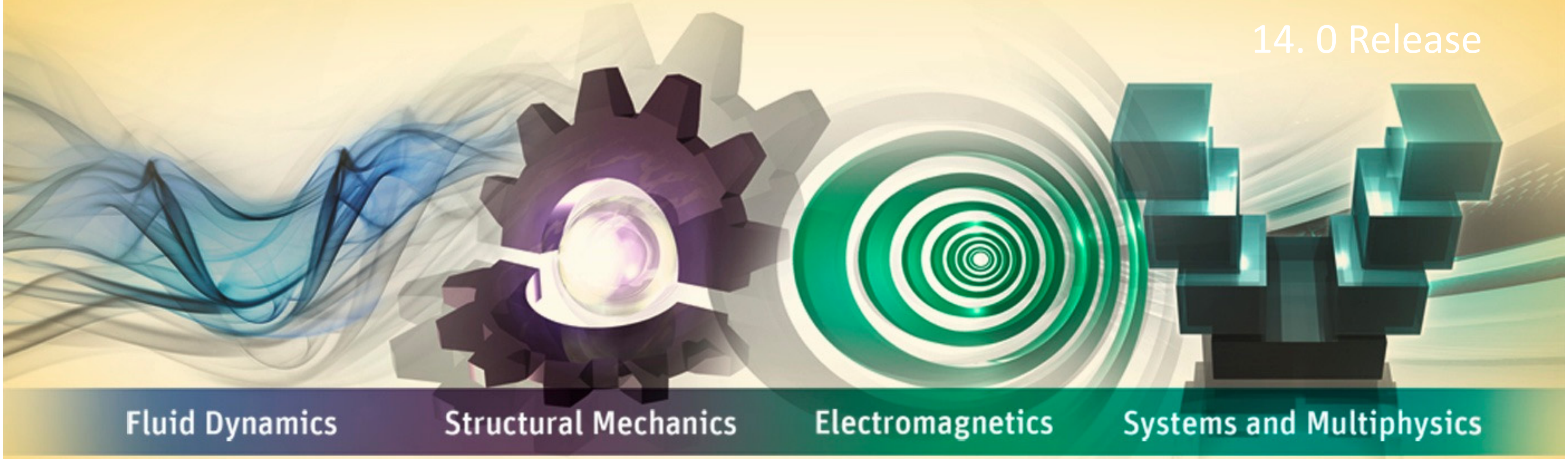
ANSYS

Realize Your Product Promise™

Lecture 7
Heat Transfer



Introduction to ANSYS FLUENT

Introduction

Lecture Theme:

Heat transfer has broad applications across all industries. All modes of heat transfer (conduction, convection – forced and natural, radiation, phase change) can be modeled in FLUENT and solution data can be used as input for one-way thermal FSI simulations.

Learning Aims:

You will learn:

- How to treat conduction, convection (forced and natural) and radiation in FLUENT
- How to set wall thermal boundary conditions
- How to export solution data for use in a thermal stress analysis (one-way FSI)

Learning Objectives:

You will be familiar with FLUENT's heat transfer modeling capabilities and be able to set up and solve problems involving all modes of heat transfer

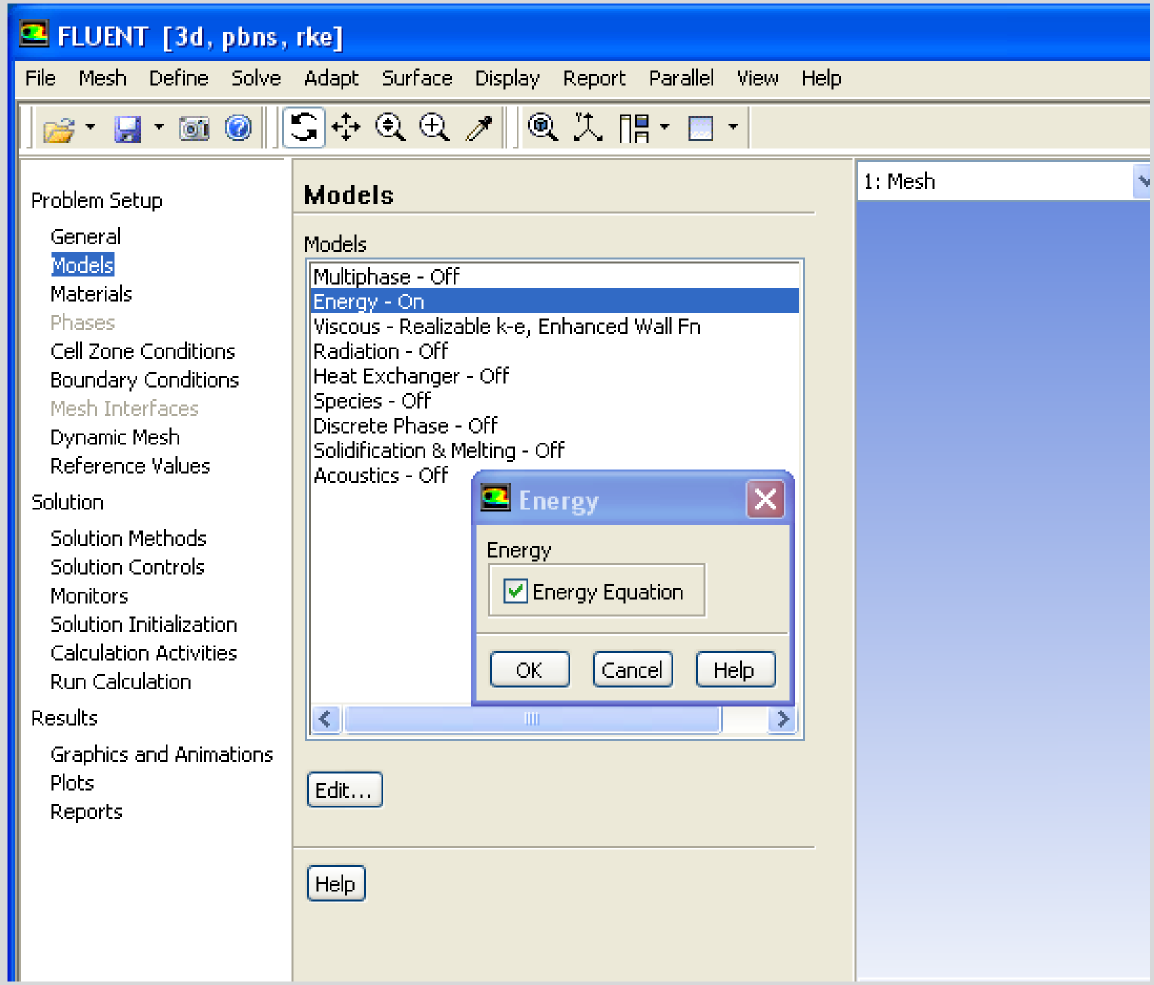
Heat Transfer Modeling in FLUENT

- **All modes of heat transfer can be taken into account in the CFD simulation :**
  - Conduction
  - Convection (forced and natural)
  - Fluid-solid conjugate heat transfer
  - Radiation
  - Interphase energy source (phase change)
  - Viscous dissipation
  - Species diffusion

Enabling Heat Transfer

- **To model heat transfer, the energy equation must be activated**

Define > Models > Energy = ON



Energy Equation – Introduction

- **Energy transport equation:**

$$\underbrace{\frac{\partial(\rho E)}{\partial t}}_{\text{Unsteady}} + \underbrace{\nabla\cdot[\mathbf{V}(\rho E + p)]}_{\text{Convection}} = \nabla\cdot\left[\underbrace{k_{\text{eff}}\nabla T}_{\text{Conduction}} - \underbrace{\sum_j h_j\,\mathbf{J}_j}_{\text{Species Diffusion}} + \underbrace{\tau_{\text{eff}}\cdot\mathbf{V}}_{\text{Viscous Dissipation}}\right] + \underbrace{S_h}_{\text{Enthalpy Source/Sink}}$$

  – Energy E per unit mass is defined as:

$$E = h - \frac{p}{\rho} + \frac{V^2}{2}$$

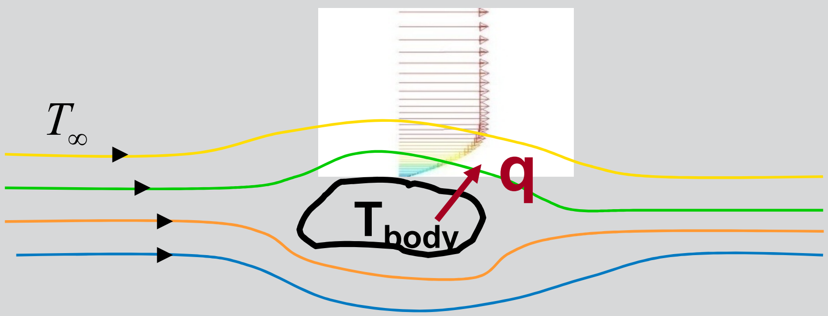
  – Pressure work and kinetic energy are always accounted for with compressible flows or when using the density-based solvers. For the pressure-based solver, they are omitted and can be added through a text command:

  – The TUI command **`define/models/energy?`** will give more options when enabling the energy equation.

Governing Equation : Convection

- **As a fluid moves, it carries heat with it ⇨ this is called convection**
  - Thus, heat transfer can be tightly coupled to the fluid flow solution
  - **Energy + Fluid flow equations activated ⇨ means Convection is computed**

- **Additionally:**
  - The rate of heat transfer is strongly dependent of fluid velocity
  - Fluid properties may vary significantly with temperature (e.g., air)
  - At walls, heat transfer coefficient is computed by the Turbulent boundary Layer Model



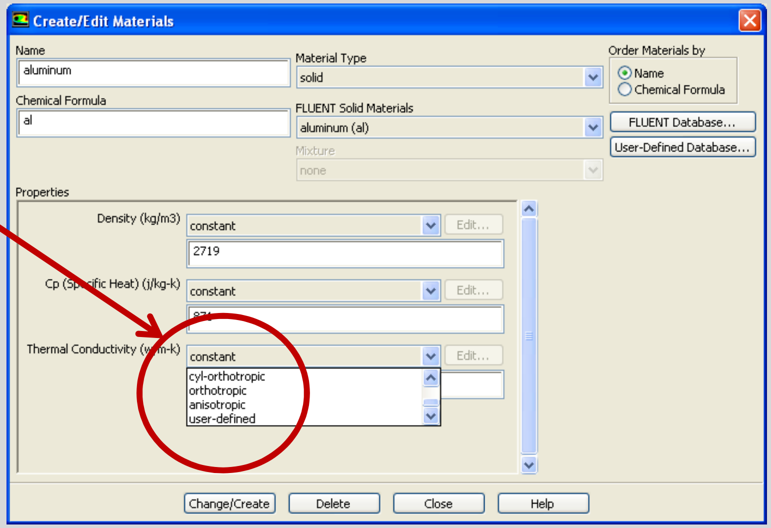
$$q = \bar{h}(T_{body} - T_\infty) = \bar{h}\Delta T$$

$\bar{h}$ = average heat transfer coefficient (W/m^2-K)

Intro. | Energy Equation | Wall BCs | Applications | 1-way Thermal FSI | Summary

Governing Equation : Conduction

- **Conduction heat transfer is governed by Fourier's Law**
- Fourier's law states that the heat transfer rate is directly proportional to the gradient of temperature
- Mathematically, $q_{\text{conduction}} = -\overset{\text{Thermal conductivity}}{k} \nabla T$
- The constant of proportionality is the **thermal conductivity (k)**
  - k may be a function of temperature, space, etc.
  - for isotropic materials, k is a constant value
  - for anisotropic materials, k is a matrix

