# CFD and Thermal Stress Analysis of Helium-Cooled Divertor Concepts

Presented by: X.R. Wang
Contributors: R. Raffray and S. Malang
University of California, San Diego

ARIES-TNS Meeting
Georgia Institute of Technology
Atlanta, GA
Dec. 12-13, 2007

# Outline

- Tools used for CFD and thermal stress analysis
- Exercise for reproducing CFD and thermal stress of ARIES-CS T-tube divertor
  - ✓CFD fluid/thermal analysis
  - ✓Coupled ANSYS thermal stress analysis
- Initial results of helium-cooled plate-type divertor
  - ✓Helium-cooled plate-type divertor design
  - ✓CFD fluid/thermal analysis
  - ✓Thermal stress analysis
- Future work

# ANSYS Workbench Integrated CAD, CFX, and ANSYS Multi-Physics Together

UC San Diego

- CFX is a computer-based CFD software for simulating the behavior of system involving fluid flow, heat transfer, and other related physical process.

- CFX delivers powerful CFD technology for all levels of complexity, and it is capable of modeling:

  Steady-state and transient flows; laminar and turbulent flows; compressible and incompressible fluids; subsonic; transonic and supersonic flows; heat transfer and thermal radiation; non-Newtonian flow; buoyancy flow; multi-phase flows; combustion; particle tracking, ect.

- A number of turbulent flow models are used in CFX to predict the effects of turbulence in fluid flow, and Standard k-ε model is one of the most popular, robust, accurate turbulent flow model accepted and used by industry.

- ANSYS Workbench provides an integration environment across:
  - ✓CAD and geometry creation,
  - ✓Simulation (CFD, and ANSYS Multi-physics),
  - ✓Optimizing tool.

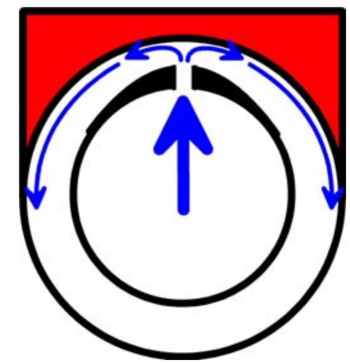
- Coupling CFX and ANSYS to Workbench eliminate data transfer errors because of using a shared geometry model within Workbench.

# Exercise of CFD and Thermal Stress Analysis for ARIES-CS T-tube Divertor

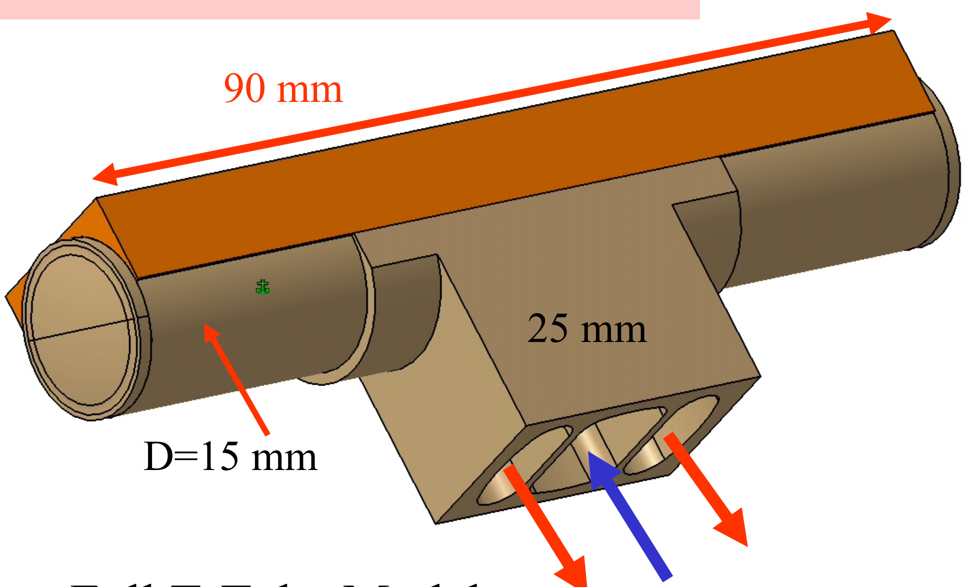
UC San Diego

- FLUENT was used to perform fluid/thermal analysis for ARIES-CS T-tube divertor(by Georgia Tech.).
- Thermal stresses were performed with Workbench (by Thomas Ihli).
- CFX and ANSYS are used to reproduce the fluid/thermal and thermal stresses for T-Tube divertor to understand the simulation process.

