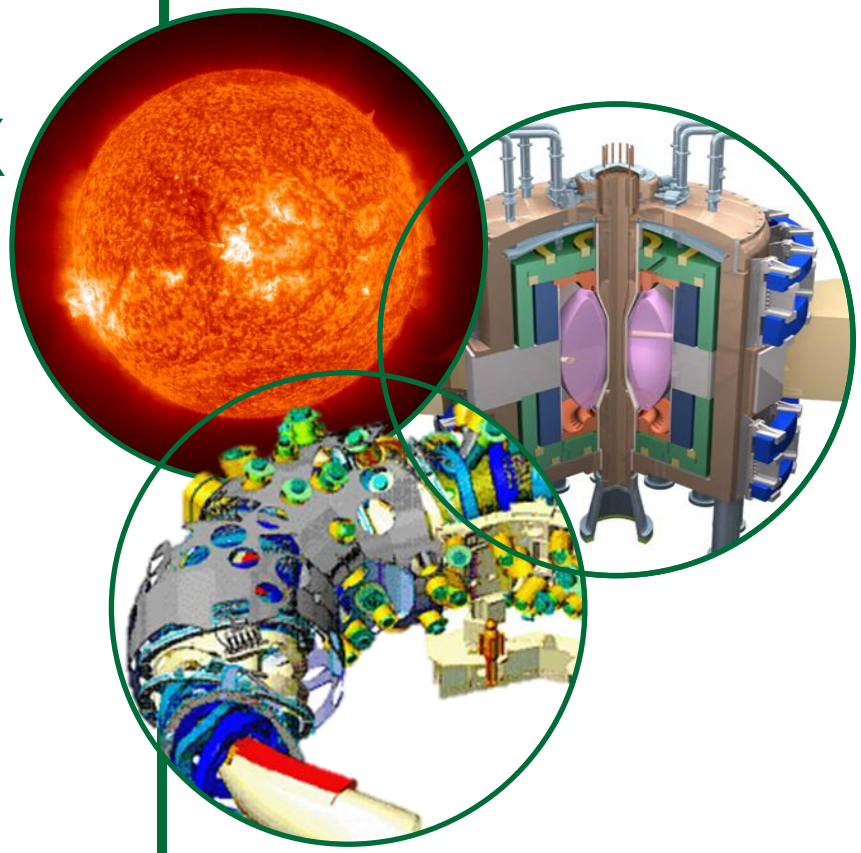
CFD Modeling of High Heat Flux Component Cooling – FNST & W7X

Joseph B Tipton, Jr, PhD

University of Evansville (Evansville, IN)

Summer 2011



U.S. DEPARTMENT OF ENERGY

OAK RIDGE NATIONAL LABORATORY

MANAGED BY UT-BATTELLE FOR THE DEPARTMENT OF ENERGY

Fusion Nuclear Science Facility (FNSF)

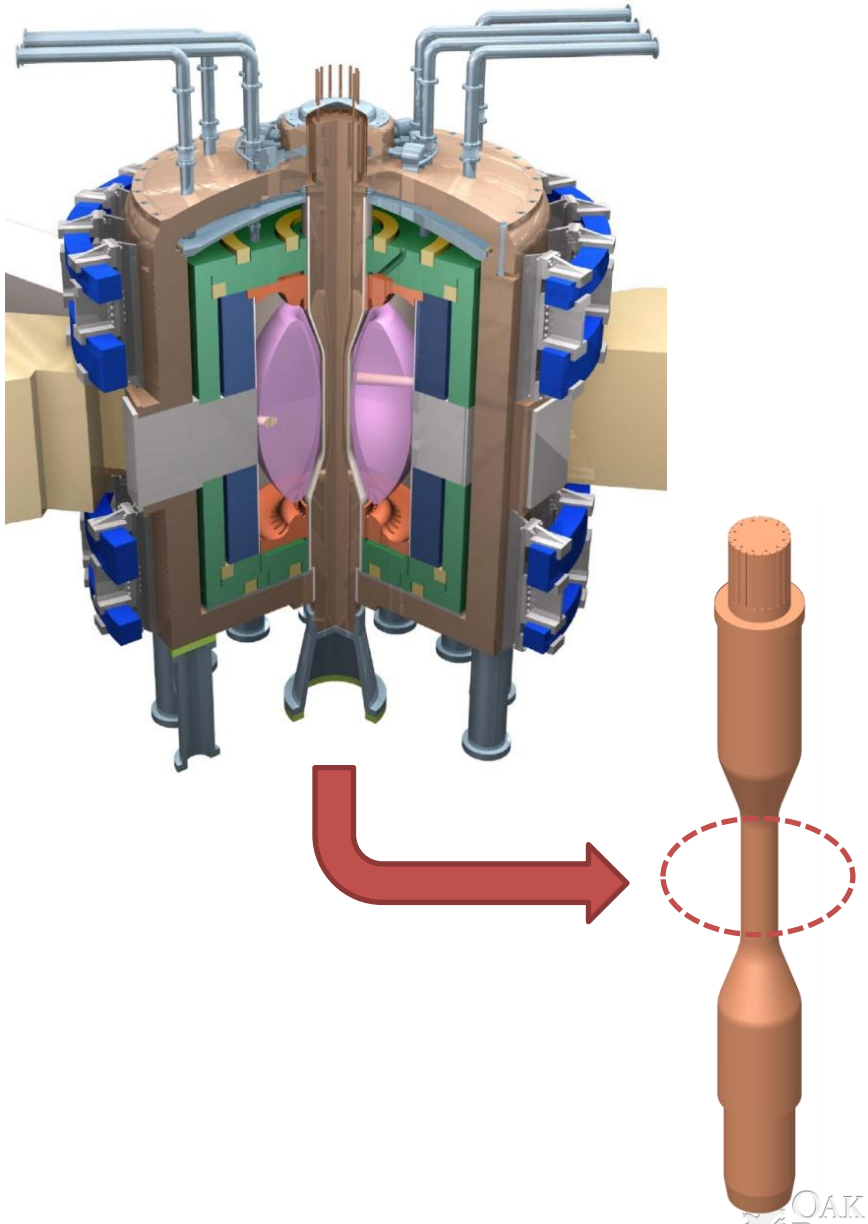
CENTERPOST THERMAL OPTIMIZATION

FNSF Centerpost Thermal Optimization

- FNSF Spherical Tokamak
- Optimization Problem Definition
- Program Flowchart
- Preliminary Results

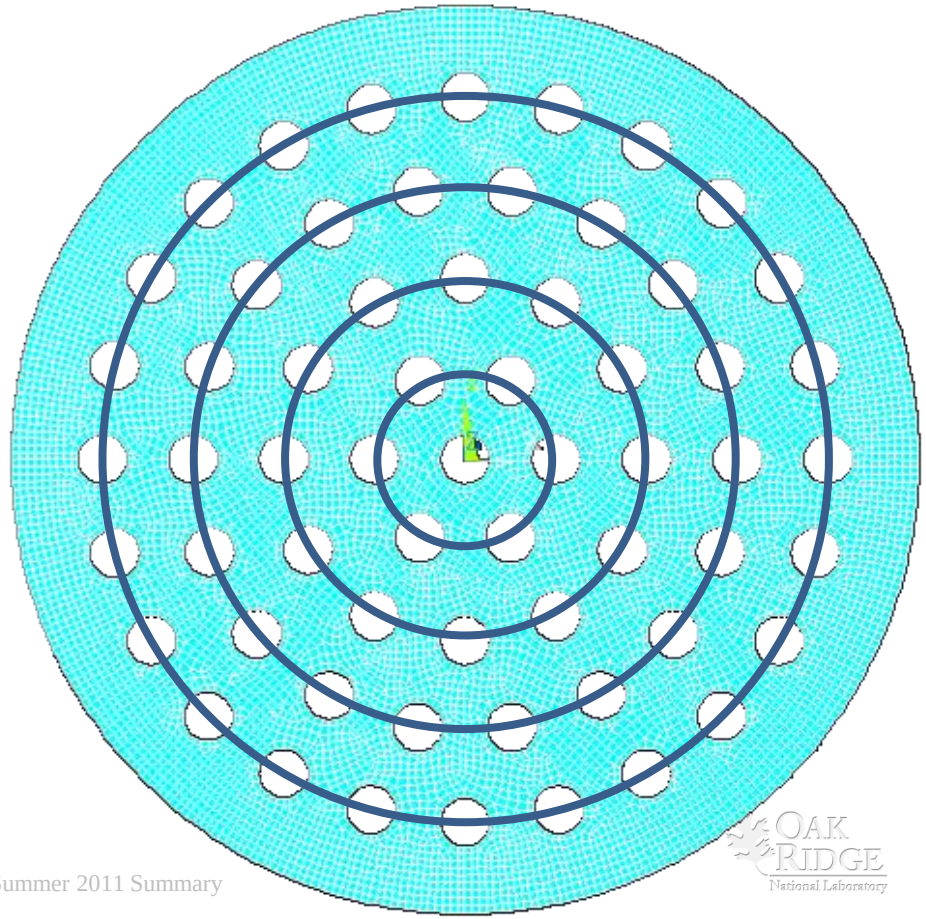
FNSF Spherical Tokamak

- The centerpost is a critical component of the spherical tokamak design.
- This will be shielded with helium cooled tungsten, and actively water cooled.
- The purpose of this study is to demonstrate the capability of optimally designing the water cooling channels in this region to minimize the peak steady-state temperature



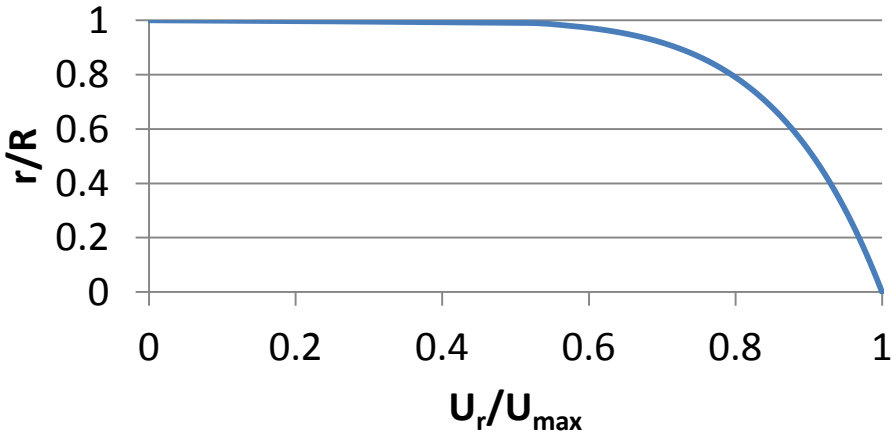
Optimization Problem Definition

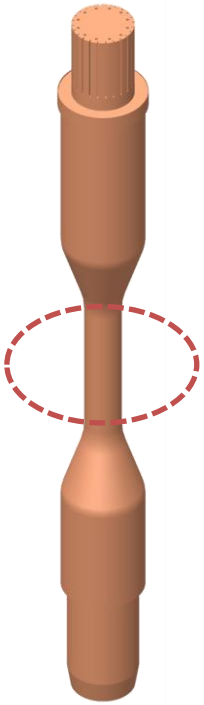
- Objective
  - Minimize maximum temperature in the cross-section
- Design Variables
  - Position of each row
  - Diameter of channel in each row
  - *Angle of orientation of each row*
  - *Number of channels in each row*
- Constraints
  - 5mm distance between channels