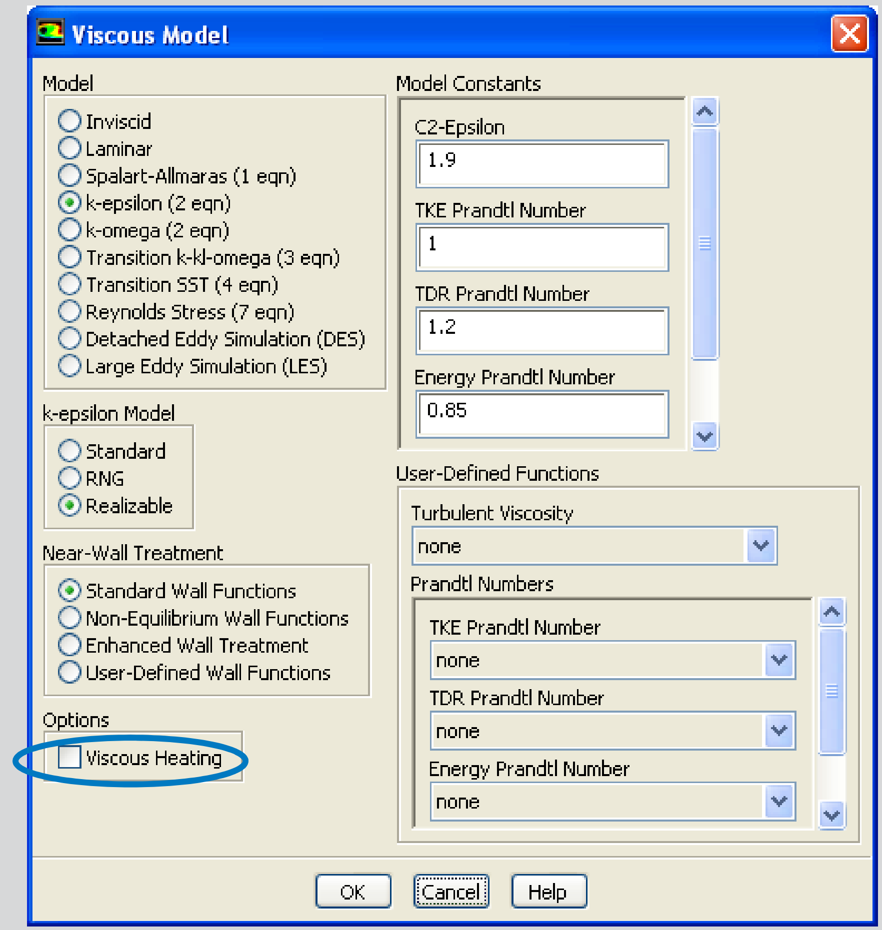


ANSYS

Governing Equation : Viscous Dissipation

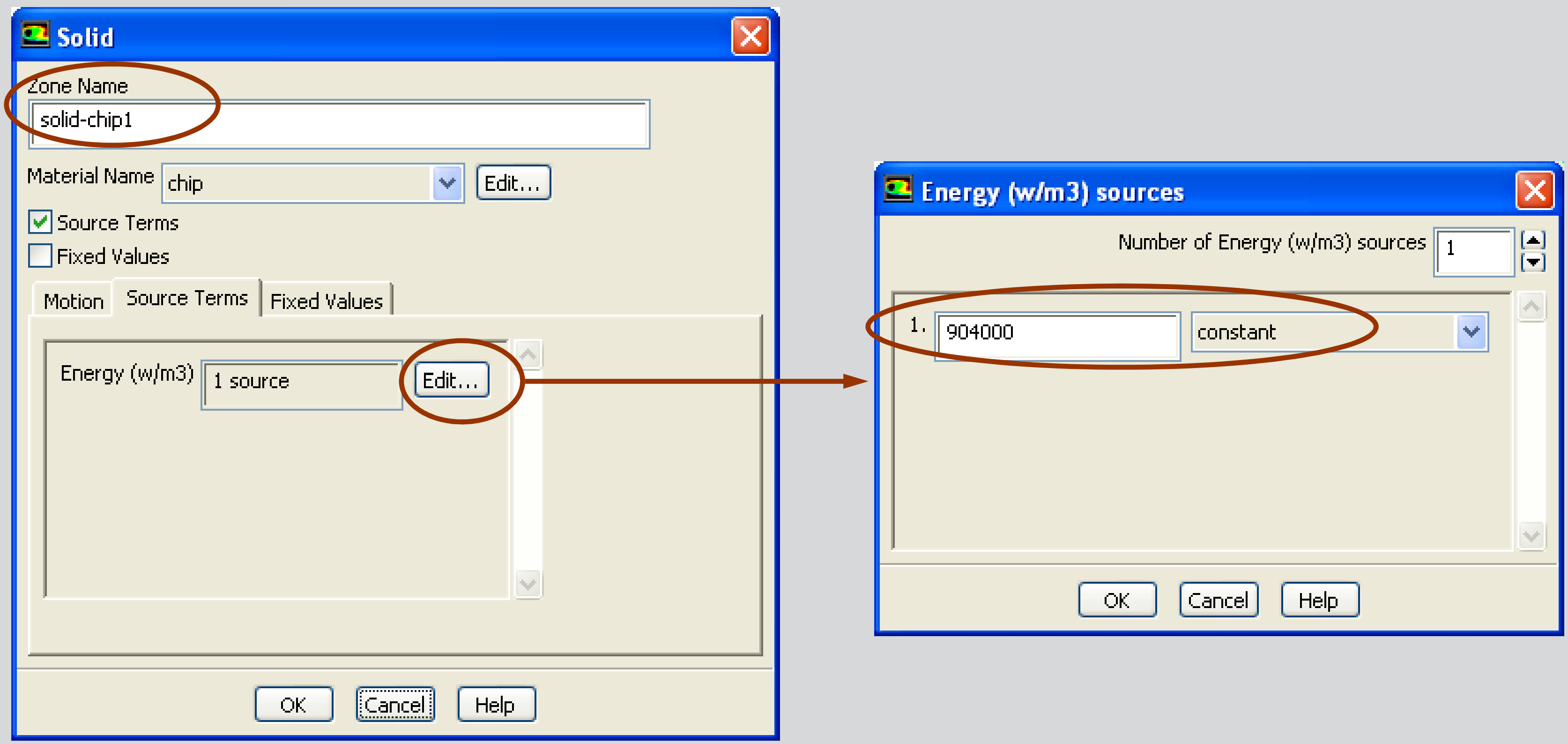
- **Energy source due to viscous dissipation:**
  - Also called viscous heating
    - Often negligible, especially in incompressible flow
  - Important when viscous shear in fluid is large (e.g., lubrication) and/or in high-velocity, compressible flows
  - Important when Brinkman number approaches or exceeds unity:

$$Br = \frac{\mu U_e^2}{k \Delta T}$$



External Energy Sources

- **Energy (heat) sources can be added to a fluid or solid zone to simulate additional external heat generation**
  - In this example an electronic ‘chip’ dissipates 2W of heat. Since its volume is small, the volumetric value entered (in W/m³) is high.



Thermal Wall Boundary Conditions

- **Five thermal conditions at Walls:**
  - **Heat Flux**
  - **Temperature**
  - **Convection** – simulates an external convection environment which is not modeled (user-prescribed heat transfer coefficient)

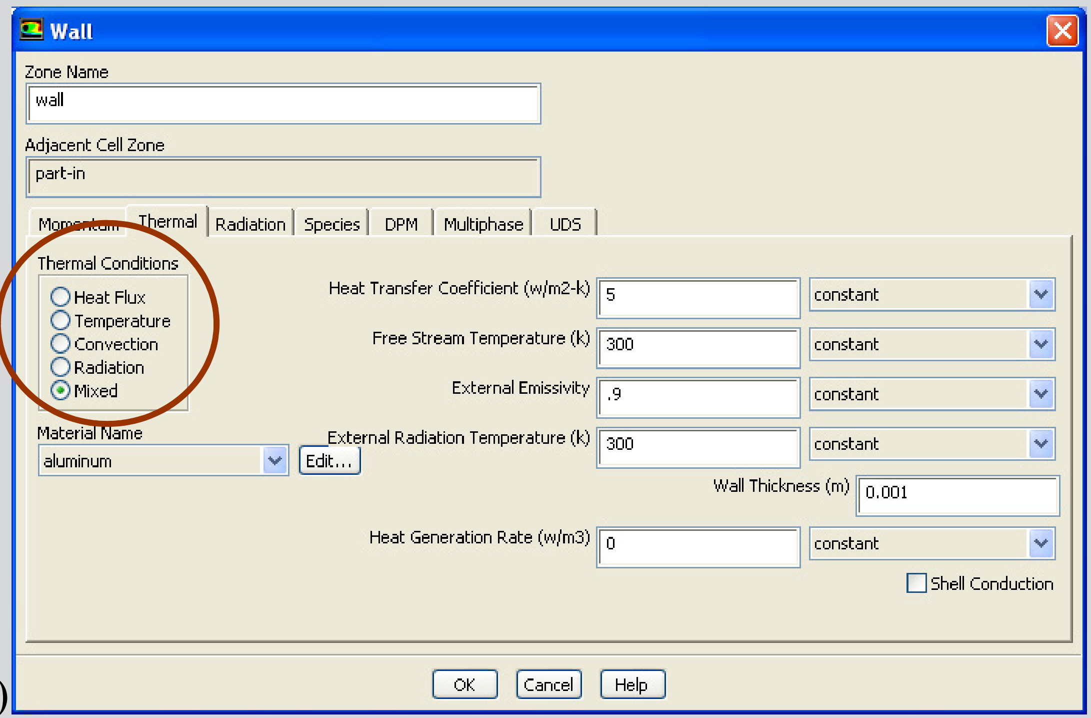
$$q_{conv} = h_{ext}(T_{ext} - T_w)$$

  - **Radiation** – simulates an external radiation environment which is not modeled (user-prescribed external emissivity and radiation temperature)

$$q_{rad} = \varepsilon_{ext}\,\sigma\,(T_\infty^4 - T_w^4)$$

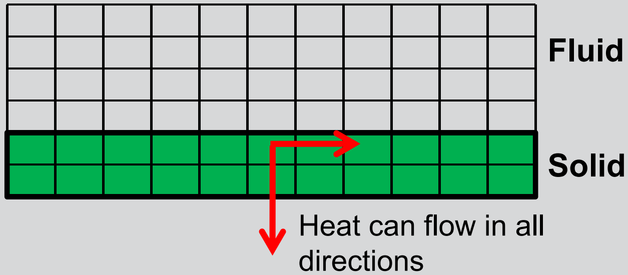
  - **Mixed** – Combination of Convection and Radiation boundary conditions

$$q_{mixed} = h_{ext}(T_{ext} - T_w) + \varepsilon_{ext}\,\sigma\,(T_\infty^4 - T_w^4)$$

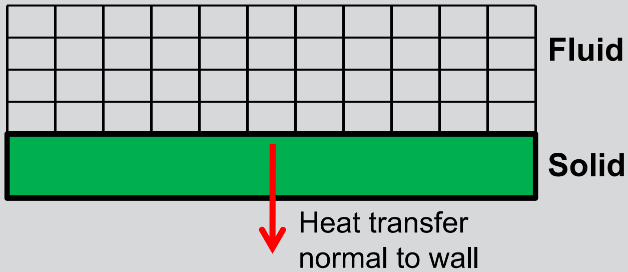


Modeling a Thin Wall

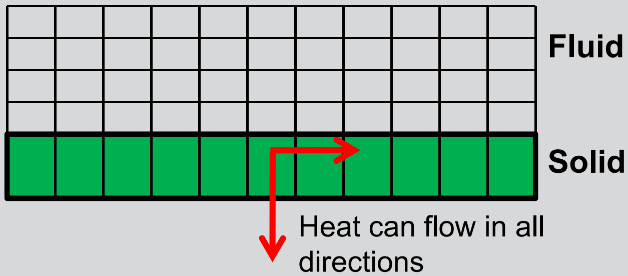
- It is often important to model the thermal effects of the wall bounding the fluid. However, it may not be necessary to mesh it.
  - Option 1
    - Mesh the wall in the pre-processor
    - Assign it as a solid cell zone
    - This is the most thorough approach



  - Option 2:
    - Just mesh the fluid region.
    - Specify a wall thickness.
    - Wall conduction will be accounted for.

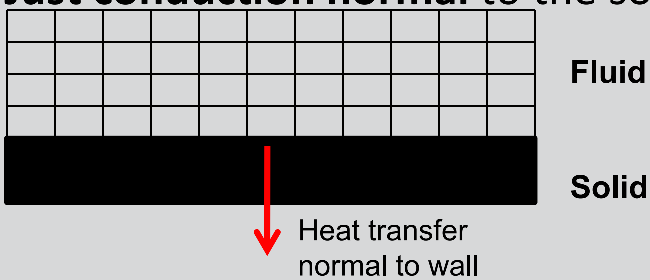


  - Option 3:
    - As option 2, but enable 'shell conduction'.
    - 1 layer of 'virtual cells' is created.

