models should be validated with acceptance test data.

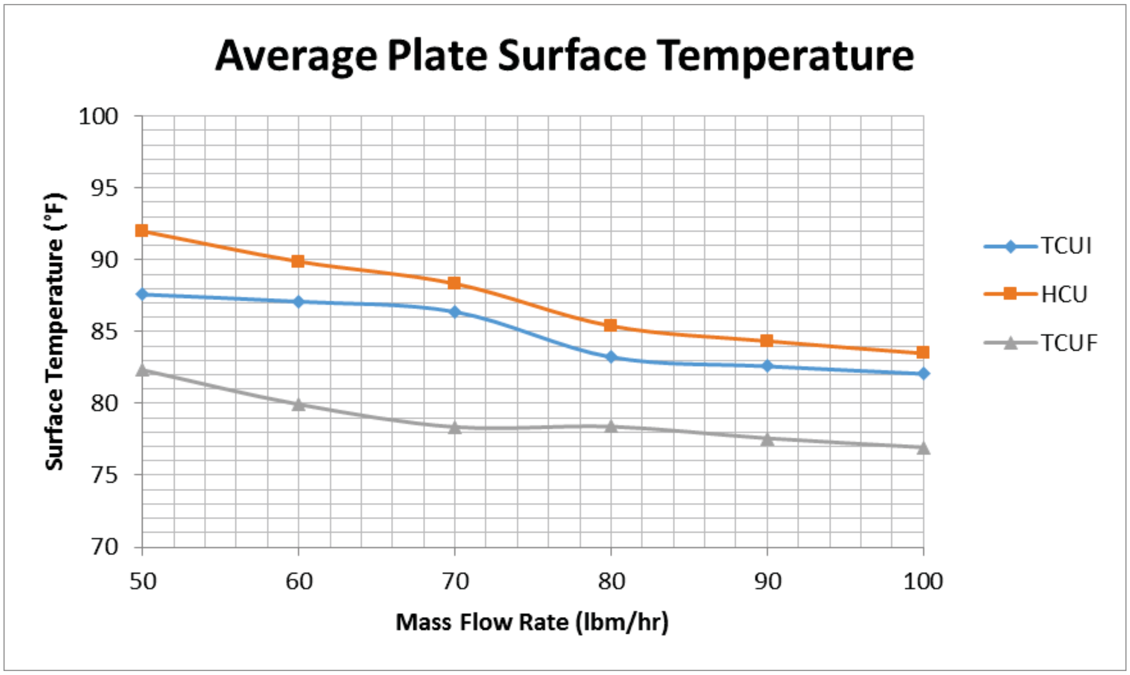


Figure 10. Average Cold Plate Temperature for Method 2

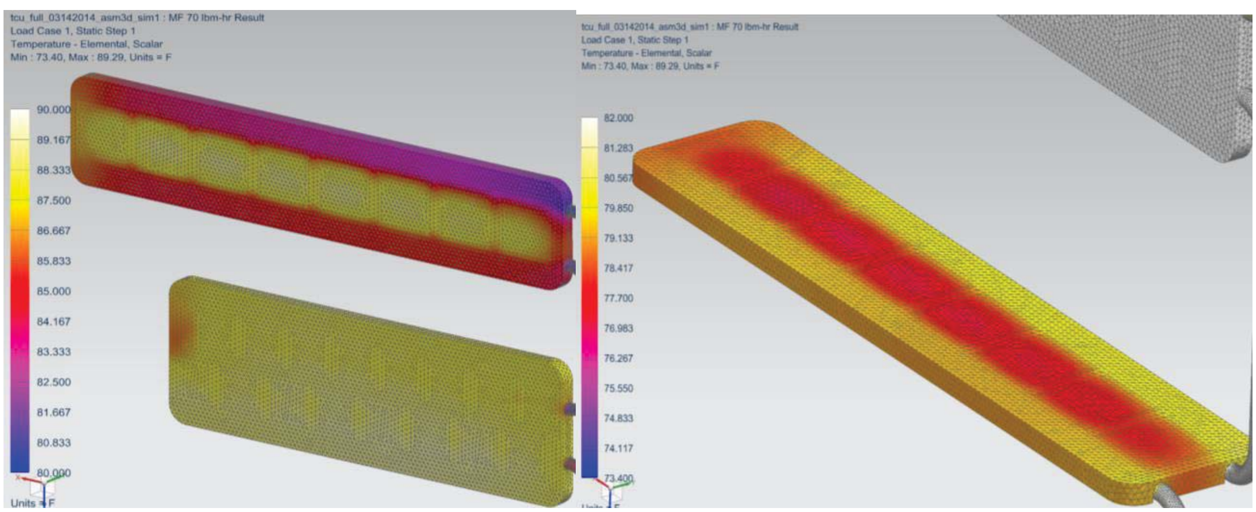


Figure 11. Temperature Contour of TCS Cold Plates for Method 2 @ 70 lbm/hr

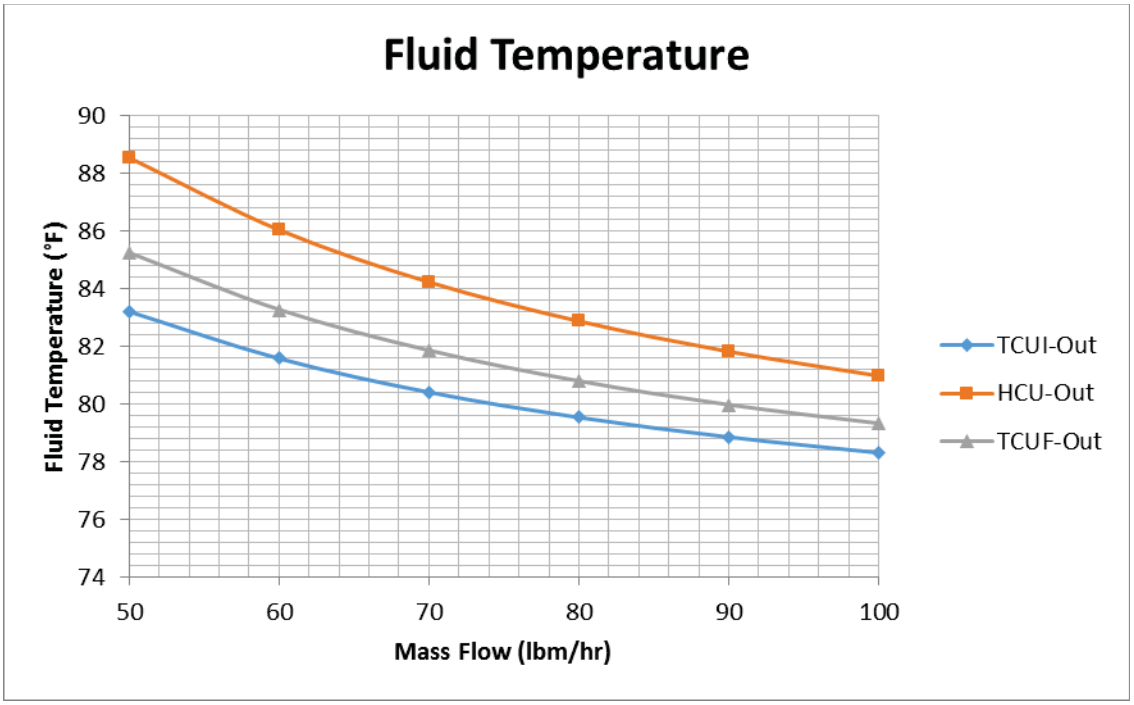


Figure 12. Fluid Temperature Exiting TCS System for Method 2

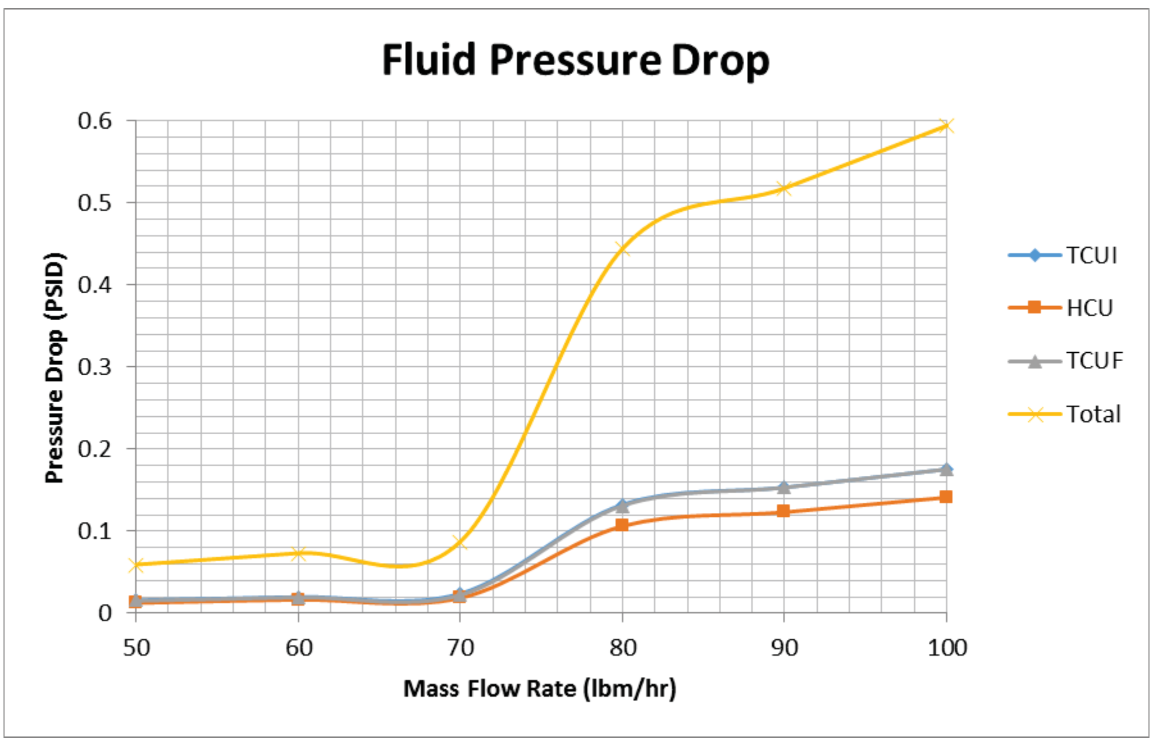


Figure 13. Fluid Pressure Drop in TCS for Method 2

TEST DATA COMPARISON

Currently integrated system testing is being completed at Orbital Technologies (Orbitec) in Wisconsin. While no final results have been published, some preliminary testing results have been provided to the author for model comparison purposes. Based on data provided the 3D CFD model used in Method was closer to experimental data. The difference in pressure drop results between the CFD model and experimental data ranged between 4% to 50% depending on flow rates, while percent difference between the 1D duct network method and experimental data were much higher.

CONCLUSIONS

The results from the two different modeling methods (1D vs. 3D flow model) produced similar fluid exit temperature results, while the cold plate temperatures and fluid pressure drop results were very different. After reviewing system requirements it was determined that a higher importance should be placed on the pressure drop results, instead of temperature. The TCS is only a single subsystem in a larger system that all must remain below a total pressure drop of 2.8 +/- 0.15 PSID. Results from the 3D flow model showed the TCS using up to 21.4% of the allowable pressure loss, while the 1D flow model used only 8.4%. For these reasons the 3D flow solver results were used for developing the prototype TCS for testing.