Optimization Problem Definition

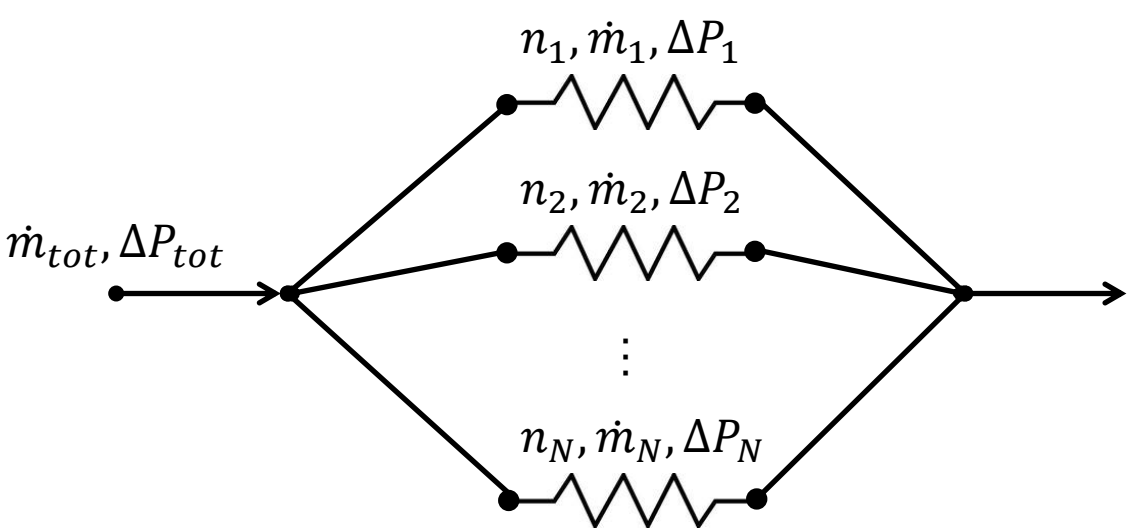
- Assumptions
  - Constant fluid properties
    - Allows fluid dynamics to be uncoupled from heat transfer
  - Fluid channel inlets
    - 1/7th Power Law Velocity Profile
      - $u(r) = u_{max}\left(1 - \frac{r}{R}\right)^{1/7}$
      - $u_{max} = \frac{\dot{m}}{2\pi\rho \int_0^R \left(1-\frac{r}{R}\right)^{1/7} r dr} = \frac{60\dot{m}}{49\pi\rho R^2}$

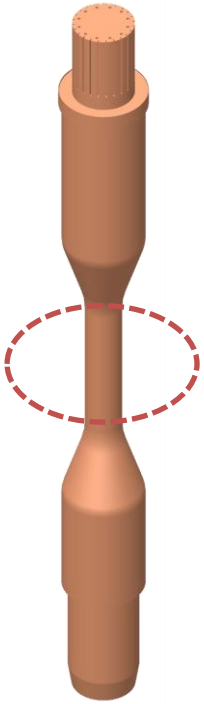




Optimization Problem Definition

- Assumptions
  - Fluid manifold
    - Large inlet manifold
    - Large outlet manifold (uniform backpressure at outlet)
    - Smooth channel walls
    - Fully developed flow
    - $f = 0.184 Re_D^{-1/5}, \; Re_D \geq 20{,}000$





$$\dot{m}_{tot} = \sum_{i=1}^{N} n_i \dot{m}_i$$

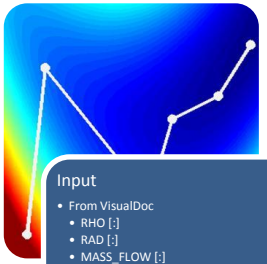
$$\Delta P_{tot} = \Delta P_1 = \cdots = \Delta P_N$$

$$\dot{m}_1 = \frac{\dot{m}_{tot}}{n_1 + \sum_{i=2}^{N} n_i \left(\frac{D_i}{D_{i-1}}\right)^{24/9}}$$

$$\dot{m}_i = \left(\frac{D_i}{D_1}\right)^{24/9} \dot{m}_1$$

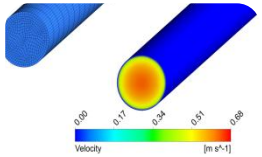
Program Flowchart

- VisualDoc optimization software interfaces with ANSYS Multiphysics Workbench via Python scripting language:

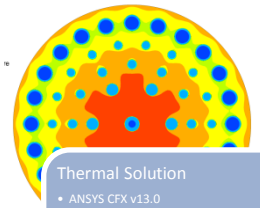


Input
- From VisualDoc
  - RHO [:]
  - RAD [:]
  - MASS_FLOW [:]
- (Using Assumptions)

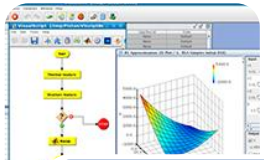
Mesh Gen
- ANSYS APDL v13.0
- Solid
- 1 Channel for every row
  - y+ ≈ 100
  - > 10 elements across channel



Fluid Solution
- ANSYS CFX v13.0
- Solve fluid flow for 1 channel in every row
- Copy fluid flow solutions to create rows of channels



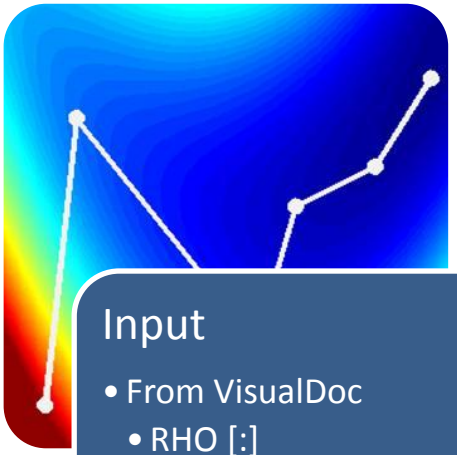
Thermal Solution
- ANSYS CFX v13.0
- Import solid mesh
- Import and duplicate fluid meshes
- Write material library and expressions
- Write solid domain
- Write fluid channel domains
- Write solid/fluid domain interface
- Write solver setup
- Input fluid solutions as initial conditions
- Disable fluid flow solver
- Solve thermal convection/diffusion



Output
- Compute maximum solid temperature
- Compute pressure drop in each channel
- Compute bulk outlet temperature in each channel

Program Flowchart

- VisualDoc optimization software interfaces with ANSYS Multiphysics Workbench via Python scripting language:

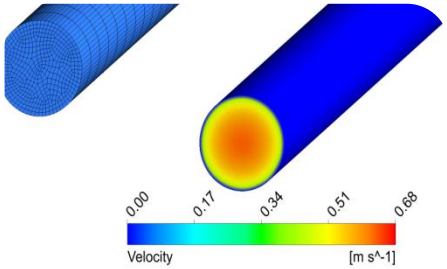


Input

- From VisualDoc
  - RHO [:]
  - RAD [:]
  - MASS_FLOW [:]
- (Using Assumptions)

Mesh Gen (ANSYS APDL)

- Solid
- 1 Channel for every row
  - y+ ≈ 100
  - > 10 elements across channel

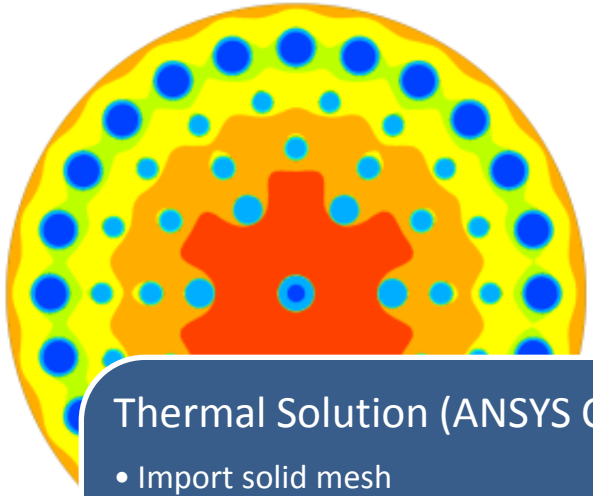


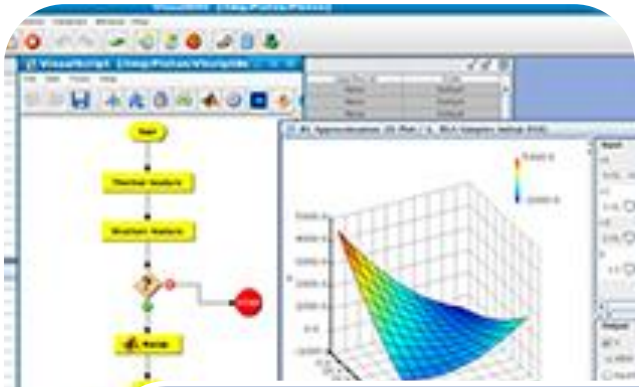
Fluid Solution (ANSYS CFX)

- Solve fluid flow for 1 channel in every row
- Copy fluid flow solutions to create rows of channels

Program Flowchart

- VisualDoc optimization software interfaces with ANSYS Multiphysics Workbench via Python scripting language:





Thermal Solution (ANSYS CFX)

- Import solid mesh
- Import and duplicate fluid meshes
- Write material library and expressions
- Write solid domain
- Write fluid channel domains
- Write solid/fluid domain interface
- Write solver setup
- Input fluid solutions as initial conditions
- Disable fluid flow solver
- Solve thermal convection/diffusion

Output

- Compute maximum solid temperature
- Compute pressure drop in each channel
- Compute bulk outlet temperature in each channel