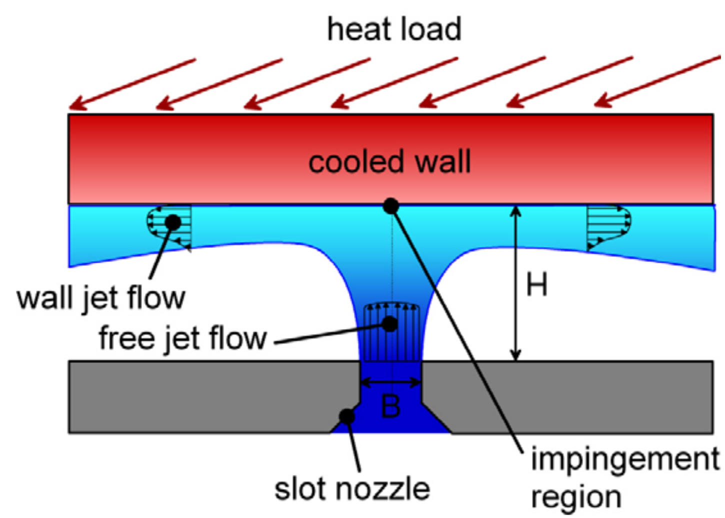
Jet cooling concept



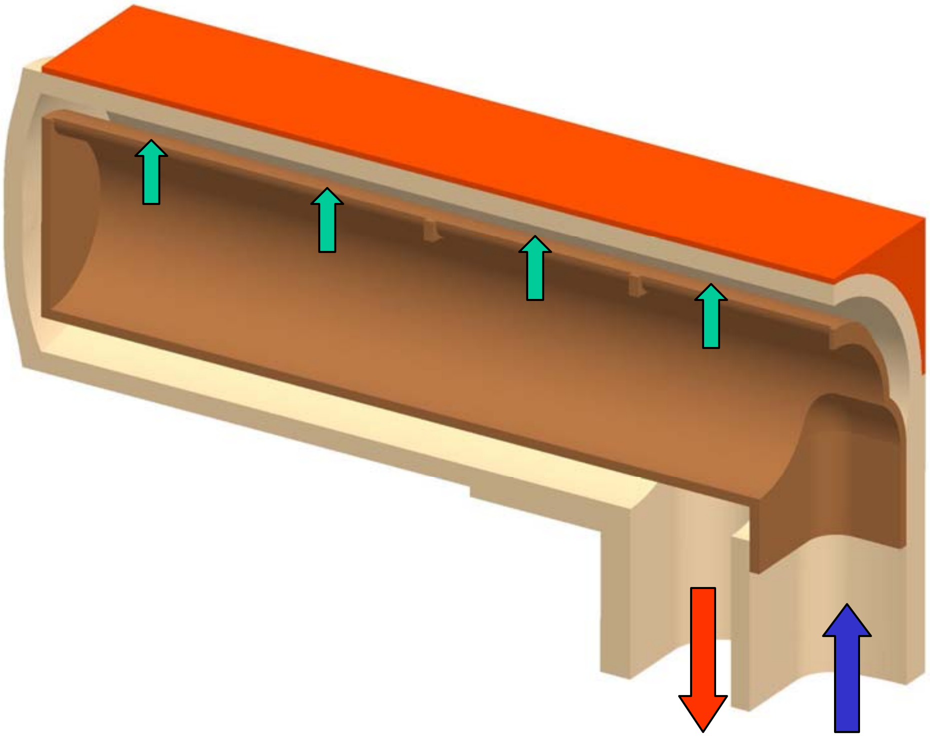
## ARIES-CS T-tube Divertor

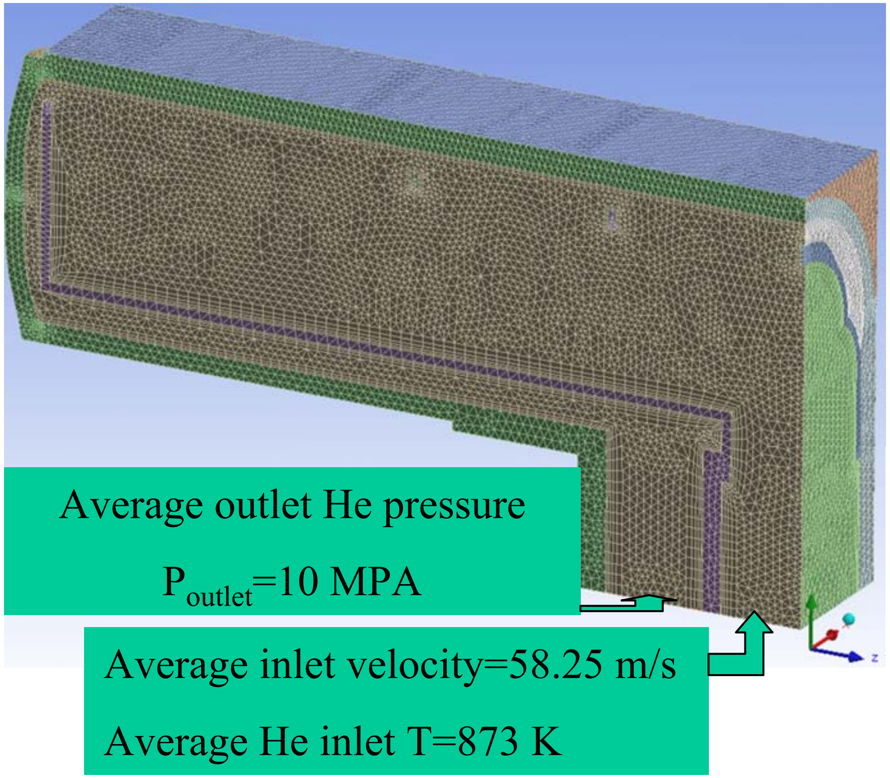



Full T-Tube Model



# CFX Element Model for Jet Cooling Simulation



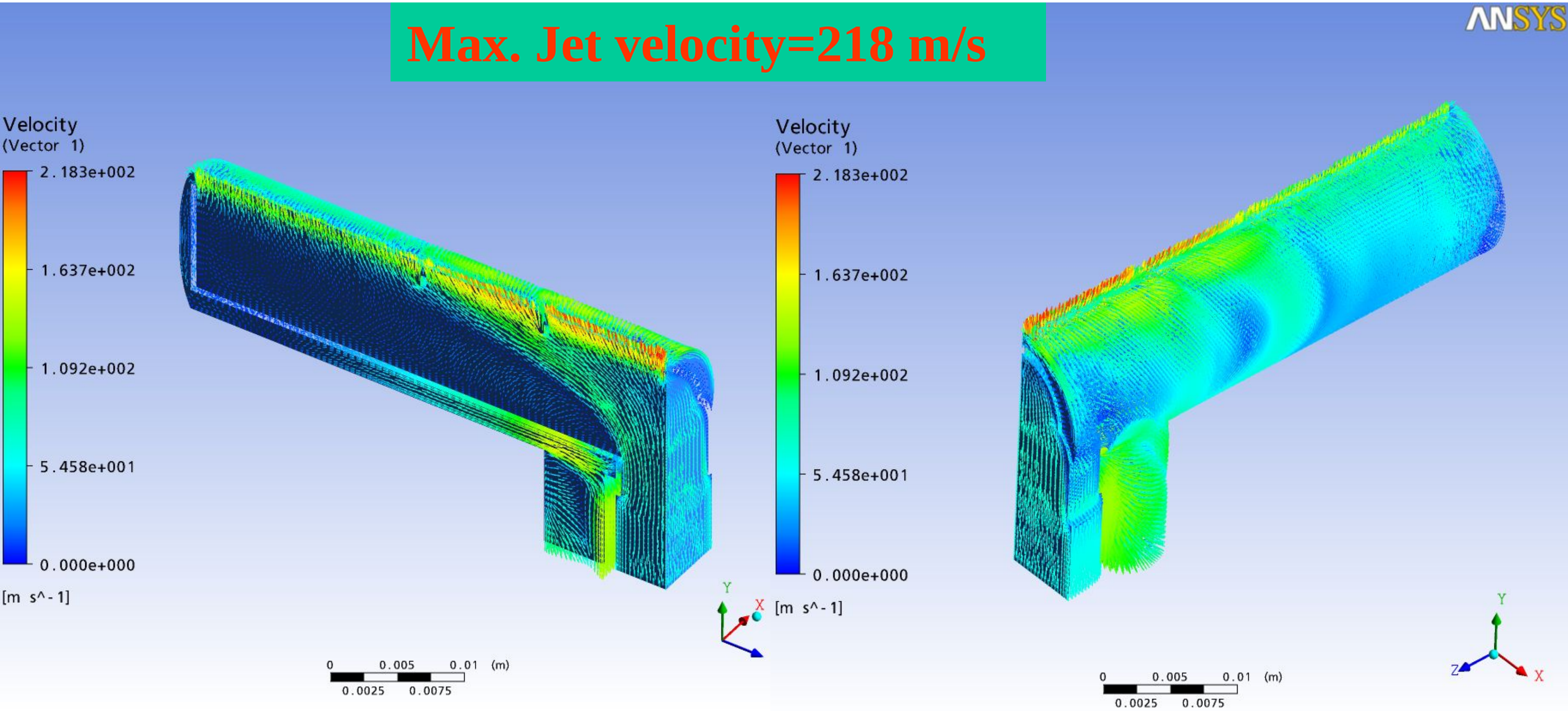



- Importing geometry from Pro/E
- 830,000 tetrahedral elements
- Turbulent flow (Re=$8.4 \times 10^4$ at inlet)
- Standard k-ε turbulent model
- Rough wall (Roughness height=20 micron)

| | |
|---|---|
| Helium pressure | 10 MPa |
| Nozzle width | 0.5 mm |
| Jet wall space | 1.2 mm |
| Heat flux at top wall | 10 MW/$m^2$ |
| Volumetric heat generation | 53 MW/$m^3$ |
| Coolant inlet/outlet temperature | 873/950 K |

