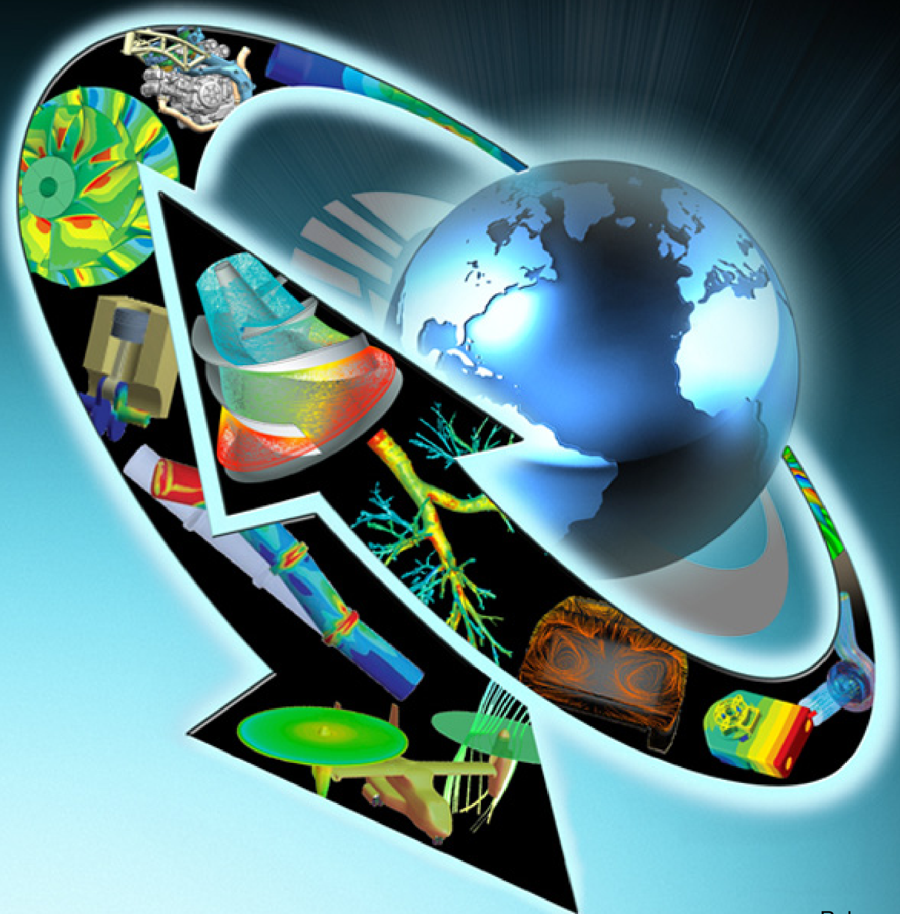
ANSYS®

Customer Training Material

Lecture 7

Heat Transfer Modeling



Introduction to ANSYS FLUENT

Introduction

- This lecture covers how the transport of thermal energy can be computed using FLUENT:
  - Convection in the fluid (natural and forced)
  - Conduction in solid regions
  - Thermal Radiation
  - External heat gain/loss from the outer boundaries of the model.

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:

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

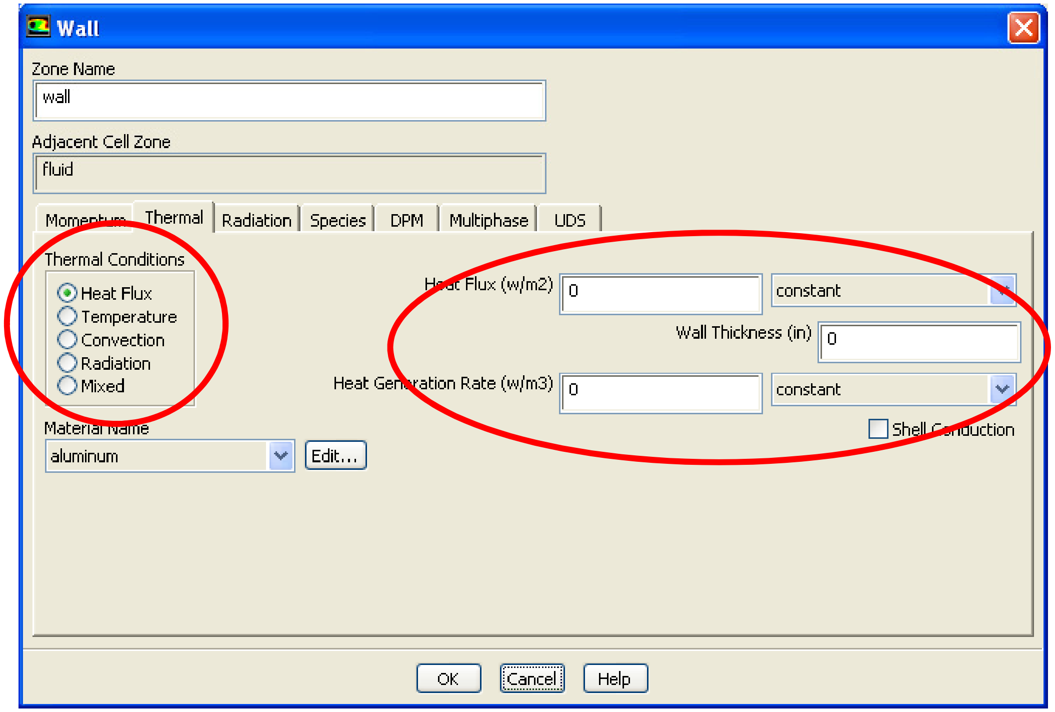
  – Pressure work and kinetic energy are always accounted for with compressible flows or when using the density-based solvers. For the pressure-based solver, they are omitted and can be added through the text command:

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

Wall Boundary Conditions

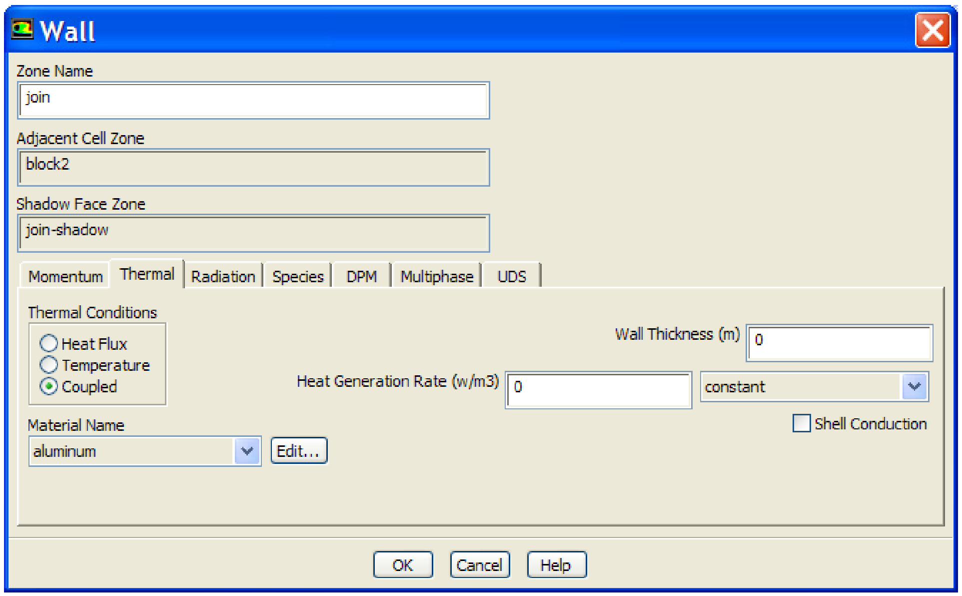
- Five thermal conditions
  - **Heat Flux**
  - **Temperature**
  - **Convection** – simulates an external convection environment which is not modeled (user-prescribed heat transfer coefficient).
  - **Radiation** – simulates an external radiation environment which is not modeled (user-prescribed external emissivity and radiation temperature).
  - **Mixed** – Combination of Convection and Radiation boundary conditions.

- Wall material and thickness can be defined for 1D or shell conduction calculations.

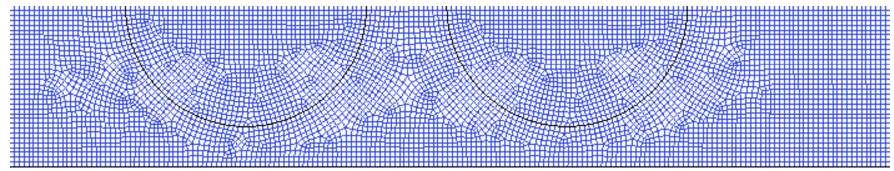


Conjugate Heat Transfer

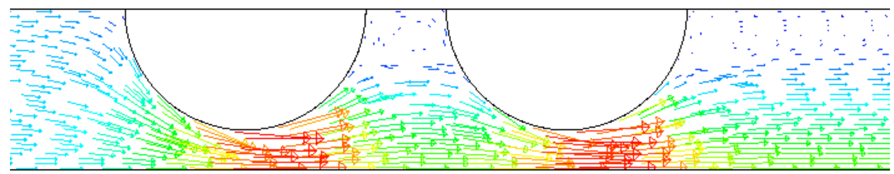
- In this example both fluid and solid zones are being solved for.
- Note there is an internal wall boundary condition on the interface, with a ‘coupled’ thermal condition. This wall will also have a partner ‘join-shadow’. Some properties like emissivity can be given different values on different sides of the wall.