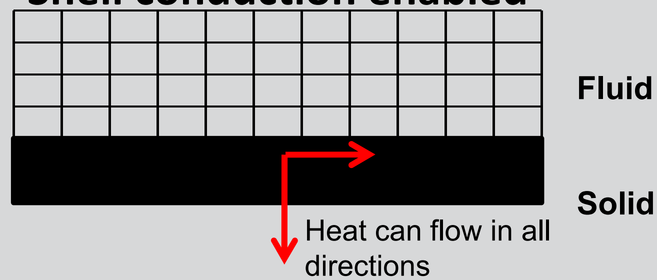


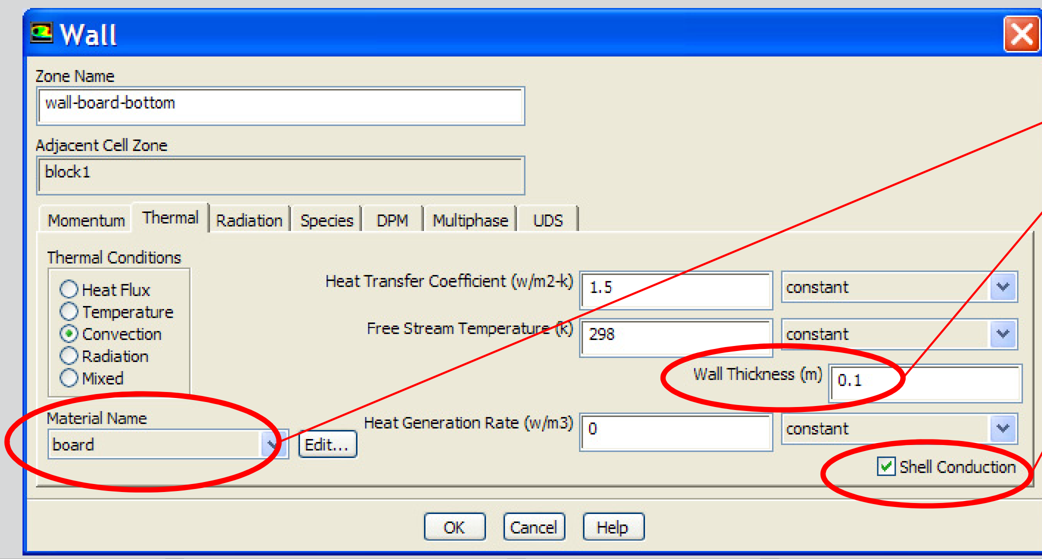
Modeling a Thin Wall

- For Options 2 and 3 on the last slide (in which it is not necessary to mesh the solid in the pre-processor), the setup panel looks like this:
- **Option 2:**
  - **Just conduction normal** to the solid



- **Option 3:**
  - **Shell conduction enabled**



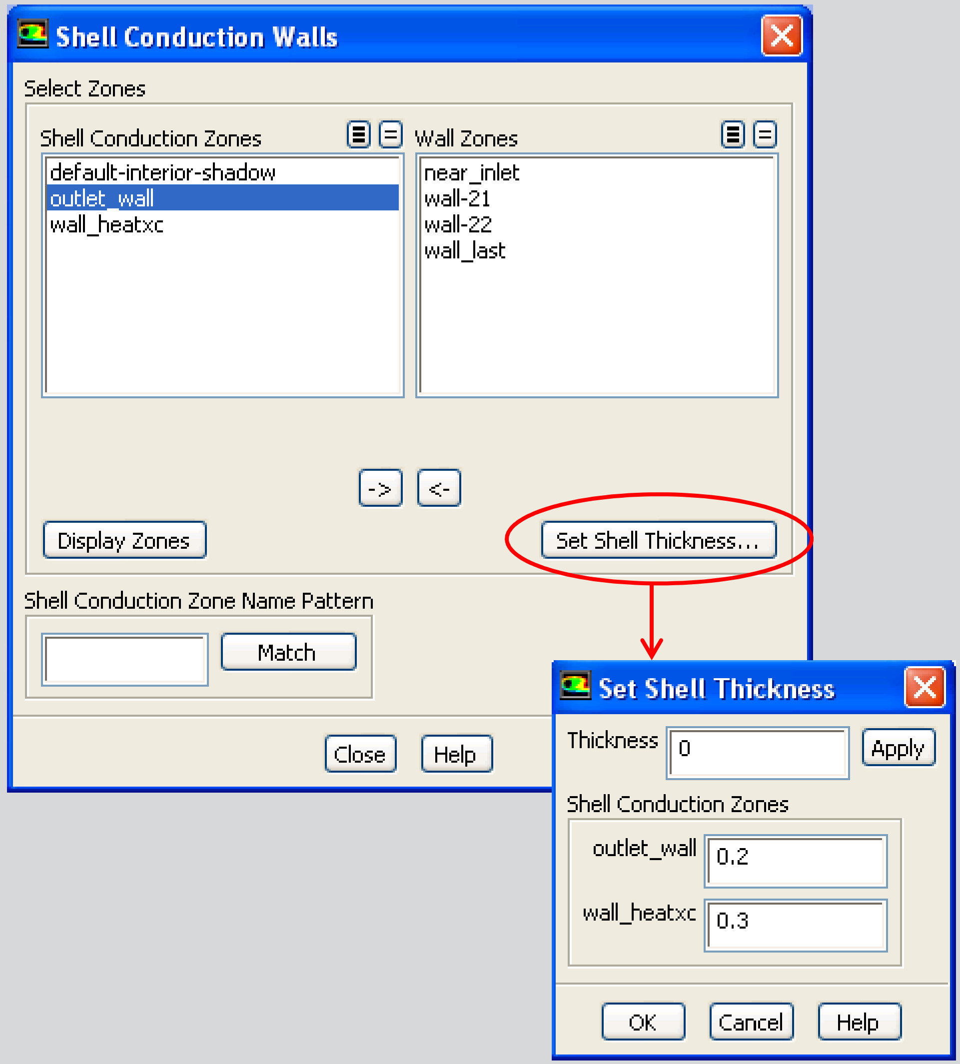


- In both cases, a material and wall thickness are enabled
- To add the virtual cells (Option 3), enable shell conduction.

  Note these virtual cells cannot be exported for FSI

Managing Shell Conduction Walls

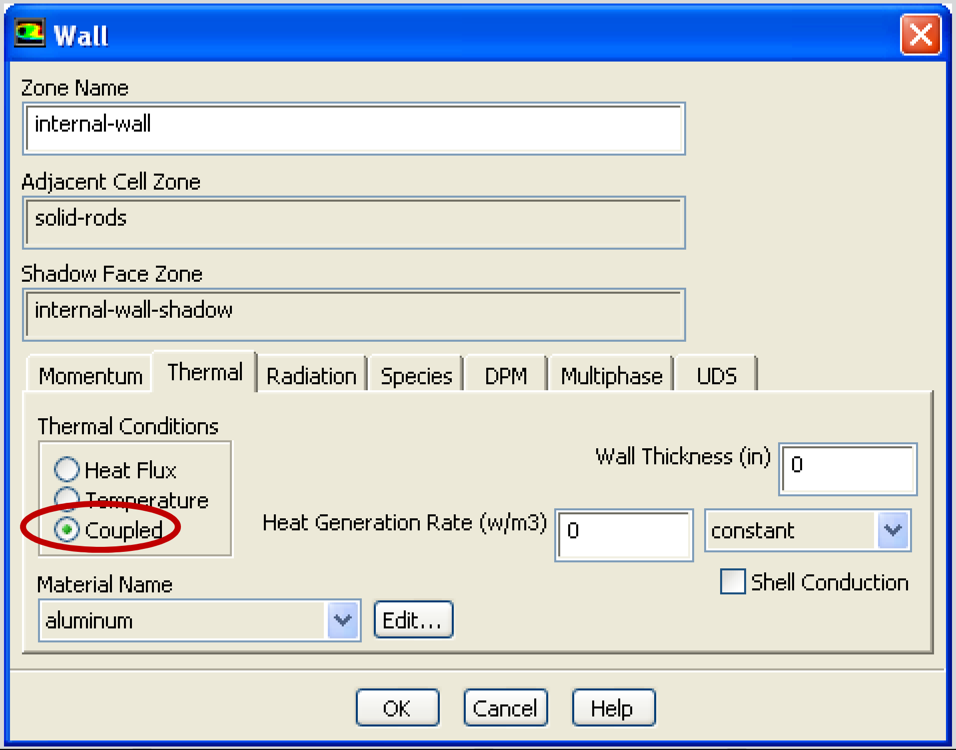
- The Shell Conduction Walls dialog box (Define > Shell Conduction Walls …) provides a means to manage all shell conduction boundaries
  - It is still possible to define shell conduction in the boundary conditions panel for individual walls.

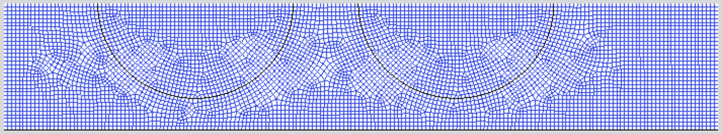


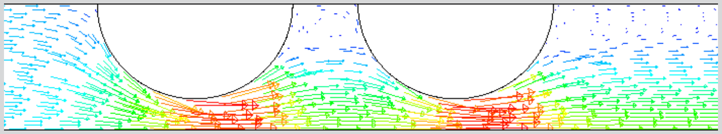
- In the Set Shell Thickness Panel
  - If you want to set the same thickness for all walls, enter the value in Thickness and click Apply
  - If you want walls to have different thicknesses, enter the value for each wall and click OK

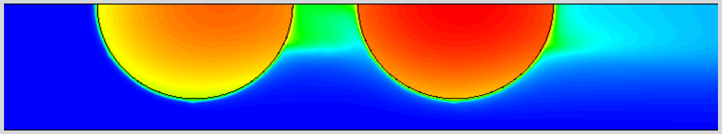
Conjugate Heat Transfer (CHT)

- **At Fluid/Solid or Fluid/Fluid interface, a wall / wall_shadow is created automatically by FLUENT while reading the mesh file**
  - By default energy is balanced automatically on the two sides of the walls
  - Possibility to uncouple and to specify different thermal conditions on each side




Grid

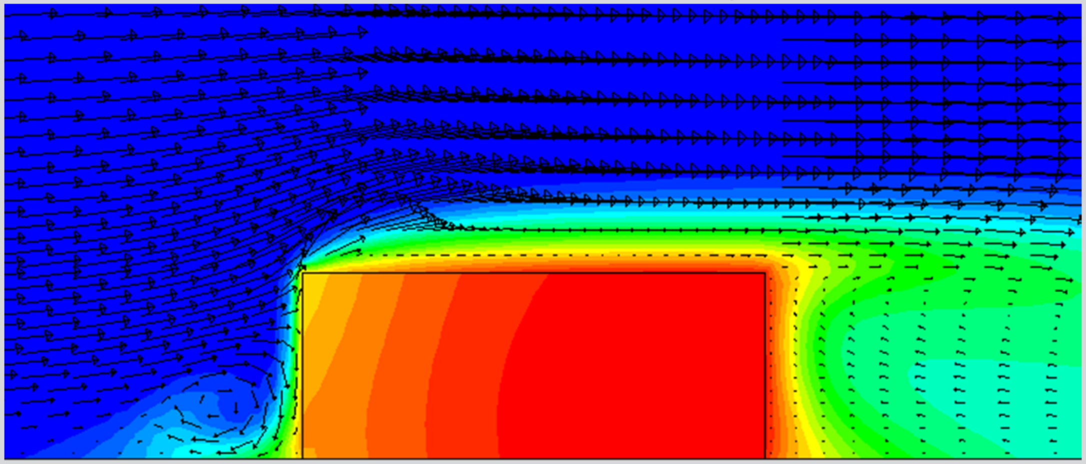

Velocity Vectors


Temperature Contours

Coolant Flow Past Heated Rods

Convection

- Convection heat transfer results from fluid motion.
  - Heat transfer rate can be closely coupled to the fluid flow solution.
  - The rate of heat transfer is strongly dependent on fluid velocity and fluid properties.
  - Fluid properties may vary significantly with temperature.
- There are three types of convection
  - Natural convection: fluid moves due to buoyancy effects
  - Boiling convection: body is hot enough to cause fluid phase change
  - Forced convection: flow is induced by some external means



Flow and heat transfer past a heated block

Example: When cold air flows past a warm body, it draws away warm air near the body and replaces it with cold air

Heat Transfer Coefficient

- In general, h is not constant but is usually a function of temperature gradient.
- There are three types of convection.
  - **Natural Convection** – Fluid moves due to buoyancy effects

$$h \propto \Delta T^{1/4}, \; h \propto \Delta T^{1/3}$$

(Laminar) (Turbulent)

