

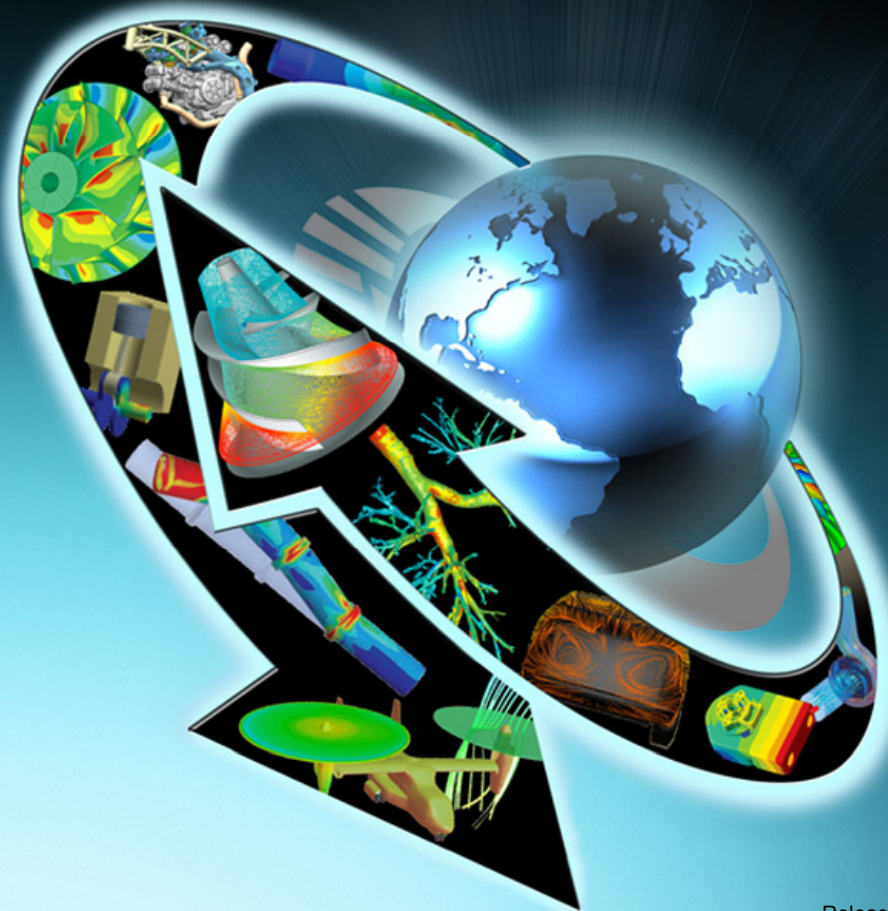


Customer Training Material

Lecture 1

Introduction

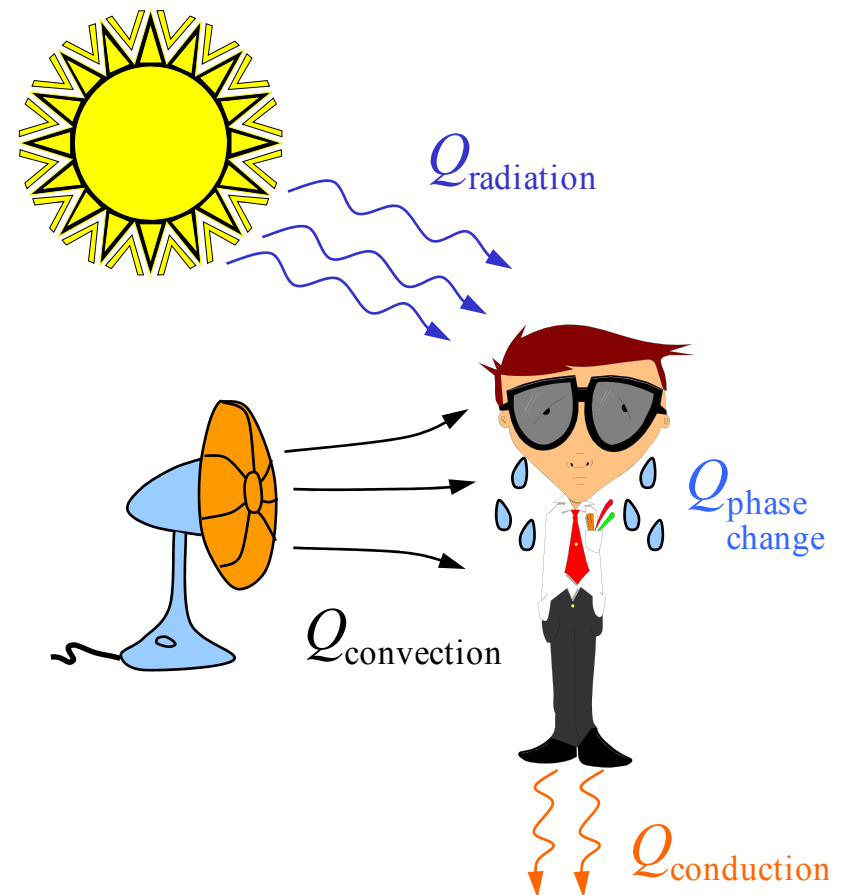
Heat Transfer Modeling using ANSYS FLUENT