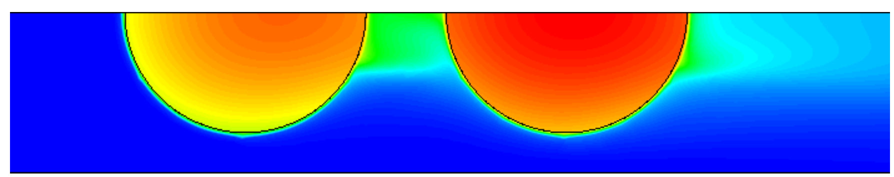
Coolant Flow Past Heated Rods



Grid

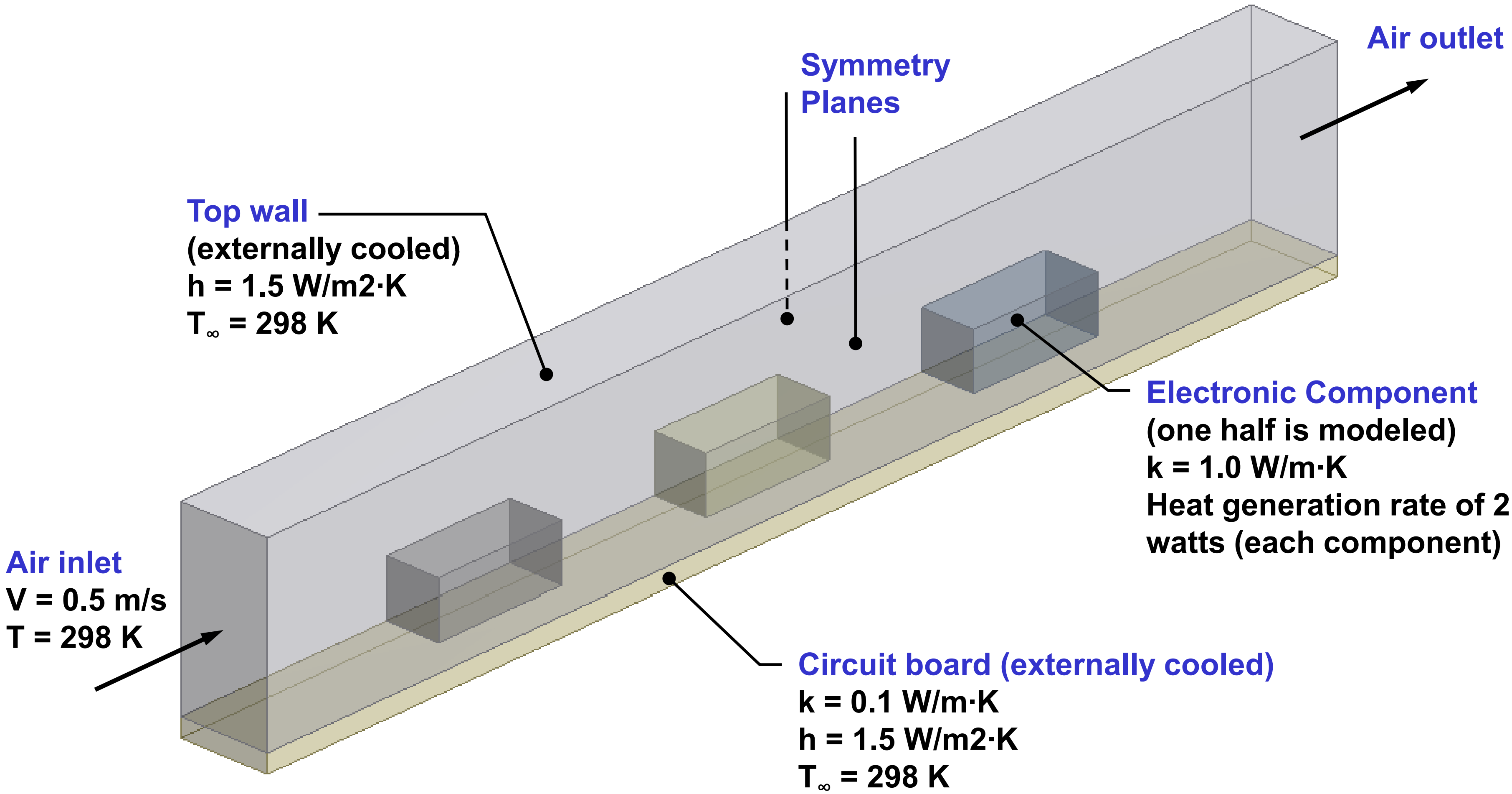


Velocity Vectors



Temperature Contours

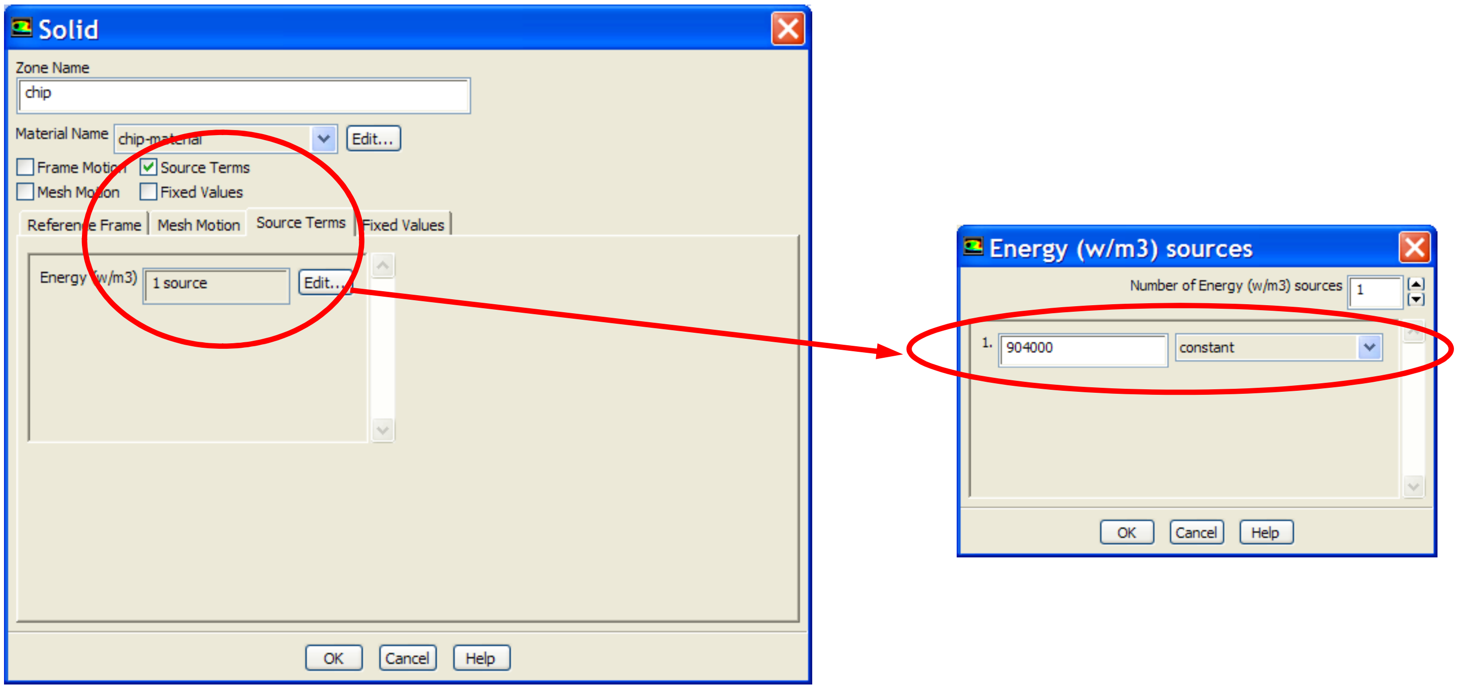
Conjugate Heat Transfer Example



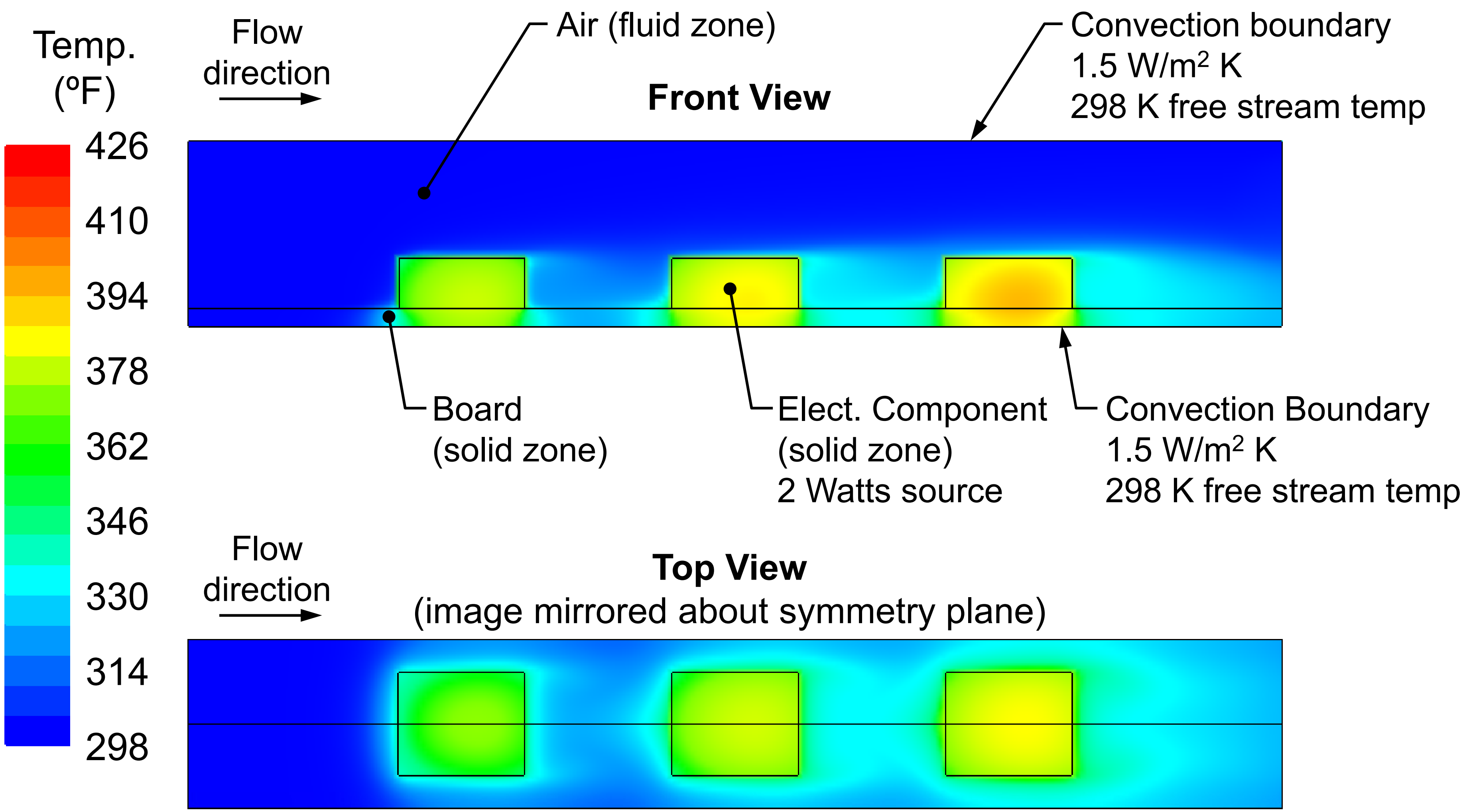
Problem Setup – Heat Source

Customer Training Material

- A volumetric heat source is applied to the ‘solid’ cell zone of the chip.
- This is applied as a source term to the cell zone
- Note the units are W/m³, volume is small so value is high.