# Example CFX Thermal Results Shown Helium Velocity Distribution

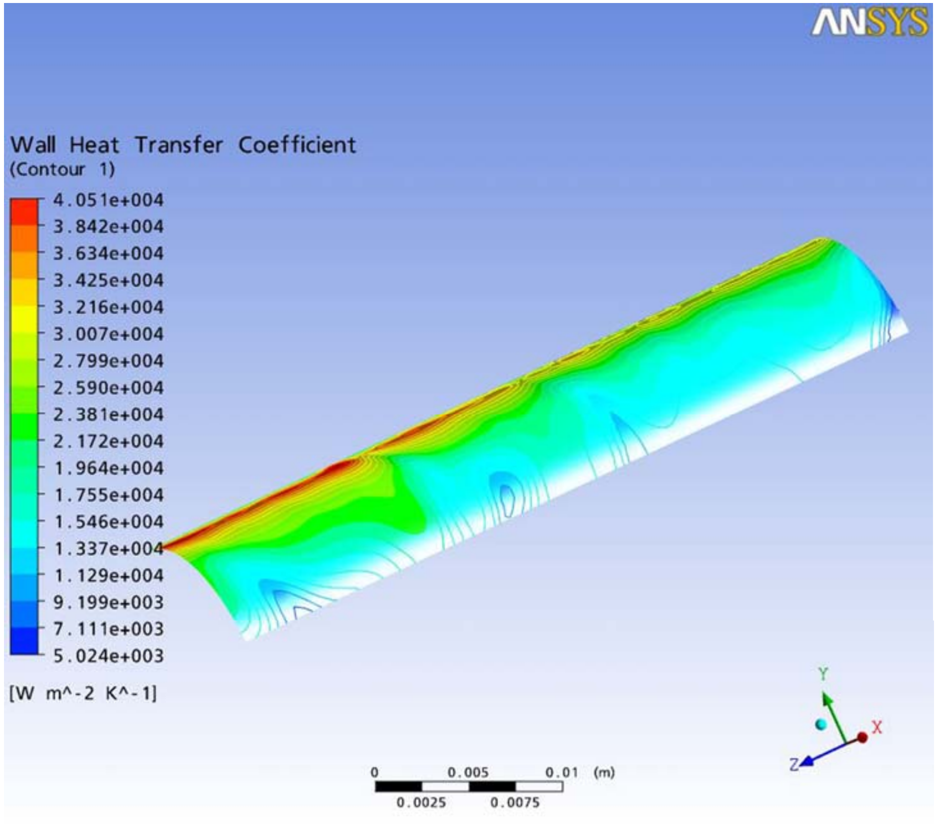
UC San Diego



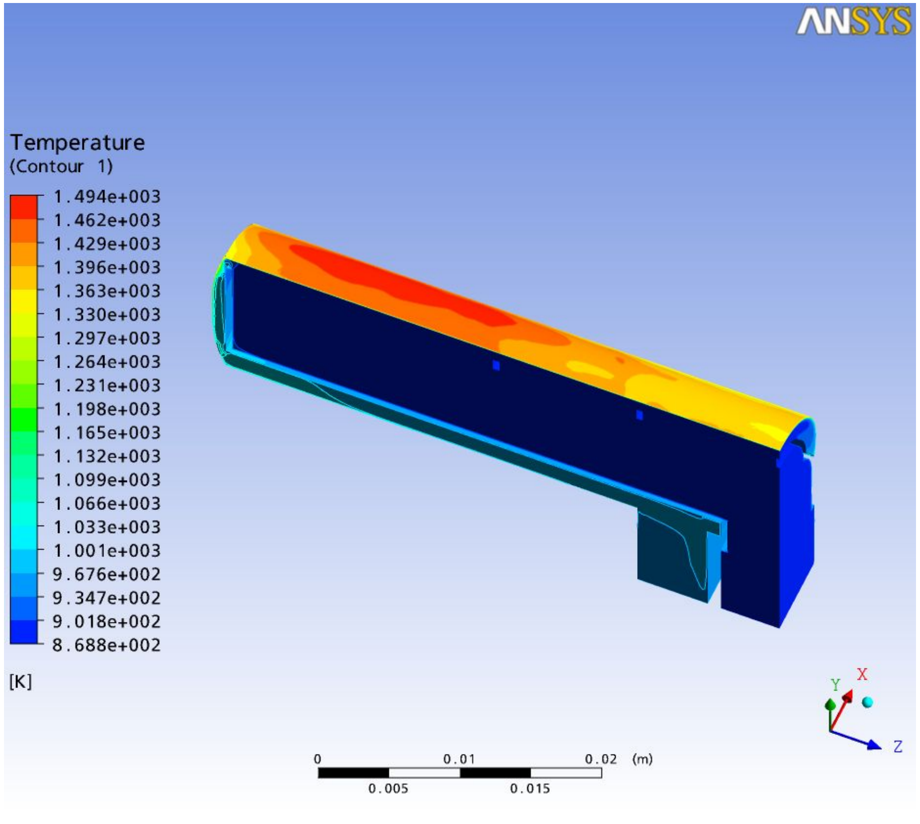

Flow Distribution

Flow Distribution

# Example CFX Thermal Results Shown Wall HTC and He Temperature



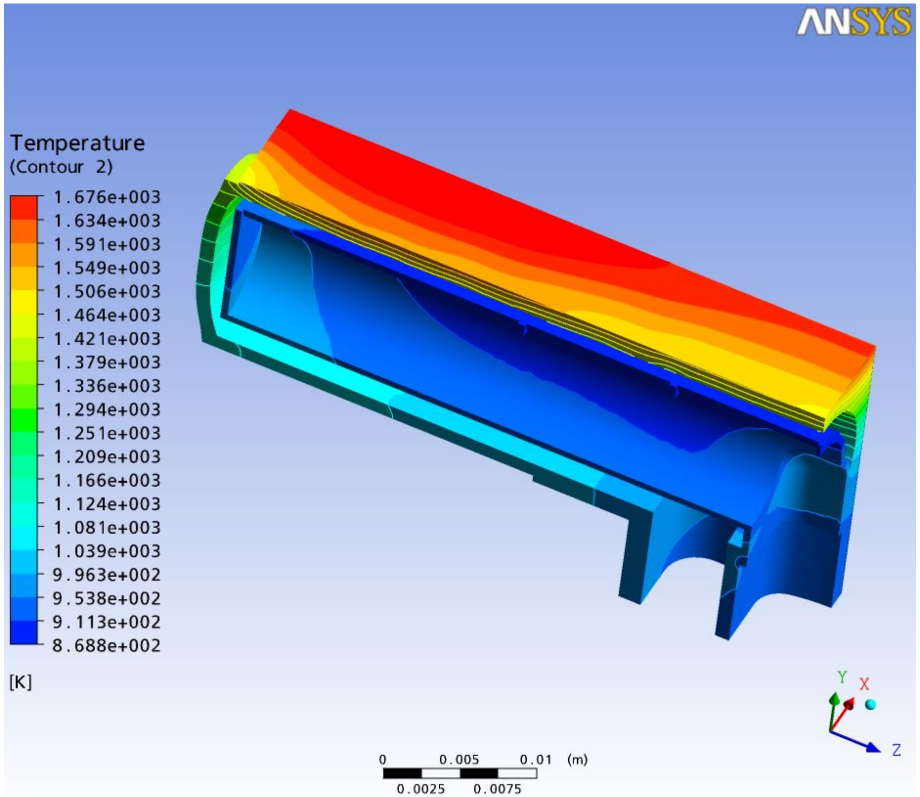

Wall Heat Transfer Coefficient

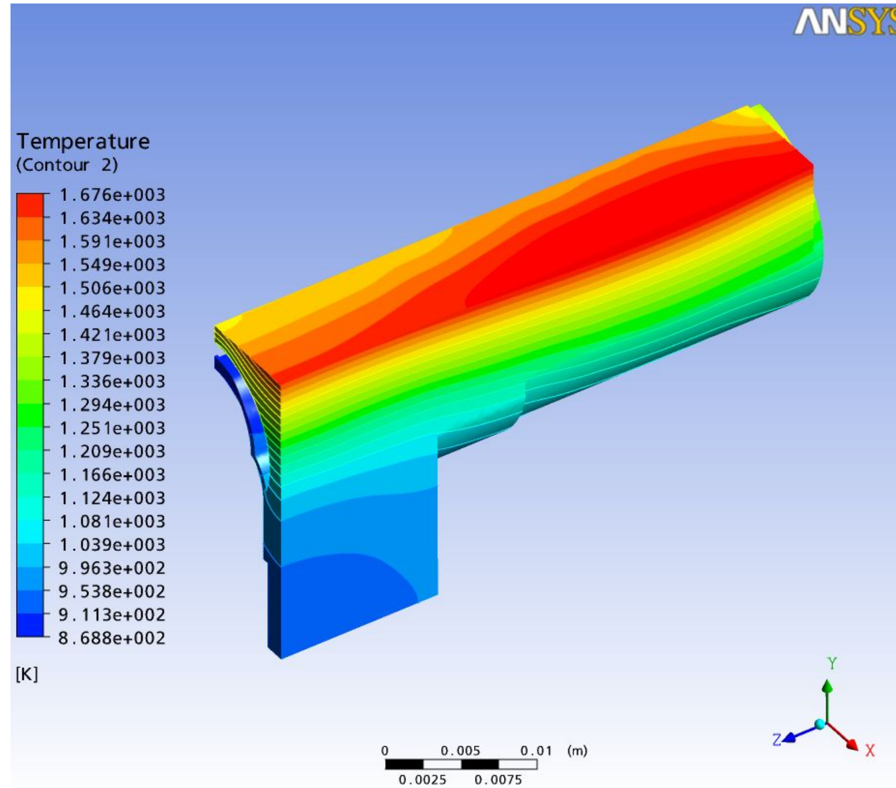



He Temperature Distribution

# Example CFX Thermal Results Shown Temperature Distribution in Structure

UC San Diego




Temperature Distribution in Solid Structure

# CFX Thermal Results Shown Good Consistence with ARIES-CS Results

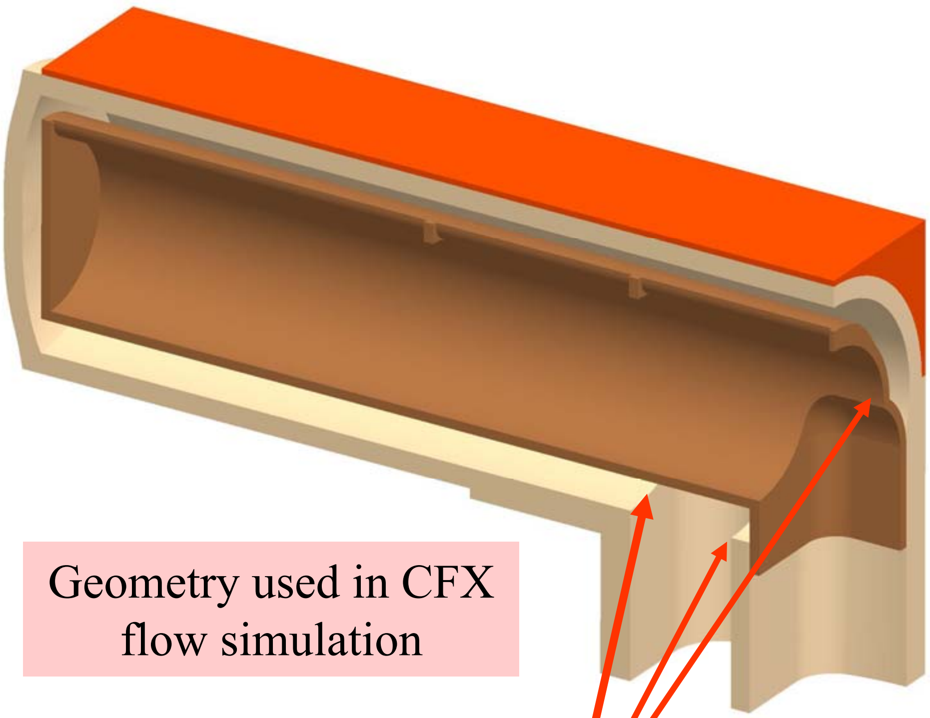
UC San Diego

| | Wall Roughness 0 micron | Wall Roughness 10 micron | Wall Roughness 20 micron | ARIES-CS Results (FLUENT) |
|---|---|---|---|---|
| **Max T at Tile [K]** | 1738 | 1705 | 1676 | 1699 |
| **Max T [K] at Tube/Tile Interface** | 1546 | 1512 | 1494 | 1523 |
| **Max Jet Velocity [m/s]** | 214.4 | 216.3 | 218.4 | 216 |
| **Wall Heat Transfer Coefficient [W/m²K]** | $2.78 \times 10^4$ | $3.76 \times 10^4$ | $4.05 \times 10^4$ | $3.74 \times 10^4$ |
| **ΔP [Pa]** | $1.4 \times 10^5$ | $1.5 \times 10^5$ | $1.6 \times 10^5$ | $1.1 \times 10^5$ |

The column highlighted by yellow color present the results more close to ARIES-CS results.

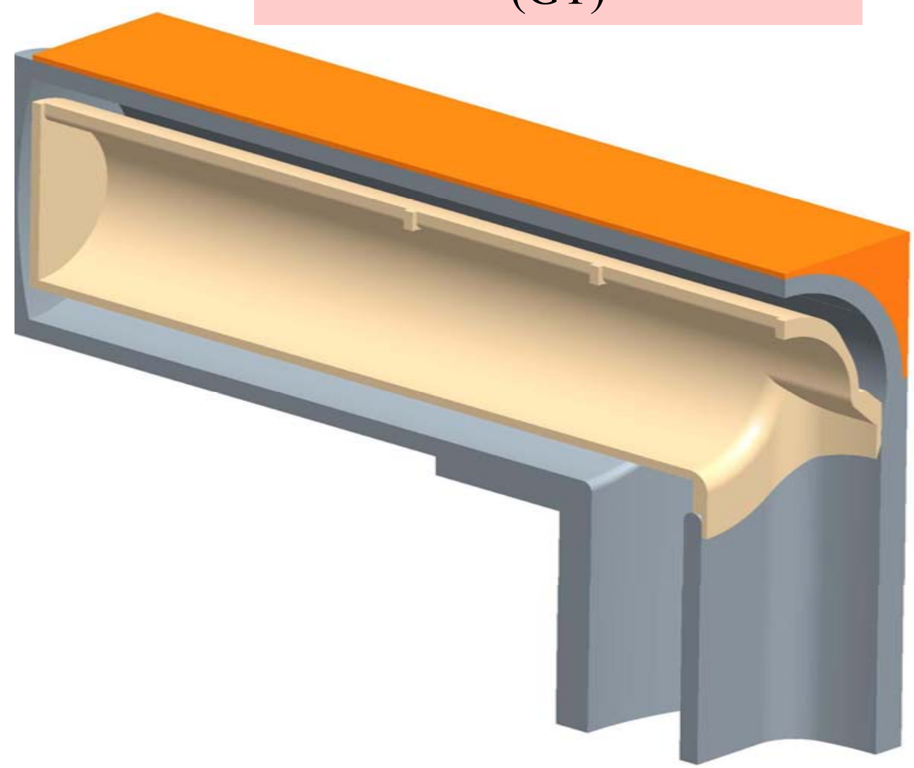
# Details of T-Tube Geometry May Affect Accurate of CFX Pressure Result

Geometry used in FLUENT flow simulation (GT)

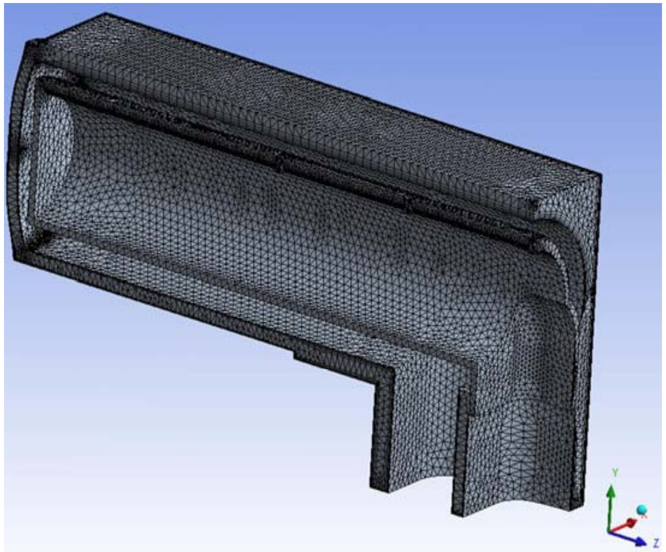


Geometry used in CFX flow simulation

Edges are not rounded

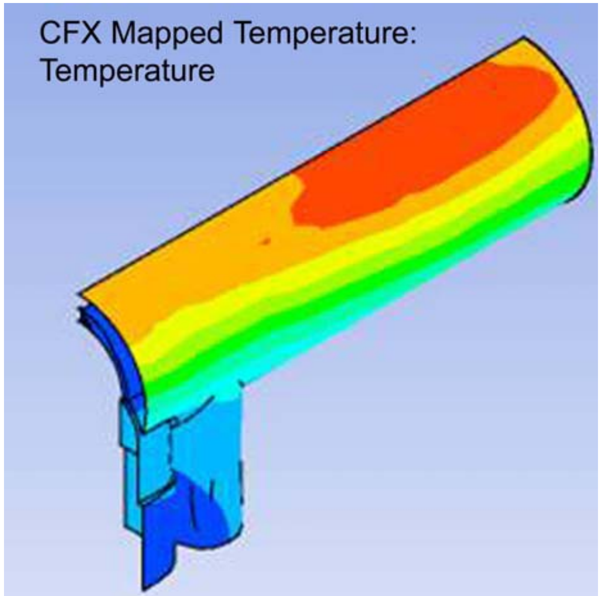


# CFX Results Mapped to ANSYS FEA Model for Repeating Thermal Analysis

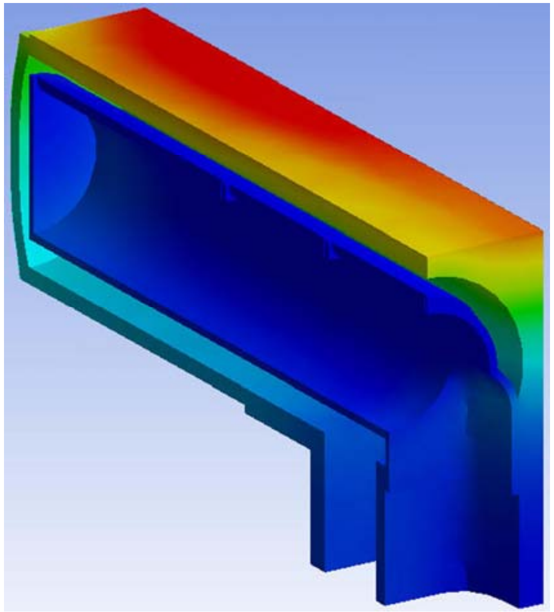



ANSY FEA Model

(Fluid suppressed)




Mapped CFX thermal results (wall temperature or wall HTC ) to FEA.

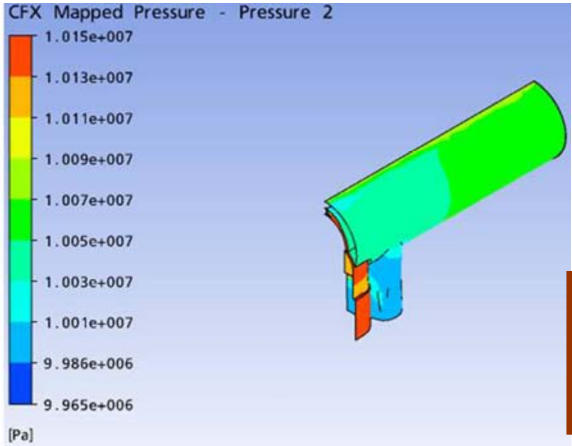


Repeated thermal analysis with ANSYS

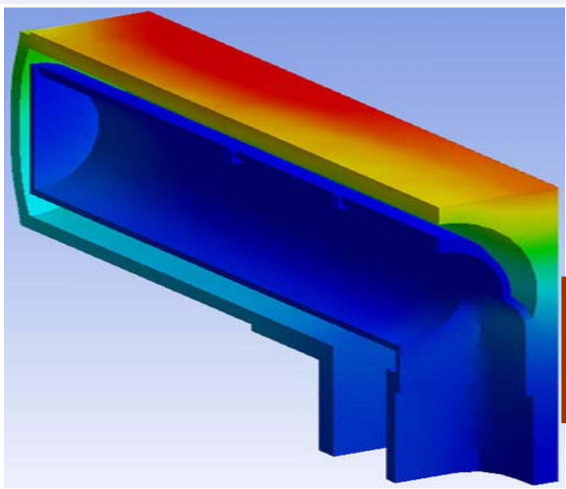
➢The volumetric temperatures from ANSYS FEA modeling are available for the subsequent structure simulation.

