Natural-Convection Liquid-Hydrogen Absorber R&D

K. Cassel, M. Boghosian, E. Almasri

Fluid Dynamics Research Center,
Mechanical, Materials and Aerospace Engineering Department,
Illinois Institute of Technology,
Chicago, IL.

S. Ishimoto, K. Yoshimura

KEK,
Tsukuba, Japan

\(LH_2\)-Absorber R&D Review Meeting
Illinois Institute of Technology
September 26, 2001
Two approaches under consideration:

1) External cooling loop (traditional approach).
   \[ \Rightarrow \] Bring the \( LH_2 \) to the coolant (heat removed in an external heat exchanger).

2) Combined absorber and heat exchanger.
   \[ \Rightarrow \] Bring the coolant (\( i.e. \) He) to the \( LH_2 \) (remove heat directly within absorber).
Advantages/disadvantages of an external cooling loop:

+ Has been used for several $LH_2$ targets (e.g. SLAC E158).
+ Easy to regulate bulk temperature of $LH_2$.
+ Is likely to work best for small aspect ratio ($L/R$) absorbers.
  - May be difficult to maintain uniform vertical flow through the absorber.

Advantages/disadvantages of a combined absorber/heat exchanger:

+ Takes advantage of natural convection transverse to the beam path.
+ Flow in absorber is self regulating, *i.e.* larger heat input $\Rightarrow$ more turbulence $\Rightarrow$ enhanced thermal mixing.
+ Is likely to work best for large aspect ratio ($L/R$) absorbers.
  - More difficult to ensure against boiling at very high Rayleigh numbers.
Heat Exchanger Analysis:

Energy balance between \( LH_2 \) and coolant (\( He \)).

• Parameters:
  
  \( T_i = \) coolant inlet temperature
  \( T_o = \) coolant outlet temperature
  \( T_{LH_2} = \) bulk temperature of \( LH_2 \)
  \( A = \) surface area of cooling tubes
  \( h_{LH_2} = \) convective heat transfer coefficient of \( LH_2 \)
  \( h_{He} = \) convective heat transfer coefficient of \( He \)
  \( \Delta x = \) thickness of cooling tube walls
  \( k_w = \) thermal conductivity of cooling tube walls
  \( c_p = \) specific heat capacity of \( He \)

• Rate of heat transfer:

\[
\dot{q} = - A \frac{(T_o - T_i)}{\left( \frac{1}{h_{LH_2}} + \frac{\Delta x}{k_w} + \frac{1}{h_{He}} \right) \ln \left( \frac{T_{LH_2} - T_i}{T_{LH_2} - T_o} \right)}
\]

• Mass flow rate of \( He \):

\[
\dot{m}_{He} = \frac{\dot{q}}{c_p (T_o - T_i)}.
\]

\( h_{He} \Rightarrow \) from appropriate correlation (flow through a tube).

\( h_{LH_2} \) and \( T_{LH_2} \Rightarrow \) from CFD simulations (no correlations for natural convection with heat generation).
Computational Fluid Dynamics (CFD):

Features of the CFD Simulations:

• Provides average convective heat transfer coefficient and average $LH_2$ temperature for heat exchanger analysis.

• Track maximum $LH_2$ temperature (cf. boiling point).

• Determine details of fluid flow and heat transfer in absorber.

  ⇒ Better understanding leads to better design!

Non-Dimensional Parameters:

• Rayleigh number:

  $$Ra = GrPr = \frac{g\beta \dot{q}''' D^5 \rho^2 c_p}{32k^2 \mu}$$

  $\dot{q}'''$ = volumetric heat generation

  $D$ = diameter of absorber

  $\beta$ = coefficient of thermal expansion of $LH_2$

  $c_p$ = specific heat capacity of $LH_2$

  $k$ = thermal conductivity of $LH_2$

  $\rho$ = density of $LH_2$

  $\mu$ = viscosity of $LH_2$

• Nusselt number:

  $$Nu = \frac{h_{LH_2} D}{k}$$
Sample CFD Results: $Ra = 1.6 \times 10^{15}$

Streamlines:

Temperature Distribution:
CFD Results from FLUENT:

Average Nusselt Number vs. Rayleigh Number:

\[ \text{Nu}_{\text{laminar}} = 0.8114 \cdot \text{Ra}^{0.1931} \]
\[ \text{Nu}_{\text{turbulent}} = 0.3079 \cdot \text{Ra}^{0.2184} \]
\[ \text{Nu} = 0.5754 \cdot \text{Ra}^{0.1979} \]
\[ \text{Nu}_{\text{JSME}} = 0.5042 \cdot \text{Ra}^{0.2126} \]

Average Convective Heat Transfer Coefficient vs. Rayleigh Number:

\[ h_{\text{avg., laminar}} = 0.5501 \cdot \text{Ra}^{0.1931} \]
\[ h_{\text{avg., turbulent}} = 0.1871 \cdot \text{Ra}^{0.2027} \]
\[ h = 0.3968 \cdot \text{Ra}^{0.1938} \]
\[ h = 0.3901 \cdot \text{Ra}^{0.1979} \]
Non-Dimensional Average Temperature vs. Rayleigh Number:

\[ T^{*}_{\text{avg}} = \left( \frac{T_{\text{avg}} - T_w}{Q'' D^2/\kappa} \right) \]

Non-Dimensional Maximum Temperature vs. Rayleigh Number:

\[ T^{*}_{\text{max}} = \left( \frac{T_{\text{max}} - T_w}{Q'' D^2/\kappa} \right) \]
CFD Results from FLUENT (cont’d):

Maximum Temperature vs. Rayleigh Number:
Sample Heat Exchanger Analysis:

Absorber parameters (single-flip lattice):

\[ L = 0.3 \, \text{m} \]
\[ R = 0.2 \, \text{m} \]
\[ \dot{q} = 150 \, \text{W} \]
\[ \Rightarrow Ra = 7.25 \times 10^{13} \]

Heat exchanger parameters (\( LH_2 \) and \( He \) at 2 atm):

\[ T_i = 14 \, \text{K} \]
\[ T_o = 15 \, \text{K} \]
\[ T_{LH_2} = 18.5 \, \text{K} \text{ (from CFD results)} \]
\[ h_{He} = 1,580 \, \text{W/m}^2\text{K} \]
\[ h_{LH_2} = 210 \, \text{W/m}^2\text{K} \text{ (from CFD results)} \]

Results:

Required heat transfer area: \( A = 0.20 \, \text{m}^2 \)

Mass flow rate of \( He \): \( \dot{m}_{He} = 0.028 \, \text{kg/s} \) (3.9 l/s)
Effect of Heater

The heater is necessary to:

1) Provide heating when the beam is off.
   ⇒ Maintain bulk temperature of $LH_2$.
   ⇒ Induce convection rolls prior to beam incidence.

2) Reduce thermal stratification in bottom portion of absorber.
   ⇒ In one case, with a heat flux from the heater equal to 24% of the beam power, the average convective heat transfer coefficient was increased by 30%.

⇒ Is it worth it?
CFD with FLUENT: Issues and Challenges

• The beam is currently being modeled as a steady Gaussian distribution.
  ⇒ What is the effect of pulsing the beam?

• The $LH_2$ flow in the absorber is at very high Rayleigh number ($Ra$):
  ⇒ Very small-scale turbulence (physically advantageous, but computationally challenging).
  ⇒ Highly unsteady fluid flow and heat transfer.
  ⇒ Need very small computational grids and time steps.

• Pushing the limits of FLUENT.
  ⇒ Developing our own CFD code for this application.
FLUENT vs. KWC Code:

FLUENT Simulations:

• Up to approximately 30,000 grid points practical.
• All turbulence is modeled using RANS models.
• Unsteady solver.
• Easier to do more complex geometries.

Our Navier-Stokes Code:

• Up to approximately 2,000,000 grid points practical.
  ⇒ Enables more accurate calculations at higher Rayleigh numbers.
• All turbulence is calculated using DNS.
• Unsteady solver.
• Easier to add complex physics (e.g. pulsed beam).
• Has been used to solve highly complex and unsteady 2-D flows.

Graduate Students:

• M. Boghosian: Completed FLUENT simulations M.S. thesis. Working full time at Gamma Technologies, Inc.
• E. Almasri: Converting our code to simulate flow in absorber.
Summary

• FLUENT has been pushed as far as it will go.

• Our code is being adapted to simulate the flow in the absorber.
  ⇒ Will allow for more accurate simulations at higher Rayleigh numbers.
  ⇒ Investigate influence of pulsed beam on fluid dynamics and heat transfer.

• More work is needed to determine if operating heater during beam incidence is advantageous.