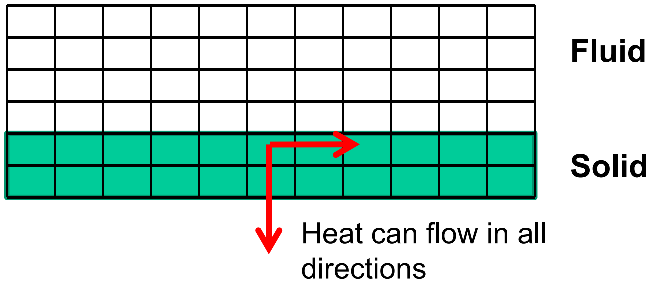
Temperature Distribution (Front and Top View)



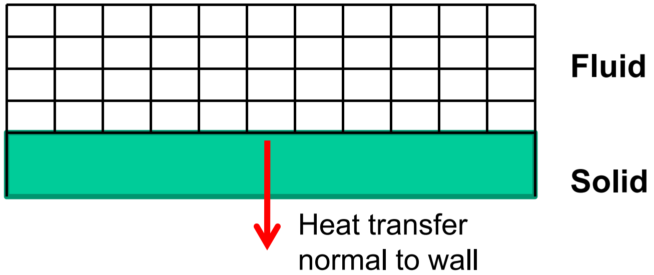
Modelling a Thin Wall

- It is often important to model the thermal effects of the wall bounding the fluid. However, it may not be necessary to mesh it.

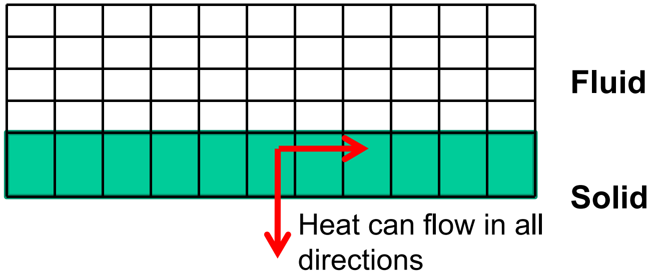
- Option 1: (as last example):
  - Mesh the wall in the pre-processor
  - Assign it as a solid cell zone
  - This is the most thorough approach



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



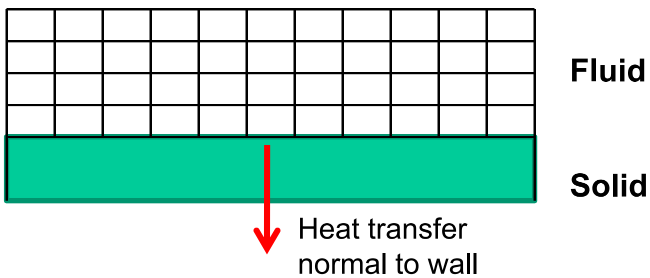
- Option 3:
  - As option 2, but enable 'shell conduction'.
  - 1 layer of 'virtual cells' is created.
  - These affect the result, but cannot be post-processed



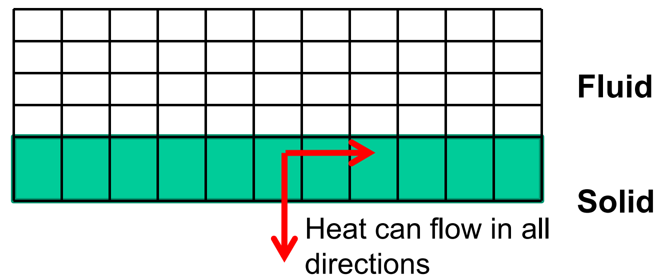
Modelling a Thin Wall

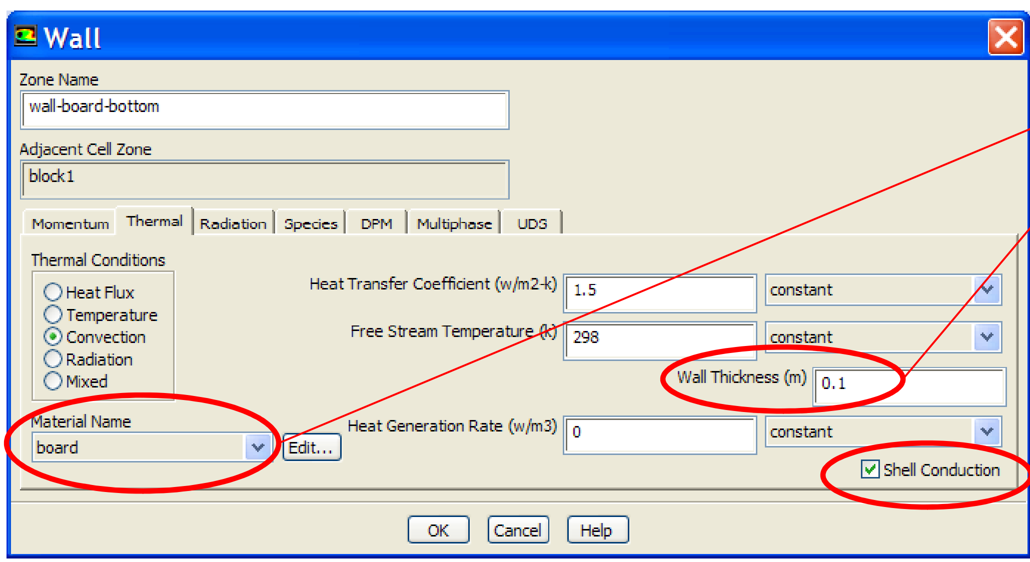
- For option 2 and option 3 on the last slide (in which it is not necessary to mesh the solid in the pre-processor), the setup panel looks like this:

- Option 2:
  - **Just conduction normal** to the solid



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





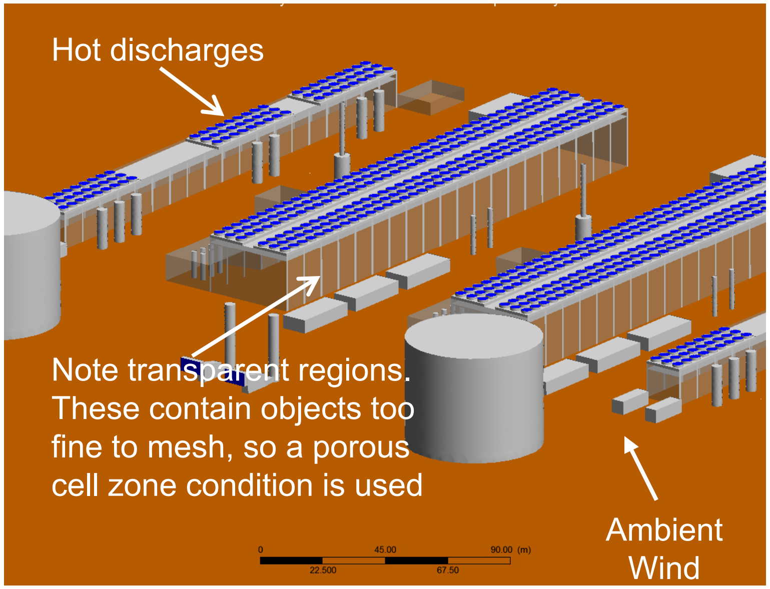
- In both cases, a material and wall thickness are enabled

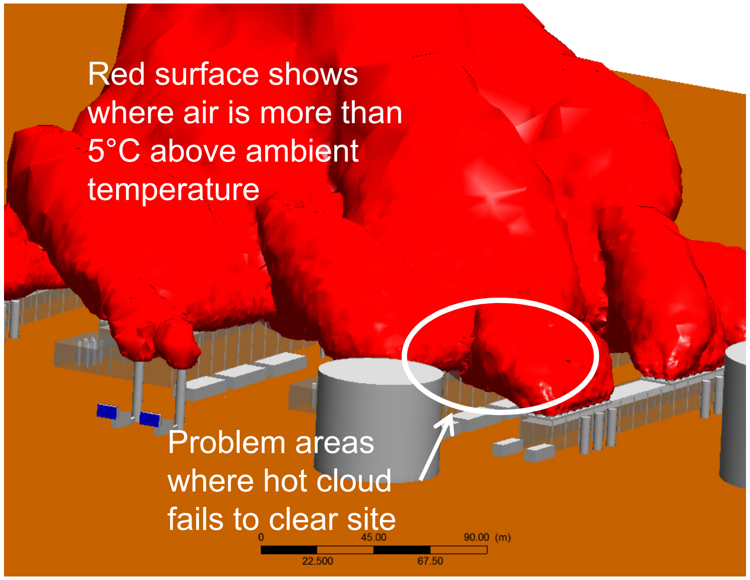
- To add the virtual cells (Option 3), enable shell conduction.

  Note these virtual cells cannot be post-processed (or exported for FSI)

Natural Convection

- 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 [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.

- Note that mathematically, whatever is set for ρ_o will cancel itself out when integrated across the domain. However careful choice of ρ_o will make a big difference to the rate of convergence (in some cases whether the model will even converge or not).

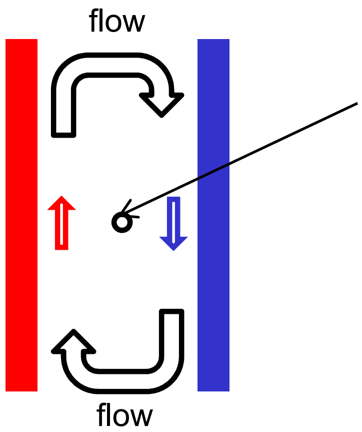
  – For enclosed problems, pick value of ρ_o that represents a typical mean density in the flow. For external (dispersion) problems select ρ_o for the ambient flow,

  – **Remember to define gravity vector**

- Illustration coming on next slide….