# CFX Pressure Load Transferred to ANSYS for Mechanical Analysis

UC San Diego

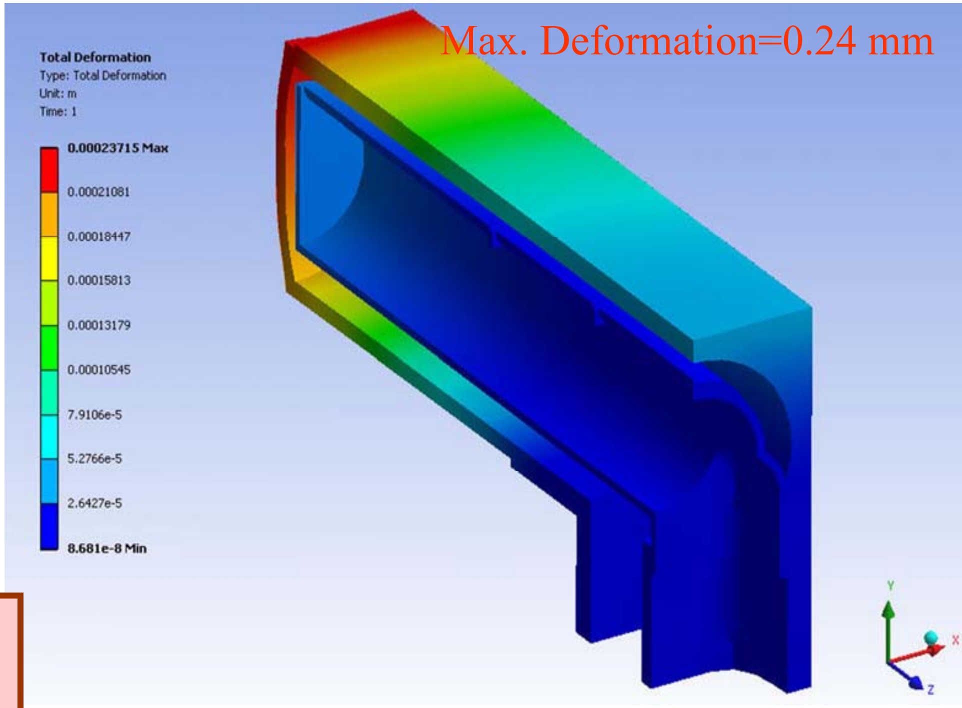



Pressure load from CFX



Thermal loads from ANSYS

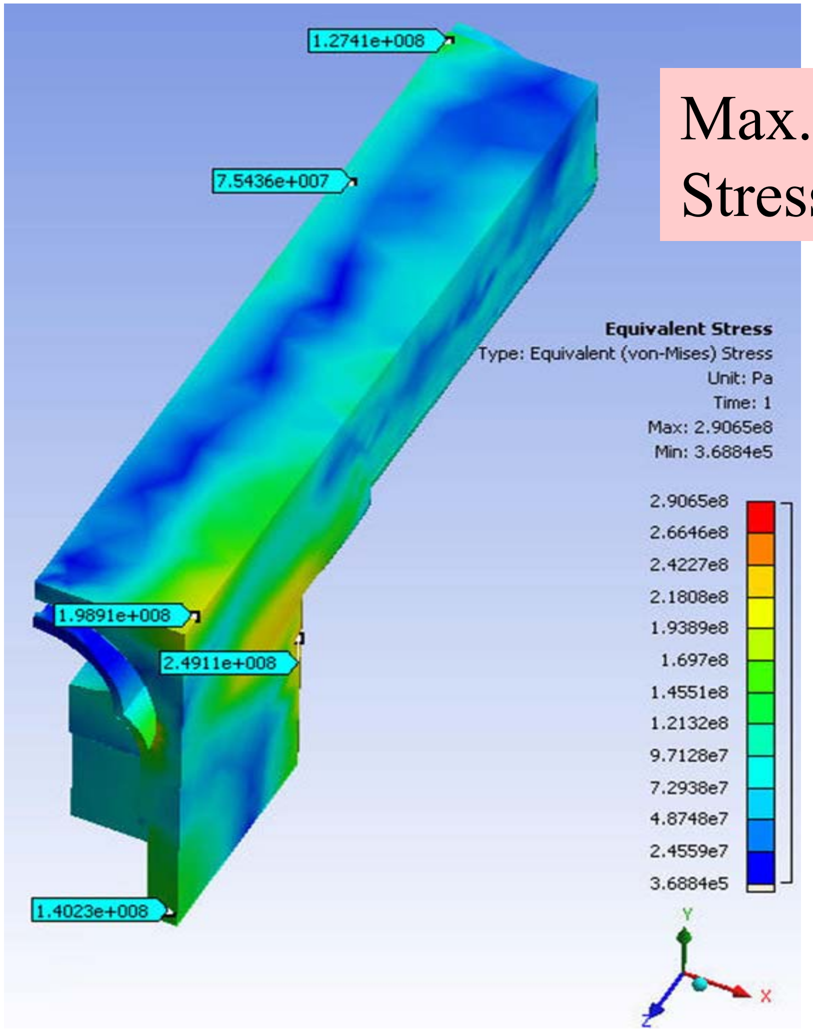
## ANSYS Mechanical Analysis



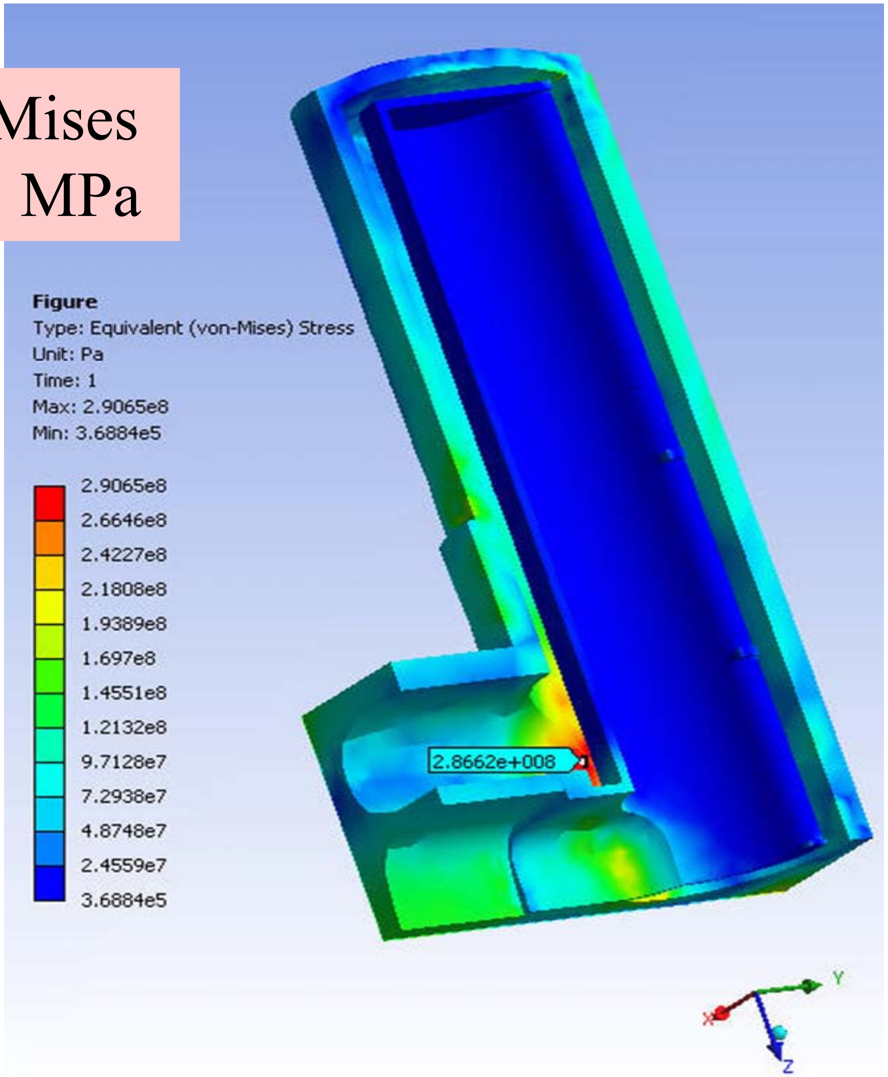

B.Cs:

- ✓Symmetry B.Cs
- ✓0 displacement in plane (x-z) at the bottom of inlet/outlet

# Example ANSYS Results Shown Thermal Stresses in Structure

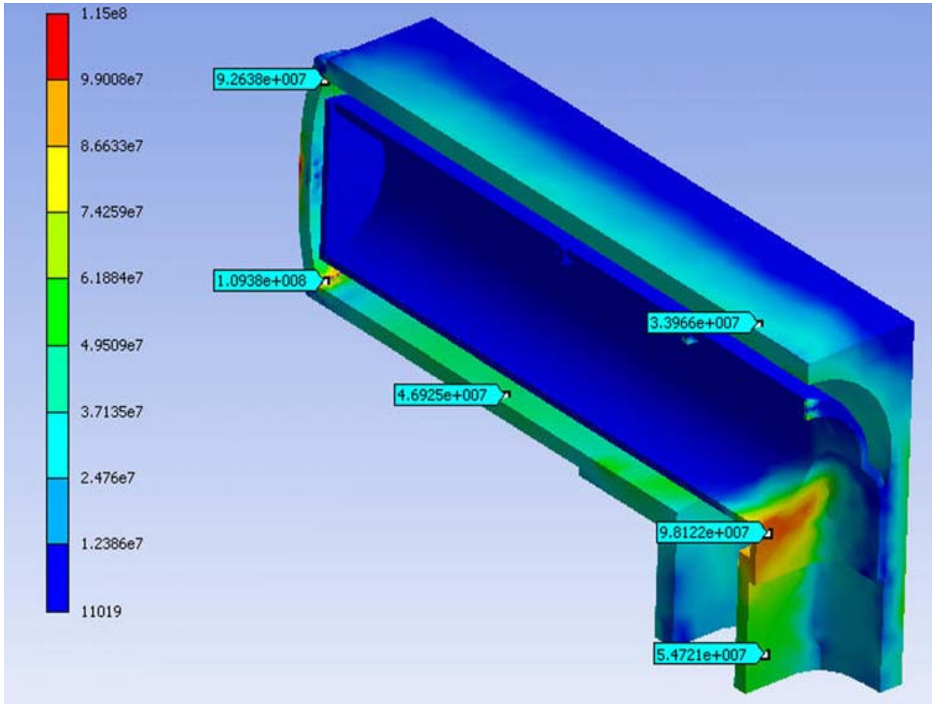



Max. von-Mises Stress=291 MPa

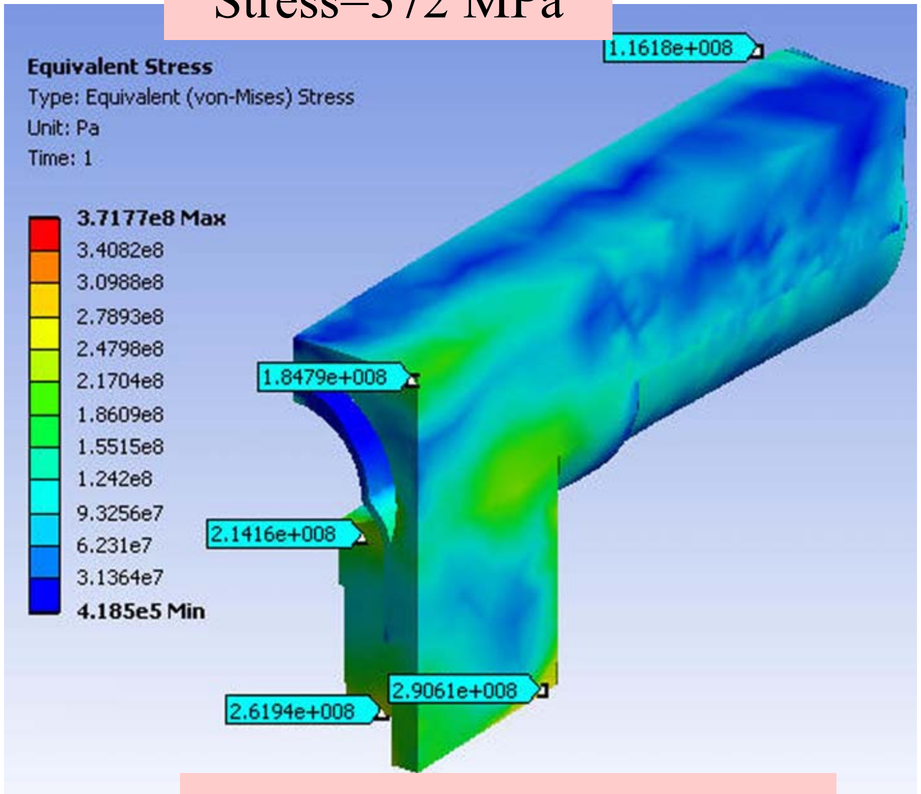


# Example ANSYS Results Shown Primary and Thermal Stresses

Max. von-Mises Stress=372 MPa




Pressure load only



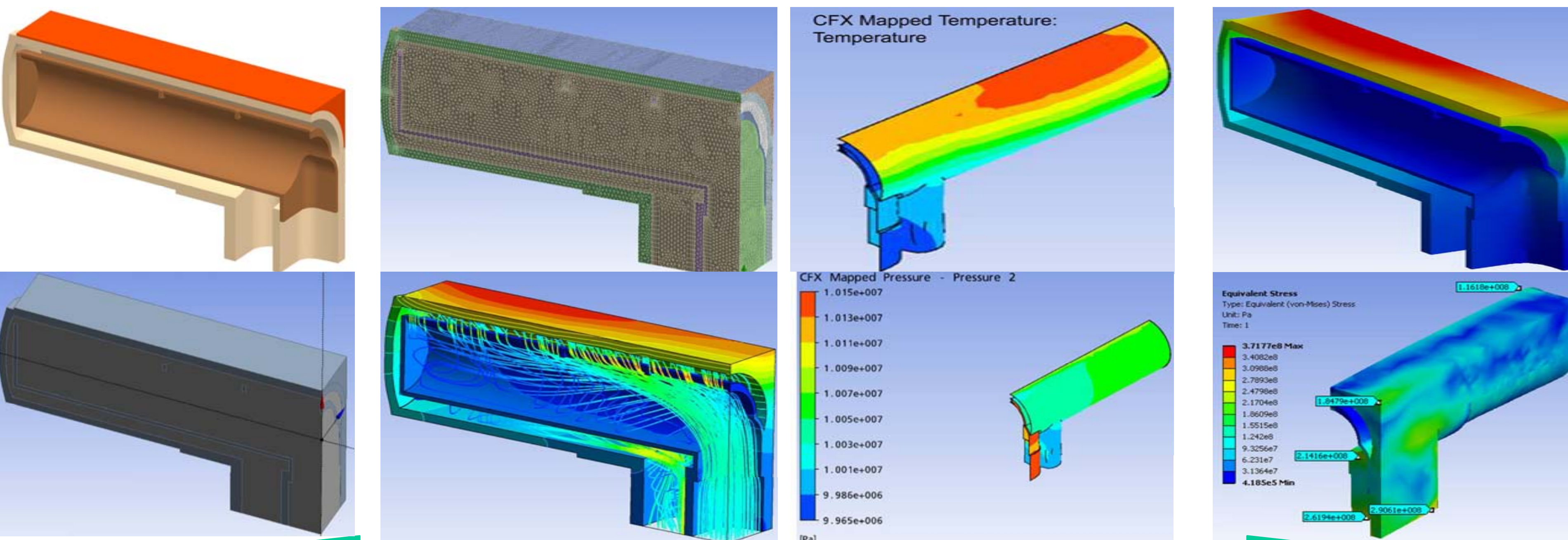

Pressure plus thermal loads

- The results are consistence with ARIES-CS (~370 MPa from Thomas Ihli).

# Summarize the Simulation Process of T-tube Divertor With CFD & ANSYS

UC San Diego

| Pro/E (CAD) Or DesignModeler in Workbench | CFX Fluid/Thermal Simulation | Mapped CFD Results to ANSYS FEA Model | ANSYS Thermal&Stress Simulation |
|---|---|---|---|

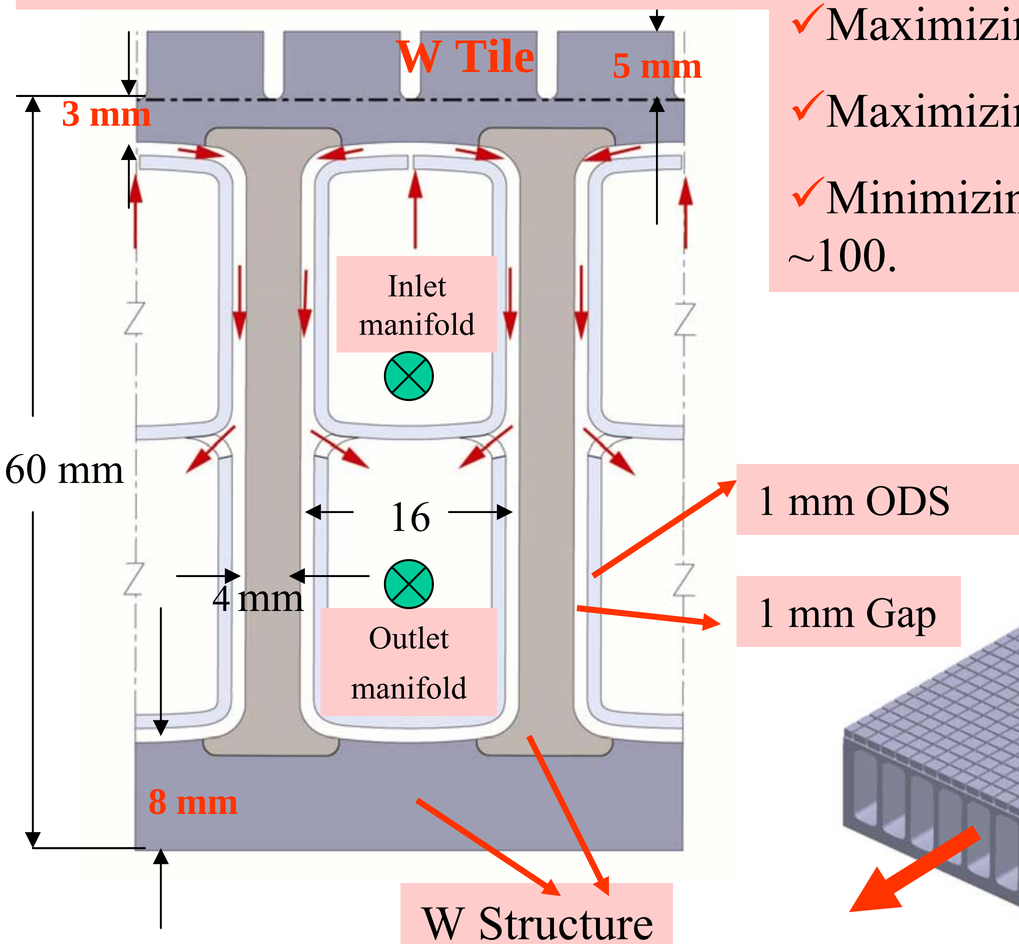


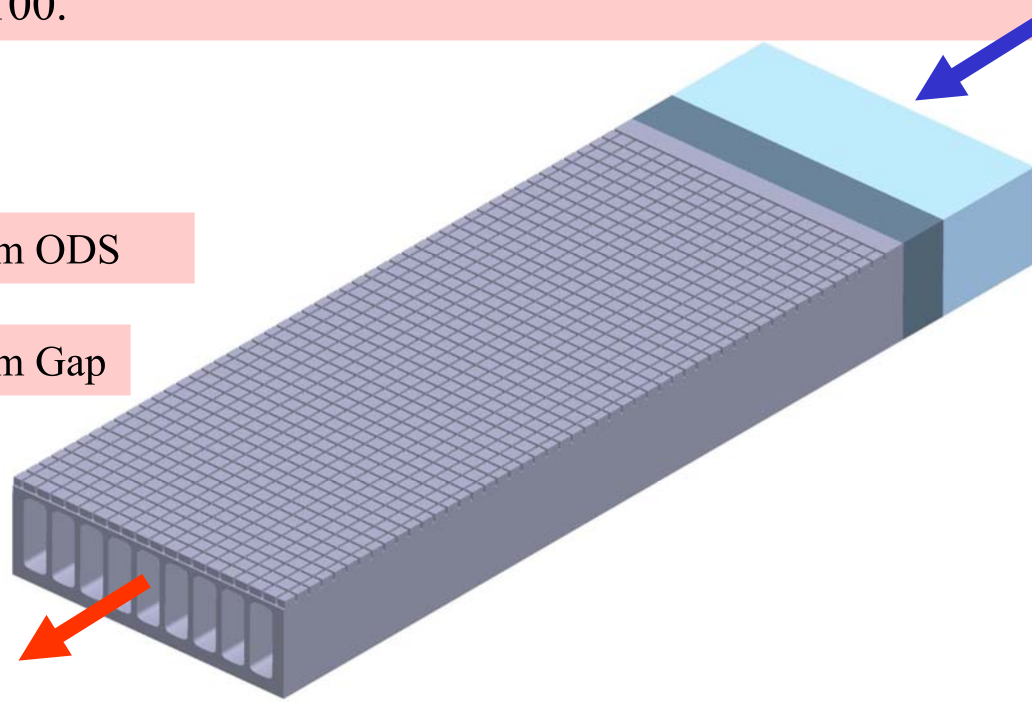

All the Processes Are Performed Within a Single and Integrated Simulation Environment: ANSYS Workbench

# Explore 10 MW/m$^2$ Helium-cooled Divertor Plate Concept

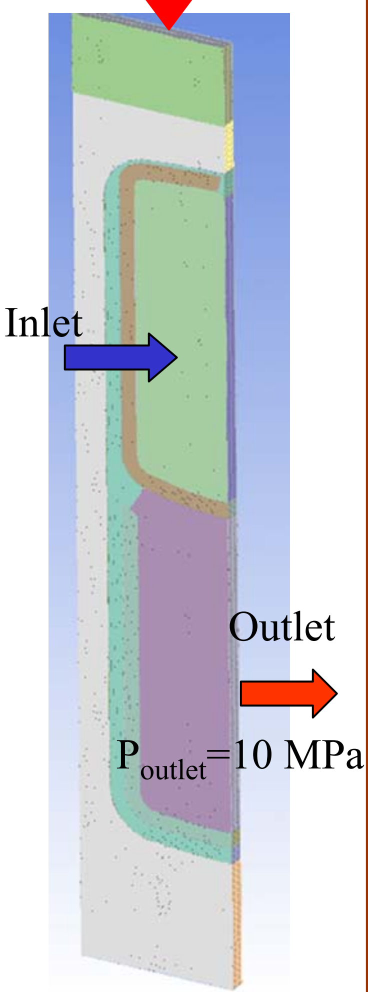
➢Try to optimize plate geometry to improve cooling performance with acceptable structure temperature (~1300 °C), stresses (3Sm=450 MPa for pure W, 401 MPa for WL10) and pumping power ($P_P/P_{th}$ <10%):

- ✓Maximizing helium velocity to ~200 m/s;
- ✓Maximizing HTC in the range of 30~50 kW/m$^2$K;
- ✓Minimizing front and back temperature difference to ~100.