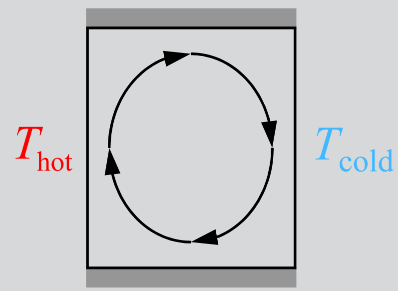
  - **Forced Convection** – Flow is induced by some external means.

$$h = f(\Delta T)$$

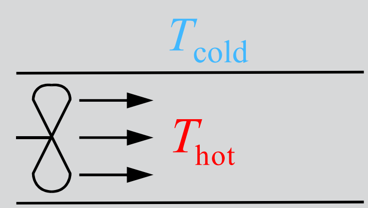
  - **Boiling Convection** – Body is hot enough to cause fluid phase change

$$h \propto \Delta T^2$$

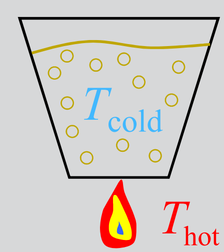
Typical values of h ($W/m^2 \cdot K$)



4 – 4,000



10 – 75,000



300 – 900,000

Natural Convection: Gravity-Reference Density

- **Momentum equation along the direction of gravity (z in this case)**

$$\frac{\partial(\rho W)}{\partial t}+\nabla\cdot(\rho \mathbf{U} W)=\mu\nabla^2 W-\frac{\partial P_{abs}}{\partial z}+\rho g$$

- **In FLUENT, a variable change is done for the pressure field as soon as gravity is enabled.**

$$P'=\left(P_{abs}-P_{operating}\right)-\rho_0 g z$$

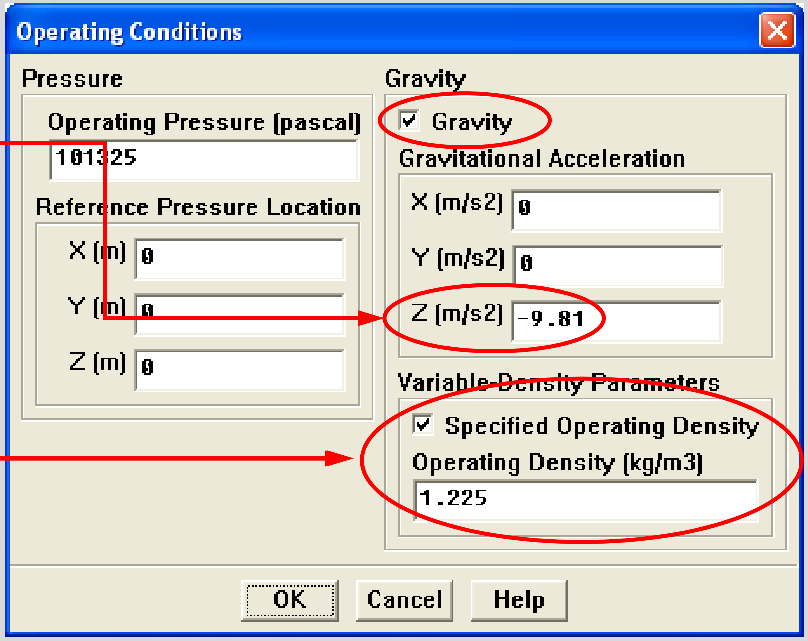
  - **Hydrostatic reference pressure head and operating pressure are removed from pressure field**

- **Momentum equation becomes**

$$\frac{\partial(\rho W)}{\partial t}+\nabla\cdot(\rho \mathbf{U} W)=\mu\nabla^2 W-\frac{\partial P'}{\partial z}+(\rho-\rho_0)g$$

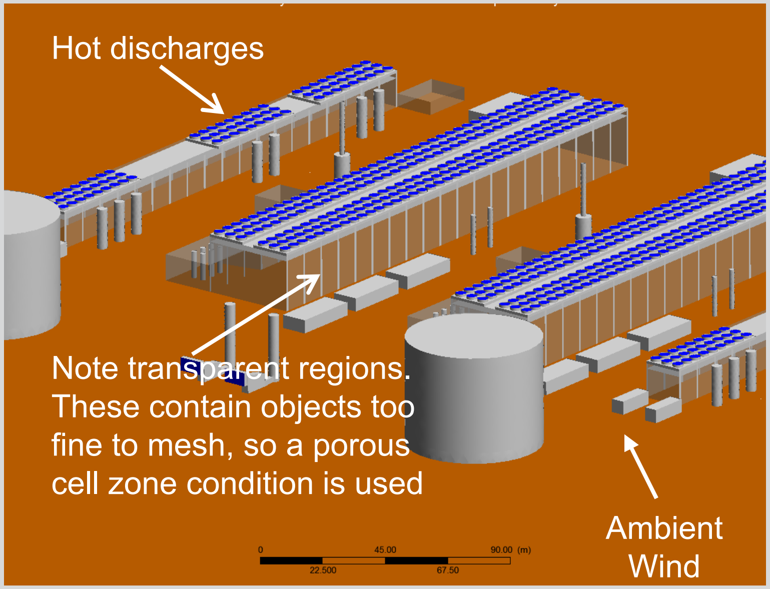
where P' is the static gauge pressure used by FLUENT for boundary conditions and post-processing.

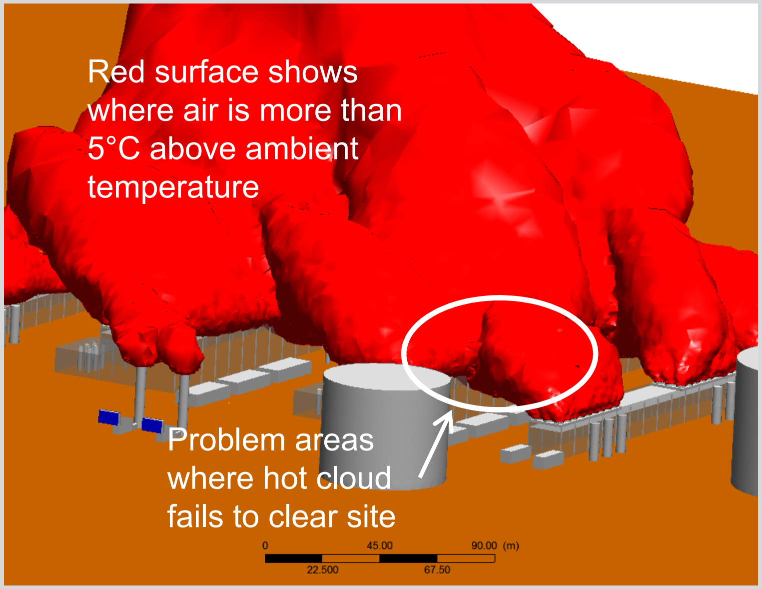
- **This pressure transformation avoids round off error and simplifies the setup of pressure boundary conditions**



Natural Convection in an Open Domain (1/2)

- Many heat transfer problems (especially for ventilation problems) include the effects of natural convection.
- As the fluid warms, some regions become warmer than others, and therefore rise through the action of buoyancy.
- This example shows a generic LNG liquefaction site, several hundred metres across. Large amounts of waste heat are dissipated by the air coolers (rows of blue circles). The aim of the CFD simulation is to assess whether this hot air rises cleanly away from the site.



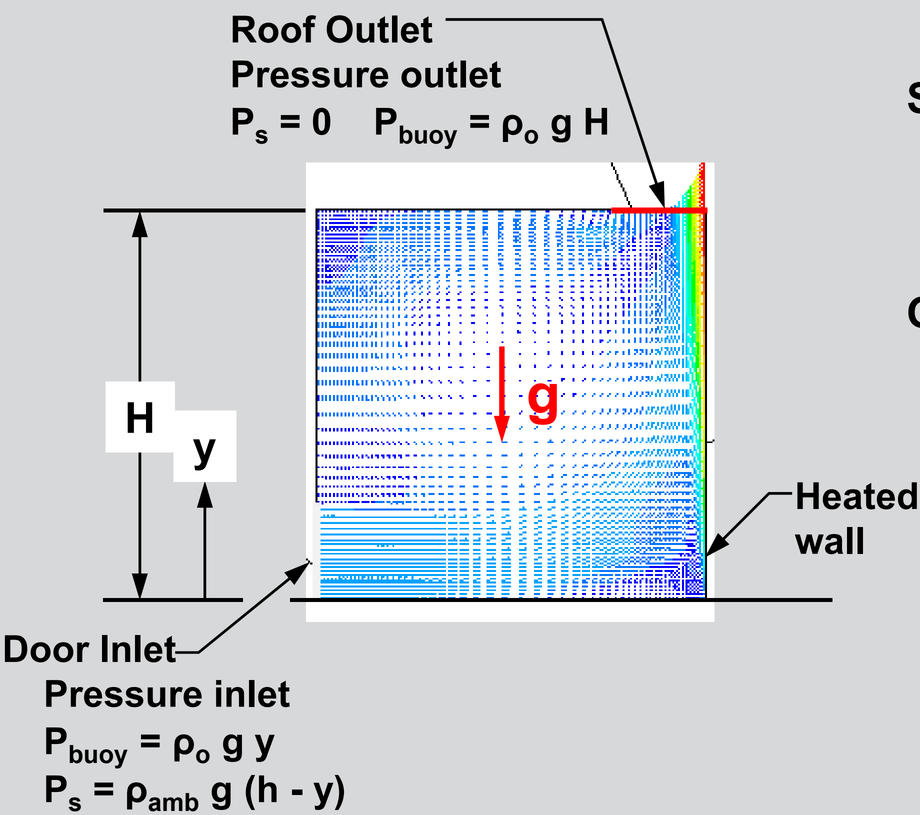


Natural Convection in an Open Domain (2/2)

- The underlying term for the buoyant force in the momentum equations is
  $(\rho - \rho_0)g$ where ρ is the local density and ρ_o a reference density
- The reference density, ρ_o is set on the 'Operating Conditions' panel.
  - Strongly recommended: **ρ_o = Ambient density**
- The pressure profile at the boundaries is dependant on the value of ρ_o, because the value entered in the boundary conditions panel corresponds to the modified pressure, P' (= P – $\rho_o\, g\, z$)
- If the computational domain contains pressure inlets and outlets connected to the same external environment, ρ_o should be set equal to the ambient density and a constant pressure of 0 Pa specified for inlets and outlets.
  - See next slide

Selecting the Reference Density

- Example – Door and roof vents on a building with heated wall
  - The roof static pressure is set to 0 while the door static pressure must be given a hydrostatic head profile based on the height of the building.



So, the correct pressure BCs are:

$$\left(P_s'\right)_{top} = \rho_0\, g\, H$$

$$\left(P_s'\right)_{bot} = \rho_0\, g\, y + \rho_{amb}\, g\left(H - y\right)$$

Or, equivalently,

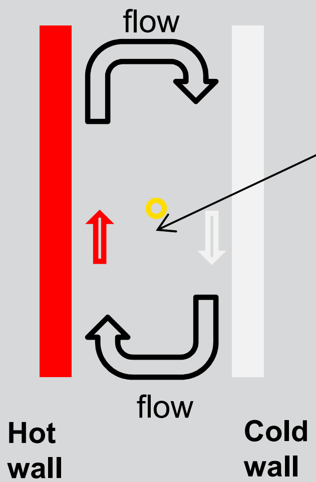
$$\left(P_s'\right)_{top} = 0$$

$$\left(P_s'\right)_{bot} = \left(\rho_{amb} - \rho_0\right) g \left(H - y\right)$$

Note: In this case, if you can set the reference density equal to the external ambient density then the hydrostatic component can be ignored:

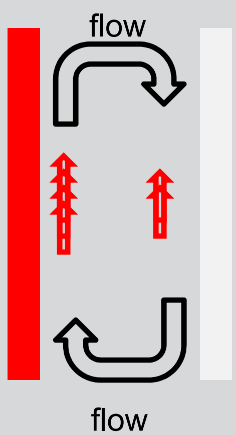
Natural Convection in a Cavity

– The choice of ρ_o could be arbitrary in a cavity but has an impact on convergence.



Well posed simulation

- ρ_o set to a value in the middle of the cavity
- Near the hot wall, the buoyant force term will be upwards, whilst at the cold wall this term will be downwards.
- This will encourage the correct flow field from the start, and should converge easily.