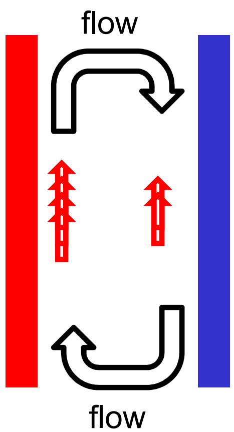
Natural Convection [3]

– E.g. consider the forces acting on this flow between a hot and cold wall



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.



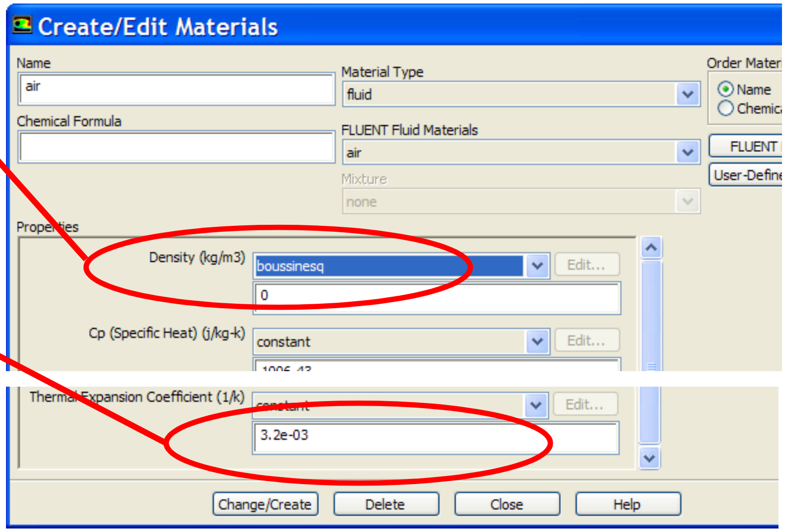
Badly posed simulation

- 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
- And define a thermal expansion coefficient β, (value in standard engineering texts)
- Buoyant force is computed from

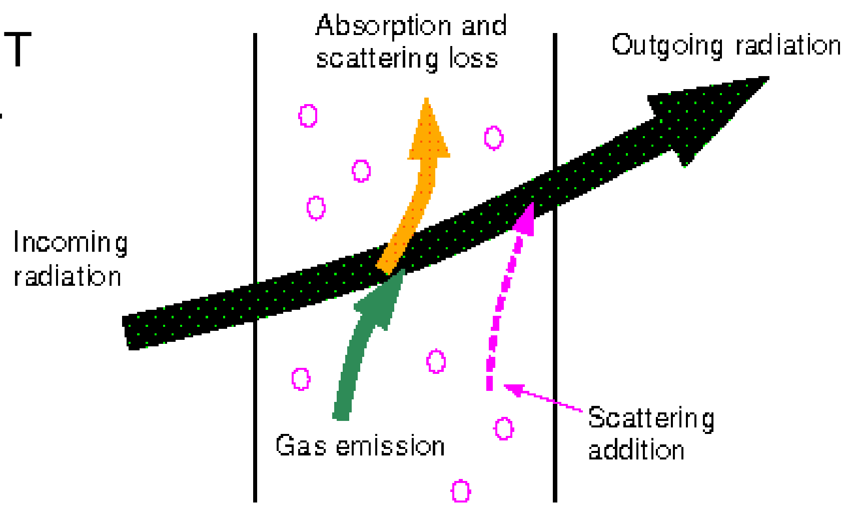
$$(\rho - \rho_0)g = -\rho_0 \beta (T - T_0) g$$



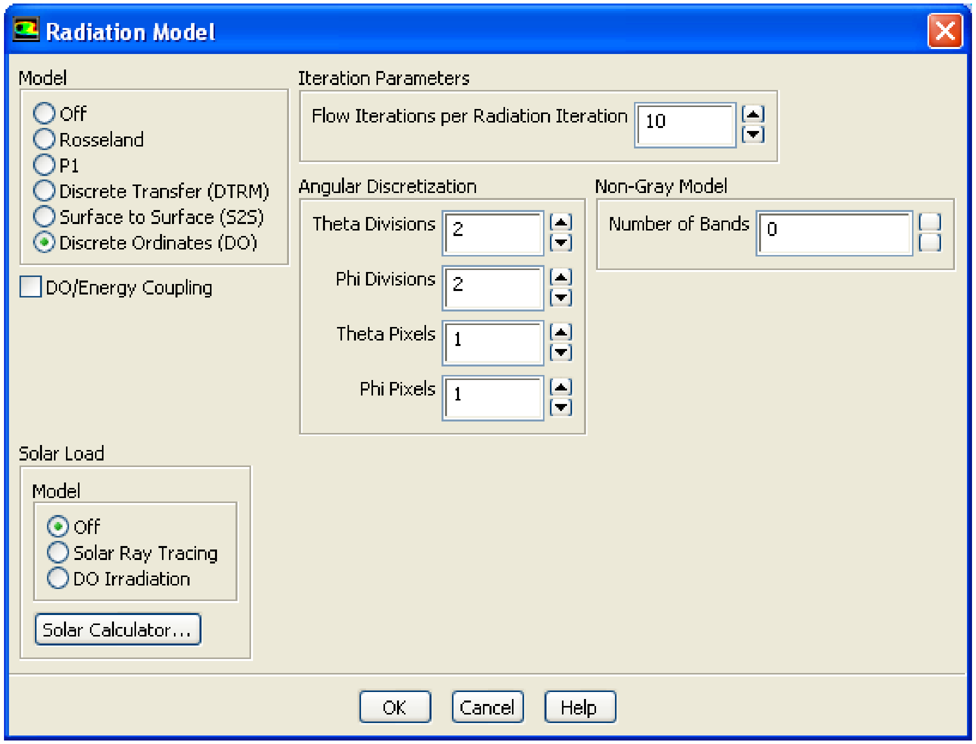
- The same comments as on the previous slides (for setting the reference density ρ_o) apply here for setting the reference temperature T_o - set in the Operating Conditions panel.

Radiation

- Radiation effects should be accounted for when $Q_{rad} = \sigma\left(T_{max}^4 - T_{min}^4\right)$ is of comparable magnitude as the convection and conduction heat transfer rates.
  - σ is the Stefan-Boltzmann constant, 5.67×10^{-8} W/(m^2·K^4)
- 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.
  - These equations are often solved separately from the fluid flow solution; however, they can be coupled to the flow.
- Radiation intensity, *I*(r,s), is directionally and spatially dependent.
- Five radiation models are available in FLUENT (see the Appendix for details on each model).
  - Discrete Ordinates Model (DOM)
  - Discrete Transfer Radiation Model (DTRM)
  - P1 Radiation Model
  - Rosseland Model
  - Surface-to-Surface (S2S)



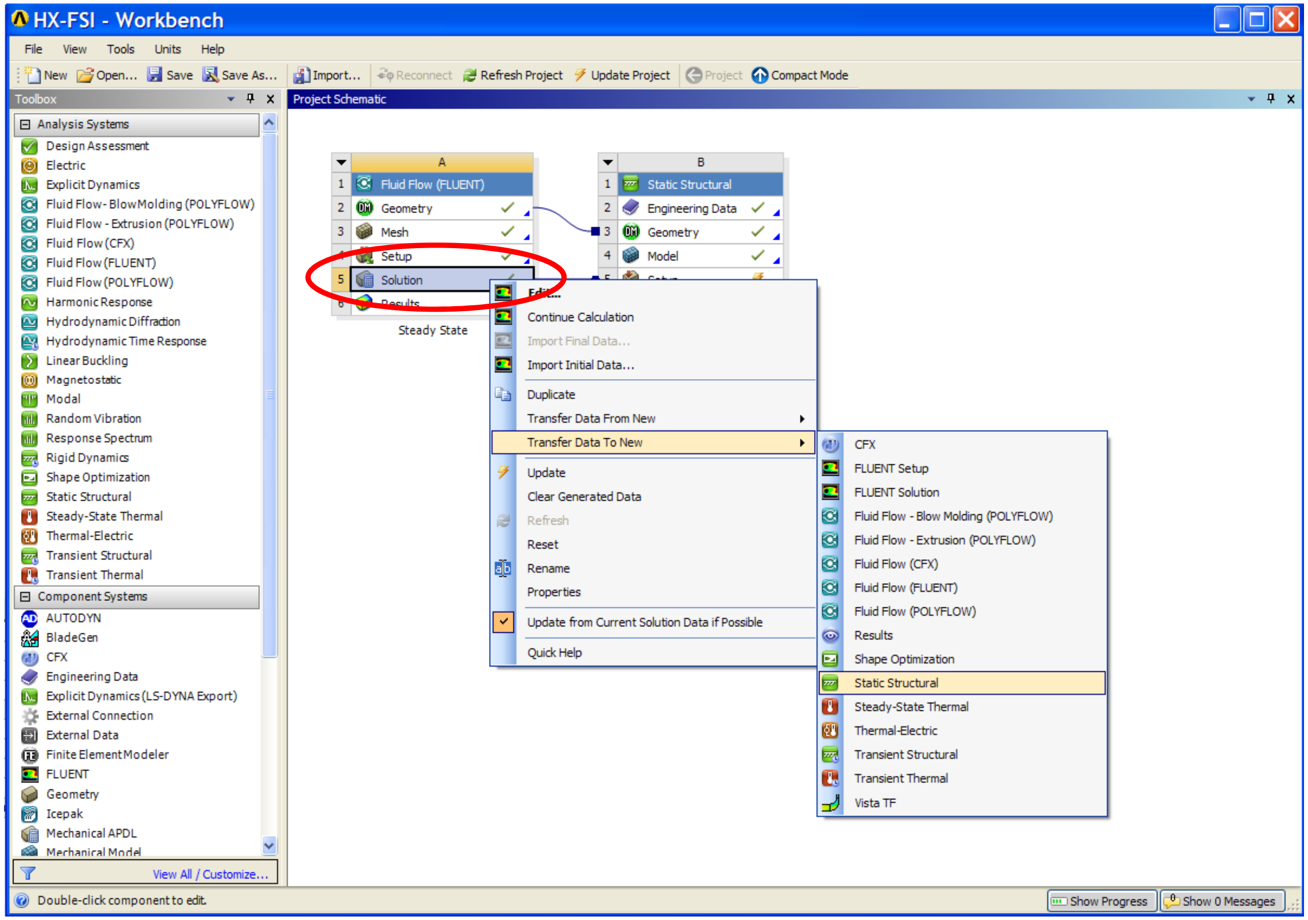
Selecting a Radiation Model



- Some general guidelines for radiation model selection:
  - **Computational effort**
    - P1 gives reasonable accuracy with the least amount of effort.
  - **Accuracy**
    - DTRM and DOM are the most accurate.
  - **Optical thickness**
    - Use DTRM/DOM for optically thin media ($\alpha L << 1$)
    - Use P1 for optically thick media.
    - Use S2S for zero optical thickness.
  - **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**
    - DTRM/DOM with a sufficiently large number of rays/ ordinates is most appropriate.