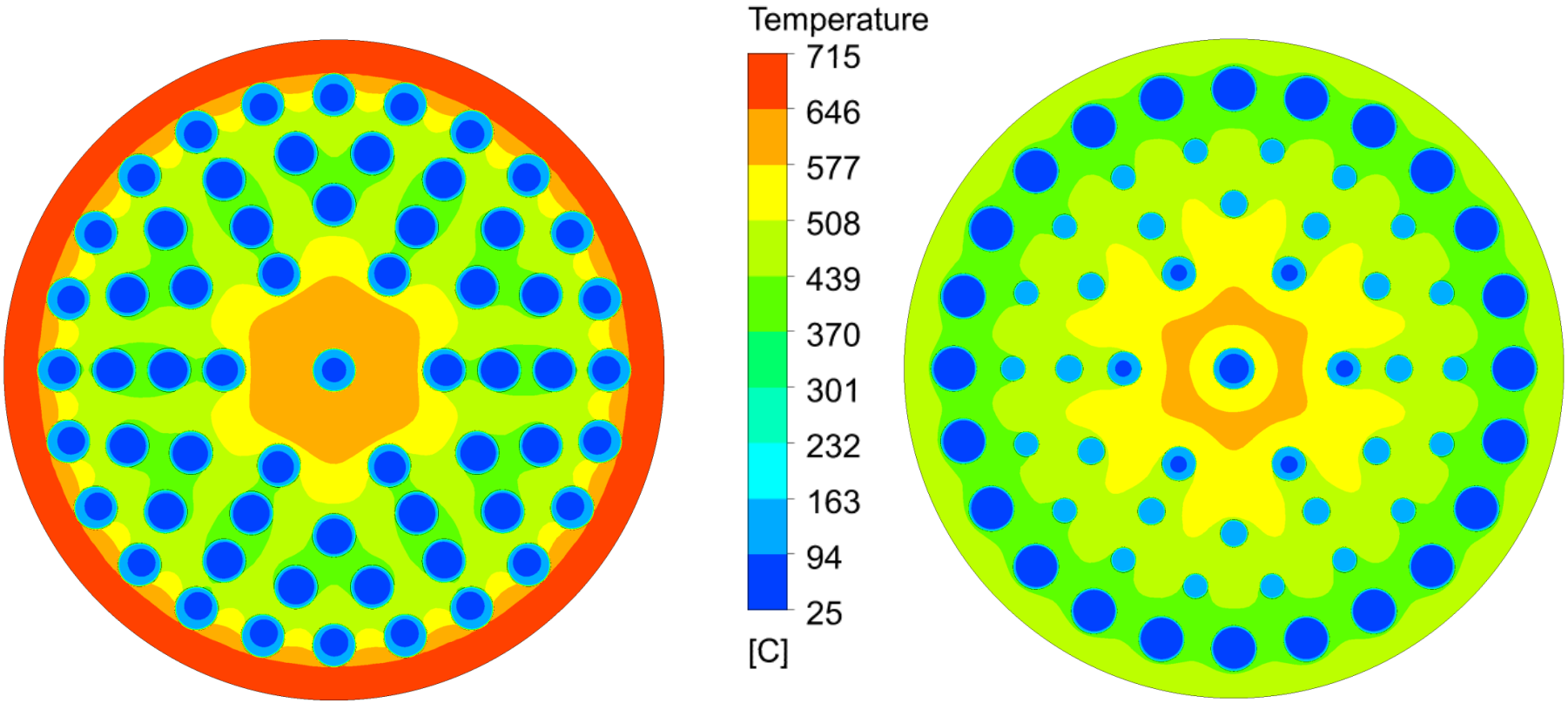
Preliminary Results

Initial Condition

- Maximum temperature occurs in area of maximum nuclear heating.

+ 2 Iterations

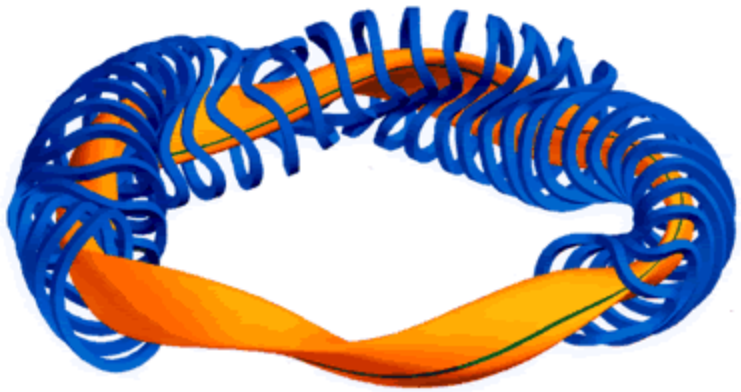
- Maximum temperature has decreased ~ 200 [C]

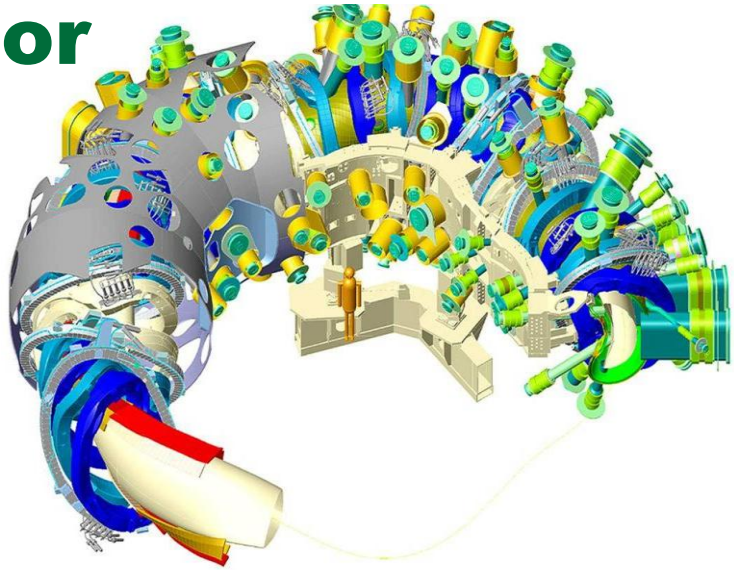


Wendelstein 7-X Stellerator

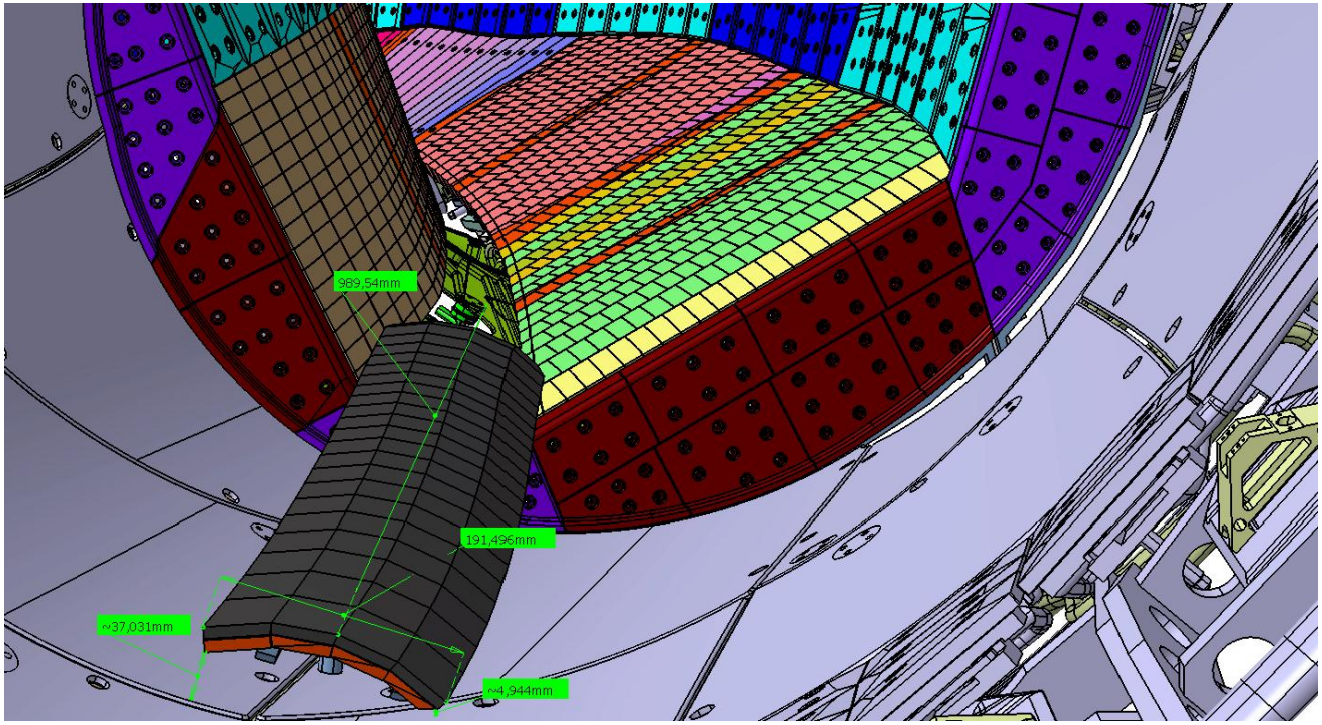
SCRAPER ELEMENT THERMAL ANALYSIS

Wendelstein 7-X Stellerator





A protective element is needed to protect the diverter from plasma during configuration changes.

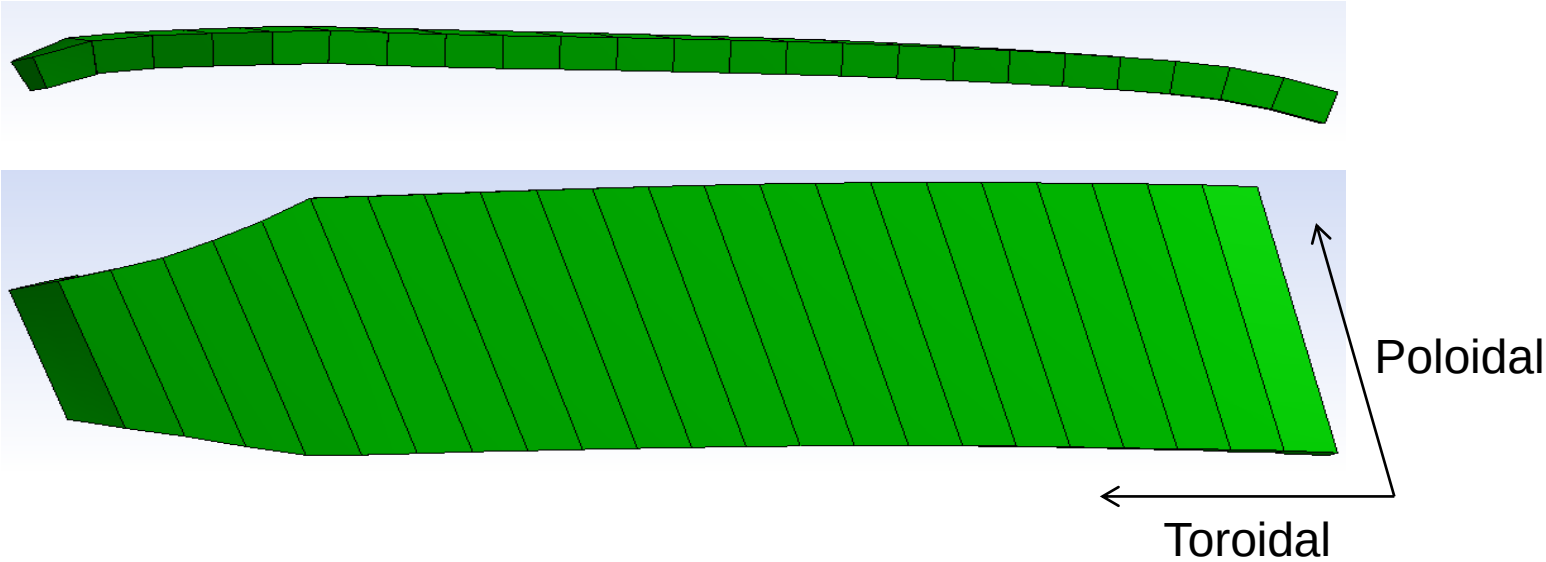


W7X Scraper Element Thermal Analysis

- Design Parameters
- Design Constraints
- Design Variables
- Design Objective
- Design Point Analysis
- Detailed CFD Analysis

