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

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

- 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, etc.

- 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

- 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.

ARIES-CS T-tube Divertor

Full T-Tube Model
Importing geometry from Pro/E
- 830,000 tetrahedral elements
- Turbulent flow (Re=8.4x10^4 at inlet)
- Standard k-ε turbulent model
- Rough wall (Roughness height=20 micron)

Average outlet He pressure: \( P_{\text{outlet}} = 10 \text{ MPA} \)
Average inlet velocity: 58.25 m/s
Average He inlet T: 873 K

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

Max. Jet velocity = 218 m/s

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

Temperature Distribution in Solid Structure
CFX Thermal Results Shown Good Consistence with ARIES-CS Results

<table>
<thead>
<tr>
<th></th>
<th>Wall Roughness 0 micron</th>
<th>Wall Roughness 10 micron</th>
<th>Wall Roughness 20 micron</th>
<th>ARIES-CS Results (FLUENT)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max T at Tile [K]</td>
<td>1738</td>
<td>1705</td>
<td>1676</td>
<td>1699</td>
</tr>
<tr>
<td>Max T [K] at Tube/Tile Interface</td>
<td>1546</td>
<td>1512</td>
<td>1494</td>
<td>1523</td>
</tr>
<tr>
<td>Max Jet Velocity [m/s]</td>
<td>214.4</td>
<td>216.3</td>
<td>218.4</td>
<td>216</td>
</tr>
<tr>
<td>Wall Heat Transfer Coefficient [W/m²K]</td>
<td>2.78 x 10⁴</td>
<td>3.76 x 10⁴</td>
<td>4.05 x 10⁴</td>
<td>3.74 x 10⁴</td>
</tr>
<tr>
<td>ΔP [Pa]</td>
<td>1.4 x 10⁵</td>
<td>1.5 x 10⁵</td>
<td>1.6 x 10⁵</td>
<td>1.1 x 10⁵</td>
</tr>
</tbody>
</table>

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 CFX flow simulation

Edges are not rounded

Geometry used in FLUENT flow simulation (GT)
CFX Results Mapped to ANSYS FEA Model for Repeating Thermal Analysis

ANSYS 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

ANSYS Mechanical Analysis

Pressure load from CFX

Thermal loads from ANSYS

Max. Deformation=0.24 mm

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

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/m² 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 (Pp/Pth <10%):

✓ Maximizing helium velocity to ~200 m/s;
✓ Maximizing HTC in the range of 30~50 kW/m²K;
✓ Minimizing front and back temperature difference to ~100.

![Diagram of the divertor plate concept with dimensions and annotations.](image-url)
A sliced 2D Plane with 1 mm thick; Heat flux: 10 MW/m²; Volumetric heat generation: 53 MW/m³; $T_{\text{outlet}} - T_{\text{inlet}} = 740-600 \, ^\circ\text{C}(77 \text{ for T-tube})$ Mass flow rate=0.00016 kg/s (Re~2.76 \times 10^3 \text{ 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.
Example CFX Results Shown Velocity and Pressure Distribution

Max velocity = 180 m/s

Velocity (Streamline 1)

Max velocity = 180 m/s

P_{outlet} = 10 MPa

{\Delta P = 10.09 - 10.0 = 0.09 MPa}

V~0, thermal insulation

“He Thermal insulation” rises temperature at the back plate, reducing thermal stresses.
Example CFX Results Shown Wall HTC and Coolant Temperature Distribution

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

Max Coolant T = 1330 K
Example CFX Results Shown
Temperature Distribution in Divertor Plate

\[ T_{\text{max}} = 1366 \, ^\circ \text{C} \]

Temp. differences between front and back plate \( \sim 540 \, \text{K} \) (too high)

Max Tile \( T = 2058 \, \text{K} \)
# CFX Fluid/Thermal Parametric Study

<table>
<thead>
<tr>
<th></th>
<th>Jet Wall Space 1.0 mm</th>
<th>Jet Wall Space 0.25 mm</th>
<th>Jet Wall Space 0.25 mm*</th>
</tr>
</thead>
<tbody>
<tr>
<td>Max helium velocity [m/s]</td>
<td>103.4</td>
<td>180.0</td>
<td>180.0</td>
</tr>
<tr>
<td>Pressure drop [Pa]</td>
<td>31 x 10³</td>
<td>90 x 10³</td>
<td>90 x 10³</td>
</tr>
<tr>
<td>Heat transfer coefficient [W/m²K]</td>
<td>1.849 x 10⁴</td>
<td>4.138 x 10⁴</td>
<td>4.138 x 10⁴</td>
</tr>
<tr>
<td>Max. T at front side of structure [K]</td>
<td>2004</td>
<td>1639</td>
<td>1586</td>
</tr>
<tr>
<td>Max. T at back side of structure [K]</td>
<td>1400</td>
<td>1100</td>
<td>1492</td>
</tr>
</tbody>
</table>

*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 °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

0 Displacement in y-z plane B.C.

Symmetry B.C.

From CFX

0 Displacement In x-z Plane

Without Mechanical Interaction

With Mechanical Interaction
Thermal Stresses With and Without Mechanical Interaction to W Tile

Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: Pa
Time: 1

Symm. B.C

Free thermal expansion

0 Displacement
In x-z Plane

Symm. B.C
Summary and Future Work

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/m² 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 °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 °C inlet temperature for full power, but 100 °C for zero power