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”

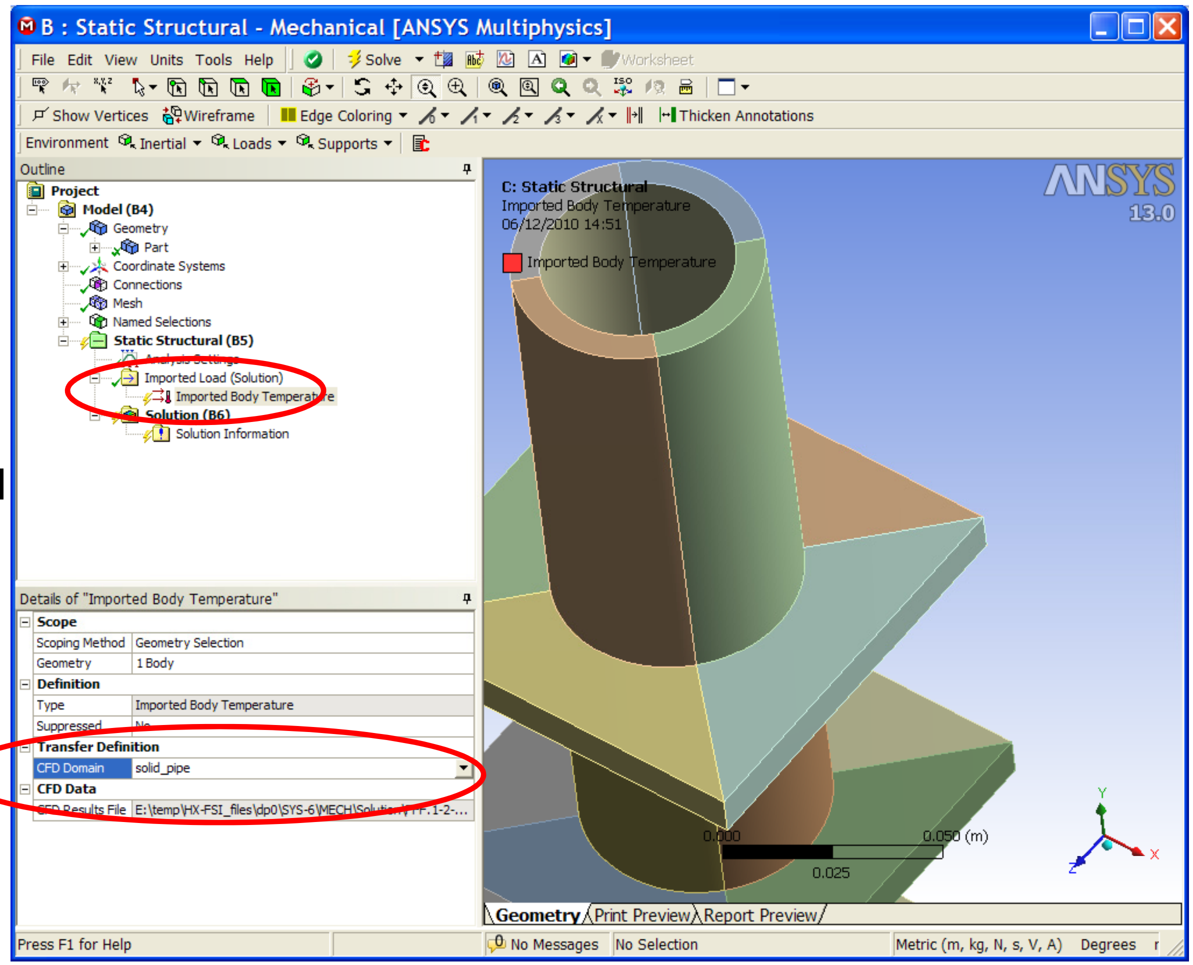
Performing a 1-way Thermal FSI Simulation

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

- Note an enhancement at R13 is that volumetric (not just surface) quantities can be transferred.

- In this heat-exchanger example, FLUENT solved the temperature for the fluids inside and outside of the pipe, as well as the thermal conduction in the solid.

  The ‘solid pipe’ temperature is interpolated from **FLUENT** to the **Mechanical** application which then performs a thermal stress analysis.

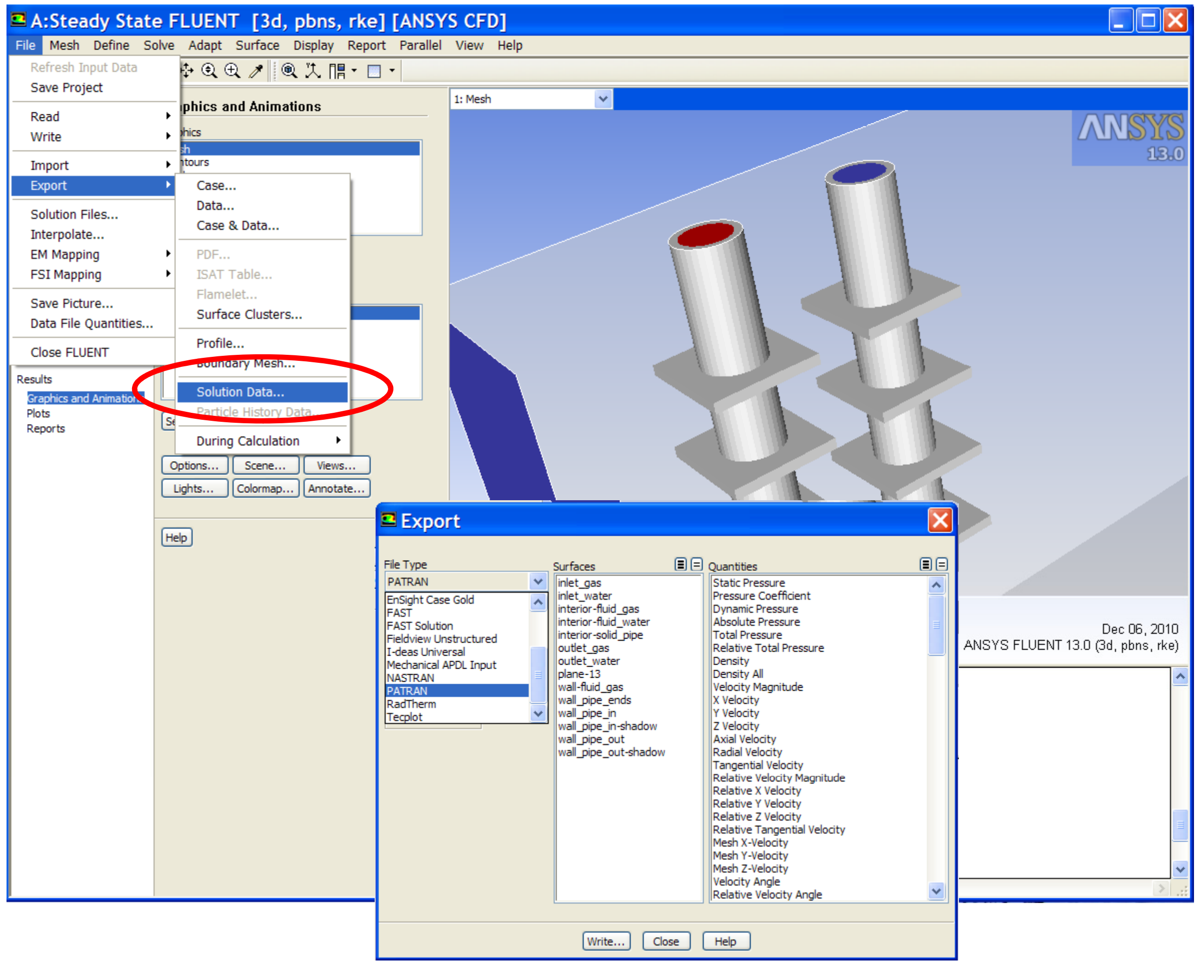


Exporting Data from FLUENT

- 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.