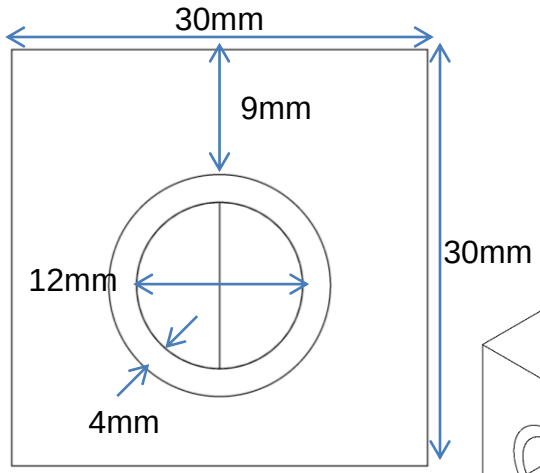
Design Parameters

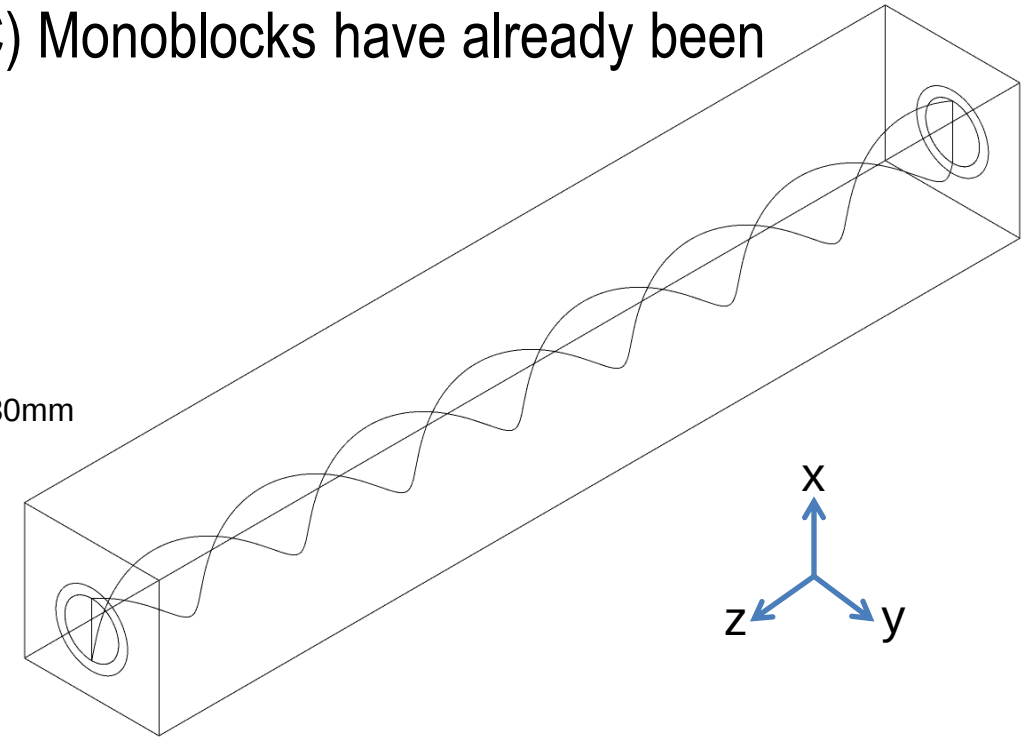
- Scraper Geometry
  - To a first approximation, the geometry will be approximated as a planar rectangle
  - Dimensions: 1 [m] (Toroidal) X 0.250 [m] (Poloidal)



Design Parameters

- Monoblock Components
  - Carbon Fiber Composite (CFC) Monoblocks have already been qualified for ITER
  - Geometry



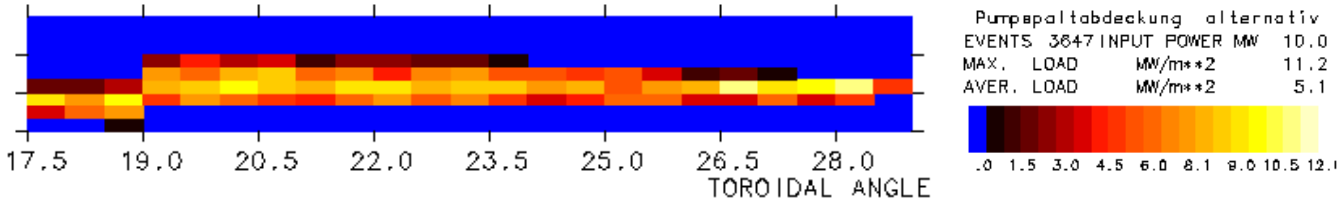


  - Twisted Tape
    - Increases margins against Critical Heat Flux (boiling "dry out")
    - Twist Ratio: $y \equiv \frac{L_{180^o Turn}}{D} = 2$
    - Thickness = 1 [mm]
    - Material = Cu

Design Parameters

- Monoblock Components
  - Materials
    - Coolant = Water
      - T_{in} = 30 [C]
      - P_{in} = 25 [bar]
      - Mass flow rate (D = 30 [mm], u = 8-10 [m/s]) = 7.120 [kg/s]
    - Pipe = CuCrZr (MATWEB Cadi Cadi 1815 Cu Alloy)
      - Density = 8890 [kg/m^3]
      - Specific Heat Capacity = 385 [J/kg-K]
      - Thermal Conductivity = 323.4 [W/m-K]
    - Monoblock = Snecma Sepcarb NB31 CFC
      - Density = 1870 [kg/m^3]
      - Specific Heat Capacity = 800 [J/kg-K]
      - Anisotropic Thermal Conductivity = (100,160,80) [W/m-K]
      - Peacock et al., *Phys. Scr.* T128 (2007) 23–28

Design Parameters

- Plasma Heat Load
  - 500 [kW/m^2] for initial steady-state operation.
  - Transient plasma “sweep” heat loading [Kisslinger, 2009-12-01]
  - 

  - Plasma “sweep” might actually stall creating a local, steady-state heat load.
  - The total integrated power input is approximately 400 [kW].
  - Results are for an older scraper geometry. Update from plasma physics is forthcoming.
  - To a first approximation, the heat load can be approximated as binary. The maximum possible 12 [MW/m^2] local flux is applied over ~13 % of the surface area to equal 400 [kW] total power input.
  -

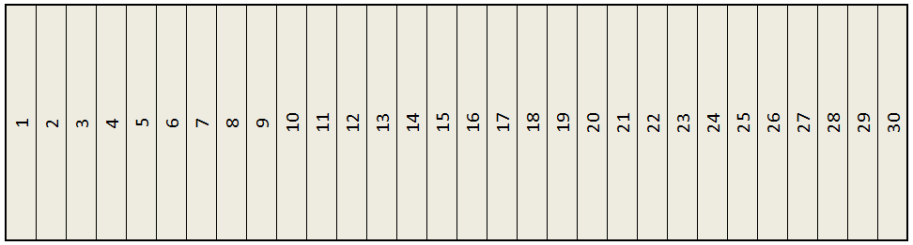
Design Constraints / Objective

- Objective
  - Note that $(T_{CFC})_{limit} \approx 1200$ [C]
  - Minimize the Maximum CFC Temperature …

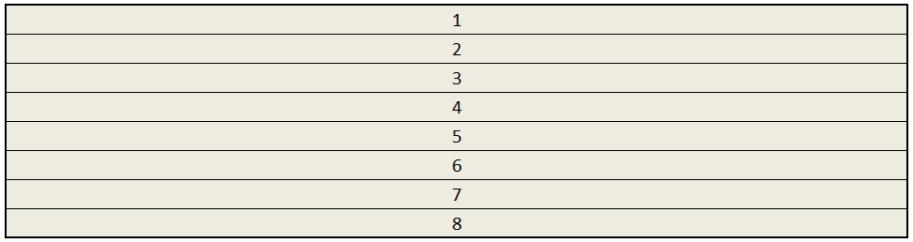
- …Subject to Constraints
  - Maximum Fluid Pressure Drop
    - $\Delta P_{max} = 14$ [bar]
  - Maximum Mean Fluid Temperature Rise
    - $(\Delta T_m)_{max} = 50$ [C]
  - Maximum Local Fluid Temperature
    - $(T_{H2O})_{max} = T_{sat}(P = 25 \text{ [bar]}) = 224$ [C]

Design Variables

- Monoblock CFC Dimensions
  - ± 2-3 mm on each side
- Channel Orientation
  - Poloidal arrangement yields 30 parallel monoblock channels:



  - Toroidal arrangement yields roughly 8 parallel monoblock channels:



- Coolant Distribution via Manifolds
  - Divide supply coolant between multiple sections of scraper element
  - Channels in each section connected by pipe bends (protected from plasma)

Design Point Analysis

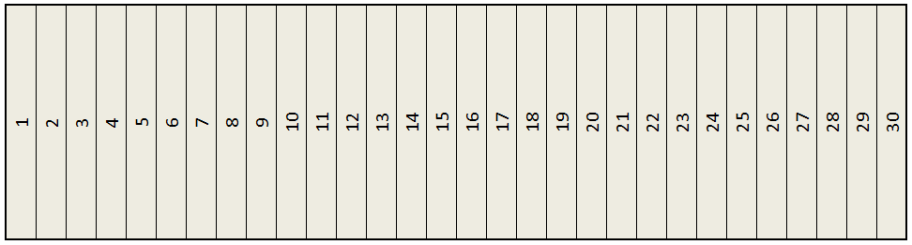
- First, use approximate hand calculations to explore design options,
  - Objective
    - Minimize the Maximum CFC Temperature … → 1-D Radial HT, Empirical correlations
  - …Subject to Constraints
    - Maximum Fluid Pressure Drop → Empirical correlations
      - ΔP_{max} = 14 [bar]
    - Maximum Mean Fluid Temperature Rise → Total energy balance
      - $(\Delta T_m)_{max}$ = 50 [C]
    - Maximum Local Fluid Temperature → Assume non-active
      - $(T_{H2O})_{max} = T_{sat}(P = 25\ [bar]) = 224\ [C]$
- Then proceed to more detailed CFD modeling