Since the completion of this analysis, integrated testing has been performed on the TCS at Orbital Technologies (Orbitec) in Wisconsin. Preliminary testing results showed that the results from the 3D flow model were similar to actual system performance. Release of the results from Orbitec are still pending.

NOMENCLATURE

APH	advanced plant habitat
EXPRESS	Expedite the Processing of Experiments for Space Station Racks
°F	degree Fahrenheit
ISS	International space station
lbm/hr	pounds of mass per hour
PCG	plant growth chamber
PSID	pound per square inch differential
TCS	thermal control system
TEC	thermoelectric cooler
TCUF	thermal control unit final
TCUI	thermal control unit initial
HCU	humidity control unit
$\boldsymbol{y}$	distance to the nearest wall
$\boldsymbol{y}^{+}$	Non-dimensional wall distance
$\boldsymbol{u}^{*}$	friction velocity
$\boldsymbol{\nu}$	kinematic viscosity

REFERENCES

[1] Siemens, "NX Version 8.5 Users Manual," 2013.

[2] Metacomp Technologies, "CFD++ Version 15 Users Guide," 2013.

[3] F. M. White, Fluid Mechanics Fifth Edition, New York: McGraw-Hill, 2003.

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

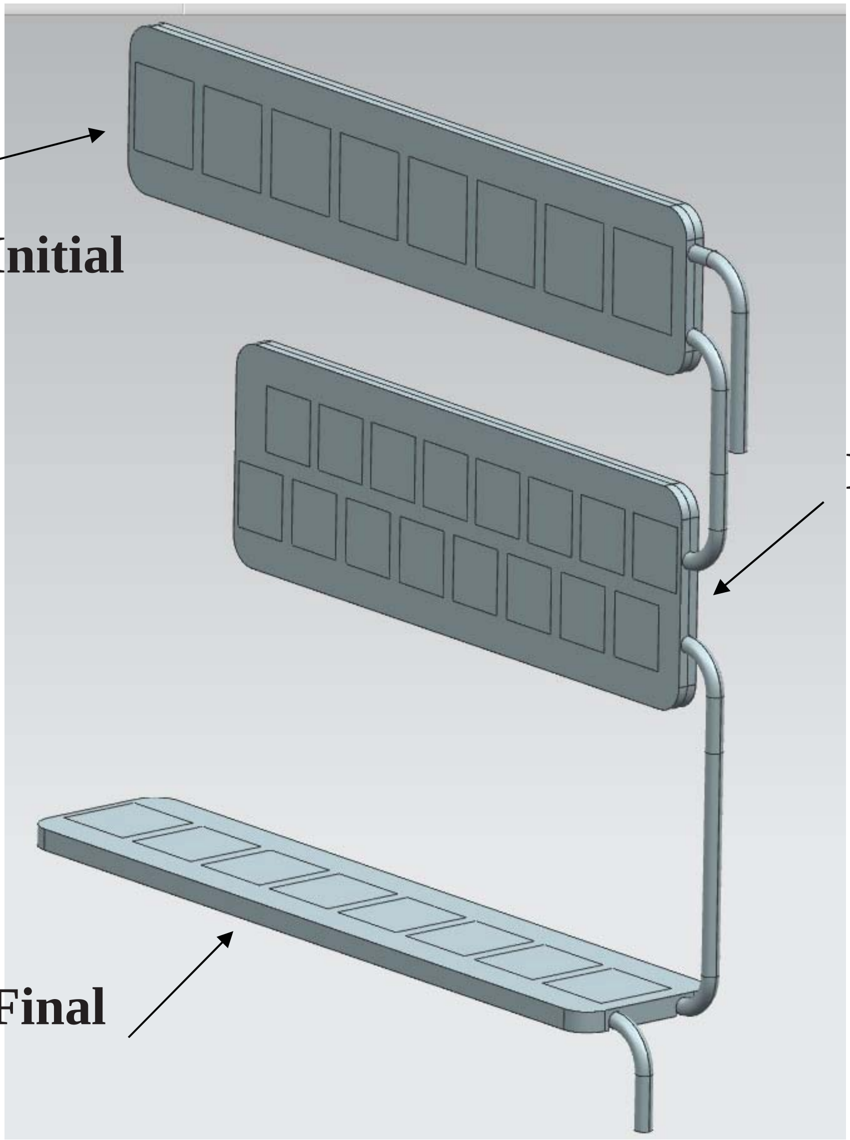
Background

- 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.

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 Fluid Dynamics (CFD) was performed in Siemens NX Thermal/Flow.
  - NX Thermal (Only): Includes 3D mesh thermal solver and 1D mesh solver for fluid flow in pipe or tubes. This may include wall thickness (if desired).
  - NX Thermal/Flow: Includes 3D mesh thermal solver and 3D CFD flow solver.
  - Both methods were used and the results will be compared in this presentation.