- **Modes of Heat Transfer**
- **Basic Heat Transfer Phenomena**
 - **Conduction**
 - **Convection**
 - **Radiation**
 - **Phase Change**
- **The Energy Equation**
- **Boundary Conditions**

Modes of Heat Transfer

- **Conduction**
 - Occurs in a medium (fluid or solid)
 - Linked to atomic and molecular vibration or electronic motion.
 - Diffusion of heat due to temperature gradient within the medium.
- **Convection**
 - Heat is transported by moving fluid.
- **Radiation**
 - Emission of energy by electromagnetic waves
- **Phase Change**
 - State of matter changes which either absorbs or emits heat



- **Modes of Heat Transfer**
- **Basic Phenomena of Heat Transfer**
 - **Conduction**
 - **Convection**
 - **Radiation**
 - **Phase Change**
- **The Energy Equation**
- **Boundary Conditions**

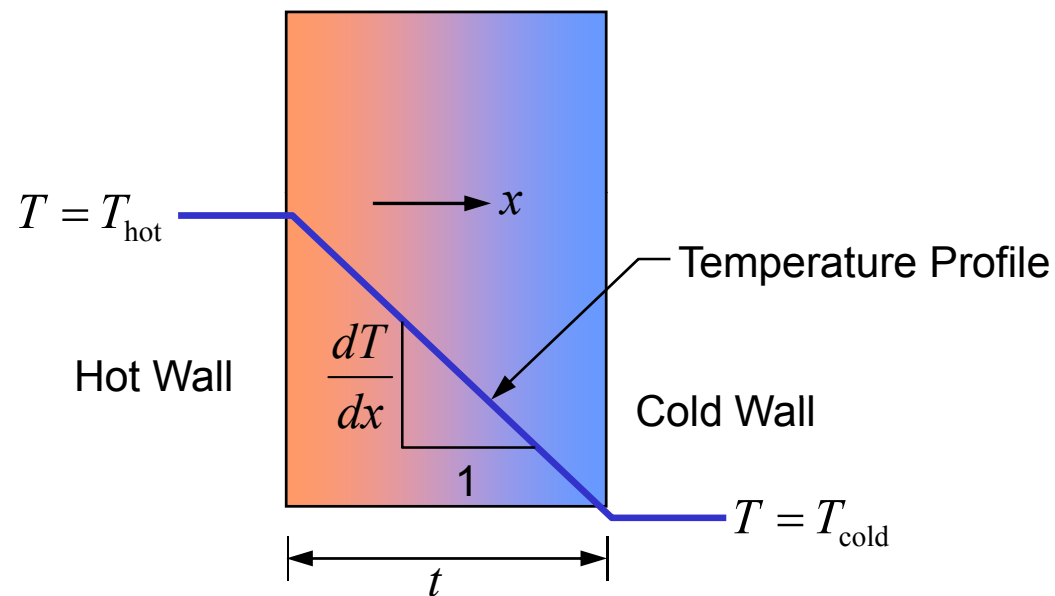
Conduction Heat Transfer

- **Conduction is the transfer of heat by molecular interaction.**
 - **Gases – Molecular velocity depends on temperature. Hot, energetic molecules collide with neighbors which increases their speed.**
 - **Solids – Molecules and the lattice structure vibrate.**
- **Fourier's Law states that heat flux is proportional to temperature gradient. Mathematically,**

$$\frac{Q}{A} = q = -k \nabla T = -k \left(\frac{\partial T}{\partial x} + \frac{\partial T}{\partial y} + \frac{\partial T}{\partial z} \right)$$

↑
Thermal conductivity
(not necessarily constant)

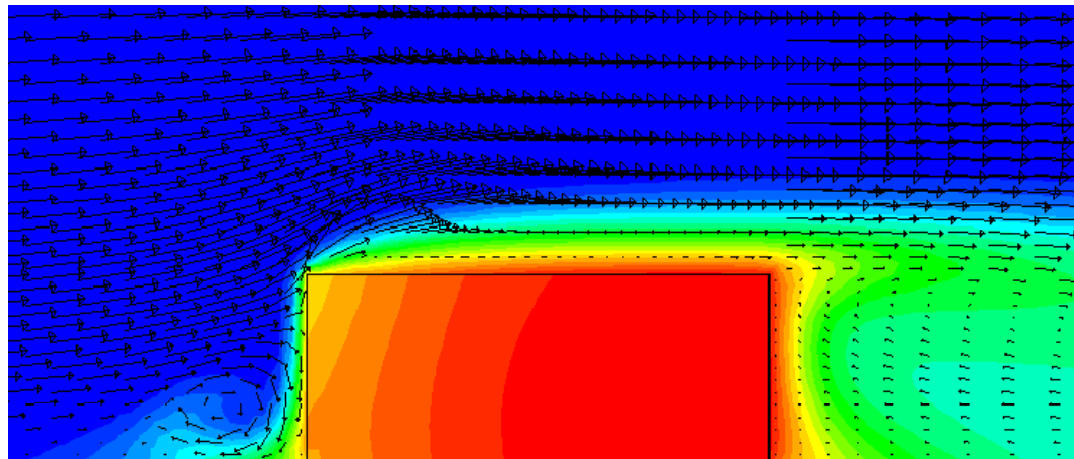
- For a simple 1D steady state conduction, the temperature profile through a slab is linear.