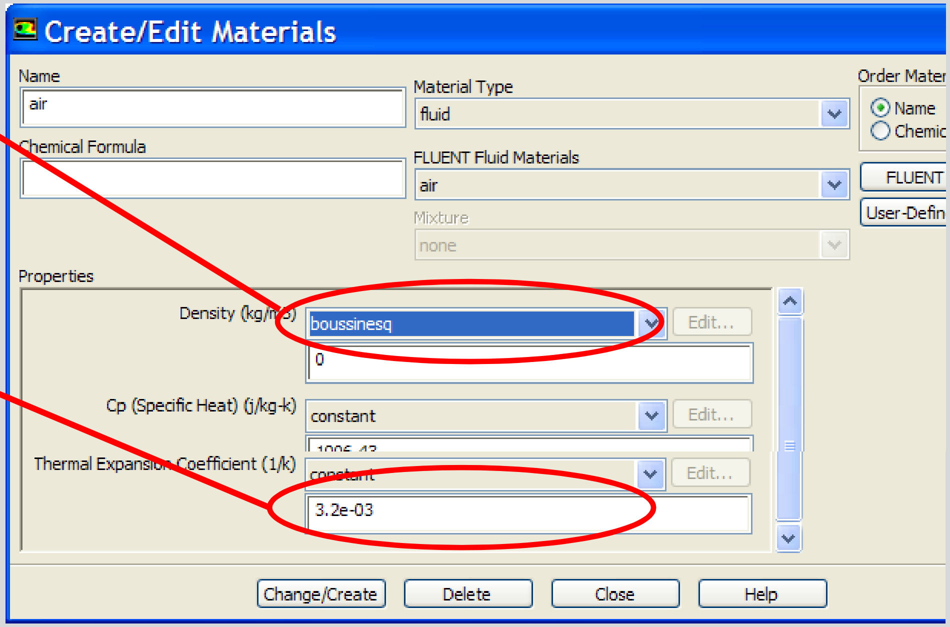


Badly posed simulation

- ρ_o set too high (equivalent to a temperature colder than at the cold wall)
- The source terms therefore produce:
  - A very high upwards force at the hot wall
  - A lesser, but still upwards, force at the cold wall.
- When converged (if it ever does!) the flow field should be the same as the top case, but convergence will be difficult.

Natural Convection – the Boussinesq Model

- A simplification can be made in some cases where the variation in density is small.
- Recall the solver must compute velocity, temperature, and pressure
- Rather than introducing another variable, density, (which adds an extra unknown, thus intensifying computational effort)



- Instead for fluid 'density' select Boussinesq.
  - Remember to enter correct value for density (do not leave the value as 0)
- And define a thermal expansion coefficient β,

  (value in standard engineering texts)

  (use slider bar to scroll to bottom of list)
- Buoyant force is computed from

$$(\rho - \rho_0)g = -\rho_0 \beta (T - T_0) g$$

- The same comments as on the previous slides (for setting the reference density ρ_o) apply here for setting the reference temperature T_o - set in the Operating Conditions panel.

Natural Convection- Tips and Tricks

- Beware of reference density:
  - Average density for a cavity (Tref= median temperature for Boussinesq model)
  - Ambient density for problems with pressure inlets and outlets (Tref= ambient temperature for Boussinesq model)

- Use Presto and Body force weighted discretization for pressure

- Requirement: $Y^+=1$ for turbulent natural convection boundary layer

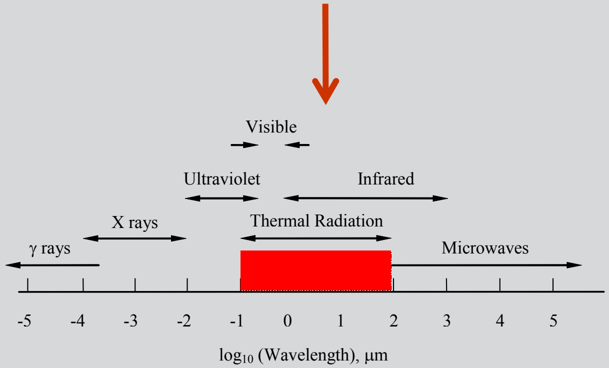
- Use pressure based pseudo transient approach for High Rayleigh number (turbulent flow)

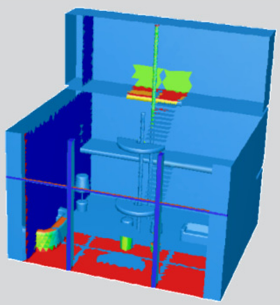
$$\Delta t \approx \sqrt{\frac{L}{g\beta\Delta T}}$$

- Use k-epsilon for buoyant stratified (Buoyancy effects for k-omega can be added by UDF and will be added as a standard feature in the future)

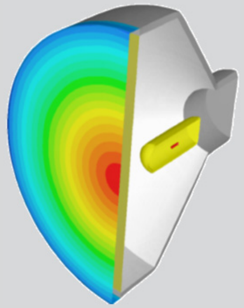
Radiation

- **Radiative heat transfer is a mode of energy transfer where the energy is transported via *electromagnetic waves***
  - **Thermal radiation covers the portion of the electromagnetic spectrum from 0.1 to 100 μm.**

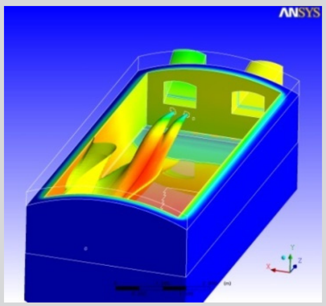




Solar load (HVAC)



Headlight



Glass furnace

- **For semi-transparent bodies (e.g., glass, combustion product gases), radiation is a *volumetric* phenomenon since emissions can escape from within bodies.**
- **For opaque bodies, radiation is essentially a *surface* phenomena since nearly all internal emissions are absorbed within the body.**

When to Include Radiation?

- **Radiation effects should be accounted for if**

$$q_{\text{rad}} = \sigma\,\varepsilon\left(T_{\max}^{4} - T_{\min}^{4}\right)$$

Stefan-Boltzmann constant
5.6704×10^{-8} W/(m^2·K^4)

is of the same order or magnitude than the convective and conductive heat transfer rates. This is usually true at high temperatures but can also be true at lower temperatures, depending on the application

- **Estimate the magnitude of conduction or convection heat transfer in the system as**

$$q_{conv} = h\left(T_{\text{wall}} - T_{\text{bulk}}\right)$$

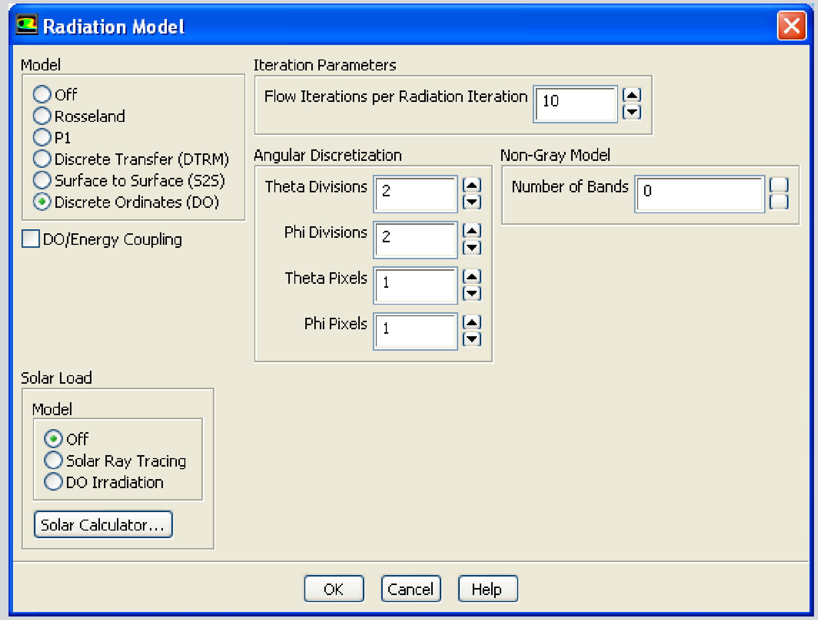
- **Compare q_{rad} with q_{conv}.**

Optical Thickness and Radiation Modeling

- **The optical thickness should be determined before choosing a radiation model**

> ***Optical Thickness ≡ $(a+\sigma_s)L$***
> ***a= absorption coefficient***
> ***σ_s=scattering coefficient (often=0)***
> ***L= mean beam length***

  - *a* Absorption Coefficient (m^{-1})
    (Note: ≠*Absorptivity* of a *Surface*)
  - *L Mean beam length (m)*
    (a typical distance between 2 opposing walls)



- **Optically thin means that the fluid is transparent to the radiation at wavelengths where the heat transfer occurs**
  – The radiation only interacts with the boundaries of the domain

- **Optically thick/dense means that the fluid absorbs and re-emits the radiation**

Choosing a Radiation Model

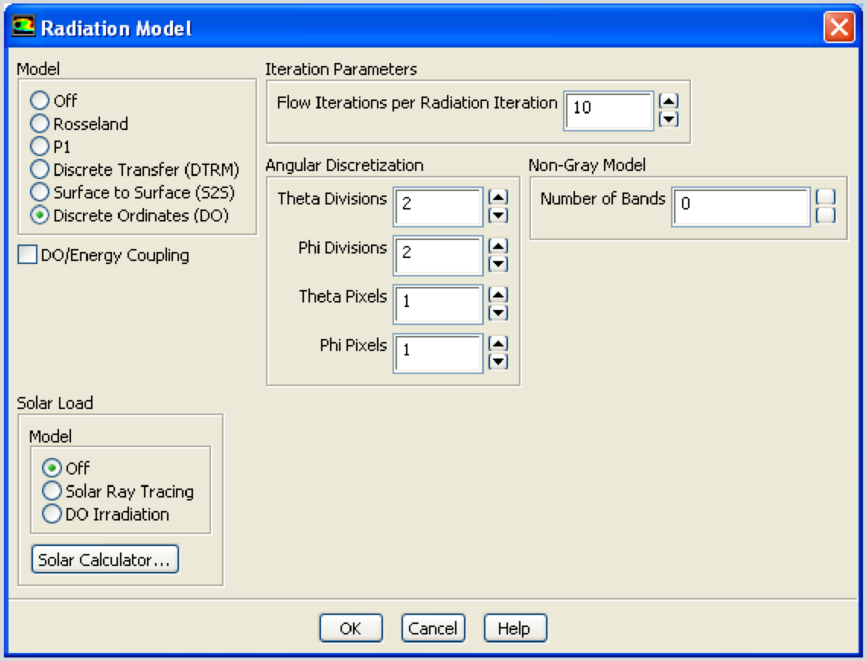
- **The radiation model selected must be appropriate for the optical thickness of the system being simulated**

Available Model	Optical Thickness
Surface to surface model (S2S)	0
Solar load model	0 (except window panes)
Rosseland	> 5
P-1	> 1
Discrete ordinates model (DO)	All
Discrete Transfer Method (DTRM)	All

- **In terms of accuracy, DO and DTRM are most accurate**
  - **S2S is accurate for optical thickness = 0**

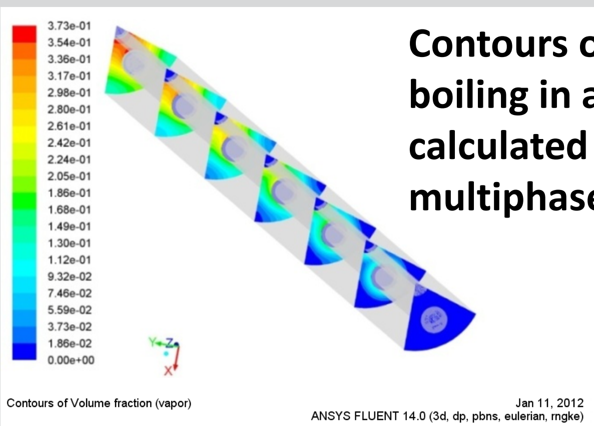
Additional Factors in Radiation Modeling

- Additional guidelines for radiation model selection:
  - **Scattering**
    - Scattering is accounted for only with P1 and DO.
  - **Particulate effects**
    - P1 and DOM account for radiation exchange between gas and particulates.
  - **Localized heat sources**
    - S2S is the best.
    - DTRM/DOM with a sufficiently large number of rays/ ordinates is most appropriate for domain with absorbing media.