Model Differences

1D flow model

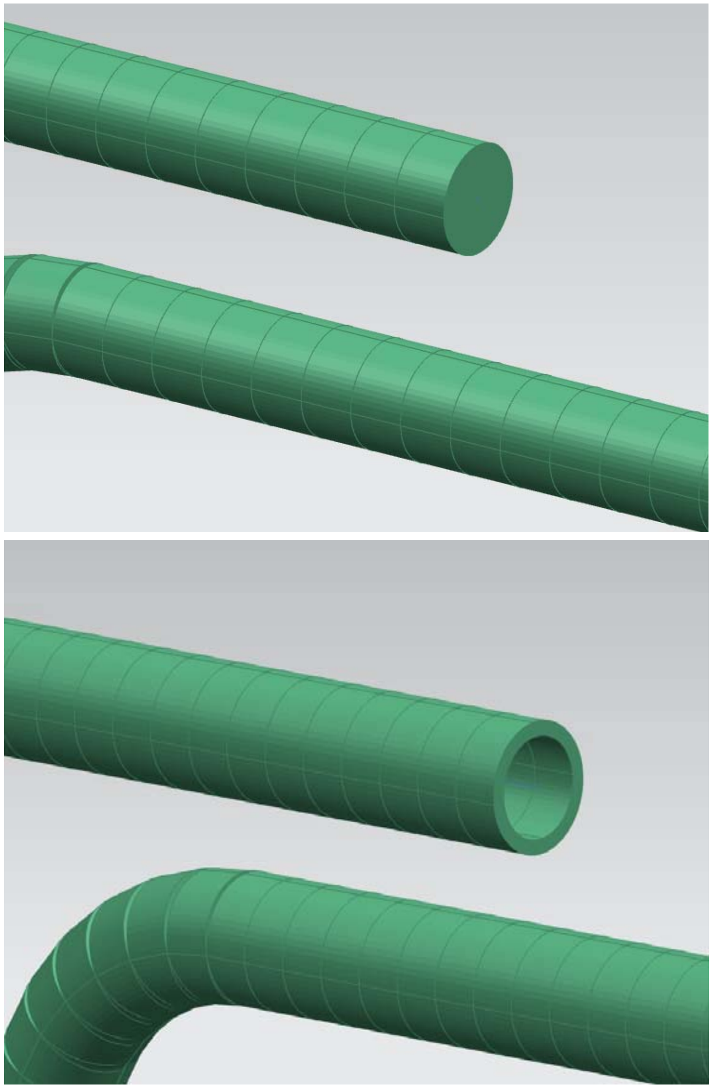
- Steady state solution
- Constant mass flow rate
- Constant heat loads (per TEC)
- Rough walled tubing
- Turbulence begins at Reynolds > 2400 (default)
- Forced convection coupling
- Single processor

3D flow model

- Steady state solution
- Constant mass flow rate
- Constant heat loads (per TEC)
- Rough walled tubing
- Turbulence begins Reynolds > 2000
- Mesh mating conditions
- Uses wall functions
- Parallel (6 processors)

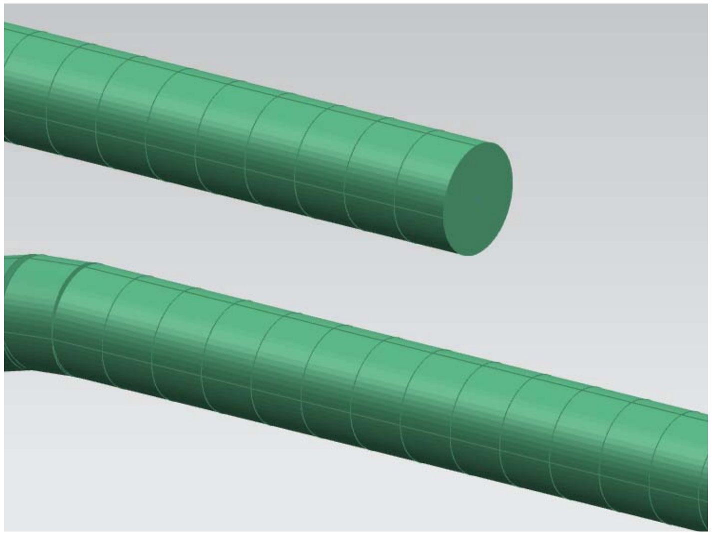
Method 1: 1D duct flow network analysis

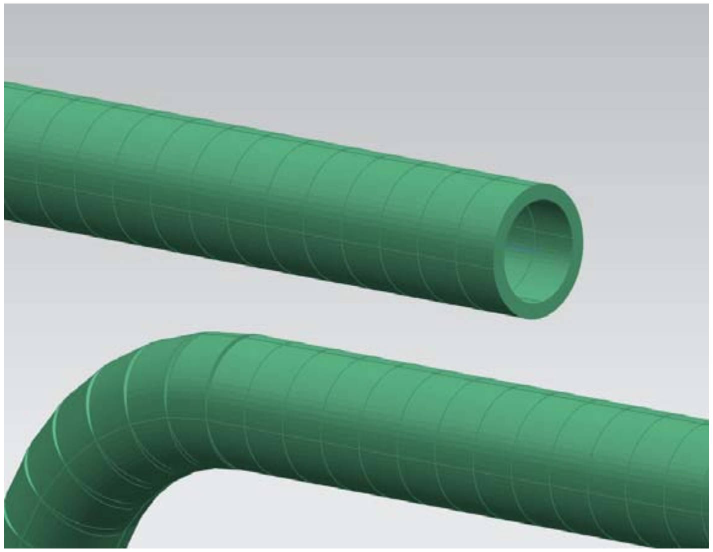
- Models fluid region as one-dimensional fluid duct.
  - Pressure
  - Velocity
  - Mass flow rate
  - Temperature
- 1D mesh
  - Fluid only
  - Fluid and tube wall
- Heat transfer (Thermal coupling)
  - Conduction, convection, radiation
  - Free, forced and user defined convection



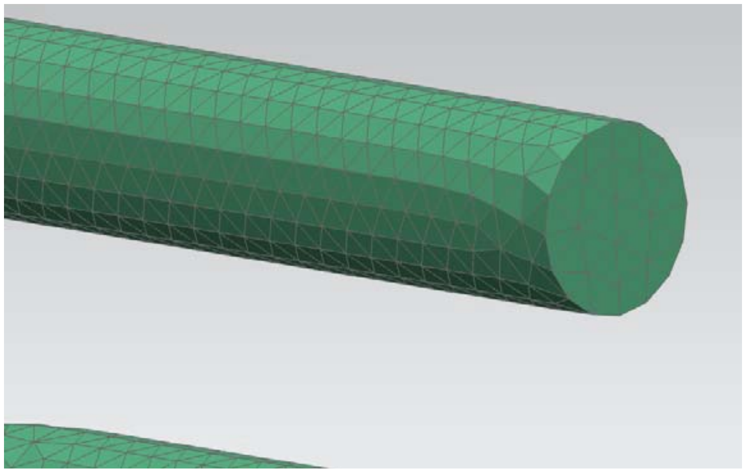
Method 1: 1D duct flow network analysis

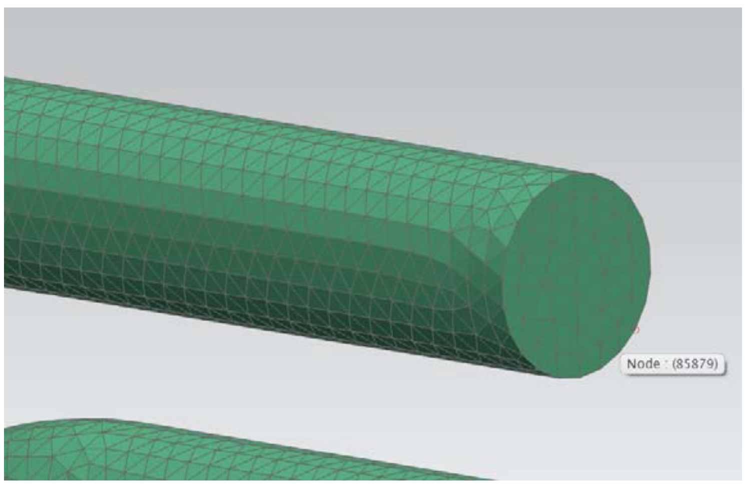
- Does not use NX Flow
  - Only NX Thermal is used
- Duct boundary conditions
  - Fan/pump
  - Opening
  - Pressure
  - Flow properties
  - Duct to 3D flow connection
    - Connects 1D duct to 3D CFD model