# CFX Element Model for Fluid & Thermal Analysis

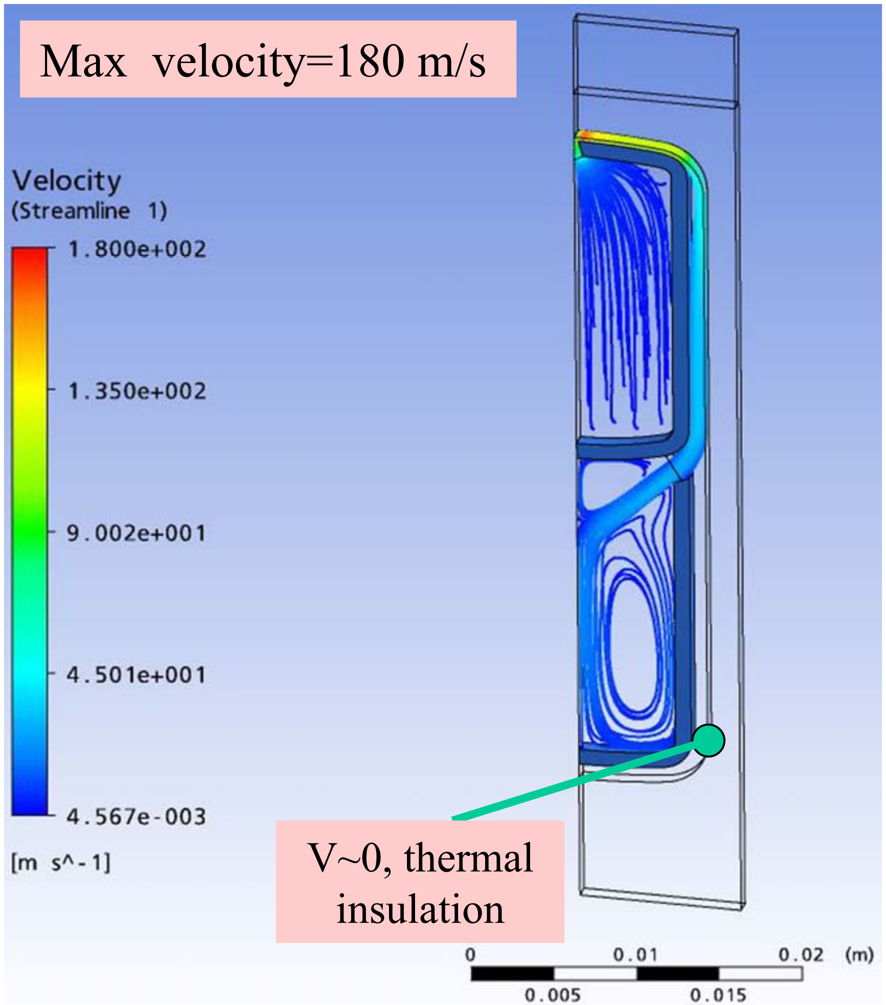


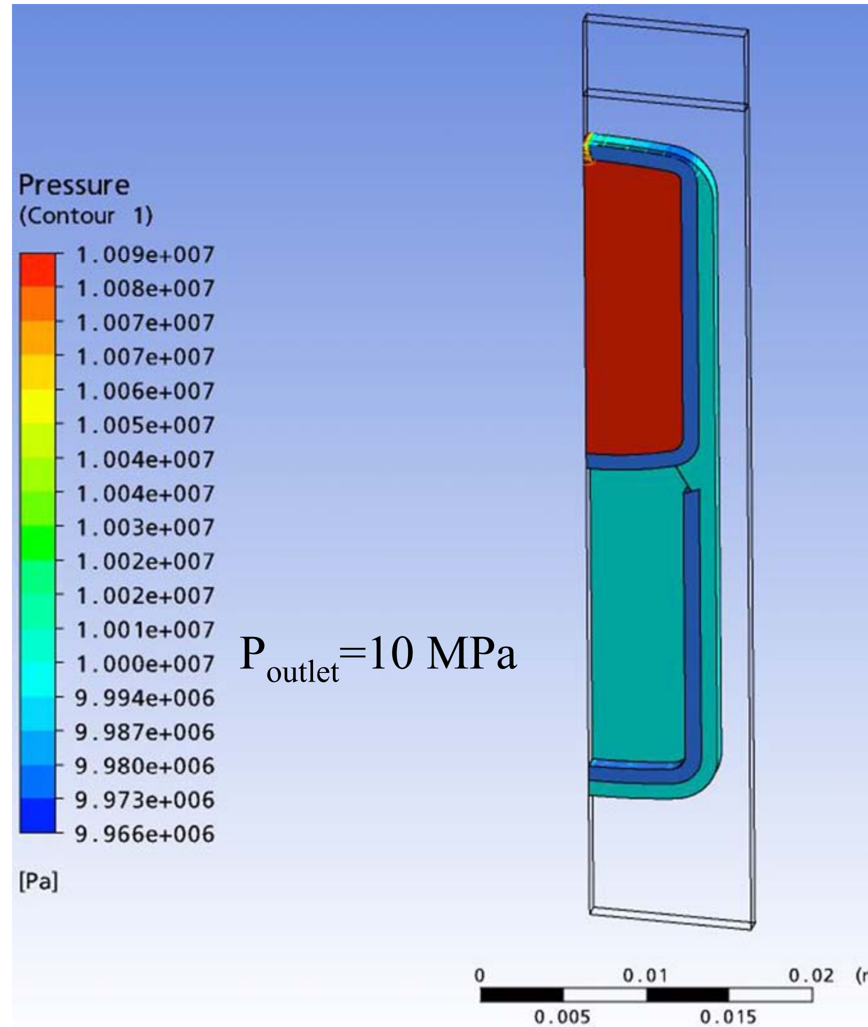

- A sliced 2D Plane with 1 mm thick;
- Heat flux: 10 MW/m$^2$;
- Volumetric heat generation: 53 MW/m$^3$;
- $T_{outlet}$-$T_{inlet}$=740-600 $^{o}$C(77 for T-tube)
- Mass flow rate=0.00016 kg/s (Re~2.76 x 10$^3$ at inlet)
- Nozzle width=0.5 mm;
- Jet wall space=0.25 mm;
- Helium outlet pressure=10 MPa;
- Standard turbulent flow model, K-epsilon with rough wall (roughness height=5 micron);
- 250,000 elements.

Mass flow rate=0.00016 kg/s

Average Inlet T=873 K

# Example CFX Results Shown Velocity and Pressure Distribution

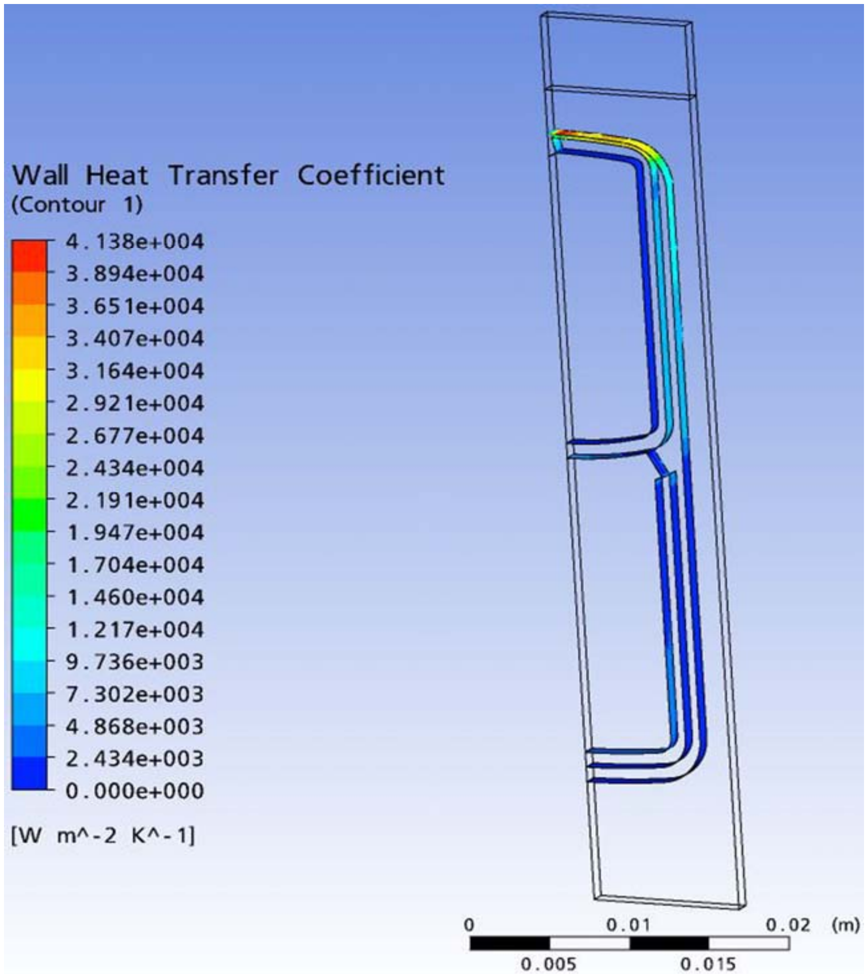


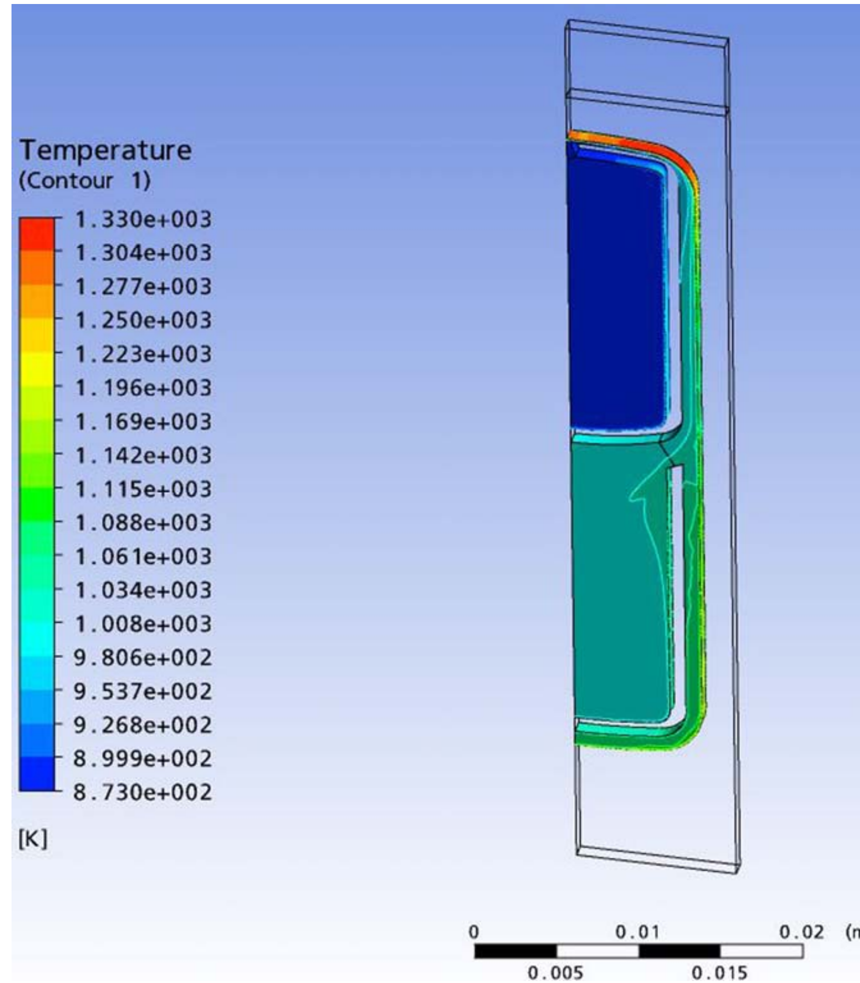




"He Thermal insulation" rises temperature at the back plate, reducing thermal stresses.