Design Point Analysis

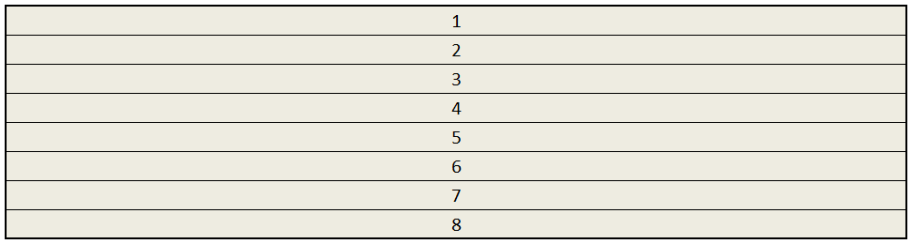
- Mean Fluid Temperature Rise
  - Simple energy balance, independent of design variables
  - $Q = \dot{m}c\left(T_{m,o} - T_{m,i}\right)$
  - $T_{m,o}$ = 64 [C]
  - ΔT = 34 [C]

Design Point Analysis

- Pressure Drop
  - Pressure drop is primarily a function of mass flow rate, twisted tape insert, and number of bend connectors.
    - The length of twisted tape insert is fixed for each alignment option.
      - » $\Delta p_{\boldsymbol{tt}} = f\,\frac{l}{D}\frac{\rho u_m^2}{2}$
      - » $f = \frac{0.0791}{Re^{0.25}}\left[\frac{\pi}{\pi-4(\delta/D)}\right]^{1.75}\left[\frac{\pi+2-2(\delta/D)}{\pi-4(\delta/D)}\right]^{1.25}\left[1+\frac{2.752}{y^{1.29}}\right]$
      - » $f = \pm\mathbf{5}\%,\ 2.5 \le y \le 10$
      - » Manglik and Bergles, JHT (1992)



$l = 0.250\ [m]$



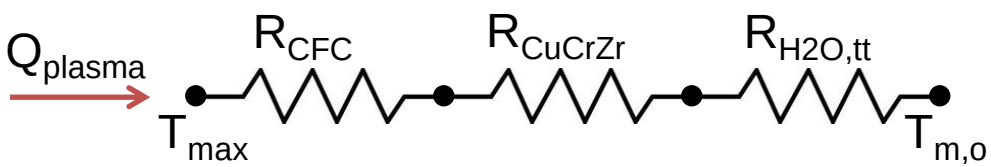
$l = 1.0\ [m]$

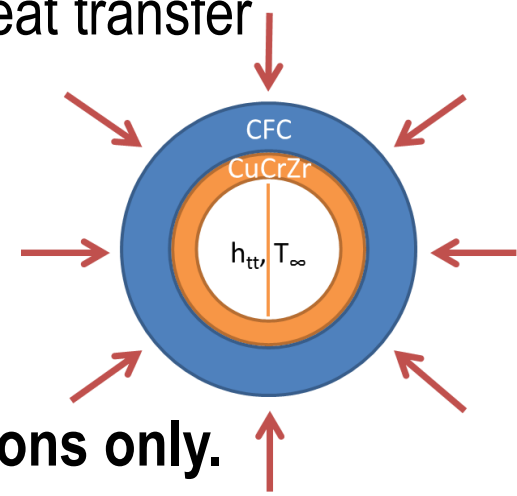
Design Point Analysis

- Pressure Drop
  - The mass flow rate and number of bend connectors will change with manifold design (number of parallel flow circuits through the scraper element).
    - » $\Delta p_{\boldsymbol{bend}} = \frac{1}{2}\rho K_B u_m^2$
    - » $f_t = (0.790\ln Re_D - 1.64)^{-2};\ 3000 \leq Re_D \leq 5 \times 10^6$
    - » For 180 [deg] bends, $K_B = 0.25\pi f_t \frac{r}{D} + 1.5K_{90}$
    - » $\frac{r}{D} = \frac{33.3/2\ mm}{12\ mm} \Rightarrow K_{90} \approx 15 f_t$
    - » Crane Flow of Fluids, TP-410 (2009)

Design Point Analysis

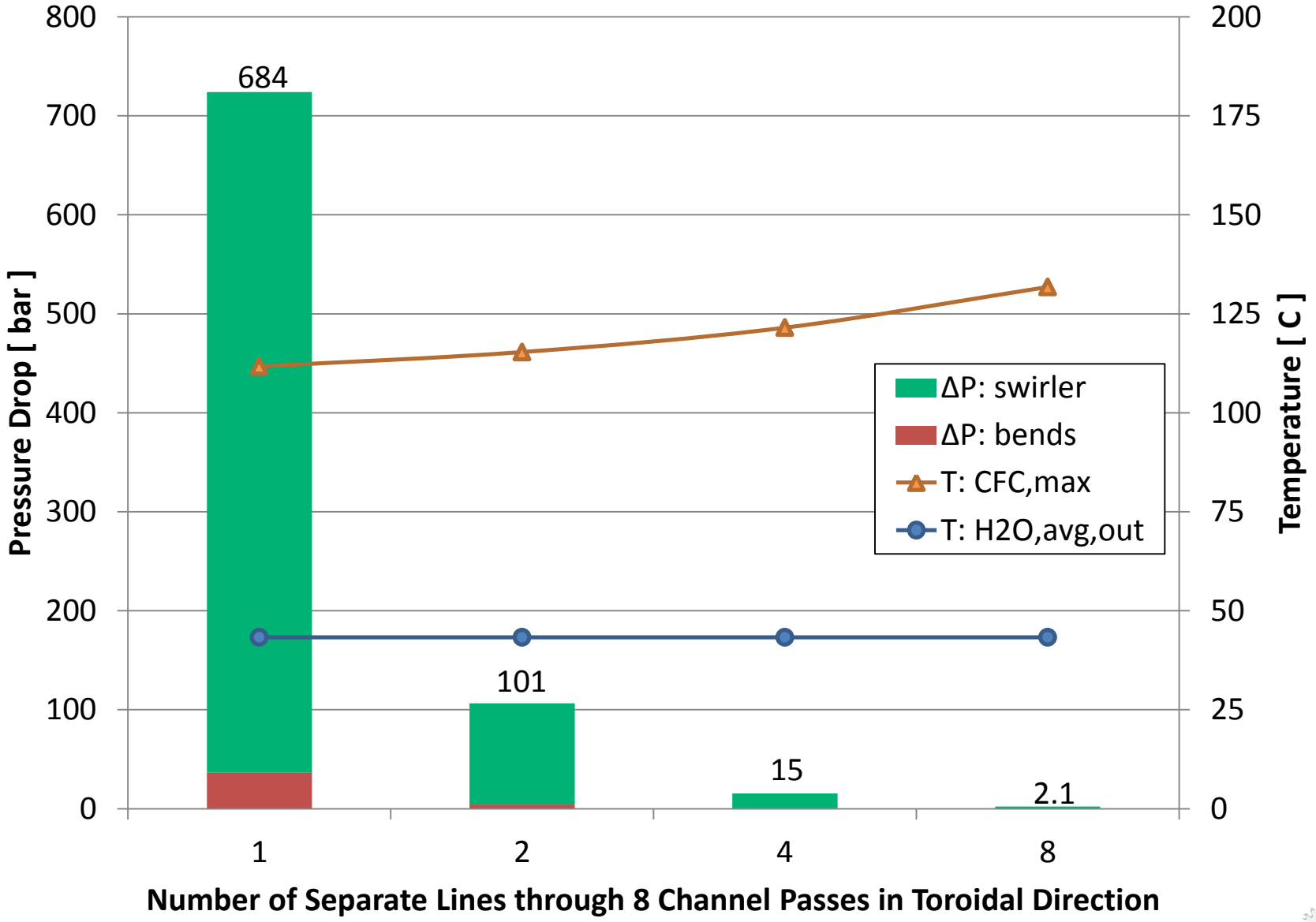
- Maximum CFC Temperature
  - Hand calculations are only possible for a 1D radial heat transfer model with uniform heat flux.