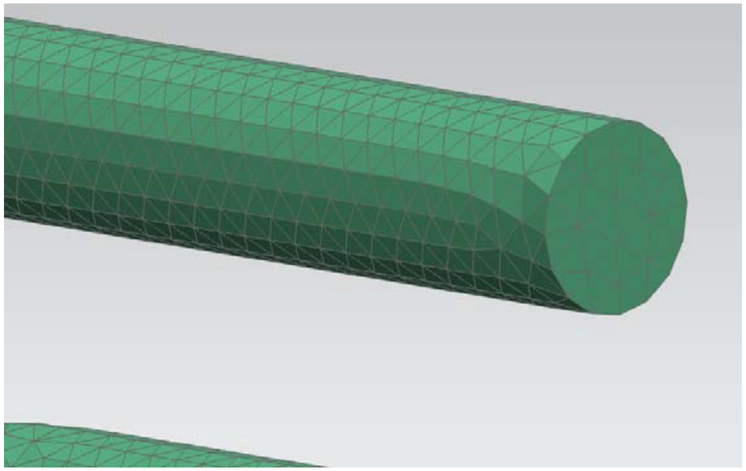
Method 2: 3D CFD Analysis



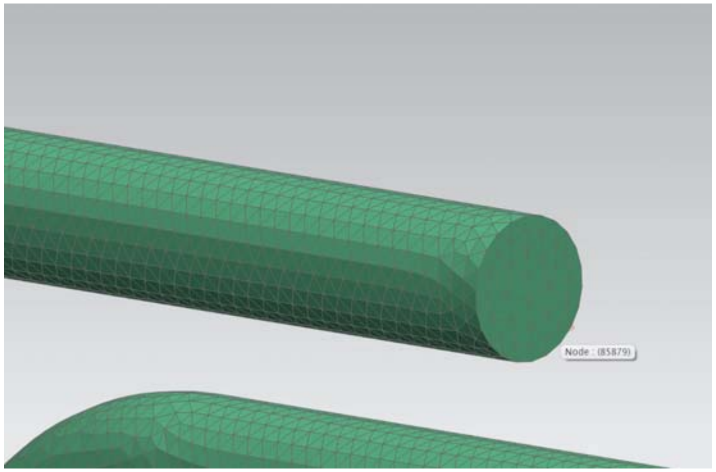


- 3D mesh
  - Structured hexahedral
    - Quad 2D face and sweep (manual)
  - Unstructured tetrahedral
    - Either automatic or manual meshing
- Turbulent model
  - *K-Omega*
  - K-Epsilon
  - *Mixing Length*
  - LES
- Both NX Flow and Thermal
- Serial & Parallel processing

Method 2: 3D CFD Analysis



- Parallel solver used fully coupled pressure velocity scheme
- All results converged for a steady state solution
- Residuals used for the analysis

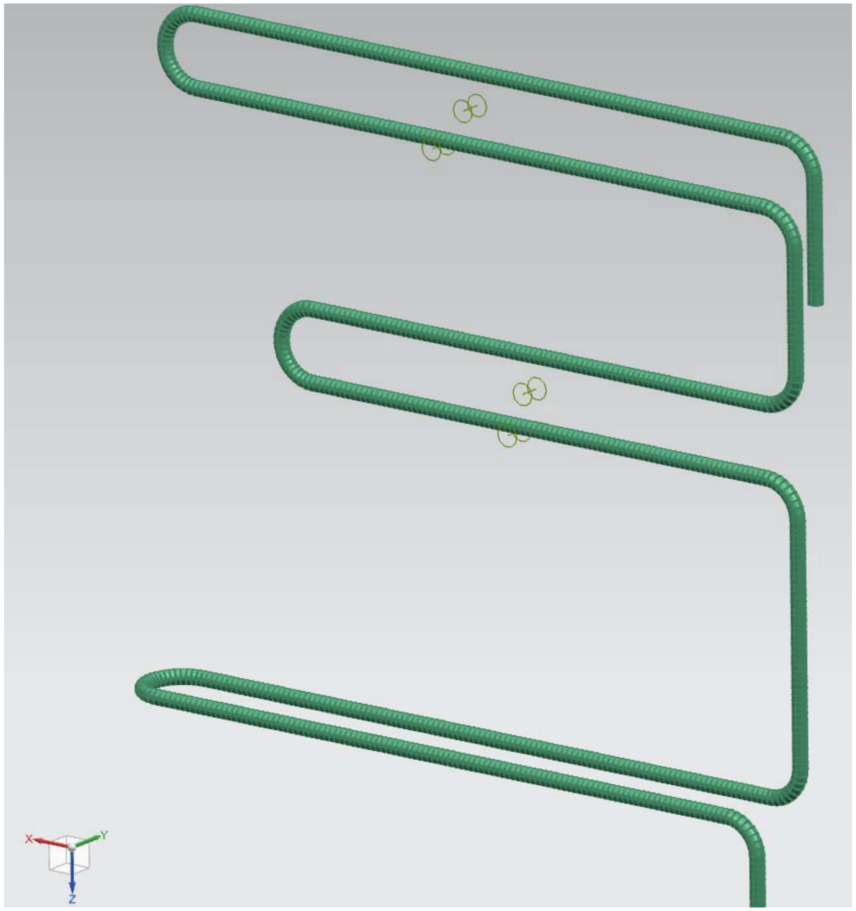


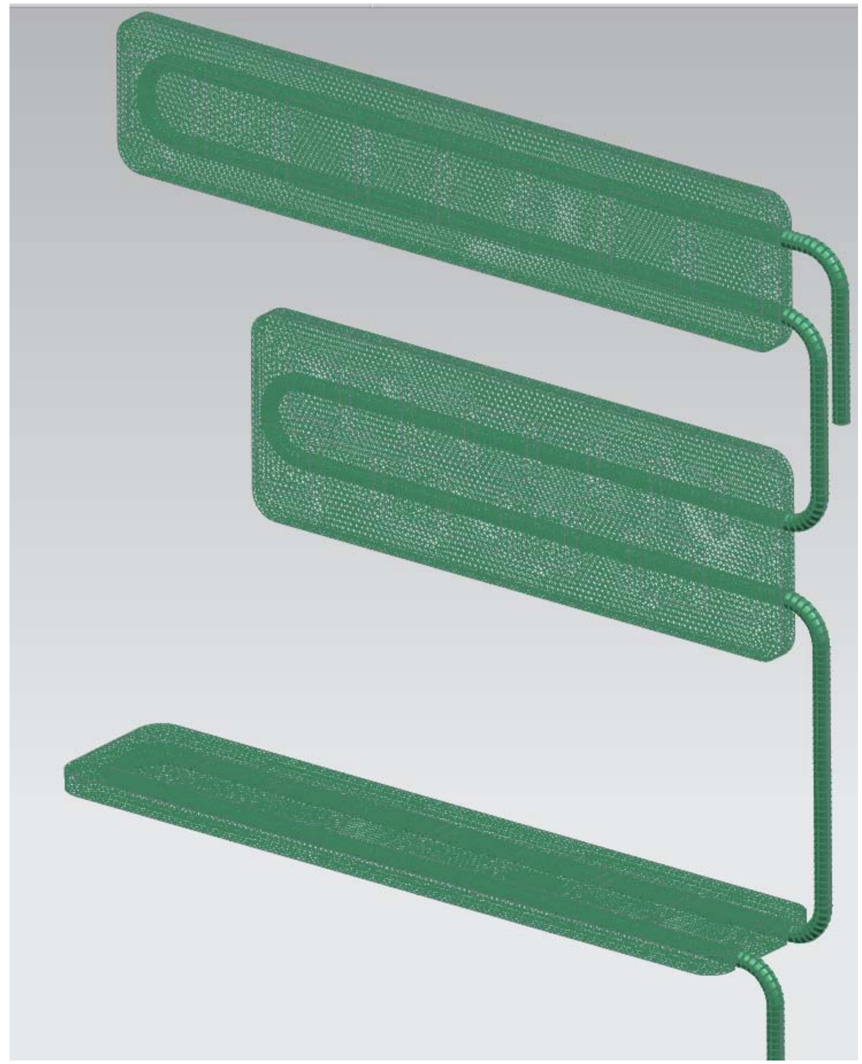
Method 1: Duct Meshing Sizing

- 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 the changing mesh size had no effect on the solution