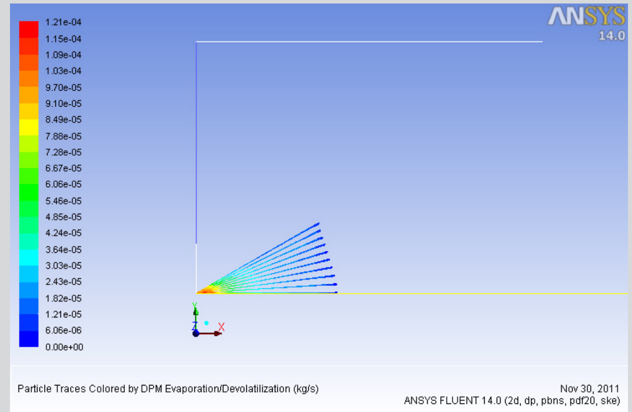


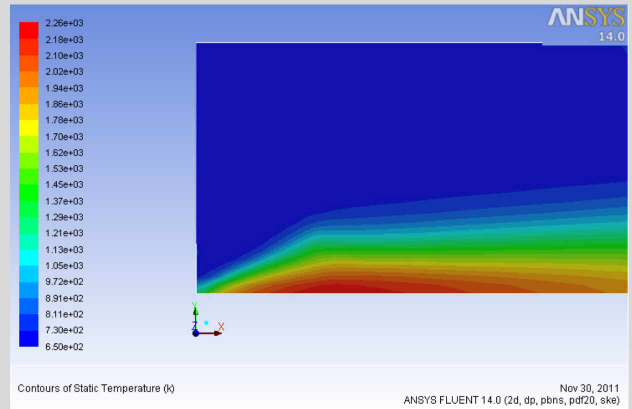
Phase Change

- **Heat released or absorbed when matter changes state**
- **There are many different forms of phase change**
  - **Condensation**
  - **Evaporation**
  - **Boiling**
  - **Melting/Solidification**
- **Multiphase models and/or UDFs are needed to properly model these phenomena**



Contours of vapor volume fraction for boiling in a nuclear fuel assembly calculated with the Eulerian multiphase model

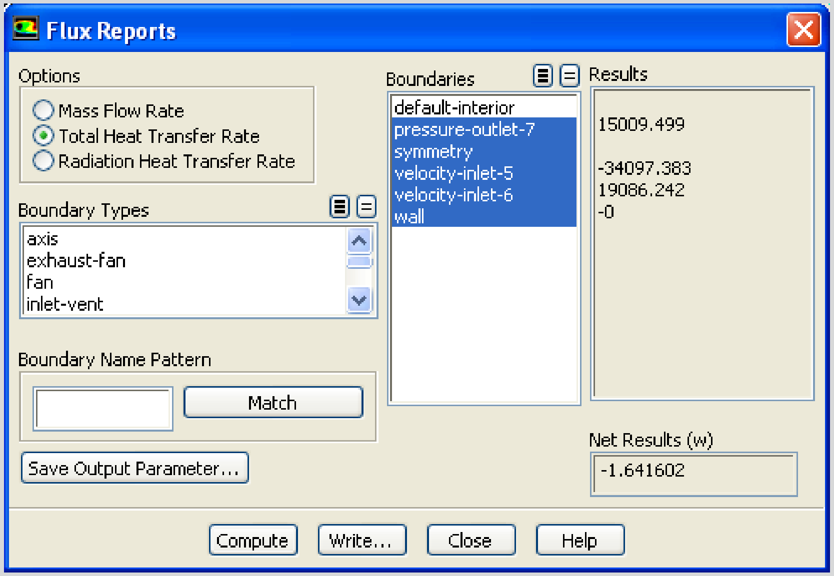




Tracks from evaporating liquid pentane droplets and temperature contours for pentane combustion with the non-premixed combustion model

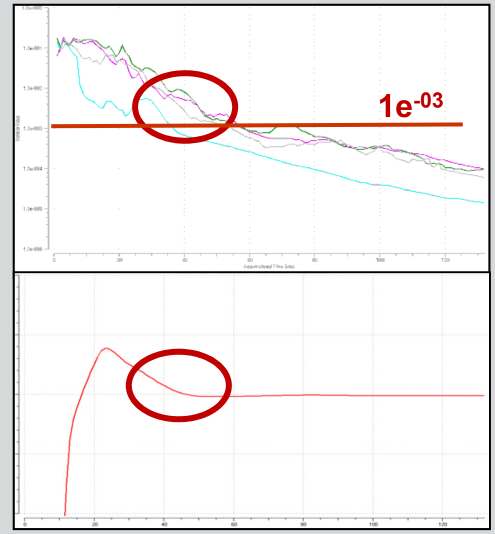
Post-Processing Heat Transfer

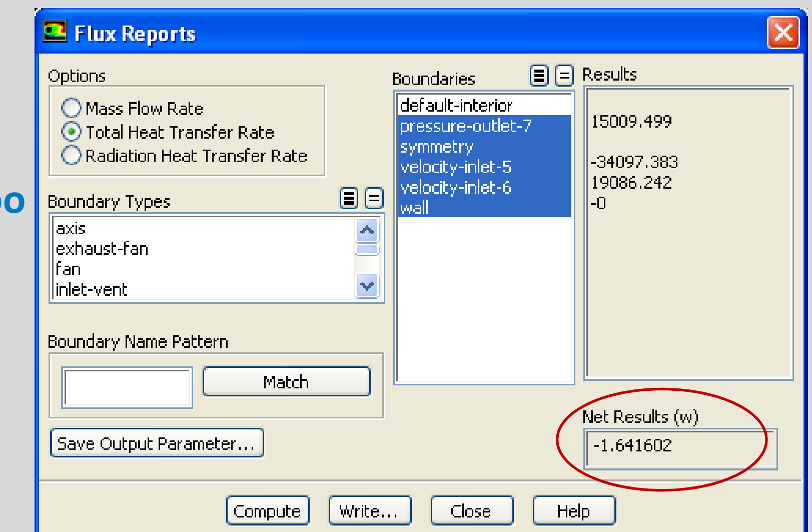
- **Heat flux report:**
  - **« Total Heat Transfer Rate »: both convective and radiative flux are computed**
    - Net heat balance should be 0 once converged
      - or opposite to all the external energy sources (UDF or constant sources, DPM)
  - **« Radiation Heat Transfer Rate », only radiative net flux is computed**
    - The sum of this flux is generally different from 0. It can represent the amount of energy that is absorbed by the media



Solution Convergence

- **When solving heat transfer problems, the double precision solver is usually needed**
- Make sure that you have allowed sufficient solution iterations for the heat imbalances to become very small, particularly when solid zones are included
- Sometimes residuals reach the convergence criteria before global imbalances trend towards zero
  - **Check the imbalance and continue iterating if it is too large**

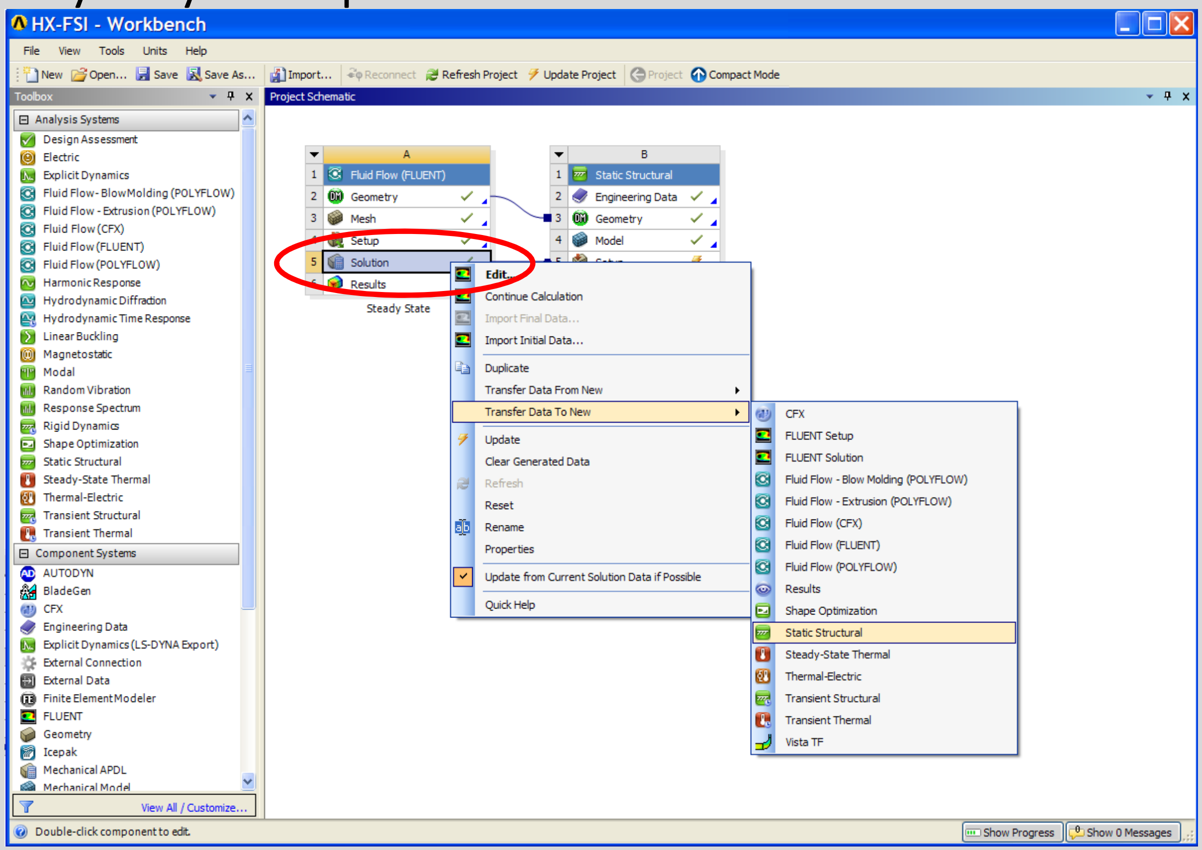




ANSYS

Performing a 1-way Thermal FSI Simulation

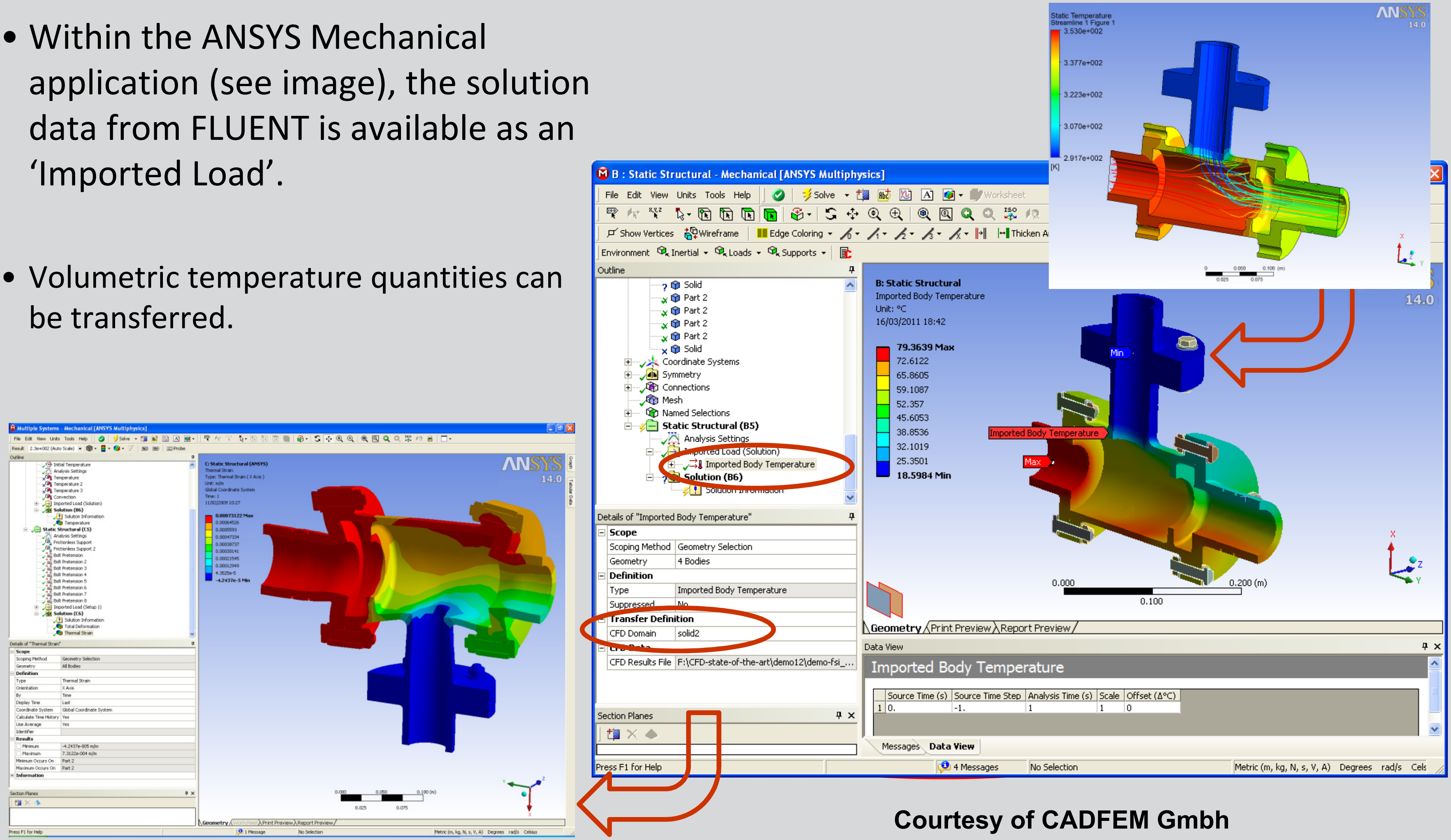
- The results of the FLUENT model can be transferred to another FE code for further analysis (for example to compute thermal stresses).
- Using Workbench, it is very easy to map the FLUENT data over to an ANSYS Mechanical simulation.
- Just right click on the "Solution" cell, then "Transfer Data To New Static Structural"



Performing a 1-way Thermal FSI Simulation

- Within the ANSYS Mechanical application (see image), the solution data from FLUENT is available as an 'Imported Load'.