$\Delta P$=10.09-10.0=0.09 MPa

# Example CFX Results Shown Wall HTC and Coolant Temperature Distribution



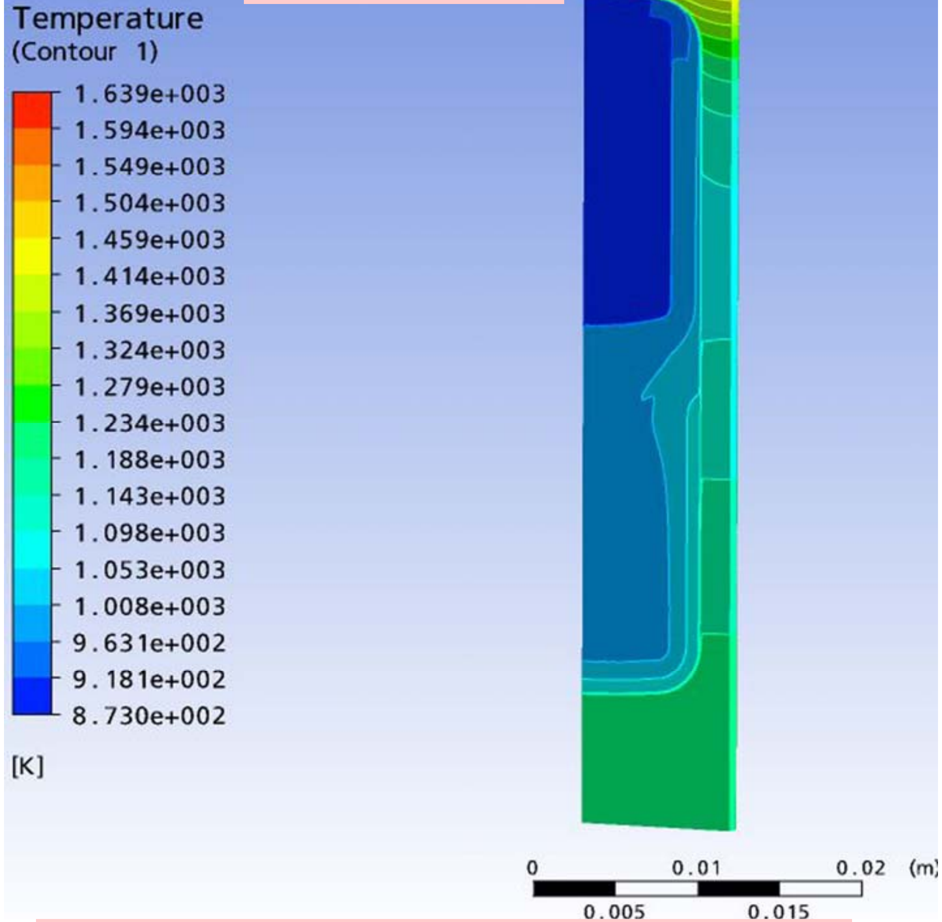




Max HTC=4.138 x $10^4$ W/m$^2$K

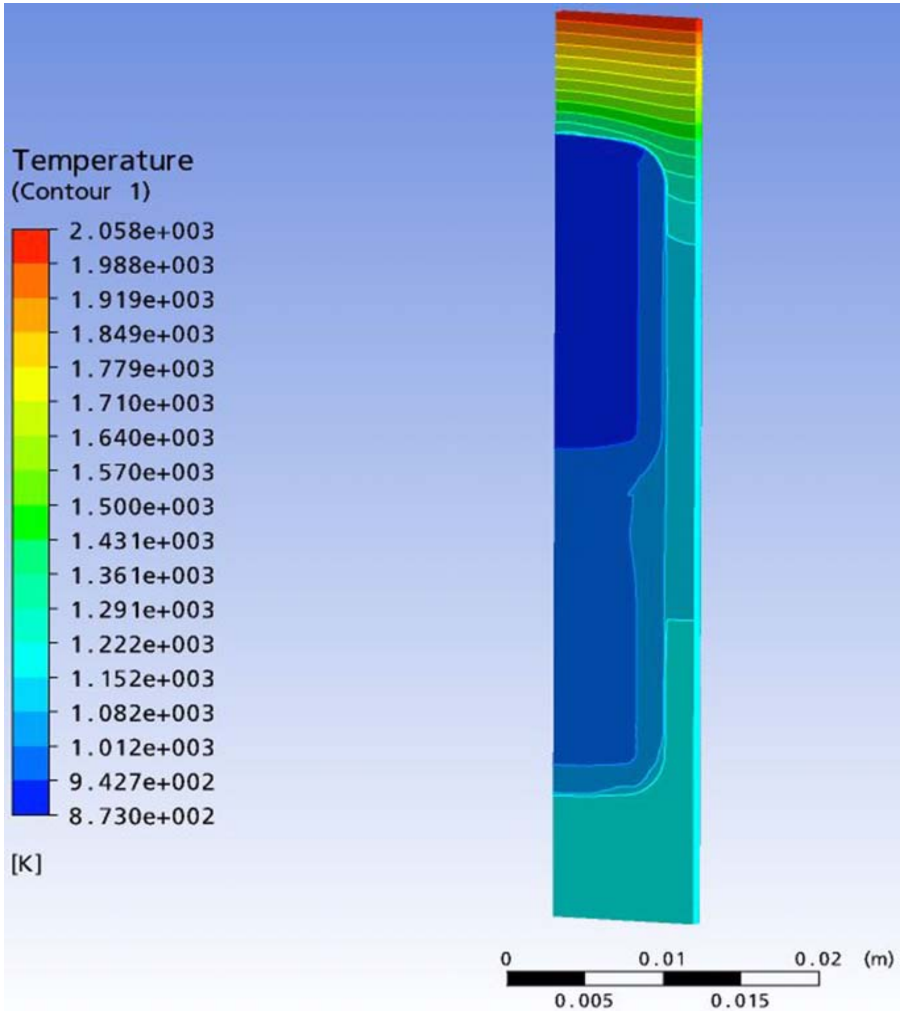
Max Coolant T=1330 K

# Example CFX Results Shown Temperature Distribution in Divertor Plate

$T_{max}$=1366 °C




Temp. differences between front and back plate ~ 540 K (too high)




Max Tile T=2058 K

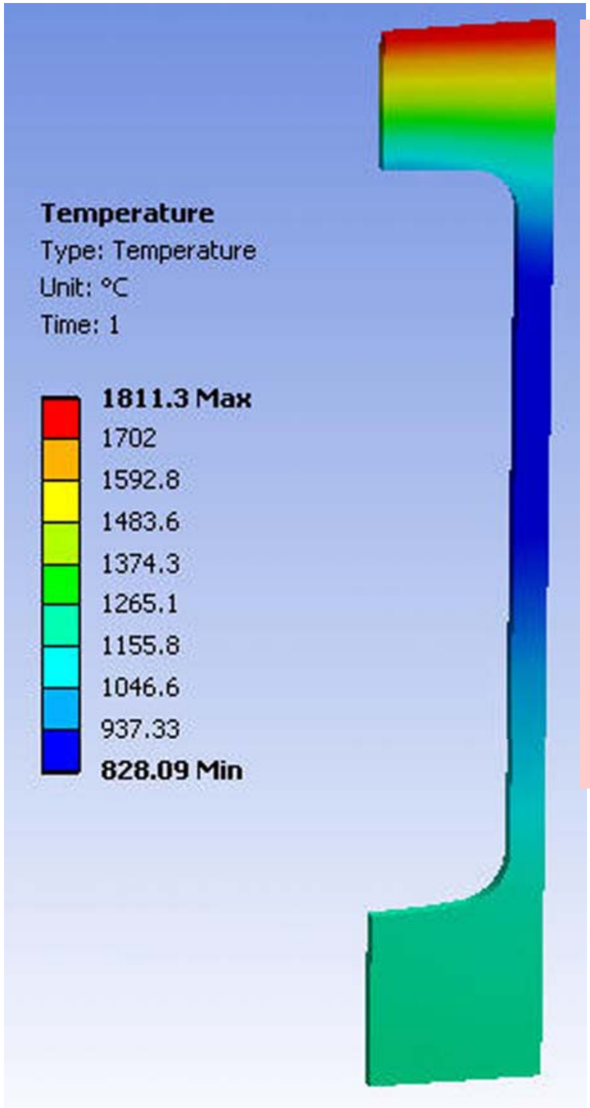
# CFX Fluid/Thermal Parametric Study

| | Jet Wall Space 1.0 mm | Jet Wall Space 0.25 mm | Jet Wall Space 0.25 mm* |
|---|---|---|---|
| Max helium velocity [m/s] | 103.4 | 180.0 | 180.0 |
| Pressure drop [Pa] | $31 \times 10^3$ | $90 \times 10^3$ | $90 \times 10^3$ |
| Heat transfer coefficient [$W/m^2K$] | $1.849 \times 10^4$ | $4.138 \times 10^4$ | $4.138 \times 10^4$ |
| Max. T at front side of structure [K] | 2004 | 1639 | 1586 |
| Max. T at back side of structure [K] | 1400 | 1100 | 1492 |

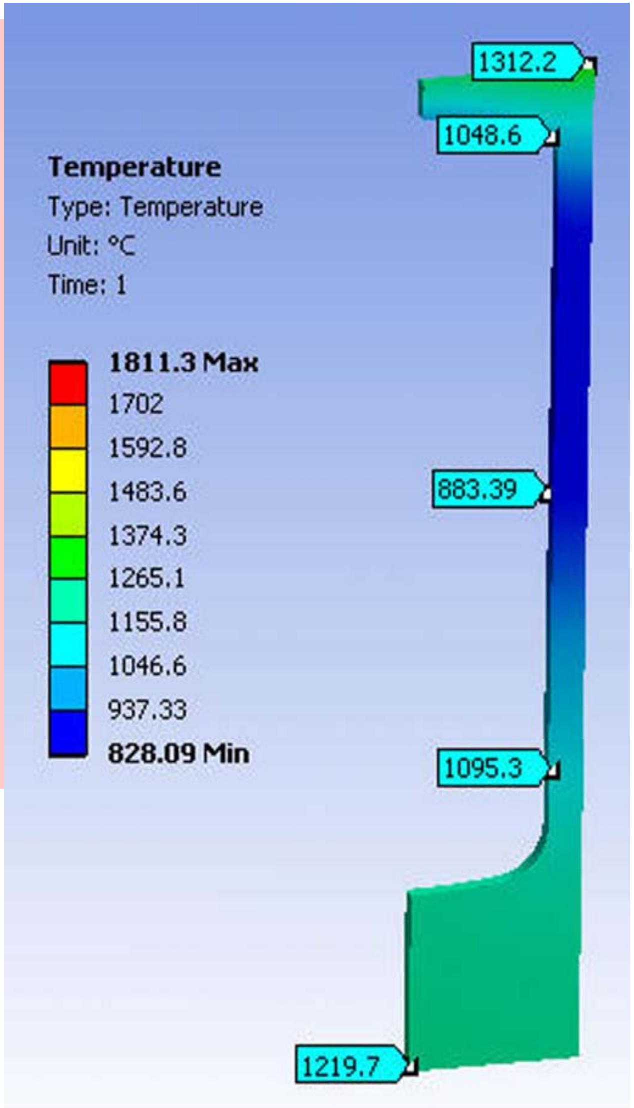
- Increasing the mass flow rate or reducing jet wall space can increase both the flow velocity and heat transfer coefficient to make temperature in W structure ~ 1300 $^o$C.

*Reduced front plate thickness from 3 to 2 mm; and increased back plate thickness from 8 to 10 mm.