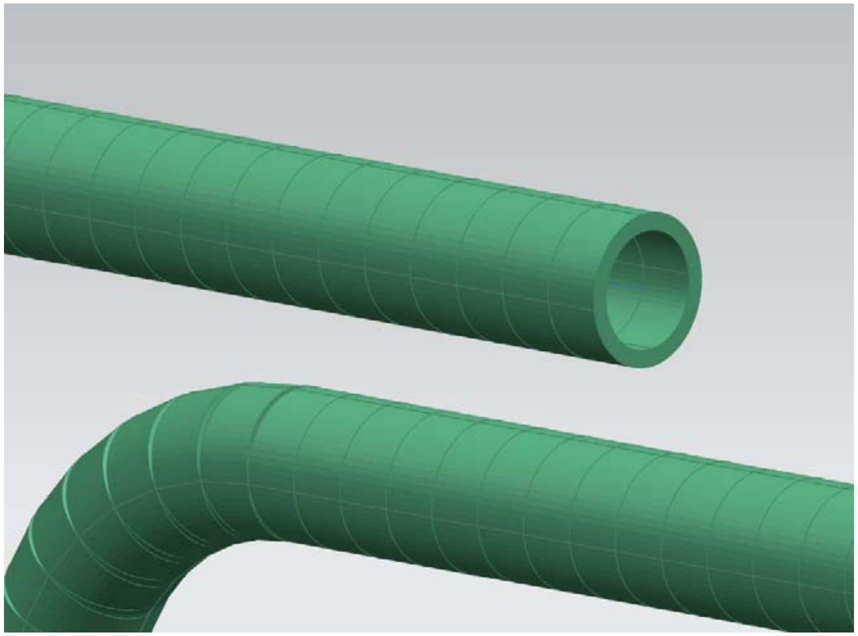
Method 1: Duct Mesh



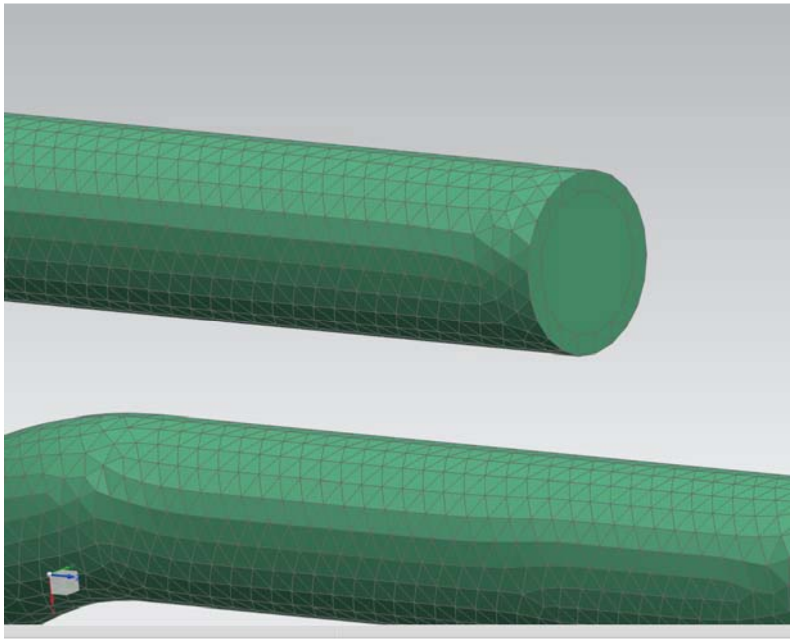


Method 1: Duct Mesh

1D duct with wall



1D duct with 3D wall



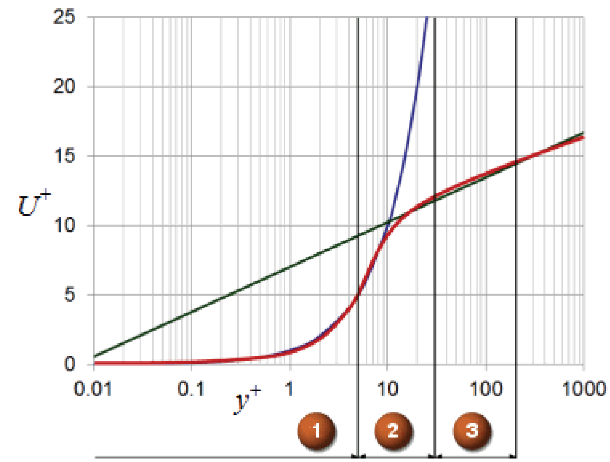
Method 2: CFD Mesh Sizing

- 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)

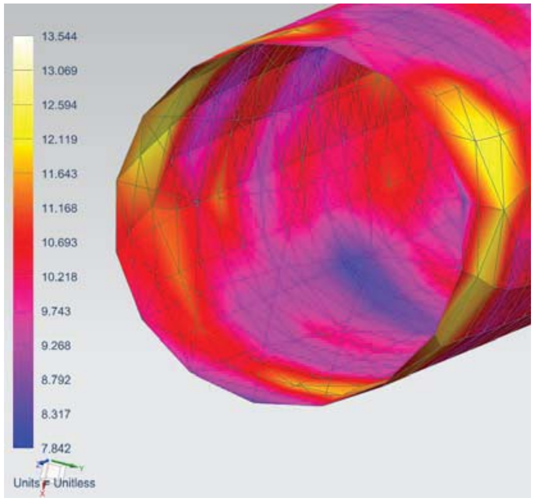
- Mesh sensitivity was tested by varying the mesh size and no effects to the solution was noticed

Method 2: CFD Mesh

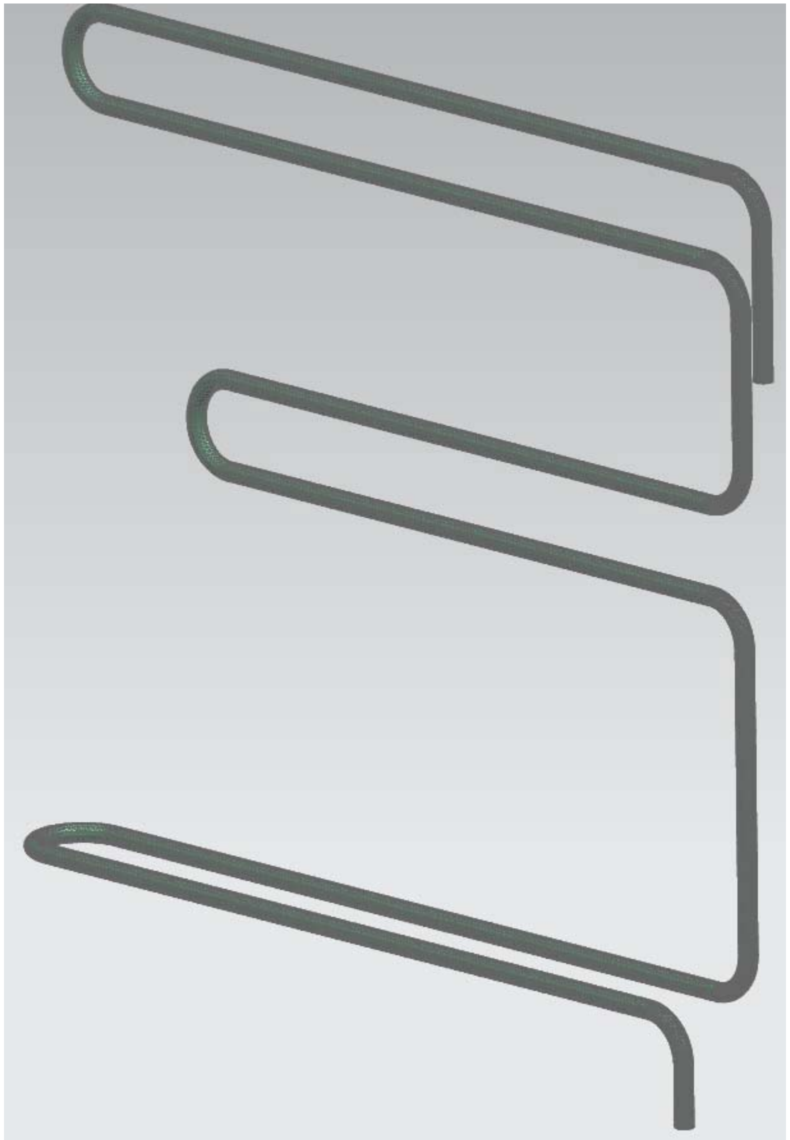
- Inner layer
  1. Viscous sublayer
  2. Buffer layer
  3. Log-law region