- Volumetric temperature quantities can be transferred.



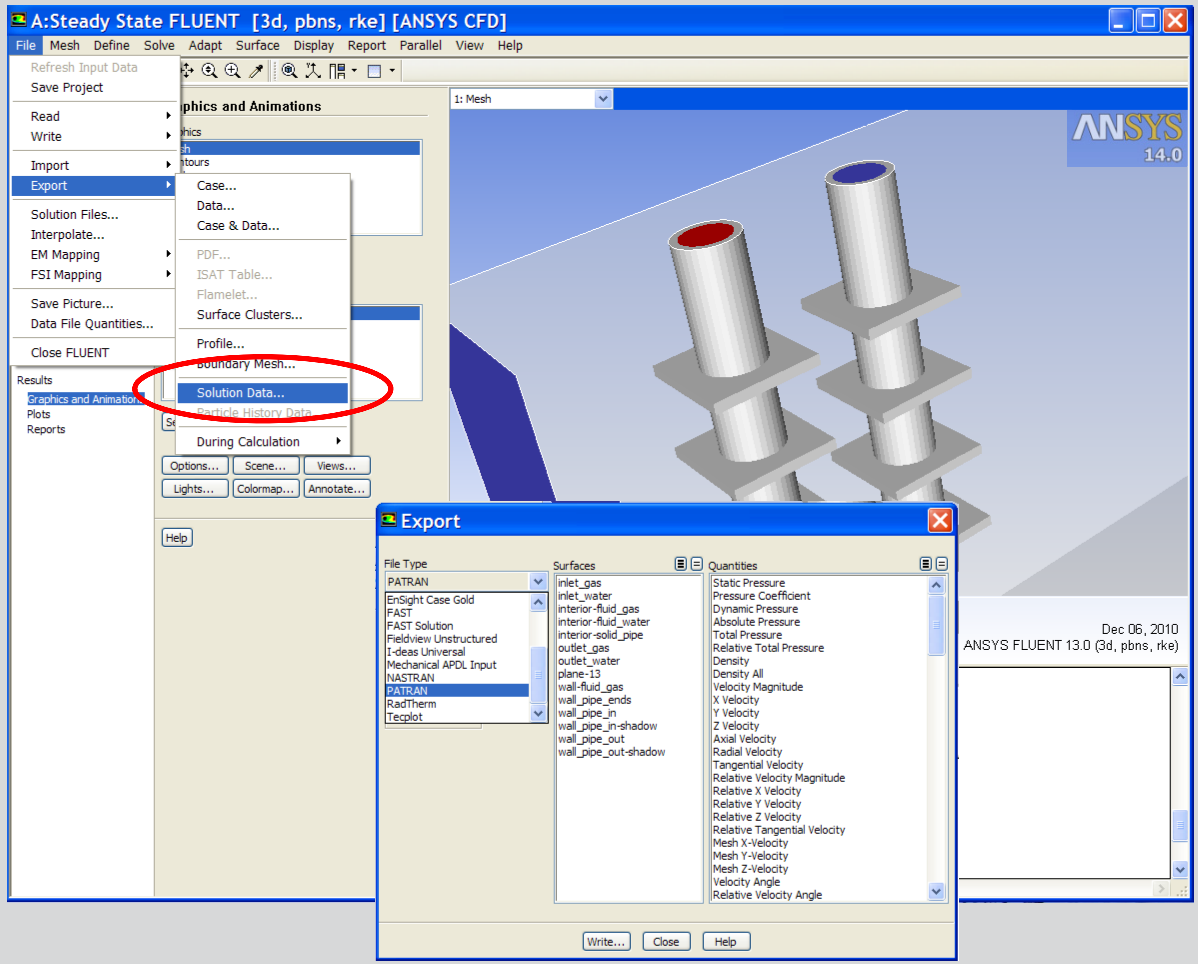
Courtesy of CADFEM Gmbh

Exporting Data from FLUENT

- FLUENT solution data can also be exported in many other formats for use in applications outside of the Workbench environment.

- These are available in the File > Export menu in FLUENT.

- Note that in this case, the data is exported at the same grid locations as the FLUENT mesh.



ANSYS

Summary

- **After activating heat transfer, you must provide :**
  - Thermal conditions at walls and flow boundaries
  - Fluid properties for energy equation

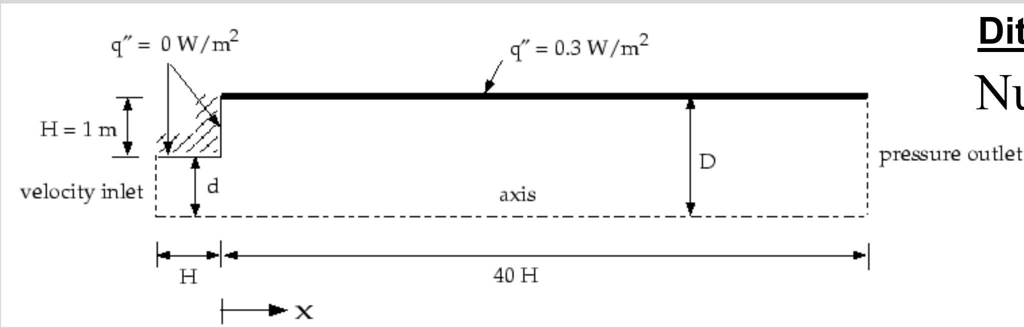
- **Available heat transfer modeling options include :**
  - Species diffusion heat source
  - Combustion heat source
  - Conjugate heat transfer
  - Natural convection
  - Radiation
  - Periodic heat transfer

- **Double precision solver usually needed to balance accurately the heat transfer rate inside the domain**

Appendix

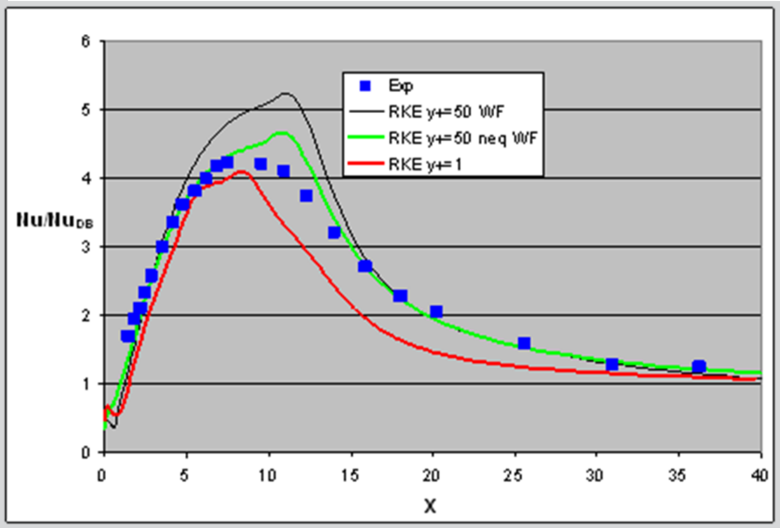
Forced Convection

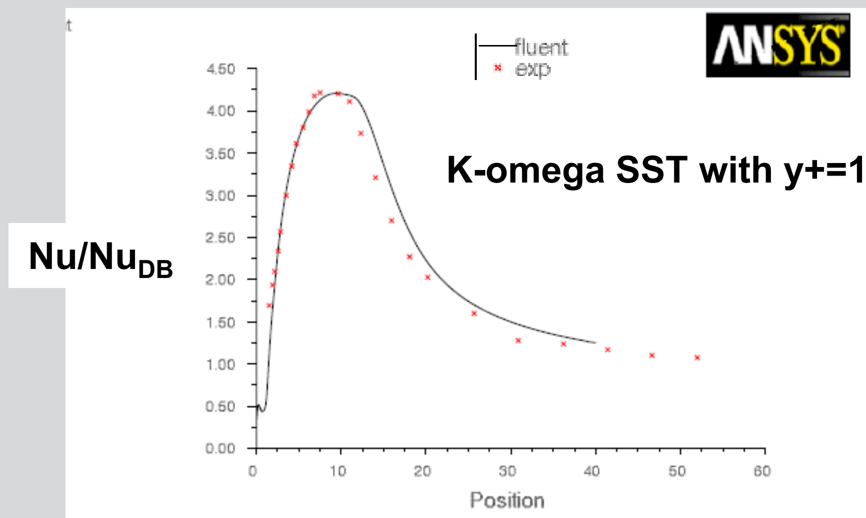
- **Forced convection results often depend on accurate resolution of turbulence**
- **Example: Baughn's Pipe Expansion Re_D= 40,750**



Dittus-Boelter correlation for a straight pipe

$$Nu_{DB} = 0.023\, Re^{0.8}\, Pr^{0.4}$$





Post-Processing Heat Transfer

- **Surface Heat Transfer Coefficient, $\mathbf{h_f}$**
  - This report is computed by using the Reference Temperature: T_{ref} specified by the User in the Reference Values panel

$$h_f = \frac{q_w}{(T_{wall} - T_{ref})}$$

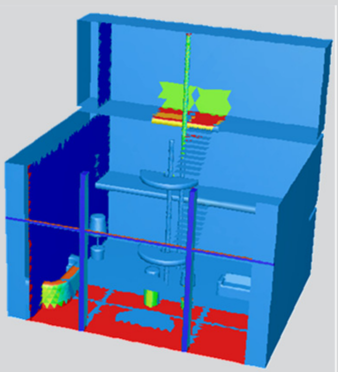
- **Wall-function-based Heat Transfer Coefficient, $\mathbf{h_{eff}}$**
  - This report is computed by using the solution of the Turbulent Boundary Layer
    - Available only when the flow is turbulent and Energy equation is enabled
    - Alternative for cases with adiabatic walls

$$h_{\text{eff}} = \frac{\rho c_p C_\mu^{1/4} k^{1/2}}{T^*} \quad \text{or} \quad h_{eff} = \frac{q_w}{(T_{wall} - T_{cell\,center})}$$

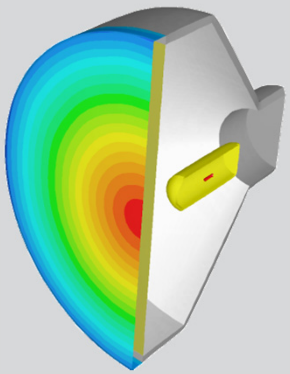
where c_P is the specific heat, k is the turbulence kinetic energy at point P, and T^* is the dimensionless temperature at point P (adjacent cell center)

Radiation

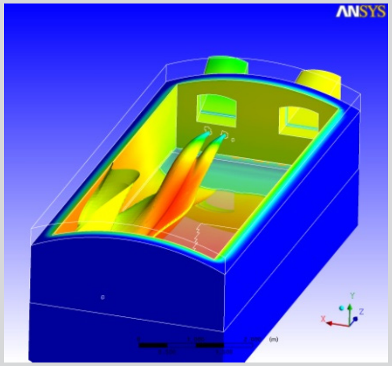
- **Thermal radiation is emission of energy as electromagnetic waves:**
  - When any object is above absolute zero it will start to emit and absorb energy
  - Spectral dependence of radiative material properties
  - Thermal radiation can occur in vacuum



Solar load (HVAC)



