Computation Fluid Dynamics for the Spallation Neutron Source

Mark Wendel
Research Engineer
Oak Ridge National Laboratory
The Spallation Neutron Source is now an operating facility.

- The SNS began operation last month (April 2006)
- At 1.4 MW it will be ~8x ISIS, the world’s leading pulsed spallation source
- SNS will be the world’s leading facility for neutron scattering
The Spallation Neutron Source is located at the Oak Ridge National Laboratory.
The mercury target is bombarded with intense proton pulses at a rate of 60 Hz.
The mercury flows through the target at a rate of 380 gpm, with peak velocities near 4 m/s.
Early FLOW-3D simulations (from 1997) led to specific target features.
Challenging problems for target flow that have been addressed, by CFX simulations.

- Liquid metal heat transfer
  - Low Prandtl number fluid creates uncertainty in using the Reynold’s analogy
  - Experimental program has shown that for narrow channels, the CFD heat transfer predictions are good using a higher turbulent Prandtl number.
Challenging problems for CFX (continued)

• Beam-induced pressure pulse
  – Upon impact, each proton pulse causes isochoric heating which creates 60 Hz pressure pulses that can lead to pitting of the target containment.
  – Compressible-liquid simulation with mercury showed the impact of the pressure pulse on the flow.
  – Fluid/Structure interaction has been done with other structural codes.
• Pressure pulse mitigation technique
  – Mitigation schemes for cavitation damage involve the injection of gas into the system.
  – Two gas injection schemes exist:
    • The first is to introduced tiny (~10 micron) bubbles into the mercury that will absorb the beam energy.
    • The second scheme is to develop a protective gas layer on the wall that prevents the pitting.
      – Initial CFX 2-component simulations have led to insights about the design of such a target.
      – Experiments are underway to collaborate CFX results.
Challenging problems for CFX (continued)

• Flow induced cavitation prediction
  – Separate from the beam-induced cavitation problem.
  – Mercury is very heavy (SG=13.5) and is prone to cavitation especially with turbulent pressure fluctuations.
  – Transient RANS calculations with very fine-grid models of a double-mitred bend were used to correlate cavitation observations for the mercury piping.
The mercury target design provides a symmetric bulk flow and window channel.
The symmetric bulk flow is typically 1–3 m/s.

- Most of the heat is deposited in bulk flow
- Heat deposited in low-speed zones is a concern (long residence times)
  - Separation bubble (Recirc zone)
  - Flow stagnation near window
- Asymmetry has been calculated and observed
- WTHL measurements confirm turbulence quantities
The proton beam deposits 1.4 MW in a small volume
Vortex shedding down the return channel shows thermal striping with $8^\circ$C amplitude.
Temperature oscillations are limited to the return channel.
Turbulence Modeling

• RANS Model was used with true transient
  – Assumes eddy viscosity calculated with the Menter SST turbulence model
  • good particularly for separated flows with heat transfer
  • Combination of k-ω and k-ε
  • Utilizes a collapsed wall layer that allows for grid convergence
Turbulence Modeling

- Turbulent Prandtl number of 2.7 was used for final simulations.
  - Based on liquid metal data
  - Higher Prandtl numbers lead to less heat transfer and higher temperatures

Relationship of Rosenhow and Cohen, reported in Viscous Flow by F. White.
Liquid Metal Heat Transfer – Tube Flow Validation

- Using the Rosenhow and Cohen correlation for Prt gives good agreement between theory (CFX and Lyon) and data.
The proton beam deposits 1.4 MW in a small mercury volume in an abrupt fashion.

- Peak energy deposition in Hg for a single pulse = 13 MJ/m³
  - Peak temperature rise is only ~ 7 K for a single pulse, but rate of rise is 107 K/s!
- Constant volume process because beam deposition time (0.7 ms) << time required for mercury to expand
  - Beam size/sound speed ~ 33 ms
- Local pressure rise is 38 MPa (380 atm compared to static pressure of 3 atm!)
Two-Dimensional CFX Compressible Liquid Simulation
Cavitation Bubble Collapse Leads to Pitting Damage

- Large tensile pressures create cavitation bubbles
  - Damage is caused by violent collapse of cavitation bubbles (microjets)

Damage in region with large pits for bare 316SS-LN diaphragm after July 2001 WNR tests
Pressure Pulse Mitigation: The Gas Wall

- Multiphase flow simulation with helium injection to liquid mercury confirms experimental data that show a gas wall
  - High surface tension (0.47 N/m)
  - High contact angle (133°)
Pressure Pulse Mitigation: The Gas Wall

- Multiphase flow simulation with helium required one week using 4 2 GHz WindowsXP machines.
Onset of Flow-Induced Cavitation

- Mercury has a high specific gravity, resulting in large-amplitude turbulent pressure fluctuations.
- Early tests with a prototypical flow loop showed pronounced cavitation in a double-mitered bend geometry for an inlet pipe.
- CFX was used to simulate the cavitation in the double-mitered bend using transient RANS.
Mitred Bends Iso-surfaces for Velocity
Mitred bends velocity contours
Mitred Bends: Cavitated State
Summary

• ANSYS-CFX has been used extensively in designing the SNS target.
• Validation of the models to experiments is comprehensive.
• For narrow channels, the Rosenhow-Cohen correlation for turbulent Prandtl number proved satisfactory.
Summary

• Two-phase flow simulations of a gas wall concept have provided some guidance to experiments, but are very expensive computationally.

• RANS transient simulations yield an extrapolation method to obtain reasonable predictions for flow-induced cavitation.
Based on CFD results, a D-shaped orifice was added to the mercury inlet.
Target Cooling – Window Coolant Channel

• Highest heat load is on window
• Cooled directly by Hg flowing through window coolant channel
Thermal loading on the target is a strong function of position.
CFX4 simulation: steady oscillation was obtained

Pressure (Pa) at monitoring point

Velocities (m/s) at monitoring point

Time step is 0.03 s, frequency is about 8 Hz content, corresponding to the largest eddies in the return channel.
A true-transient hydrodynamic solution was continued until a steady oscillation was obtained.

Pressure (Pa) at monitoring point

RMS Residuals

Velocities (m/s) at monitoring point

1.375s (0.7 Hz) periodicity

Also, 8 Hz content
Target Cooling – Window Flow

• Simpler channel flow

• Complications
  – Moderate Curvature – Taylor–Goertler vorticies (roll cells)
  – Converging/diverging channel
  – Thermal wetting was a concern

• Experimental measurements
Temperatures along the beam centerline at the window show that grid-convergence has been obtained.
CFX4 simulation: steady oscillation was obtained

Pressure (Pa) at monitoring point

Velocities (m/s) at monitoring point

Time step is 0.03 s, frequency is about 8 Hz content, corresponding to the largest eddies in the return channel.
Computed temperatures fluctuate at monitoring points.

Coarse Grid
- Separated zone
- Bulk flow near window

Medium Grid
- Separated zone
- Bulk flow near window

Fine Grid
- Separated zone
- Bulk flow near window

Reduced Time step (0.03 s)
Temperatures in target

Centerline Temperatures (°C)

Distance (m)

Temperature (°C)

2 MW

1 MW

© 2006 ANSYS, Inc.
Time-varying temperatures on centerline

Series plotted every 0.016 s

2 MW
A true-transient hydrodynamic solution was continued until a steady oscillation was obtained.

Pressure (Pa) at monitoring point

Velocities (m/s) at monitoring point

RMS Residuals

1.375s (0.7 Hz) periodicity

Also, 8 Hz content
Static Linear stresses in target at 2 MW

443 MPa (64 ksi) maximum stress intensity.