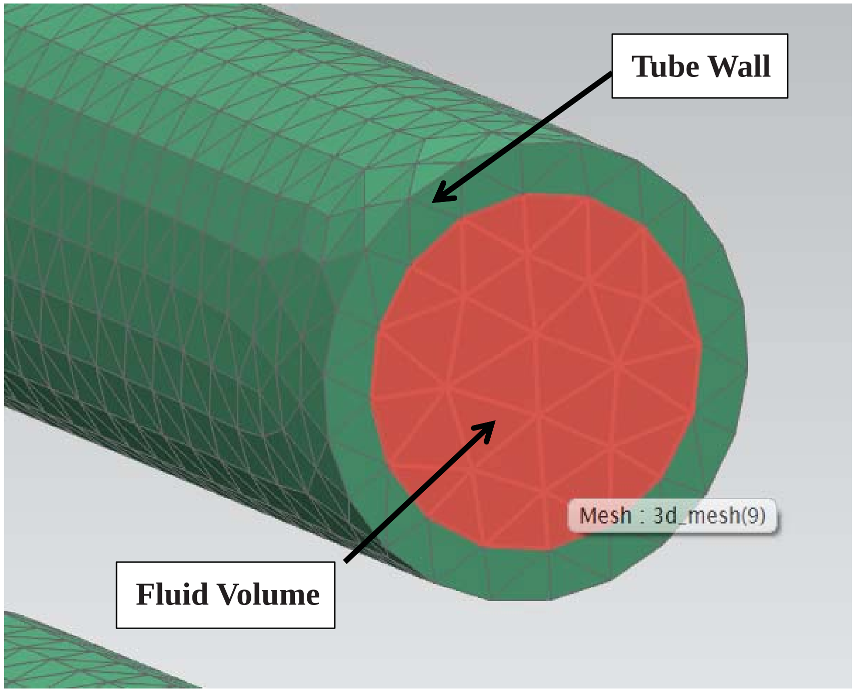


- Goal was to keep mesh large enough so y+>30
  - Actual ranged 7.8 to 13.5



3D flow model mesh

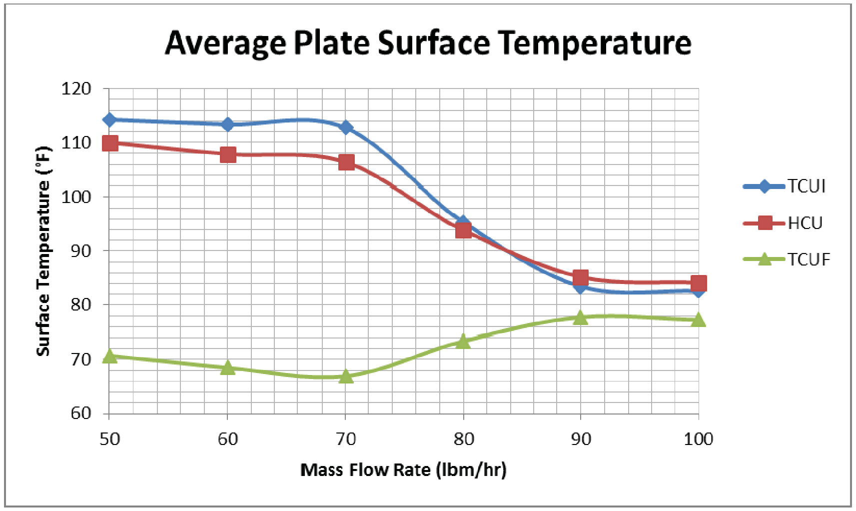


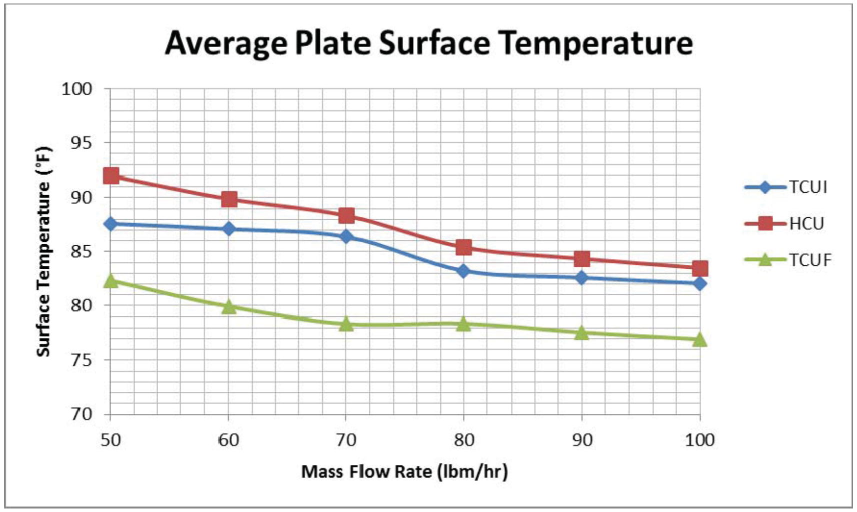


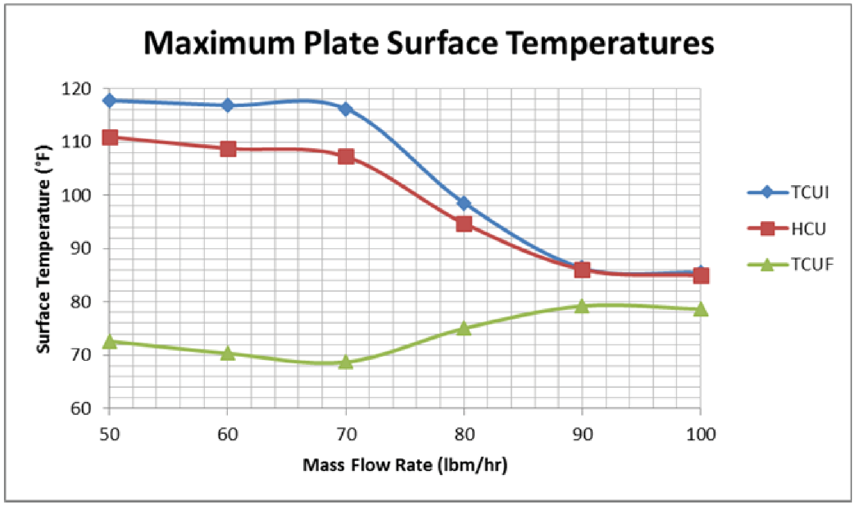
Cold Plate Surface Temperature Results

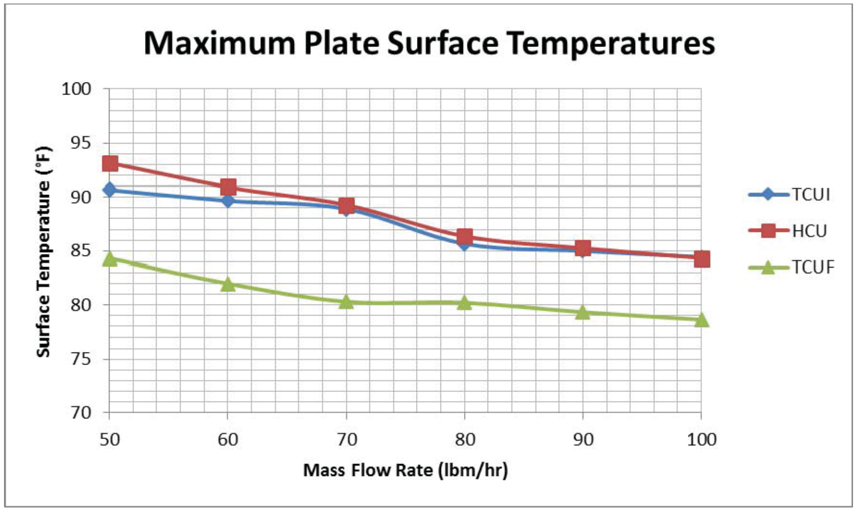
1D flow model

3D flow model



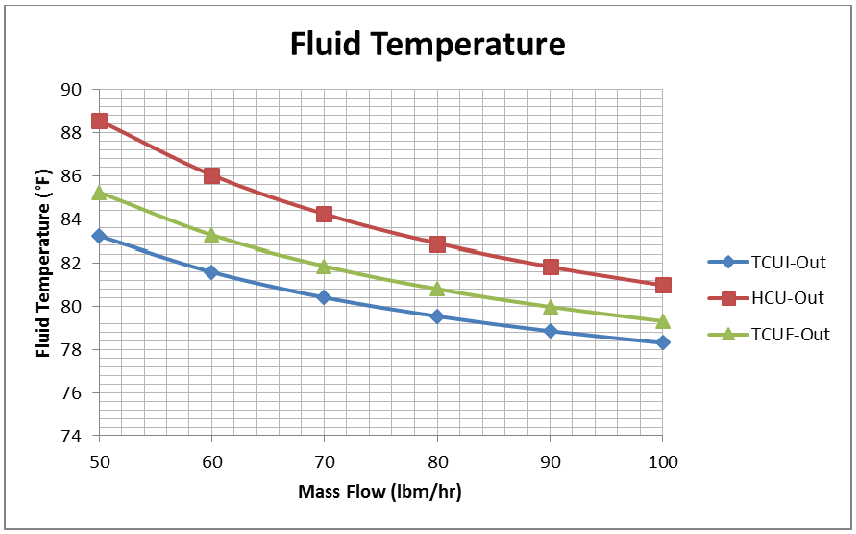




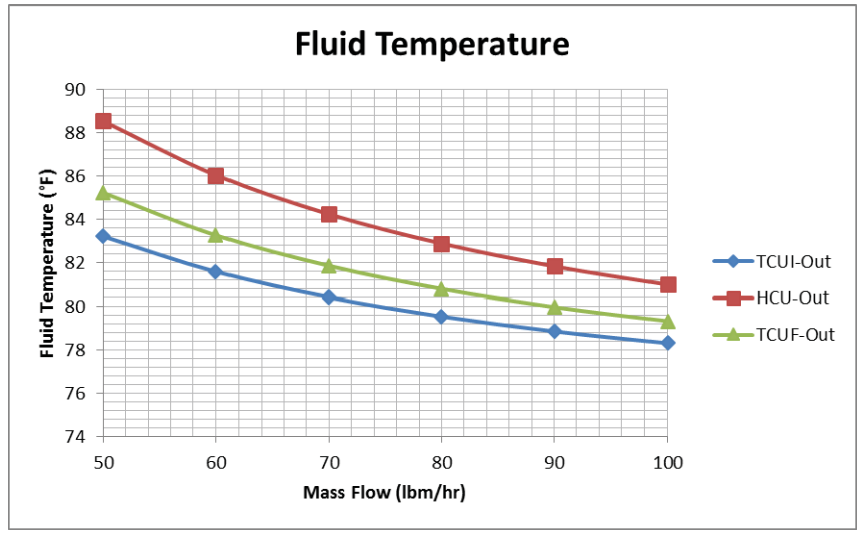


Fluid Temperature Results

1D flow model

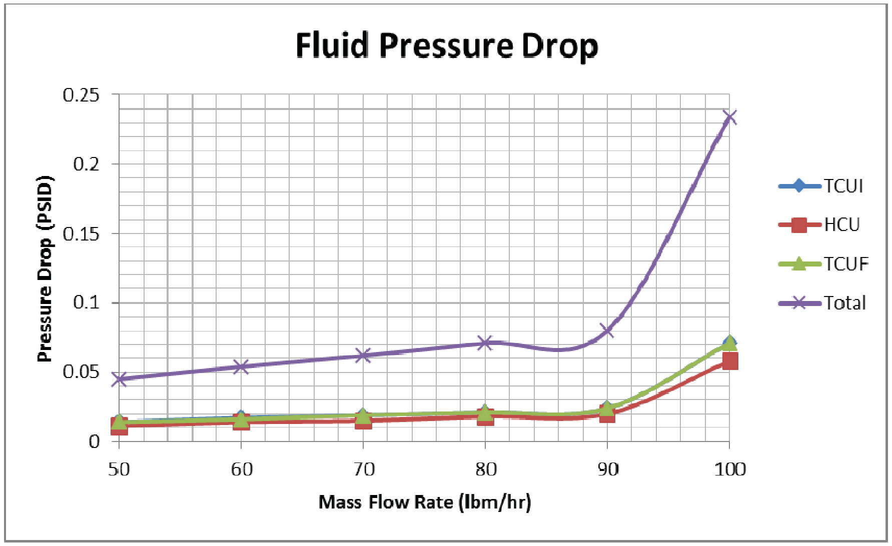


3D flow model