Headlight



Glass furnace

- **Radiation effects should be accounted for when Q_{rad} is of equal or greater magnitude than that of convective and conductive heat transfer rates**

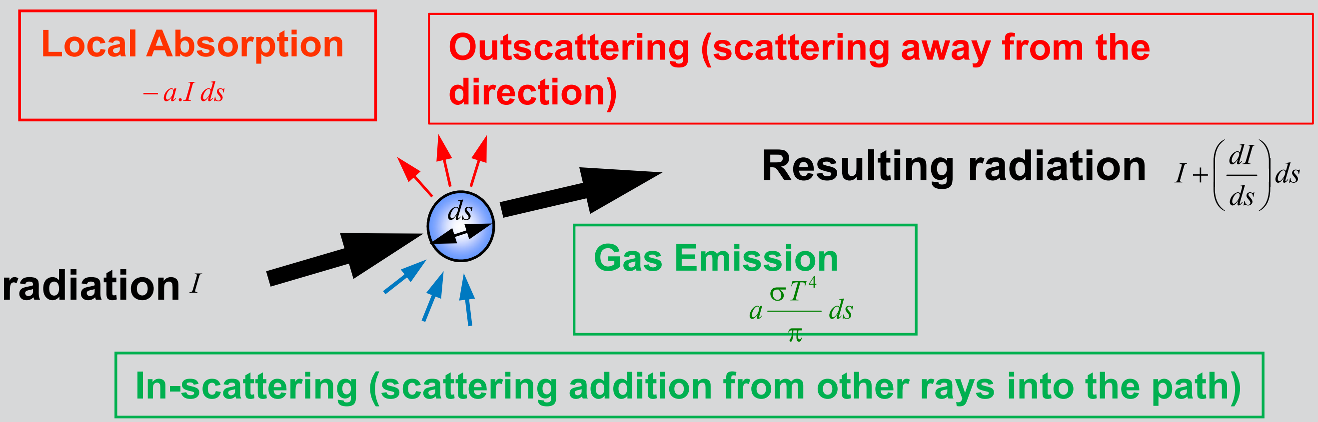
$$Q_{rad} = \sigma.\left(T_{\max}^{4} - T_{\min}^{4}\right) \geq Q_{conv} = h.\left(T_{wall} - T_{free}\right)$$

Radiation

- **To account for radiation, Radiative Intensity Transport Equations (RTEs) are solved**
  - Local absorption by fluid and at boundaries couples these RTEs with the energy equation

- **Radiation intensity is directionally and spatially dependent**

- **Transport mechanisms for radiation intensity along one given direction:**



  - Scattering often occurs when particles and droplets are present within the fluid and is often neglected

Radiation Models

- **Several radiation models are available which provide approximate solutions to the RTE**

1) Rosseland Model (Diffusion Approximation Model)

2) P-1 Model (Gibb's Model/Spherical Harmonics Model)

2) Surface-to-Surface (S2S)

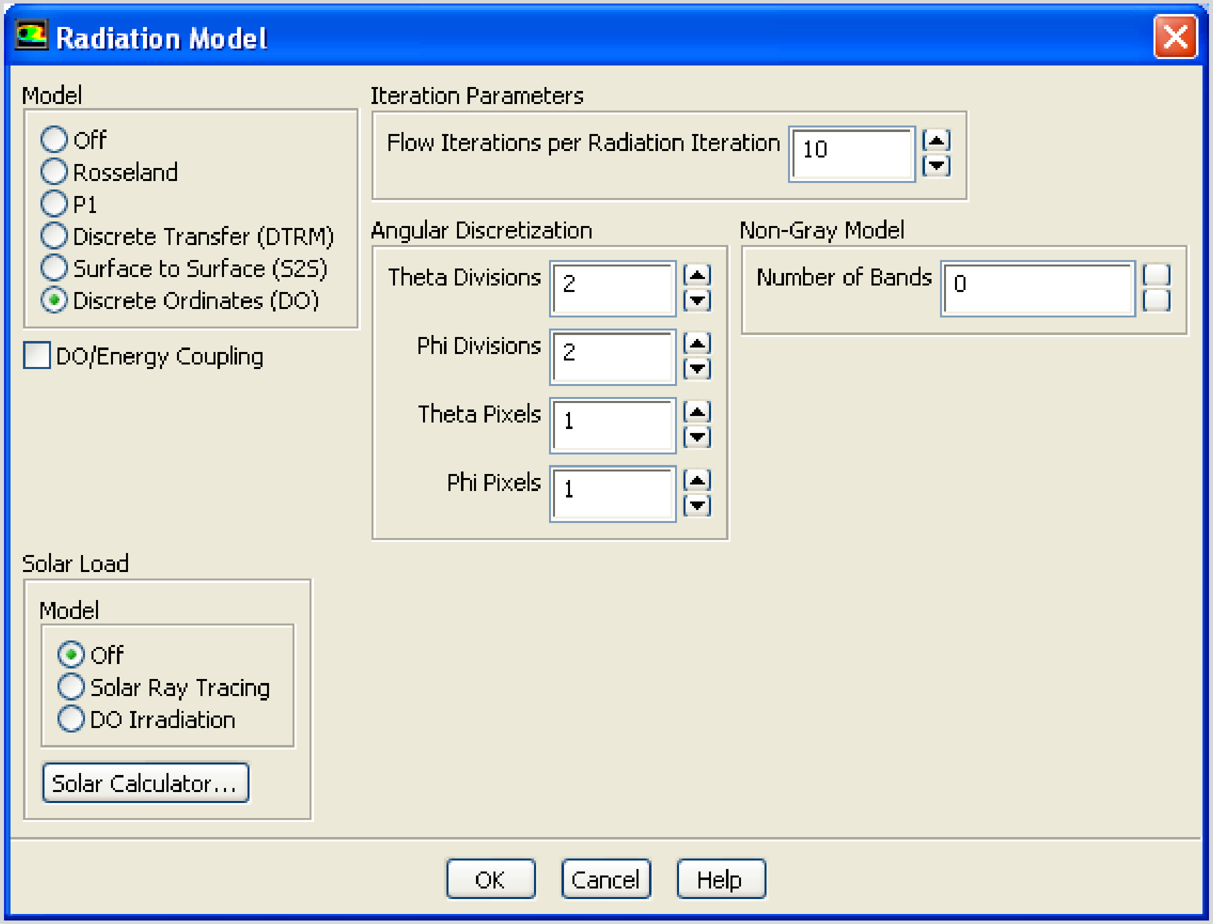
3) Discrete Transfer Radiation Model (DTRM)

4) Discrete Ordinates Model (DOM)

\+ Solar Load BC

- **Each radiation model has its assumptions, limitations, and benefits**

Radiation Models



Choosing a Radiation Model

- **For optically thick media the P1 model is a good choice**
  - Many combustion simulations fall into this category since combustion gases tend to absorb radiation
  - The P1 models gives reasonable accuracy without too much computational effort
- **For optically thin media the DOM or DTM models may be used**
  - DTM can be less accurate in models with long/thin geometries
  - DOM uses the most computational resources,
  - Both models can be used in optically thick media, but the P1 model uses far less computational resources
  - S2S is only for transparent media (Optical Thickness = 0)

Which Model is Best for My Application?

Application	Model/Method
Underhood	S2S (DOM if symmetry)
Headlamp	DOM (non-gray)
Combustion in large boilers charged with particles	DOM, DTM, P1 (WSGGM)
Combustion	DOM, DTM (WSGGM)
Glass applications	Rosseland, P1, DOM (non-gray)
Greenhouse effect	DOM
UV Disinfection (water treatment)	DOM
HVAC	Solar load model , DOM, S2S

Natural Convection

- **Natural convection has to be considered when :**
  - **Richardson number : Ri = Natural convection / Forced convection**

$$R_i = \frac{g.\beta.\Delta T.L}{U_0^{\,2}}$$

Ri = 1	$\Rightarrow$	**Free and Forced convection effects must be considered**
Ri << 1	$\Rightarrow$	**Free convection effects may be neglected**
Ri >> 1	$\Rightarrow$	**Forced convection effects may be neglected**

  - **Rayleigh number : Ra =** Buoyancy force / Losses due to viscosity and thermal diffusion

$$R_{a_x} = \frac{g.\beta.\Delta T.x^3}{\nu.a}$$

Transition Laminar – Turbulent :

It has been shown that in forced convection, the flow becomes turbulent when a critical value for Rayleigh number is reached

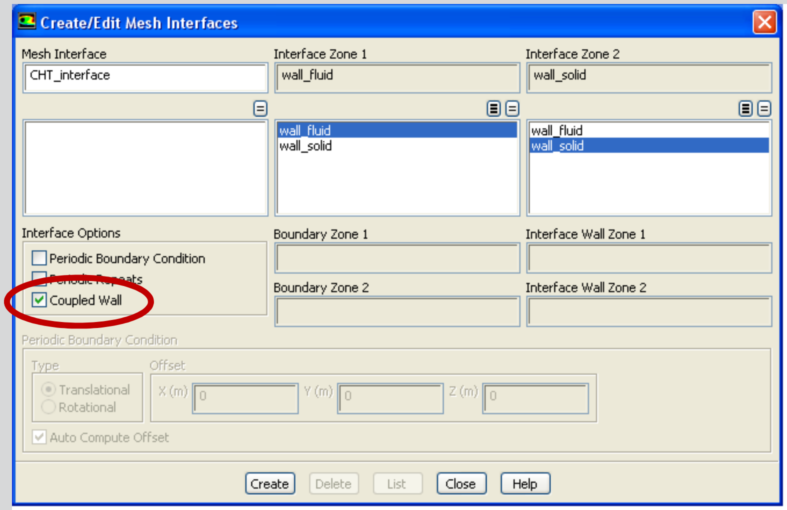
- **Ra_c is around $10e^9$**
- **but the transition zone is quite large as it varies from $10e^6 < Ra < 10e^{10}$**

Natural Convection : Closed Domain

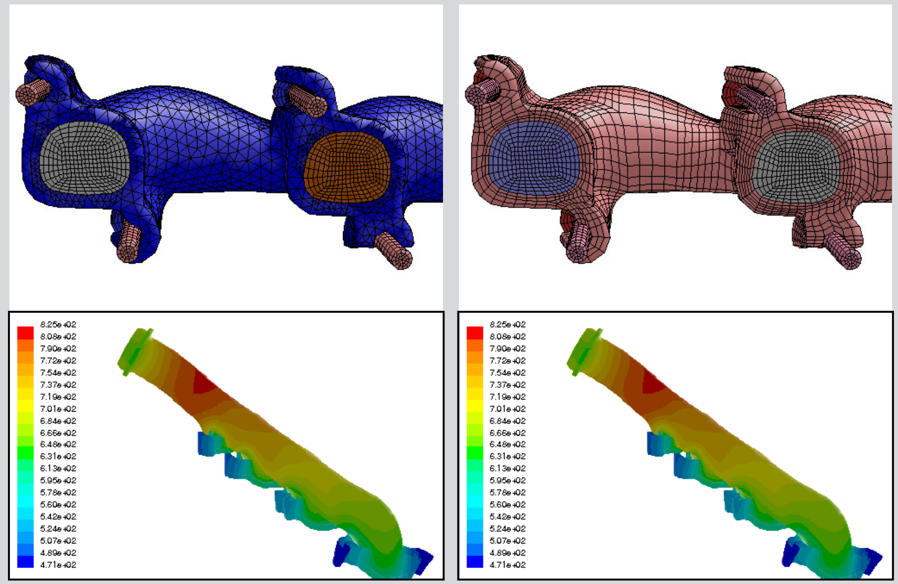
- **Natural convection problems inside closed domains :**
  - **For steady-state solver, Boussinesq model must be used**
    - Density is assumed constant
  - **For unsteady solver, Boussinesq model or ideal gas law can be used**
    - Initial Conditions define mass in the domain

Non Conformal Fluid/Solid Domain Interfaces

- **Non-conformal mesh can be used at a fluid/solid domain interface:**
  - In some cases it may be useful to use a fine mesh on the fluid zone and coarser mesh on the solid zone



Note: You can use */display/zone-grid ID* to display the shadow walls



Exporting Data from FLUENT [2]

- FLUENT also includes an FSI Mapping tool.
- Using this tool (unlike the export option on last slide) enables CFD results from FLUENT to be interpolated on to a different FEA mesh.
- First obtain the FLUENT result, then generate the FEA mesh (ABAQUS, I-deas, ANSYS, NASTRAN, PATRAN)
- Read the FEA mesh into FLUENT's FSI Mapping Tool
- FLUENT will then map the CFD results and save the interpolated results in a format the FEA code can read in.