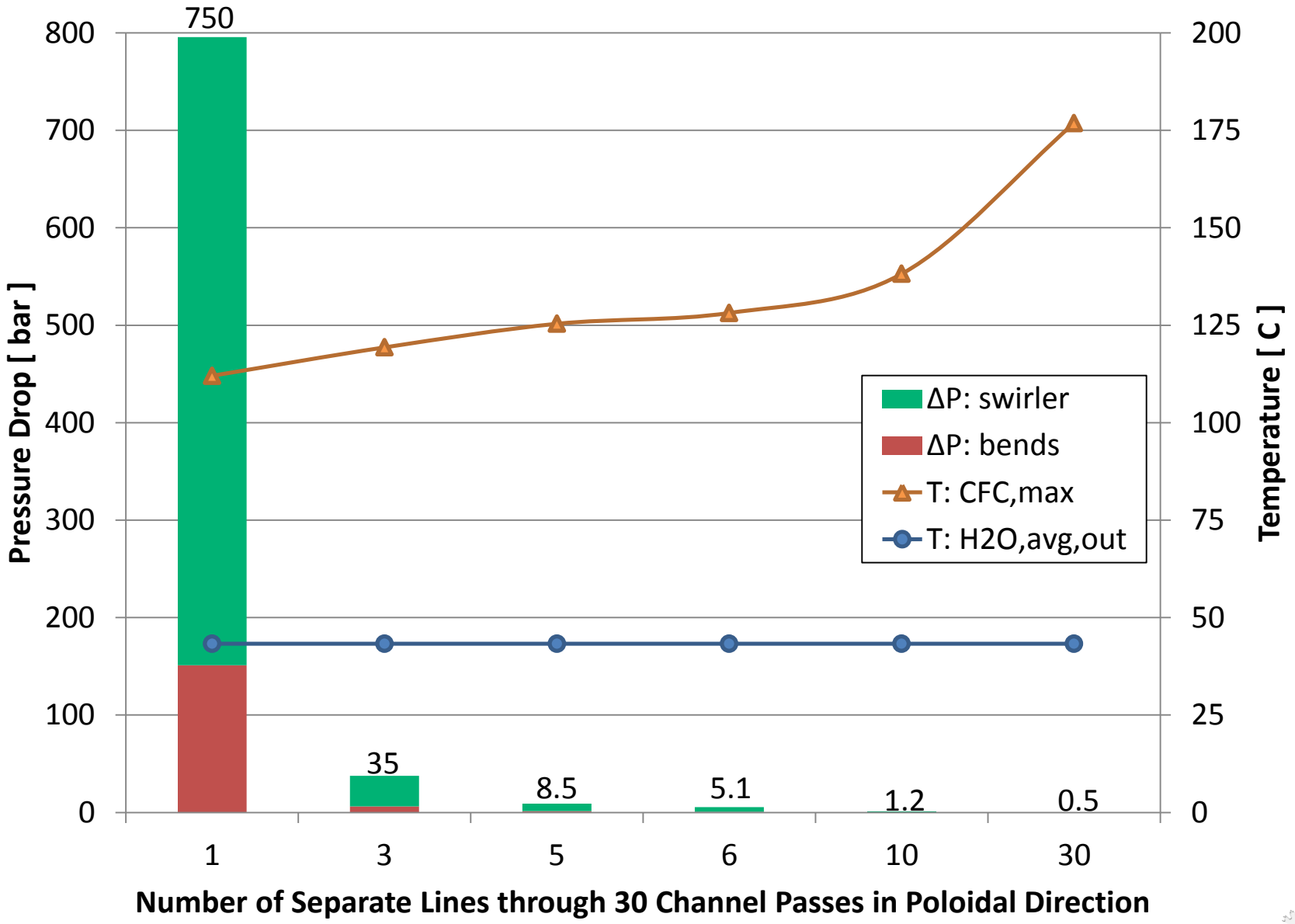


  - **Accordingly, these results are for trend comparisons only.**
  - Semi-empirical correlations are available for straight pipes with twisted tape inserts.
    - » $\frac{Nu_{tt}}{Nu_{y=\infty}} = 1 + \frac{0.769}{y}$
    - » $Nu_{y=\infty} = 0.023 Re^{0.8} Pr^{0.4} \left[\frac{\pi}{\pi - 4(\delta/D)}\right]^{0.8} \left[\frac{\pi + 2 - 2(\delta/D)}{\pi - 4(\delta/D)}\right]^{0.2} \left(\frac{\mu_{bulk}}{\mu_{wall}}\right)^{0.18}$
    - » $U_{Nu} = \pm 10\%,\ 2.5 \leq y \leq 10$
    - » Manglik and Bergles, JHT (1992)

Design Point Analysis

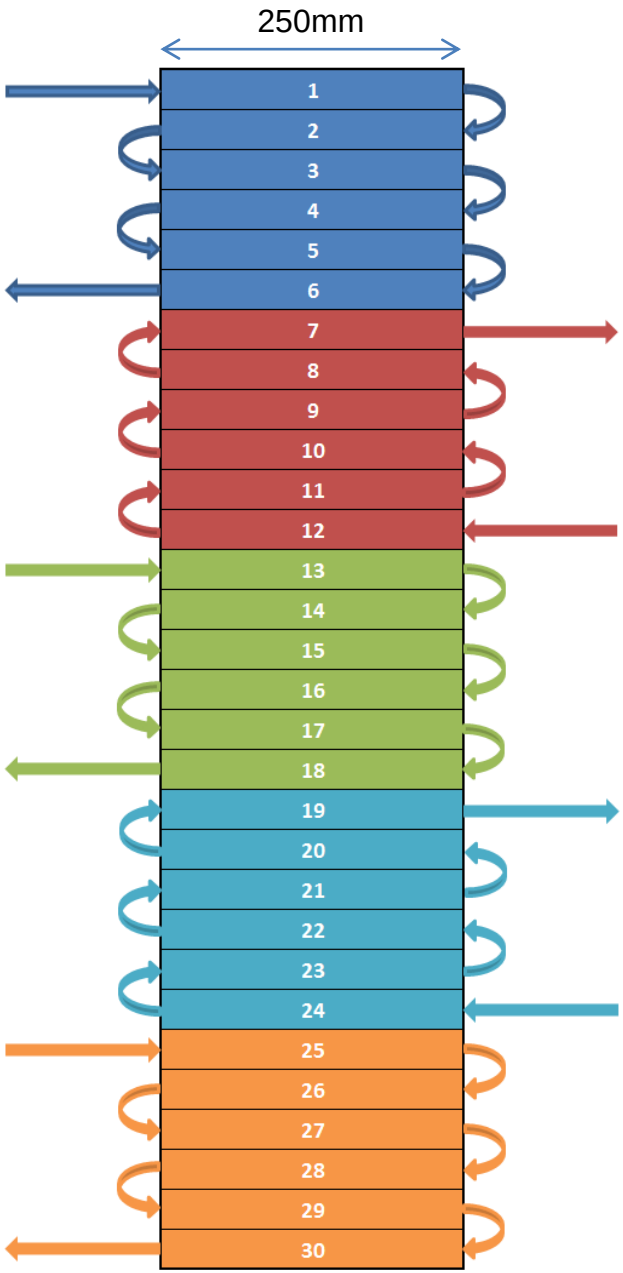


Design Point Analysis



Design Point Analysis

- Best Case
  - Five parallel flow circuits, each consisting of 6 channel passes in the poloidal direction, connected by 180 [deg] bends.
  - Given the total heat load (400 [kW]) and total mass flow rate (7.120[kg/s]), a simple energy balance gives the mean outlet water temperature as 64 [C].
  - The pressure drop among parallel flow circuits is constant (analogous to voltage drop along parallel circuits). Semi-empirical correlations indicate a pressure drop of 8.5 [bar] ± 10%.

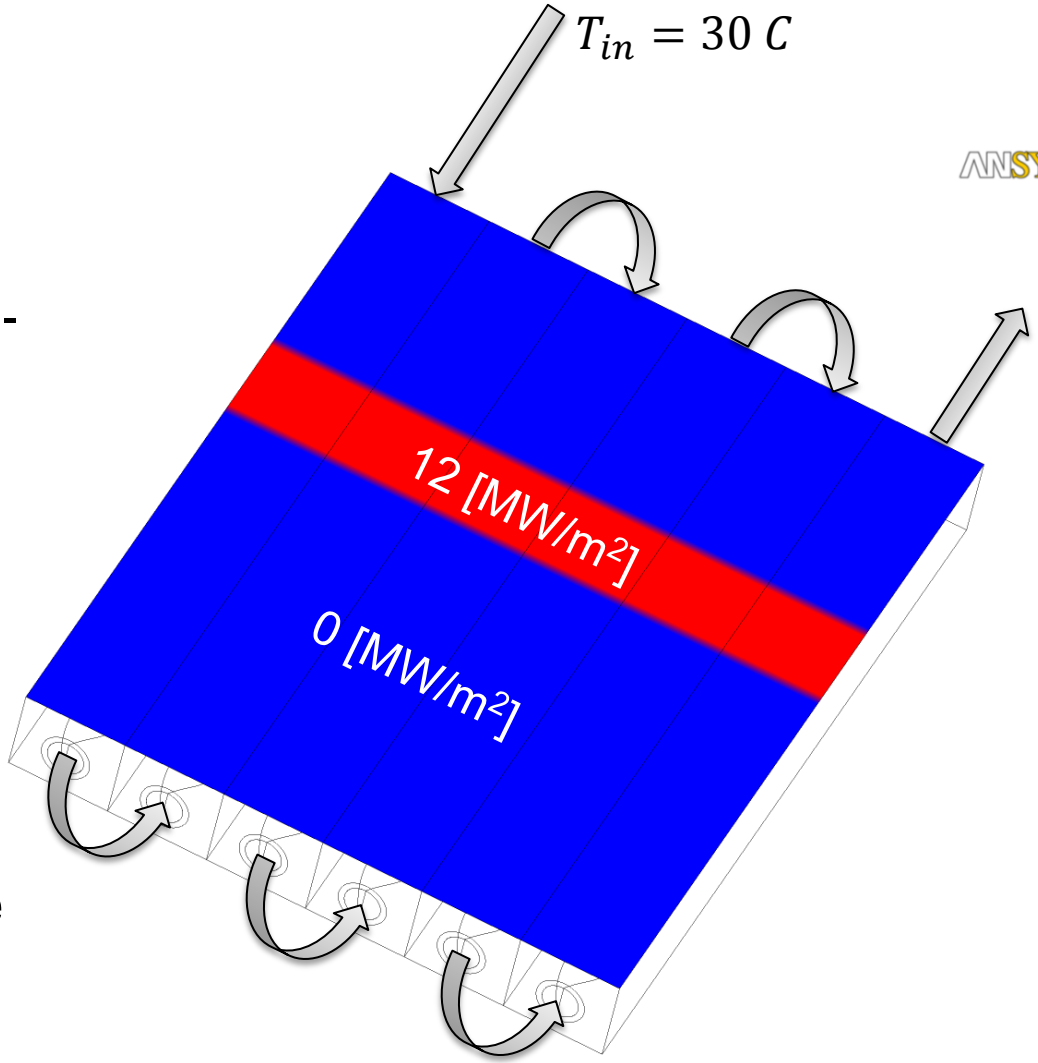


CFD Analysis

- Model Assumptions
  - Steady-state
  - Constant material properties
  - K-ε turbulence model with semi-empirical near-wall effects
  - Pipe bends replaced with grid connections
  - No heat transfer in pipe bends
  - No radiation heat transfer
  - Pipe is entirely CuCrZr
  - Adiabatic CFC walls
  - 1D heat transfer in twisted tape (thickness dimension)
  - Binary plasma heat flux (0/12 [MW/m^2]), resulting in 400 [kW] power input