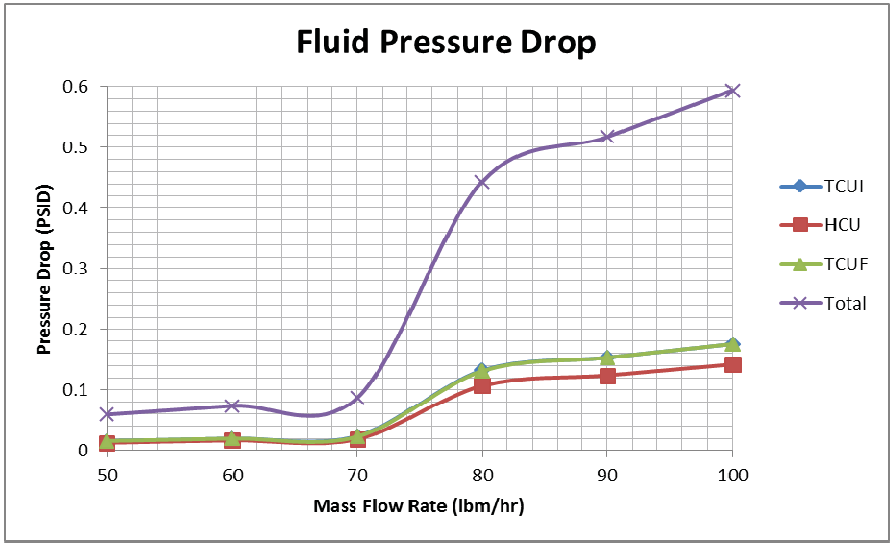


Fluid Pressure Drop Results

1D flow model



3D flow model



Conclusions

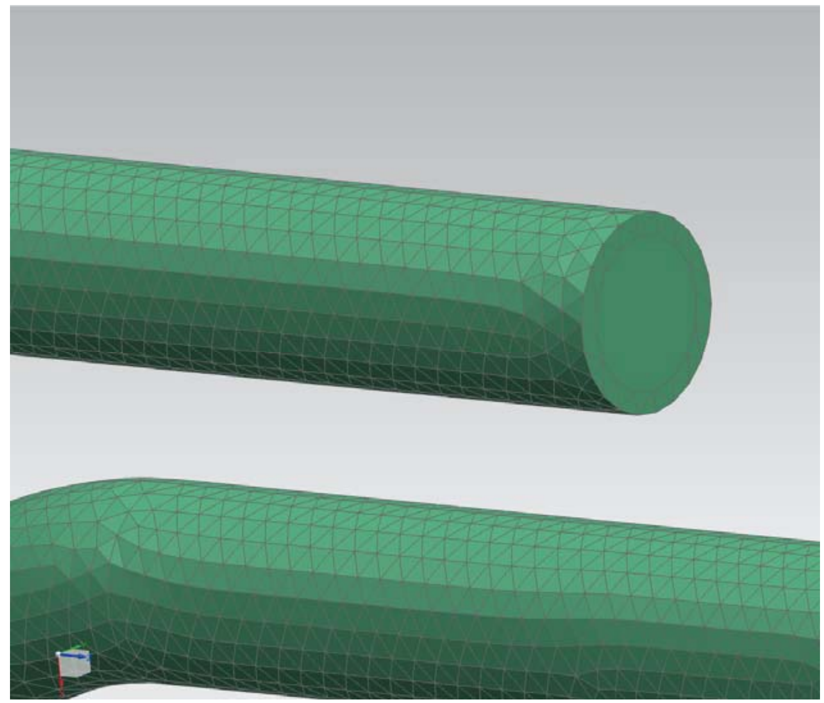
- Models produced similar fluid temperatures, but differed in pressure drop and surface temperature results.
- Testing sensitivity in Method 1 to convection coupling or Method 2 to turbulence model did not produce drastic changes
  - Grid size sensitivity also did not effect the solution
  - Additional pressure drop could be do to additional swirling velocity terms (problems would be in any 1D network model)