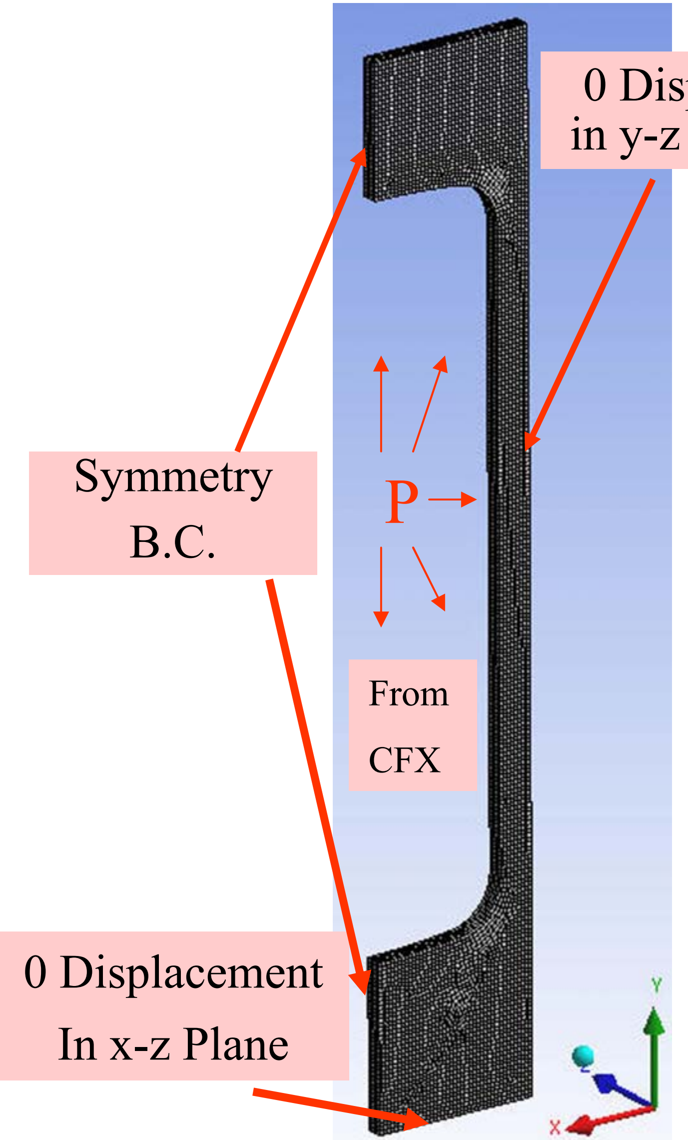
# Thermal Results Shown Improvements With Modified Geometry

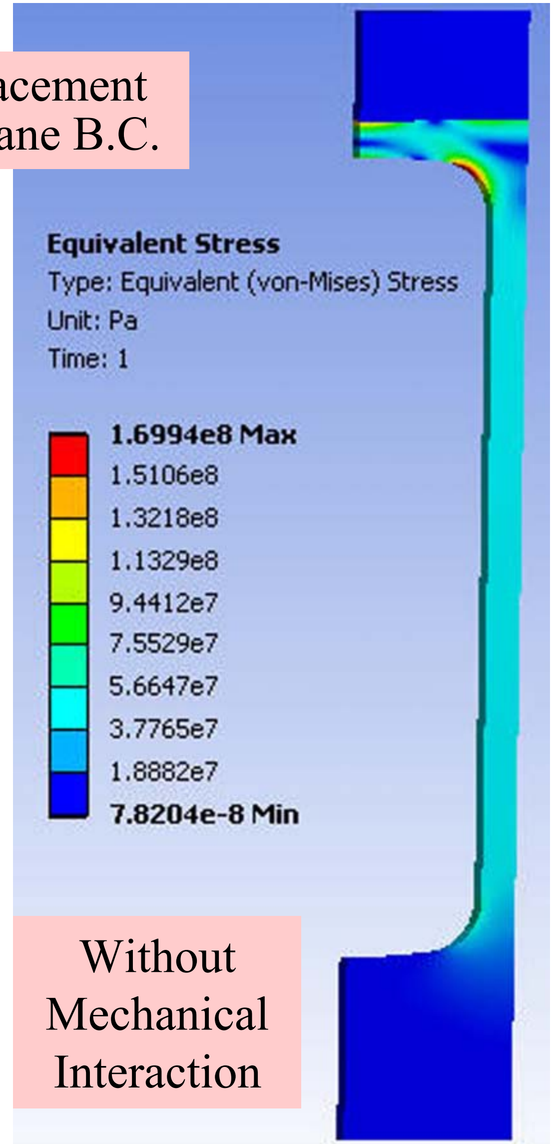


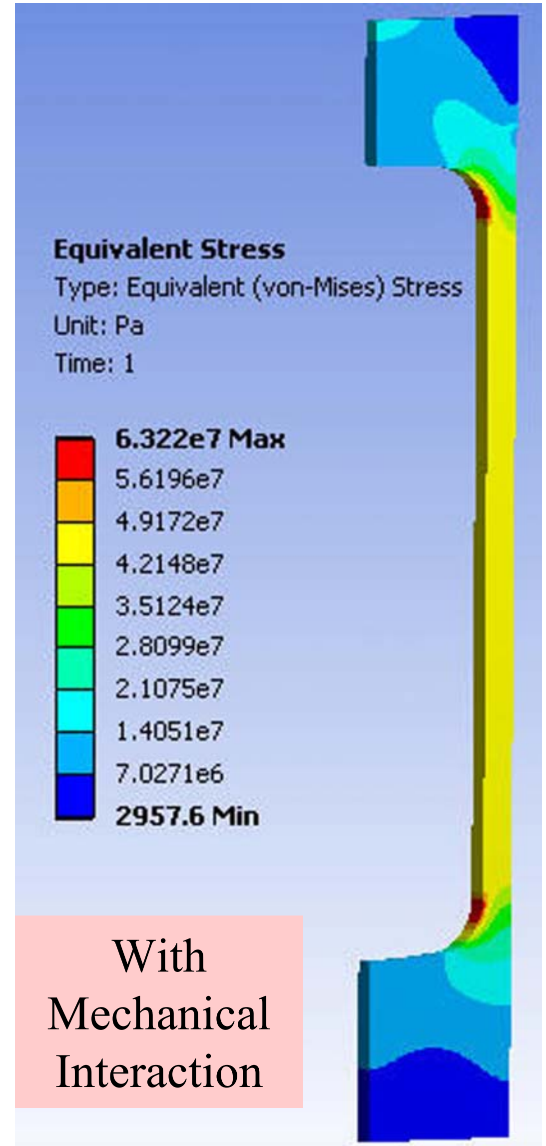

- ODS insert tube is excluded in ANSYS FEA model for thermal&stress analysis because of both sides with helium.
- Wall surface temperature from CFX is mapped to ANSYS thermal model.
- Temperature difference between the front and back plate is dropped from 540 to 100.



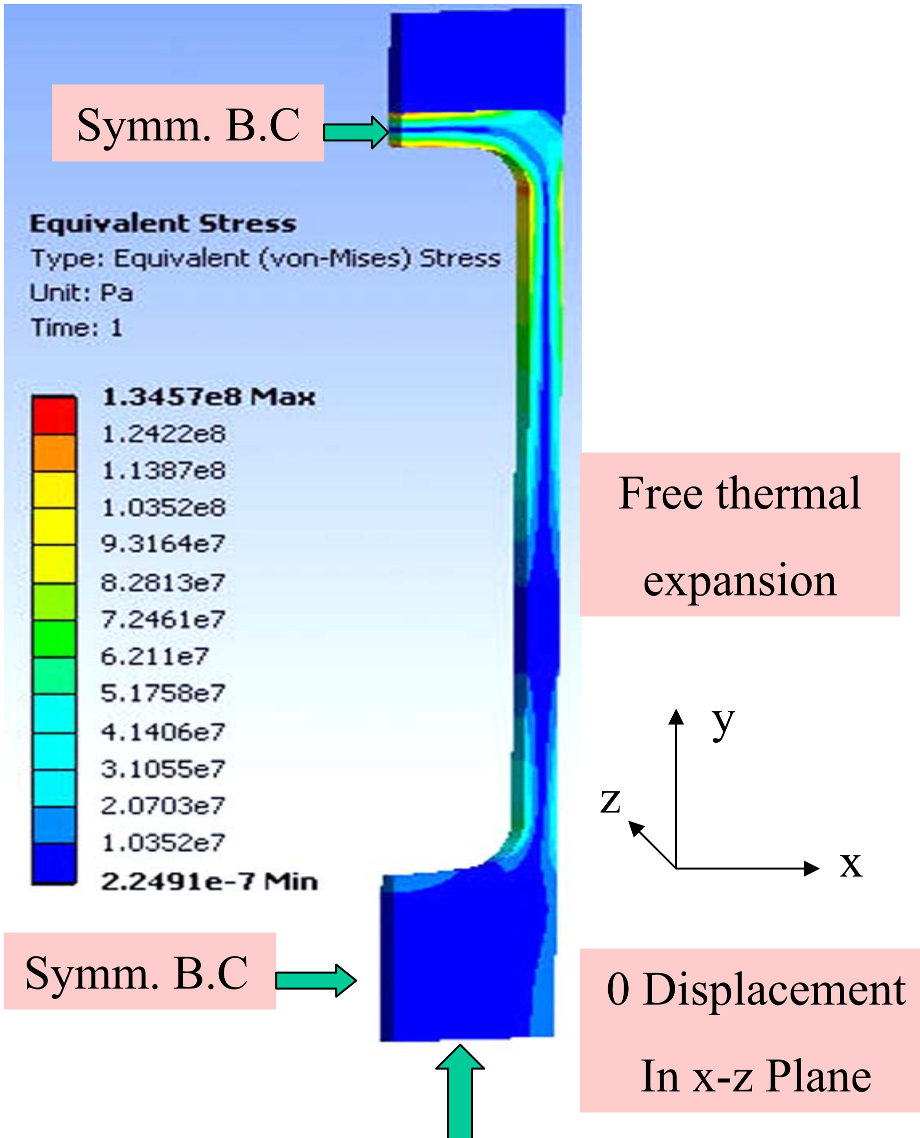
# Primary Stresses With and Without Mechanical Interaction to W Tile

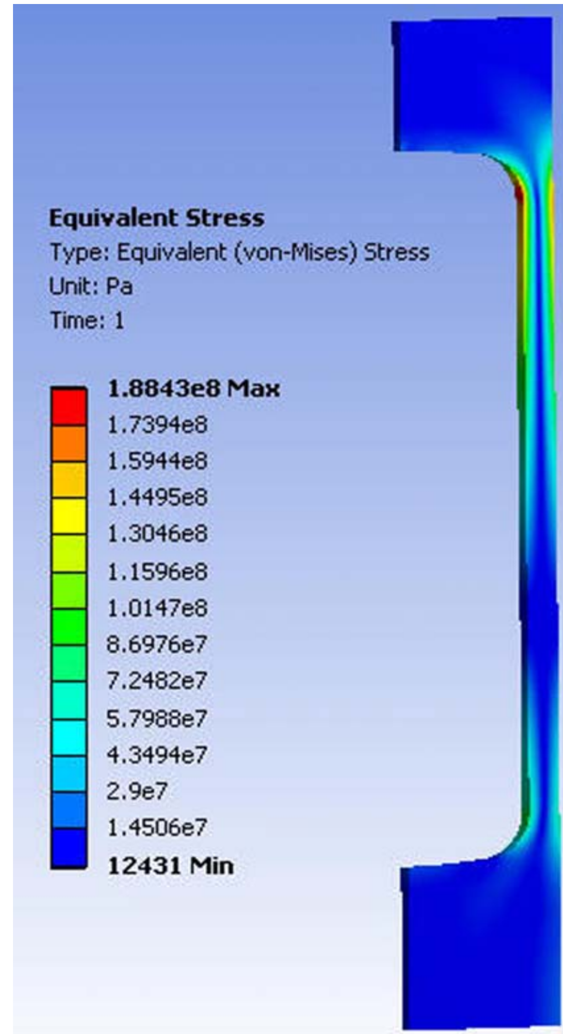


# Thermal Stresses With and Without Mechanical Interaction to W Tile



# Summary and Future Work

UC San Diego

- Fluid/Thermal and stresses of Helium-cooled ARIES-CS T-Tube divertor have been reproduced in order to establish work experience on understanding the process of the CFD and thermal stress simulation with CFX and Workbench.
- 10 MW/m$^2$ Helium-cooled plate-type divertor has initially been explored based on a sliced 2-dimensional plane (1 mm thick). The results indicate that both the temperature and stress at the plate structure are below design limits (~1300 $^{o}$C, 3Sm=450 MPa for pure W, and 401 MPa for WL10), but more detailed analysis of the thermal stresses are required.
- A real 3-dimensional plate model will be needed to simulate the jet cooling.
- A transient thermal stress analysis for the plate-type divertor will be performed for different operating condition:
  - ✓ ***Transient power cycles between full power and zero power, but constant helium inlet temperature***
  - ✓ ***Transients helium inlet temperature, 600 $^{o}$C inlet temperature for full power, but 100 $^{o}$C for zero power***