Exporting Data from FLUENT [2]

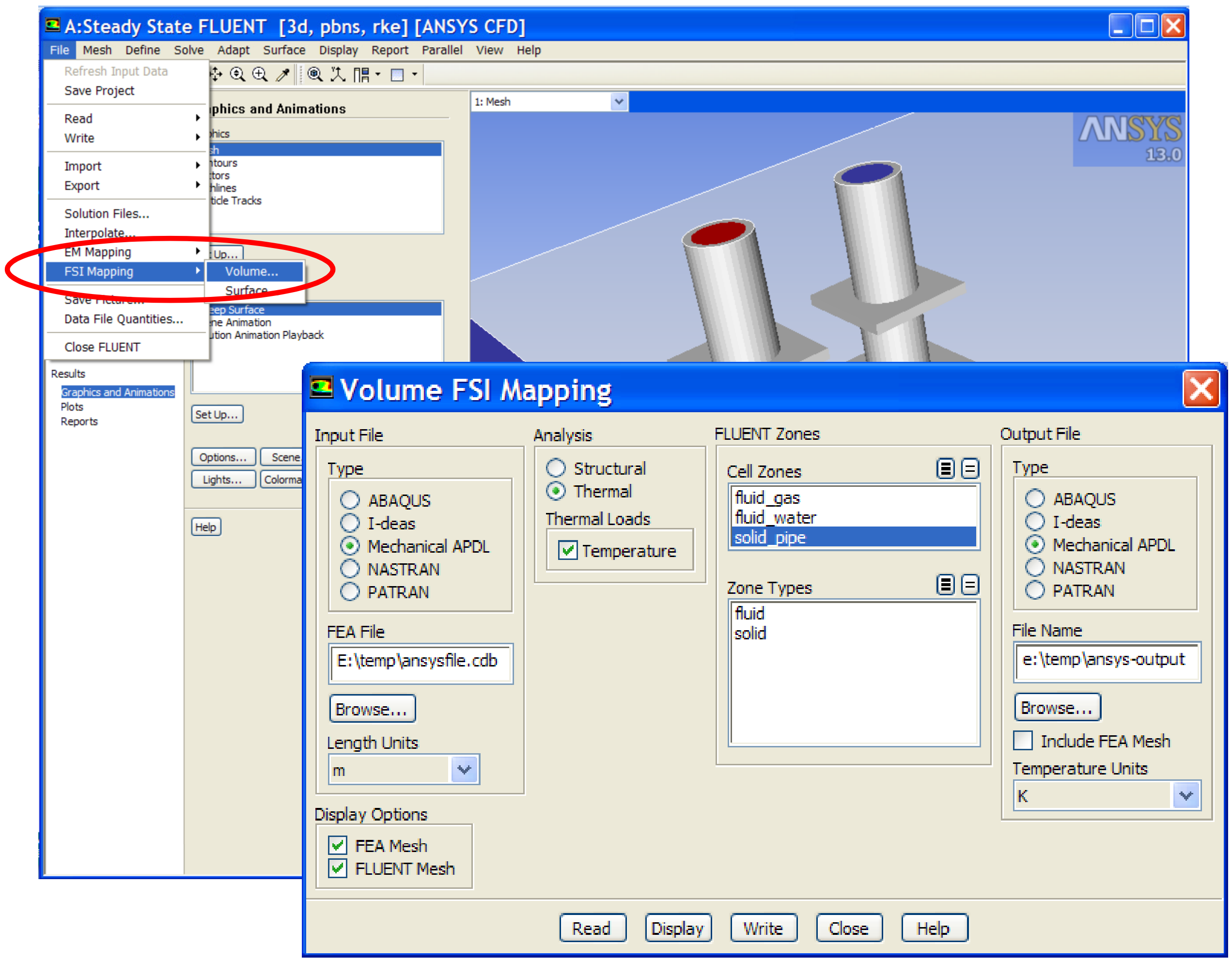
- FLUENT also includes an FSI Mapping tool.

- Using this tool (unlike the export option on last slide) enables CFD results from FLUENT to be interpolated on to a different FEA mesh.

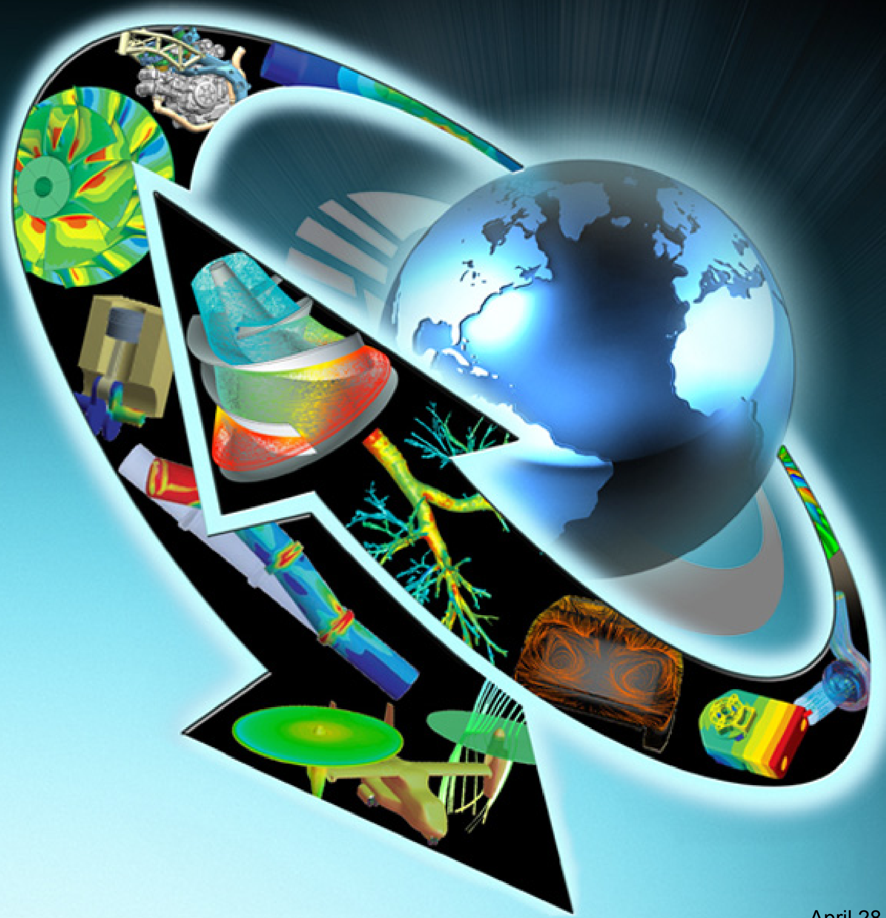
- First obtain the FLUENT result, then generate the FEA mesh (ABAQUS, I-deas, ANSYS, NASTRAN, PATRAN)

- 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.



Appendix

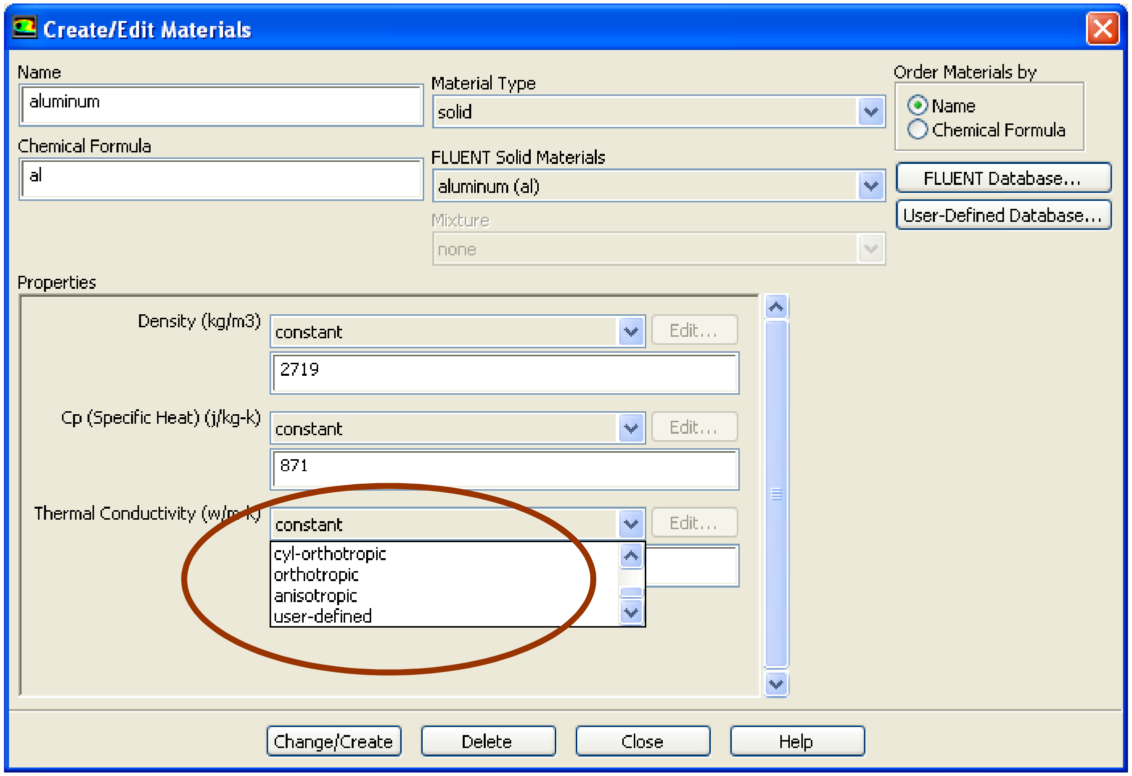


April 28, 2009
Inventory #002600

Energy Equation for Solid Regions

- Ability to compute conduction of heat through solids
- Energy equation: $$\frac{\partial(\rho h)}{\partial t} + \nabla \cdot (\mathbf{V} \rho h) = \nabla \cdot (k_{\text{eff}} \nabla T) + S_h$$
  – h is the sensible enthalpy:

$$h = \int_{T_{\text{ref}}}^{T} c_p \, dT$$

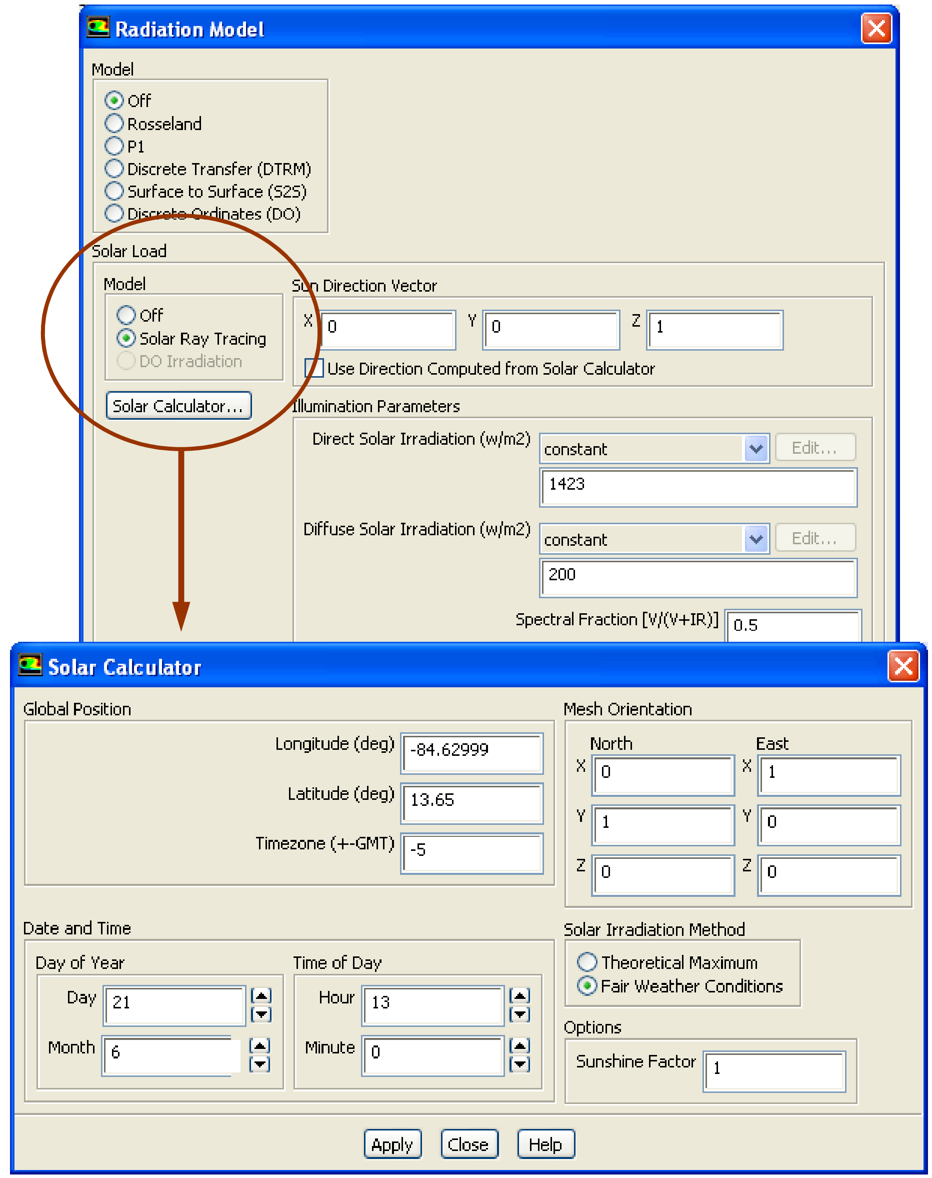


- Anisotropic conductivity in solids (pressure-based solver only)

$$\nabla \cdot \left(k_{ij} \nabla T \right)$$

Solar Load Model

- Solar load model
  - Ray tracing algorithm for solar radiant energy transport: Compatible with all radiation models
  - Available with parallel solver (but ray tracing algorithm is not parallelized)
  - 3D only
- Specifications
  - Sun direction vector
  - Solar intensity (direct, diffuse)
  - Solar calculator for calculating direction and direct intensity using theoretical maximum or "fair weather conditions"
  - Transient cases
    - When direction vector is specified with solar calculator, sun direction vector will change accordingly in transient simulation
    - Specify "time steps per solar load update"



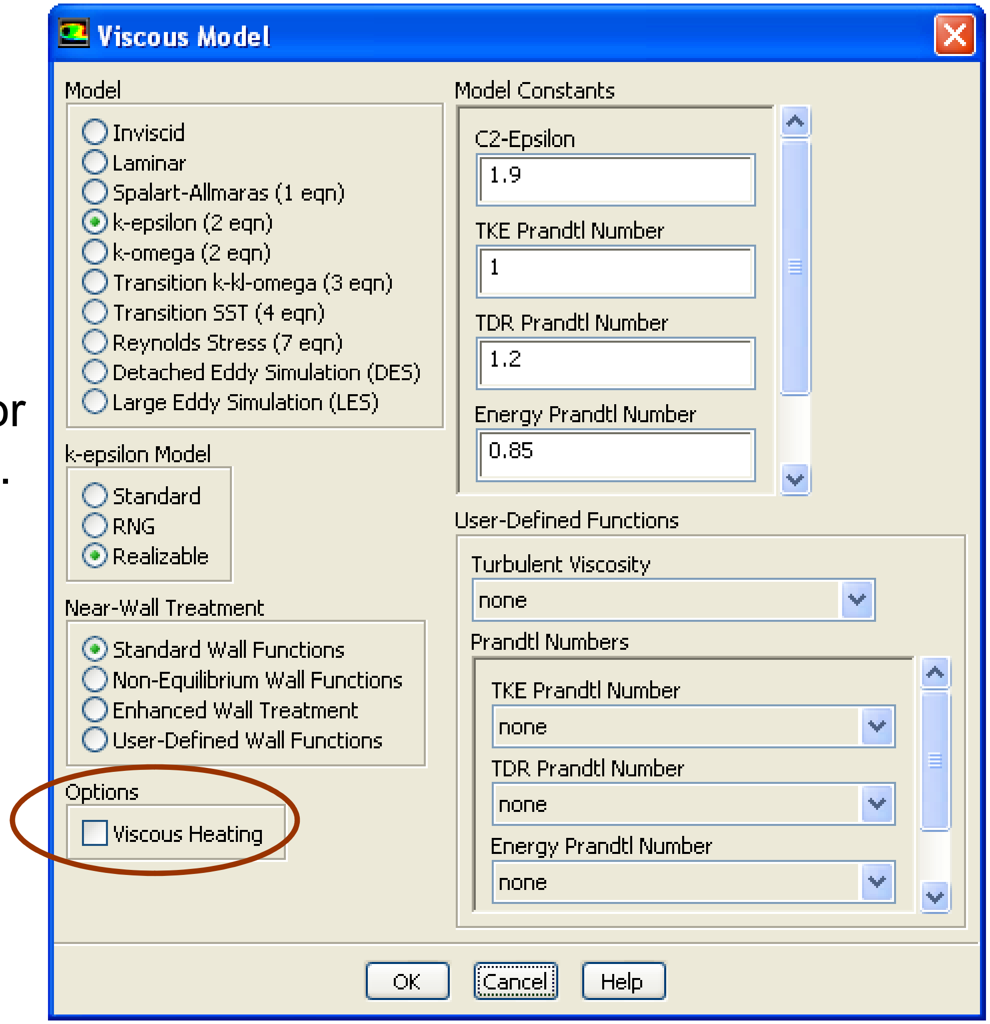
Energy Equation Terms – Viscous Dissipation

- Energy source due to viscous dissipation:

$$\nabla \cdot \tau_{\text{eff}} \cdot \mathbf{V}$$

  – Also called viscous heating.
  – Important when viscous shear in fluid is large (e.g. lubrication) and/or in high-velocity compressible flows.
  – Often negligible
    - Not included by default in the pressure-based solver.
    - Always included in the density-based solver.
  – Important when the Brinkman number approaches or exceeds unity:

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



Energy Equation Terms – Species Diffusion

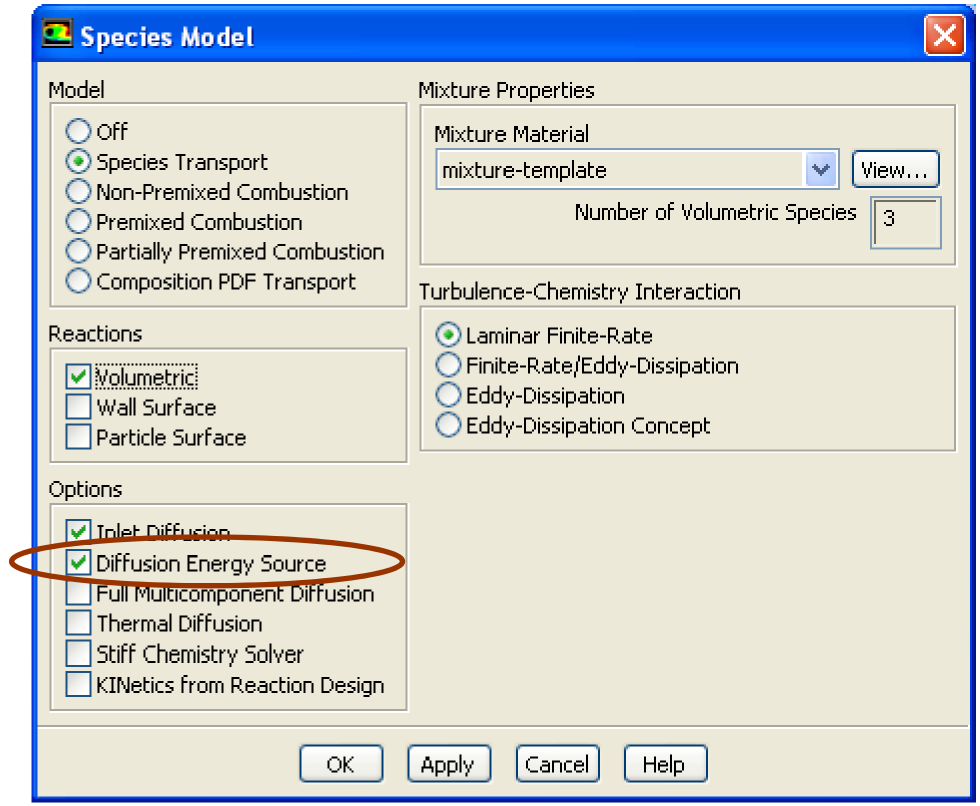
ANSYS

Customer Training Material

- Energy source due to species diffusion included for multiple species flows.

$$\nabla \cdot \sum_j h_j \mathbf{J}_j$$

  – Includes the effect of enthalpy transport
  due to species diffusion
  – Always included in the density-based solver.
  – Can be disabled in the pressure-based solver.