- Preliminary data from the integrated testing at Orbitec showed the CFD method was closer to in actual pressure drop to prototype.
  - 1 for 1 comparisons aren't available since on the spot changes were made to the prototype were made, moving the configuration away from the geometry analyzed.

Questions?

Backup

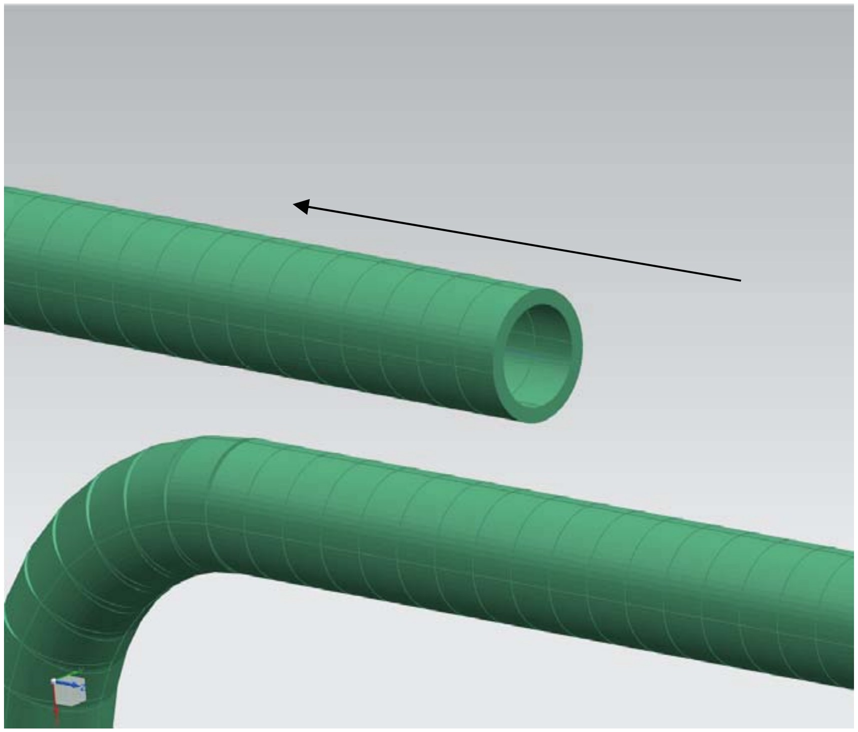
1D flow model (wall included)

1D flow model (wall excluded)

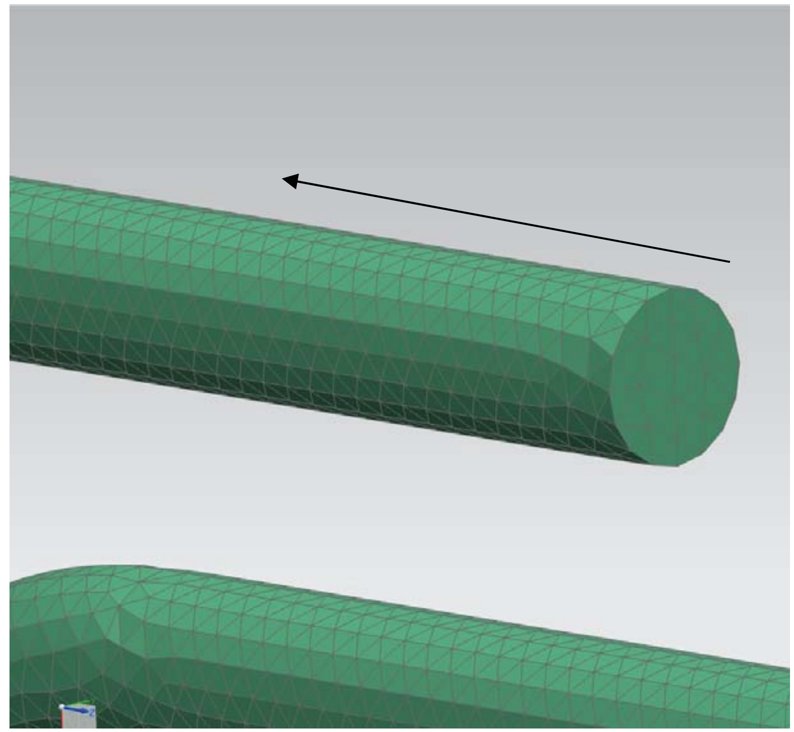


Backup Flow Model Meshes

1D flow model

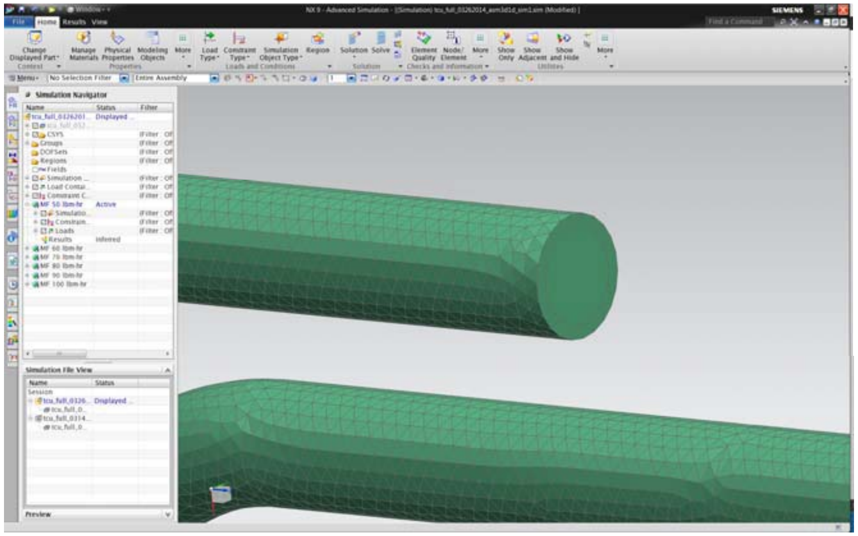


3D flow model



Backup

1D flow model



3D flow model