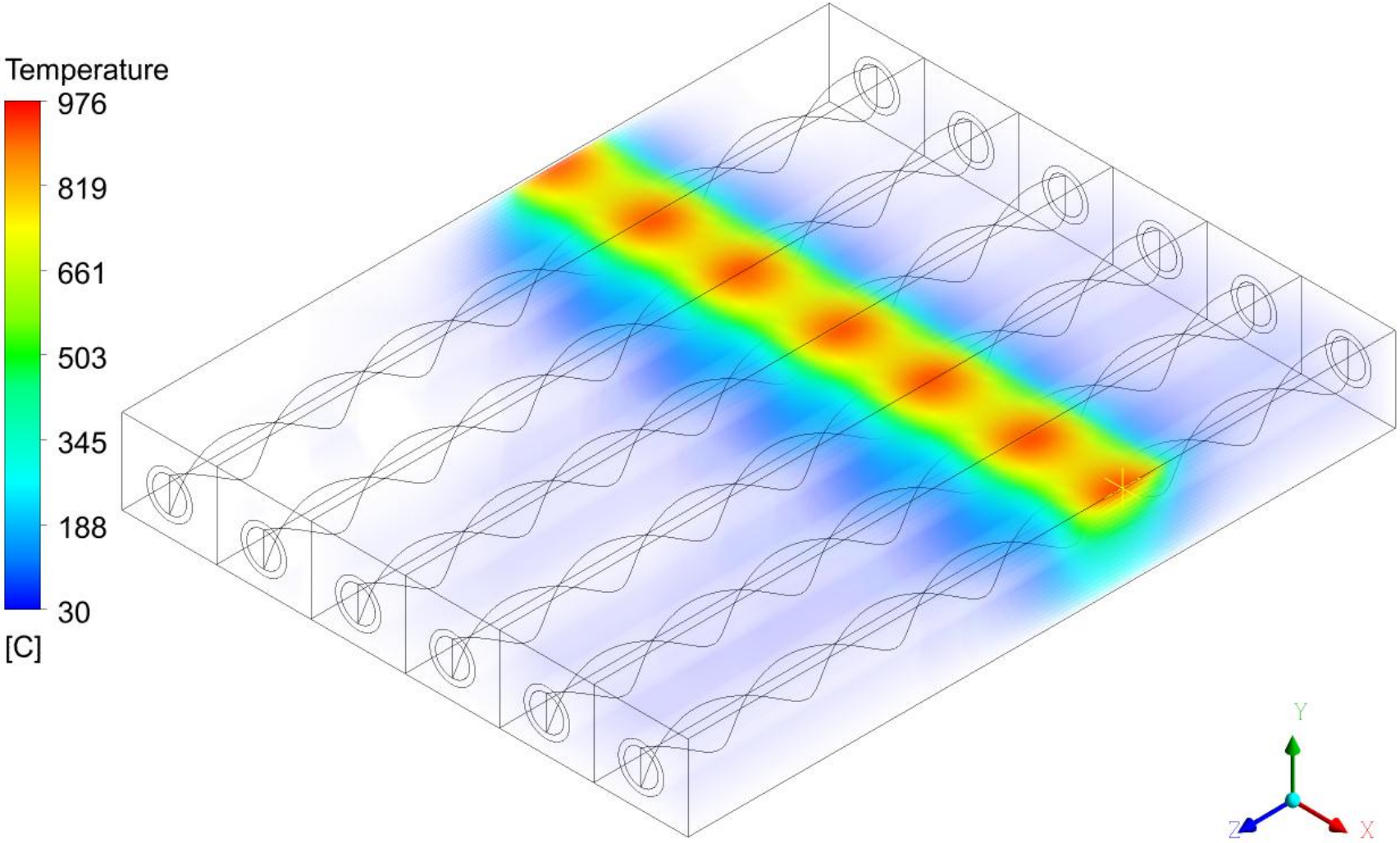
$$\dot{m}_{in} = \frac{\dot{m}_{tot}}{5} = 1.437\ \frac{kg}{s}$$

$$T_{in} = 30\ C$$



CFD Analysis

- ANSYS CFX v13.0



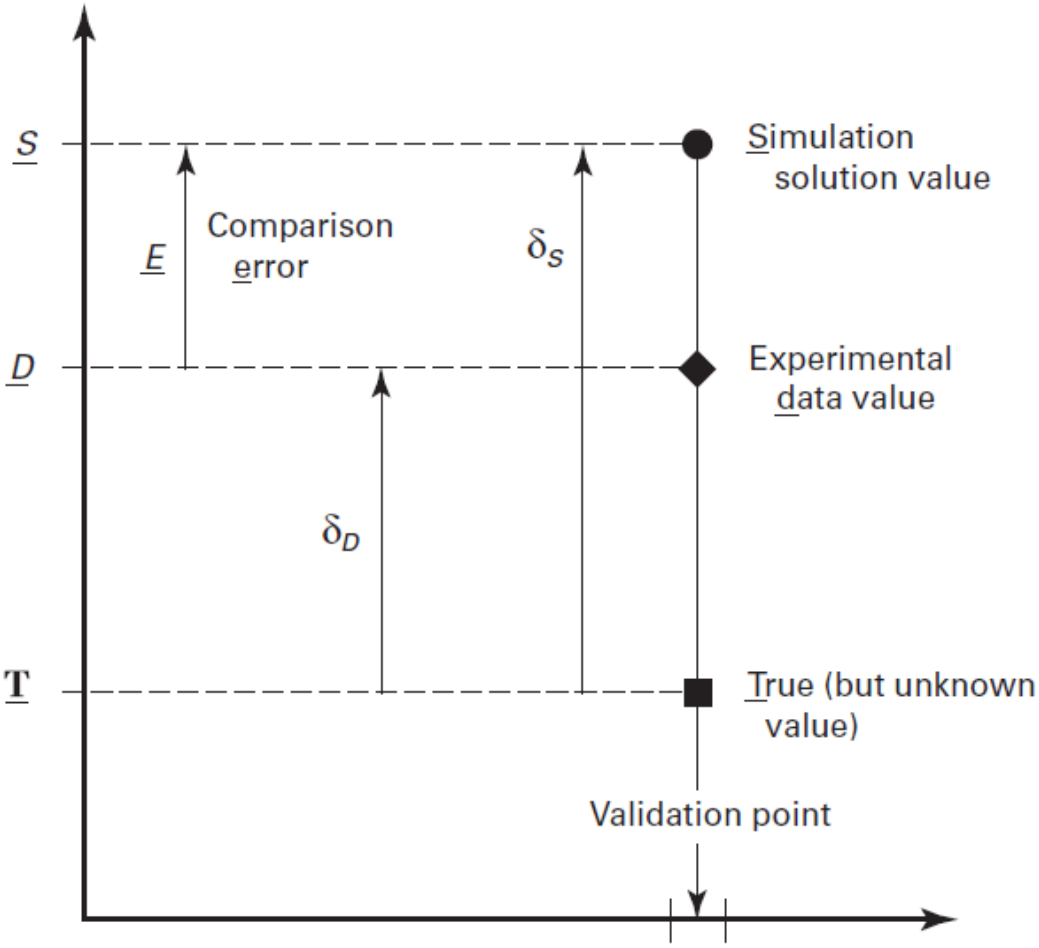
Summary

- Preliminary analysis indicates favorable results

Variable	Limit	Type	Result	Source
Pressure Drop	14 [bar]	Constraint	8.5 [bar] ± 10%	Semi-Empirical
Mean Fluid Temp Rise	50 [C]	Constraint	34 [C]	Energy Balance
Local Fluid Temp	224 [C]	Constraint	N/A	Assumed Non-active
Max CFC Temp	≈1200 [C]	Objective	976 [C] ± 10%	CFD (Grid Convergence)

ASME Validation & Verification

- How well can we model twisted tape heat transfer?



$$E = \delta_S - \delta_D$$

$$E = \delta_{model} + \delta_{num} + \delta_{input} - \delta_D$$

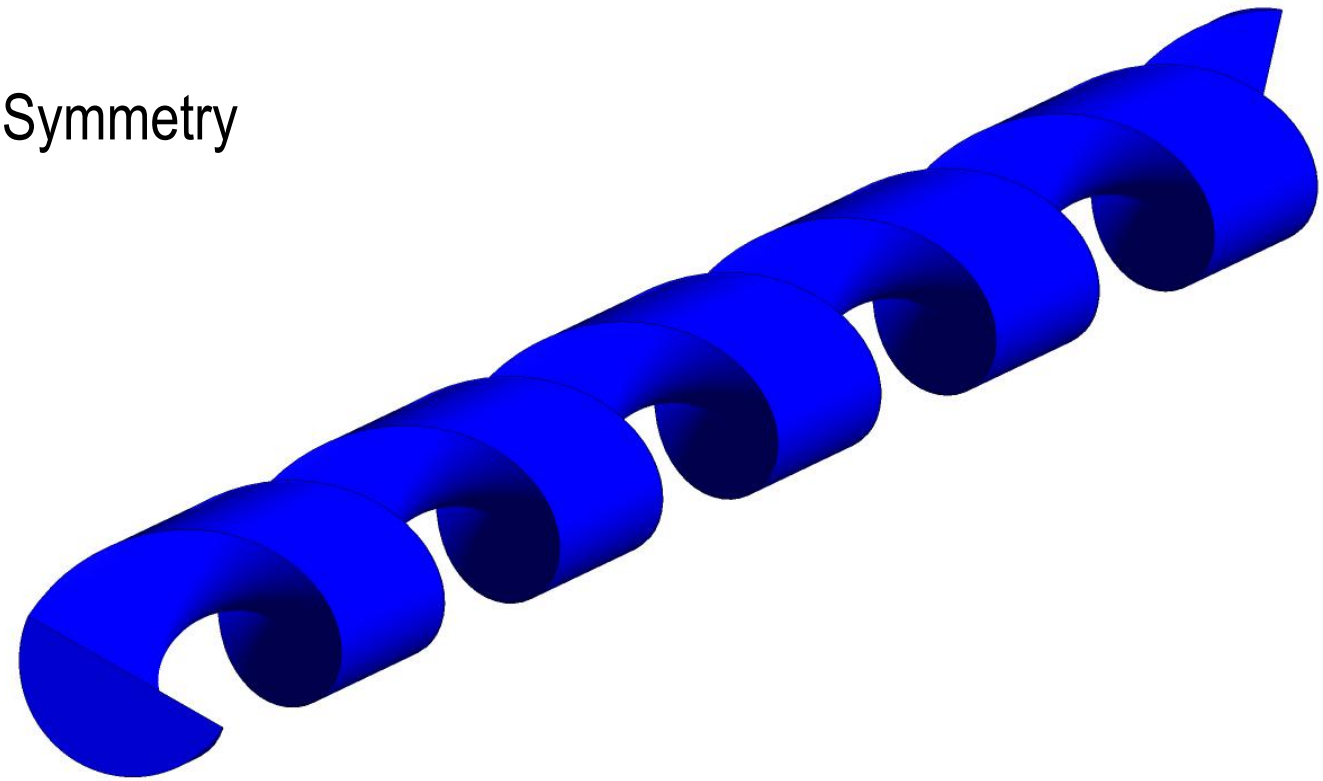
$$\delta_{model} = E - \left(\delta_{num} + \delta_{input} - \delta_D\right)$$