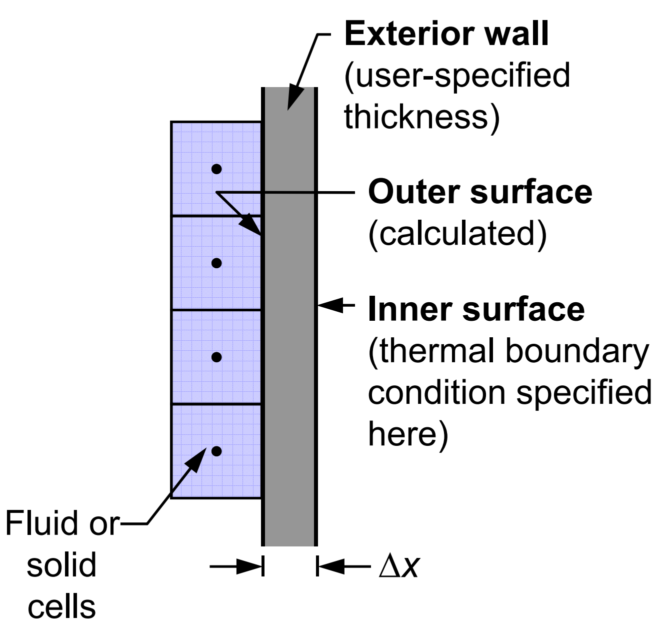
Energy Equation Terms – Source Terms

- Energy source due to chemical reaction is included for reacting flows.
  - Enthalpy of formation of all species.
  - Volumetric rate of creation of all species.
- Energy source due to radiation includes radiation source terms.
- Interphase energy source:
  - Includes heat transfer between continuous and discrete phase
  - DPM, spray, particles…

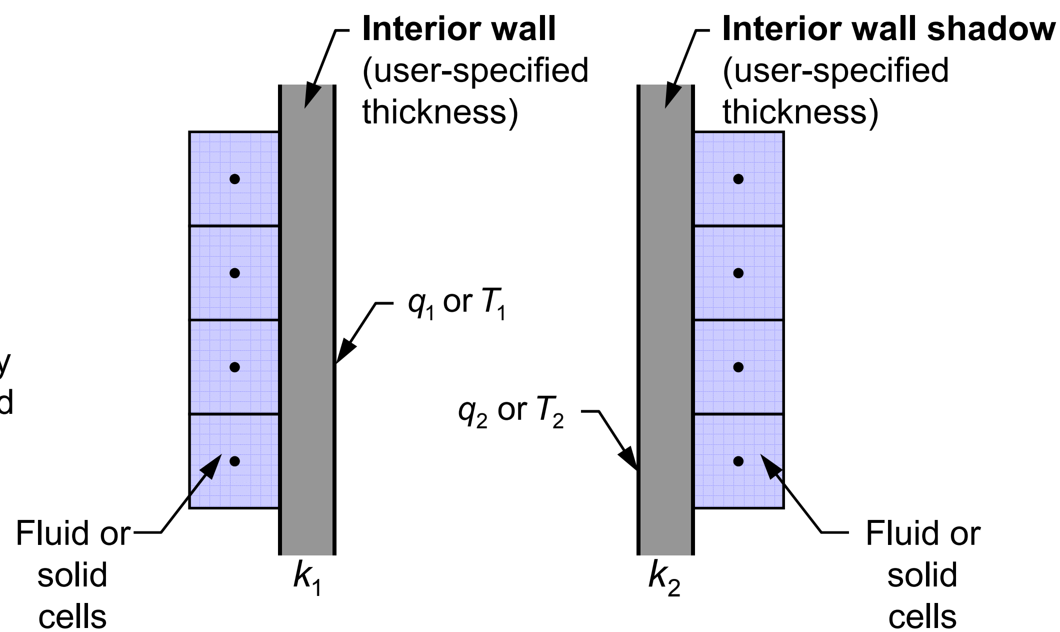
$$\frac{\partial(\rho E)}{\partial t} + \nabla \cdot [\mathbf{V}(\rho E + p)] = \nabla \cdot \left[k_{\text{eff}} \nabla T - \sum_j h_j \mathbf{J}_j + \tau_{\text{eff}} \cdot \mathbf{V} \right] + \boxed{S_h}$$

Thin and Two-Sided Walls

- In the Thin Wall approach, the wall thickness is not explicitly meshed.
- Model thin layer of material between two zones
- Thermal resistance $\Delta x/k$ is artificially applied by the solver.
- Boundary conditions specified on the outside surface.



Thermal boundary conditions are supplied on the inner surface of a thin wall



Thermal boundary conditions are supplied on the inner surfaces of uncoupled wall/shadow pairs

Discrete Ordinates Model

- The radiative transfer equation is solved for a discrete number of finite solid angles, σ_s:

$$\underbrace{\frac{\partial I}{\partial x_i} + (a + \sigma_s) I(\mathbf{r}, \mathbf{s})}_{\text{Absorption}} = \underbrace{a n^2 \frac{\sigma T^4}{\pi}}_{\text{Emission}} + \underbrace{\frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\mathbf{r}, \mathbf{s}') \Phi(\mathbf{s} \cdot \mathbf{s}') \, d\Omega'}_{\text{Scattering}}$$

- Advantages:
  - Conservative method leads to heat balance for coarse discretization.
    - Accuracy can be increased by using a finer discretization.
  - Most comprehensive radiation model:
    - Accounts for scattering, semi-transparent media, specular surfaces, and wavelength-dependent transmission using banded-gray option.

- Limitations:
  - Solving a problem with a large number of ordinates is CPU-intensive.

Discrete Transfer Radiation Model (DTRM)

- Main assumption – Radiation leaving a surface element within a specified range of solid angles can be approximated by a single ray.

- Uses a ray-tracing technique to integrate radiant intensity along each ray:

$$\frac{dI}{ds} + aI = \frac{a\sigma T^4}{\pi}$$

- Advantages:
  - Relatively simple model.
  - Can increase accuracy by increasing number of rays.
  - Applies to wide range of optical thicknesses.

- Limitations:
  - Assumes all surfaces are diffuse.
  - Effect of scattering not included.
  - Solving a problem with a large number of rays is CPU-intensive.

P-1 Model

- Main assumption – The directional dependence in RTE is integrated out, resulting in a diffusion equation for incident radiation.

- Advantages:
  - Radiative transfer equation easy to solve with little CPU demand.
  - Includes effect of scattering.
    - Effects of particles, droplets, and soot can be included.
  - Works reasonably well for applications where the optical thickness is large (e.g. combustion).

- Limitations:
  - Assumes all surfaces are diffuse.
  - May result in loss of accuracy (depending on the complexity of the geometry) if the optical thickness is small.
  - Tends to overpredict radiative fluxes from localized heat sources or sinks.

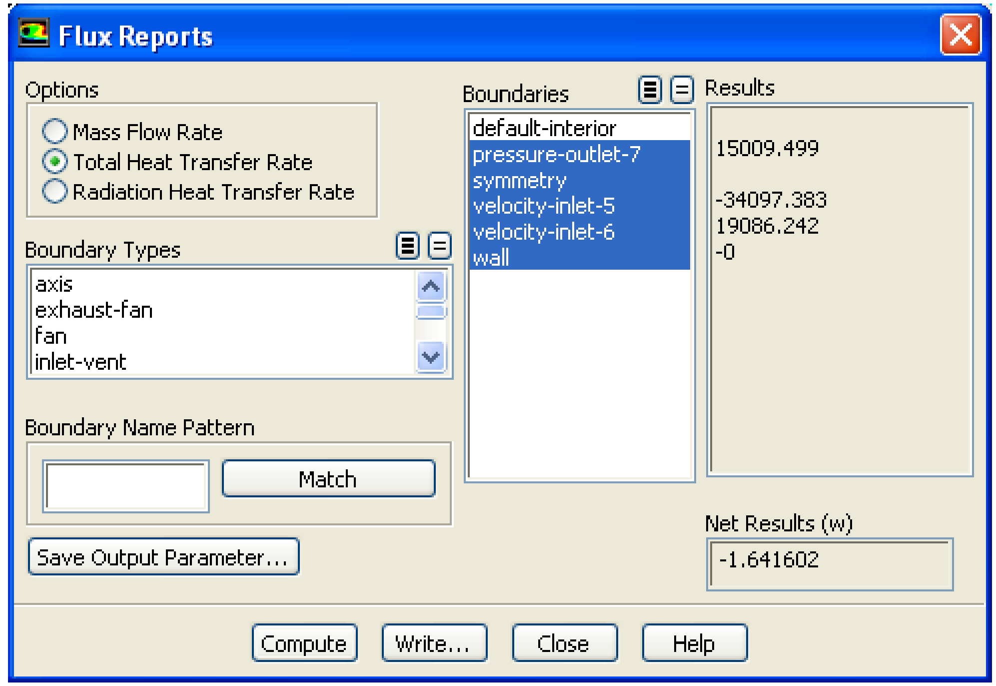
Surface-to-Surface (S2S) Radiation Model

- The surface-to-surface radiation model can be used for modeling radiation in situations where there is no participating media.
  - For example, spacecraft heat rejection system, solar collector systems, radiative space heaters, and automotive underhood cooling.
  - S2S is a view-factor based model.
  - Non-participating media is assumed.

- Limitations:
  - The S2S model assumes that all surfaces are diffuse.
  - The implementation assumes gray radiation.
  - Storage and memory requirements increase very rapidly as the number of surface faces increases.
    - Memory requirements can be reduced by using clusters of surface faces.
      - Clustering does not work with sliding meshes or hanging nodes.
  - Not to be used with periodic or symmetry boundary conditions.

Reporting – Heat Flux

- Heat flux report:
  - It is recommended that you perform a heat balance check so to ensure that your solution is truly converged.



- Exporting Heat Flux Data:
  - It is possible to export heat flux data on wall zones (including radiation) to a generic file.
  - Use the text interface:

    `file/export/custom-heat-flux`
  - File format for each selected face zone:

```
zone-name nfaces
x_f   y_f   z_f   A   Q   T_w   T_c   HTC
...
```

Reporting – Heat Transfer Coefficient

- Wall-function-based heat transfer coefficient

$$h_{\text{eff}} = \frac{\rho c_p C_\mu^{1/4} k^{1/2}}{T^*}$$

where c_P is the specific heat, k_P is the turbulence kinetic energy at point P, and T^* is the dimensionless temperature:

$$T^* = \frac{(T_w - T_P)\rho c_p k_P^{1/2}}{\dot{q}} = \begin{cases} Pr y^* + \frac{1}{2}\text{Pr}\frac{C_\mu^{1/4} k_P^{1/2} U_P^2}{\dot{q}} U_P^2 & (y^* < y_T^*) \\ \text{Pr}_t \left[\frac{1}{\kappa}\ln(EY^*) + P\right] & \\ \quad + \frac{1}{2}\frac{\rho C_\mu^{1/4} k_P^{1/2}}{\dot{q}}\left\{\text{Pr}_t U_P^2 + (\text{Pr} - \text{Pr}_t) U_c^2\right\} & (y^* > y_T^*) \end{cases}$$

– Available only when the flow is turbulent and Energy equation is enabled.

– Alternative for cases with adiabatic walls.