# 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

ILLINOIS INSTITUTE OF TECHNOLOGY

## Introduction: Approaches to Heat Removal

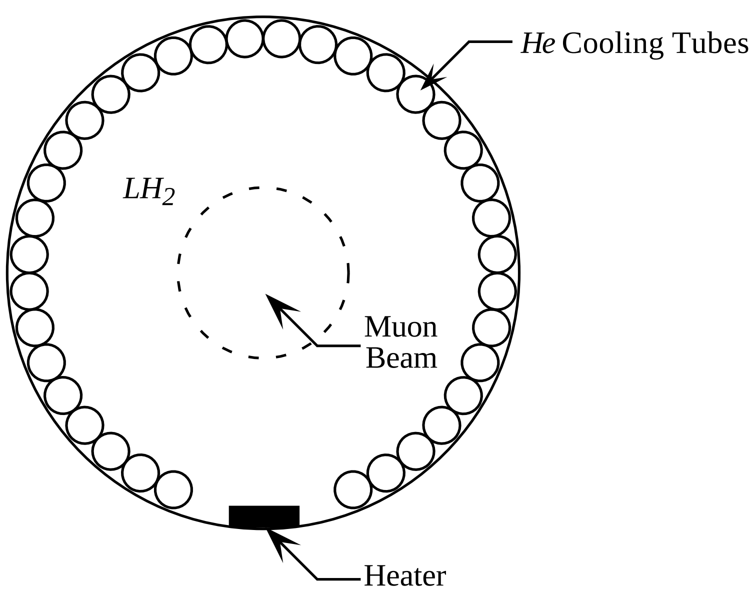
Two approaches under consideration:

1) External cooling loop (traditional approach).

   ⇒ Bring the $LH_2$ to the coolant (heat removed in an external heat exchanger).

2) Combined absorber and heat exchanger.

   ⇒ Bring the coolant (*i.e.* $He$) to the $LH_2$ (remove heat directly within absorber).



## Introduction (cont'd):

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:

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

- 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.

  $\Rightarrow$ *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 $LH_2$

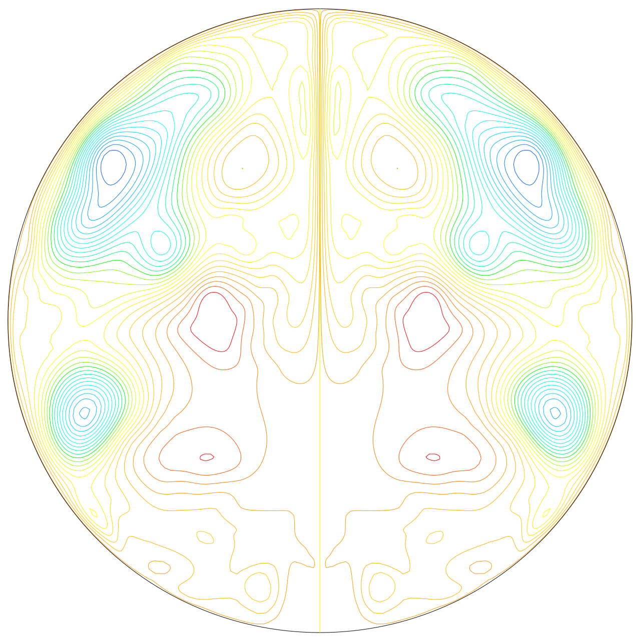
  $\mu =$ viscosity of $LH_2$

- Nusselt number:

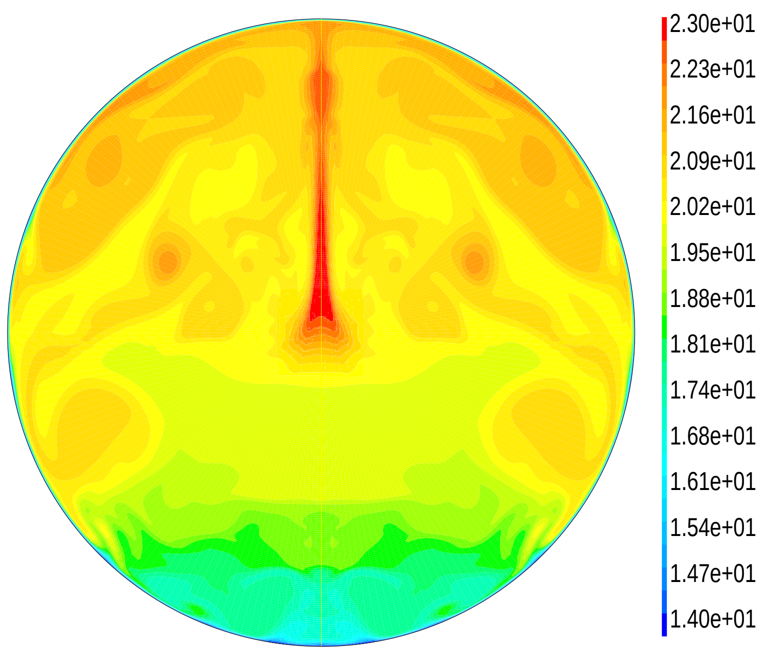
$$Nu = \frac{h_{LH_2} D}{k}$$

# **Sample CFD Results:** $Ra = 1.6 \times 10^{15}$

Streamlines:

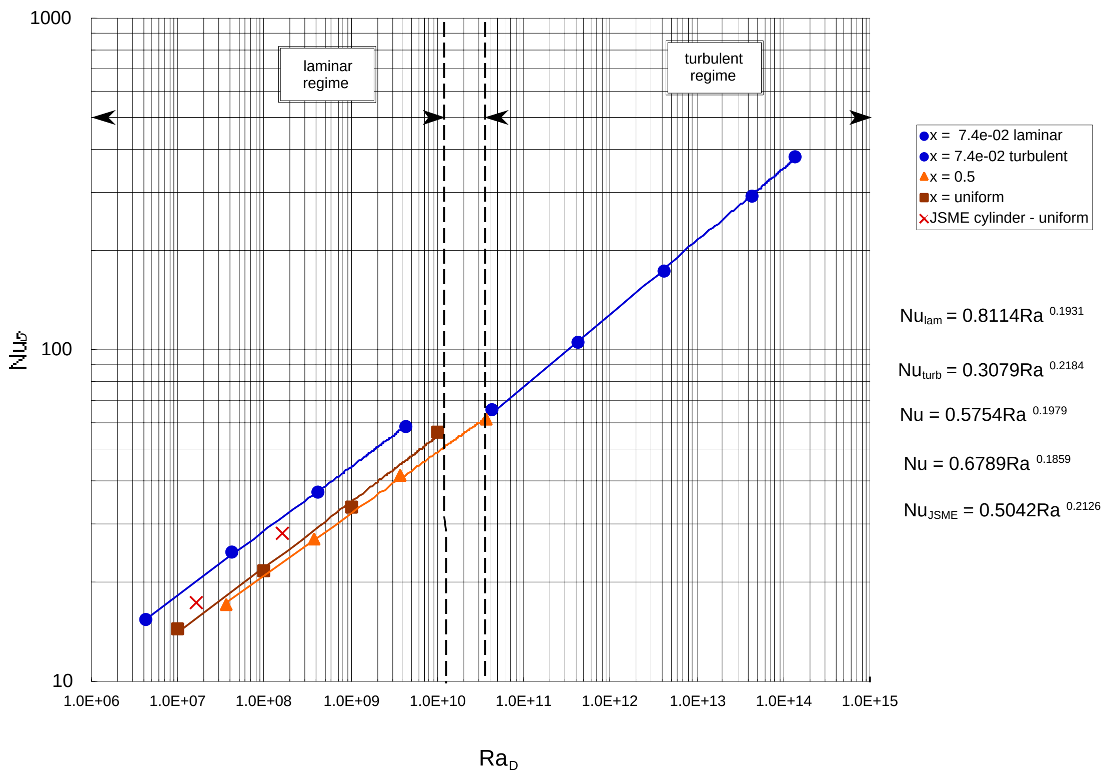


Temperature Distribution:

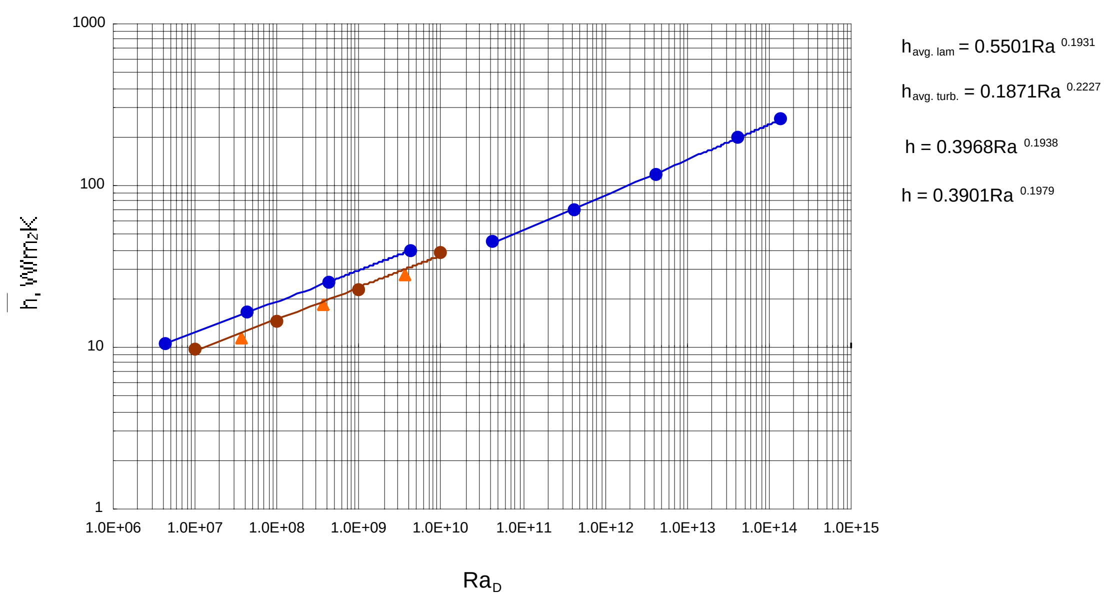


# CFD Results from FLUENT:

## Average Nusselt Number vs. Rayleigh Number:

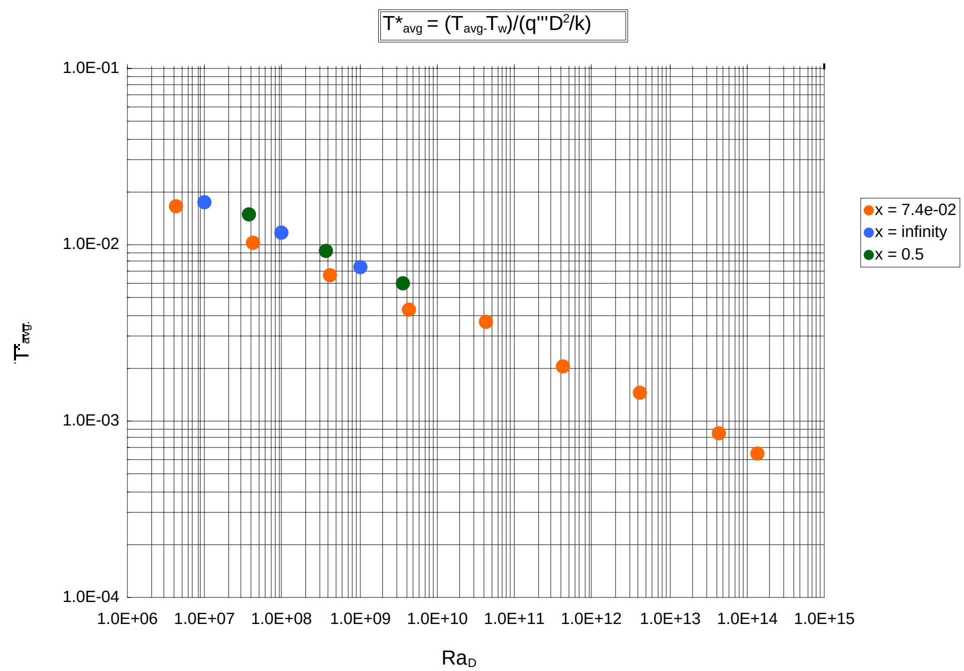



## Average Convective Heat Transfer Coefficient vs. Rayleigh Number:

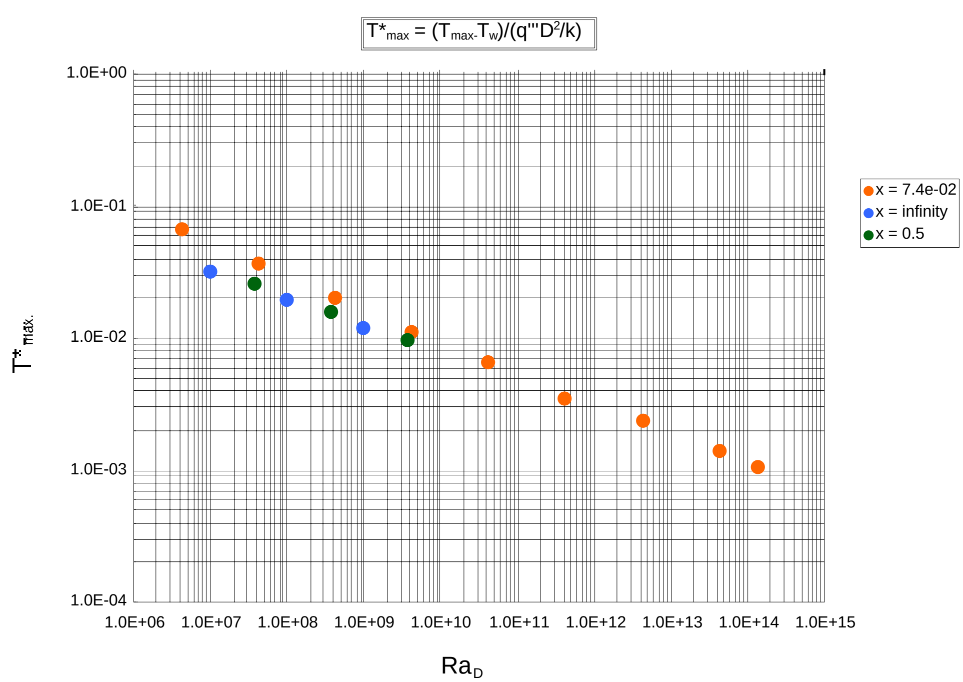


# CFD Results from FLUENT (cont'd):

## Non-Dimensional Average Temperature vs. Rayleigh Number:

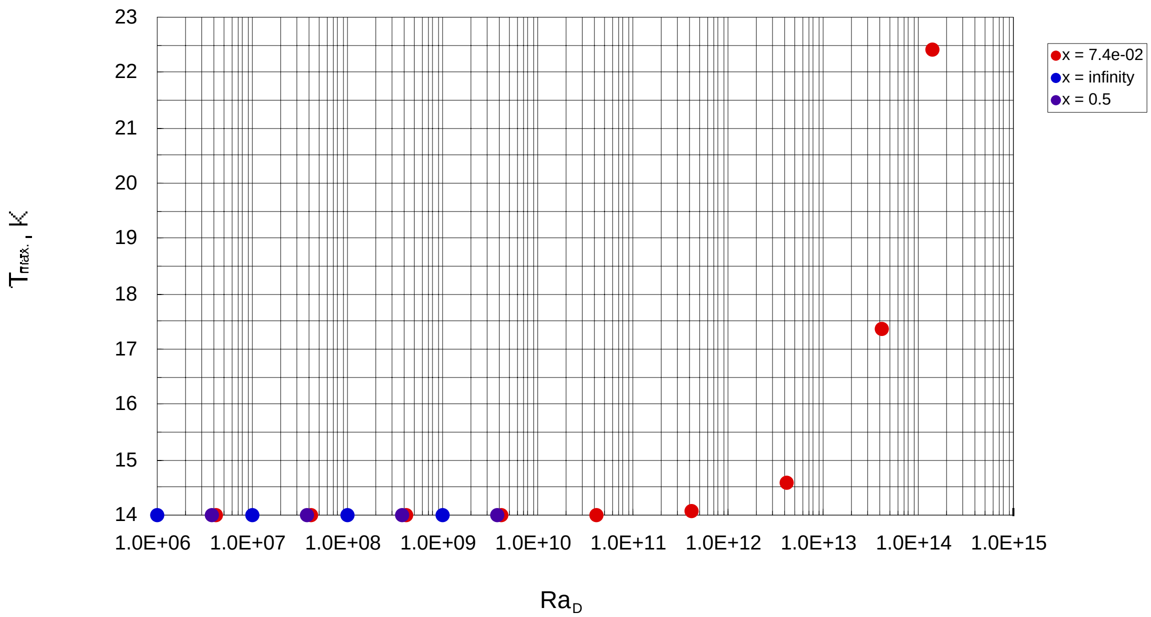



## Non-Dimensional Maximum Temperature vs. Rayleigh Number:



# CFD Results from FLUENT (cont'd):

Maximum Temperature vs. Rayleigh Number:



## Sample Heat Exchanger Analysis:

---

Absorber parameters (single-flip lattice):

$L = 0.3$ m

$R = 0.2$ m

$\dot{q} = 150$ W

$\Rightarrow Ra = 7.25 \times 10^{13}$

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

$T_i = 14$ K

$T_o = 15$ K

$T_{LH_2} = 18.5$ K (from CFD results)

$h_{He} = 1,580 \mathrm{W/m^2K}$

$h_{LH_2} = 210 \mathrm{W/m^2K}$ (from CFD results)

Results:

Required heat transfer area: $A = 0.20 \mathrm{m}^2$

Mass flow rate of $He$: $\dot{m}_{He} = 0.028$ 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.
  - $\Rightarrow$ What is the effect of pulsing the beam?
- The $LH_2$ flow in the absorber is at very high Rayleigh number ($Ra$):
  - $\Rightarrow$ Very small-scale turbulence (physically advantageous, but computationally challenging).
  - $\Rightarrow$ Highly unsteady fluid flow and heat transfer.
  - $\Rightarrow$ Need very small computational grids and time steps.
- Pushing the limits of FLUENT.
  - $\Rightarrow$ 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 advantagous.