$$\delta_{model} \in [E - u_{val}, E + u_{val}]$$

$$u_{val} = \sqrt{{u_{num}}^2 + {u_{input}}^2 + {u_d}^2}$$

ASME Validation & Verification

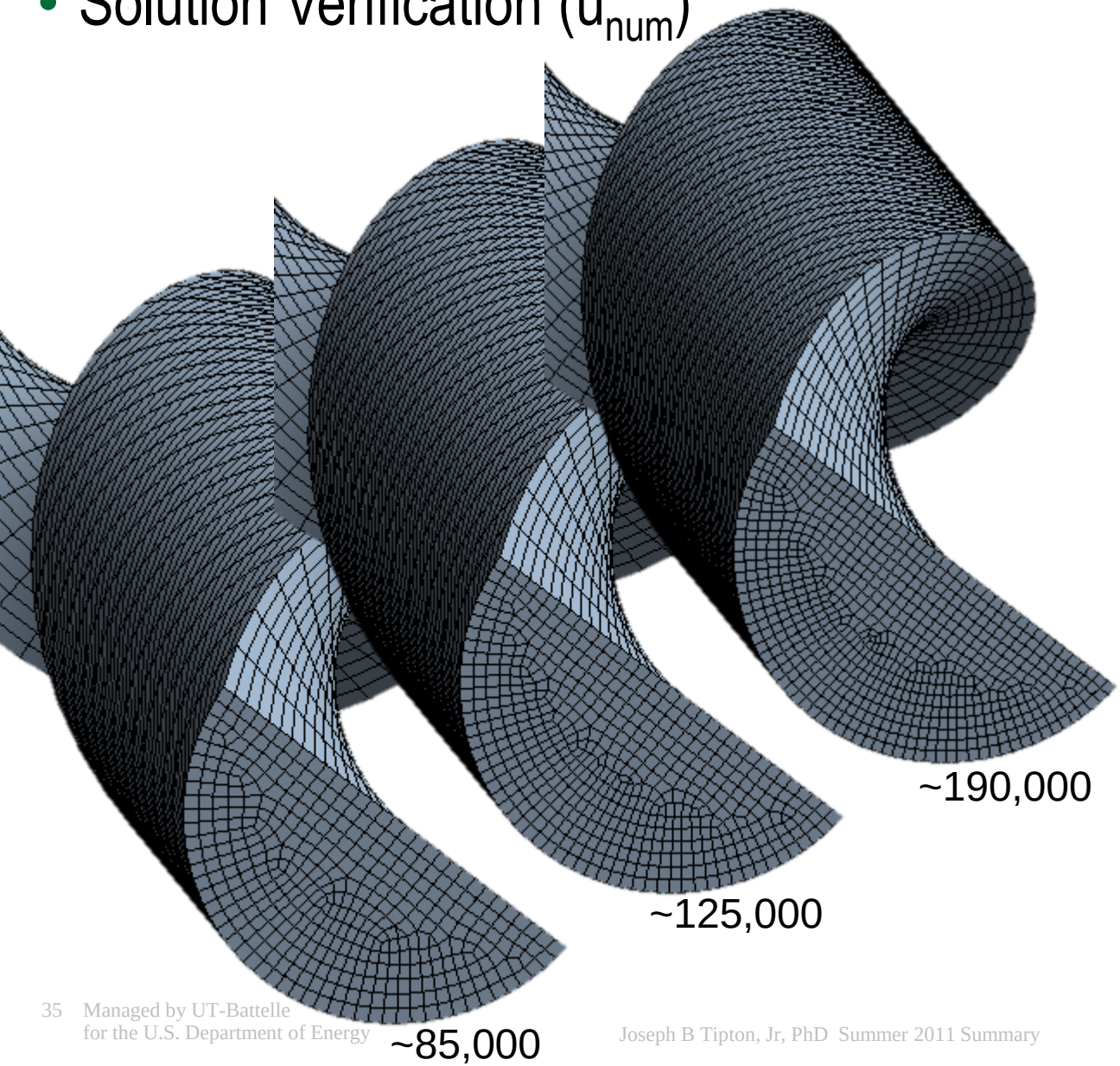
- Test Model
  - ½ Fluid Channel Symmetry
  - $T_{in} = 25\ [C]$
  - $\dot{m} = 0.12\ \left[\frac{kg}{s}\right]$
  - $q" = 3\ \left[\frac{MW}{m^2}\right]$
  - $l = 250\ [mm]$
  - $D = 12\ [mm]$



- Experimental Result Uncertainty (u_D)
  - Manglik and Bergles, JHT (1992)
  - Survey of experimental results with semi-empirical curve-fit
  - Observed variation in HTC ~ ±10% and ΔP ~ ±5%

ASME Validation & Verification

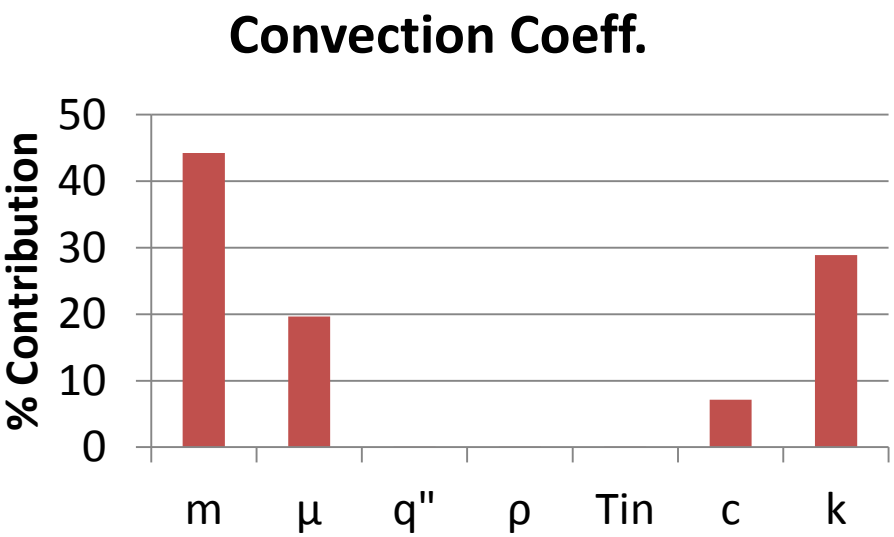
- Solution Verification (u_{num})

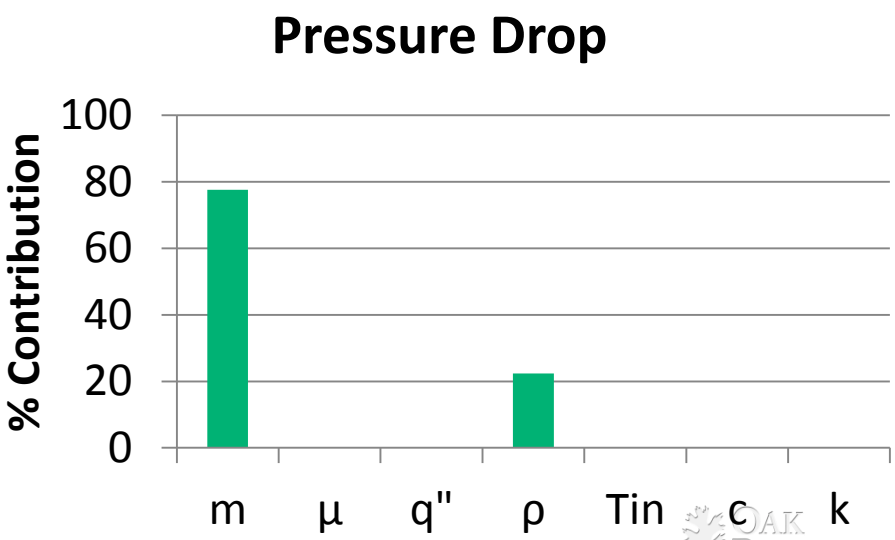


ϕ	Convection Coeff.	Pressure Drop
	[W/m²-K]	[Pa]
r_{21}	1.52	1.52
r_{32}	1.47	1.47
ϕ_1	20994	6973
ϕ_2	20663	7025
ϕ_3	20495	7097
p	1.34	1.03
ϕ_{ext}^{21}	21433	6876
GCI_{fine}^{21}	2.61%	1.74%

ASME Validation & Verification

- Input Parameter Uncertainty (u_{input})
  - Assume:
    - Solution = $f\left(T_{in}, \dot{m}, q'', \rho, \mu, k, c\right)$
    - Uncertainties of each variable are ±5%
  - Calculate sensitivities of each variable using first order, forward finite difference (ΔX_i=1%)
  - $u_{input}{}^2 = \sum_{i=1}^{n} \left(\frac{\partial S}{\partial X_i} u_{X_i}\right)^2$





ASME Validation & Verification

Convection Heat Transfer Coeff.

- Result
  - S = 20,994 [W/m^2-K]
  - D = 23,063 [W/m^2-K]
  - u_{num} = ± 2.6%
  - u_{input} = ± 6.2%
  - u_D = ± 10%
  - **u_{val} = ± 12%**
- Interpretation
  - $\delta_{model} \in [E \pm u_{val}]$
  - $\delta_{model} \in [2{,}069 \pm 2{,}519]$ **W/m^2-K**

Pressure Drop

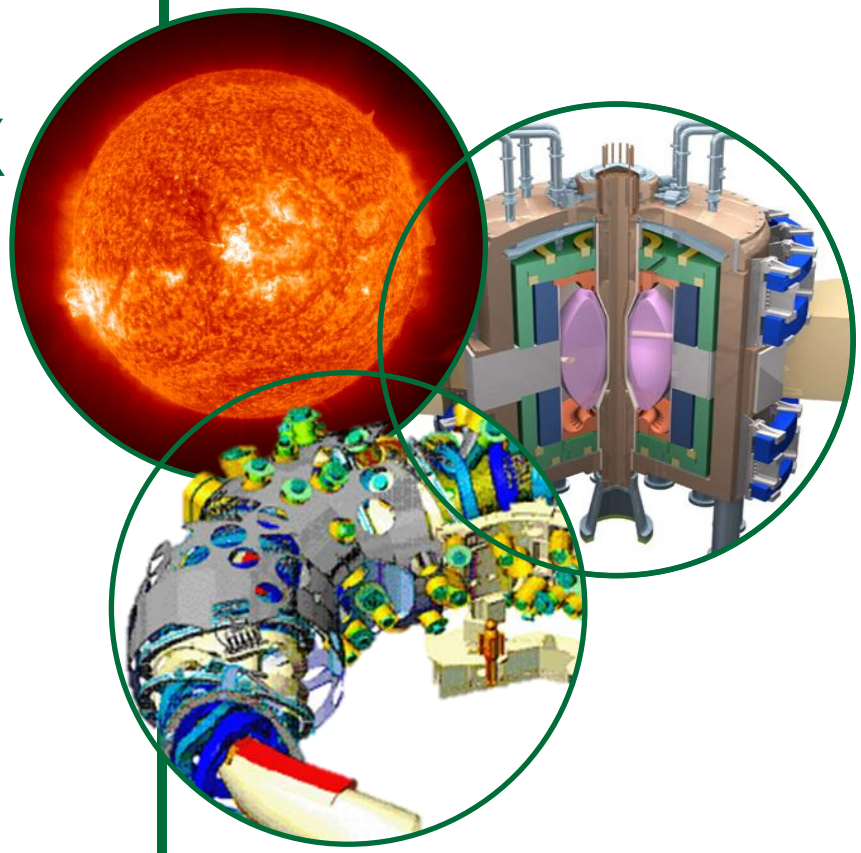
- Result
  - S = 6,973 [Pa]
  - D = 5,443 [Pa]
  - u_{num} = ± 1.7%
  - u_{input} = ± 10.5%
  - u_D = ± 5%
  - **u_{val} = ± 12%**
- Interpretation
  - $\delta_{model} \in [E \pm u_{val}]$
  - $\delta_{model} \in [1{,}530 \pm 837]$ **Pa**

CFD Modeling of High Heat Flux Component Cooling – FNST & W7X

Joseph B Tipton, Jr, PhD

Fusion Energy Division

Summer 2011



U.S. DEPARTMENT OF ENERGY

OAK RIDGE NATIONAL LABORATORY

MANAGED BY UT-BATTELLE FOR THE DEPARTMENT OF ENERGY