- This leads to the concept of thermal resistance:

$$T_{\text{hot}} - T_{\text{cold}} = R Q \qquad R = \frac{t}{k A}$$

- **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.
- **Example – When cold air flows past a warm body, it draws away warm air near the body and replaces it with cold air**



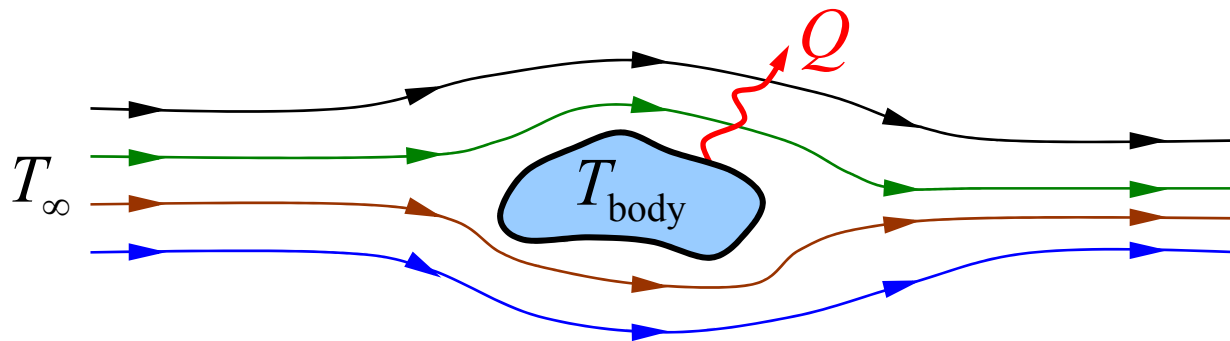
Flow and heat transfer past a heated block

Newton's Law of Cooling

- Newton's law of cooling states that

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

Average 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

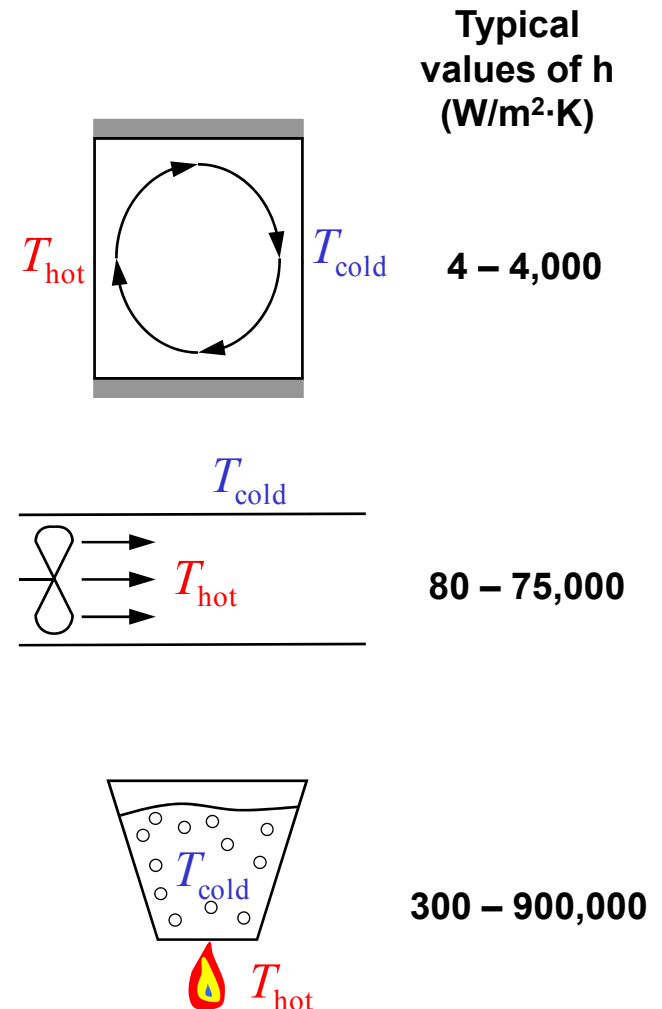
$$h \propto \Delta T^{1/4}, h \propto \Delta T^{1/3}$$

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

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

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

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



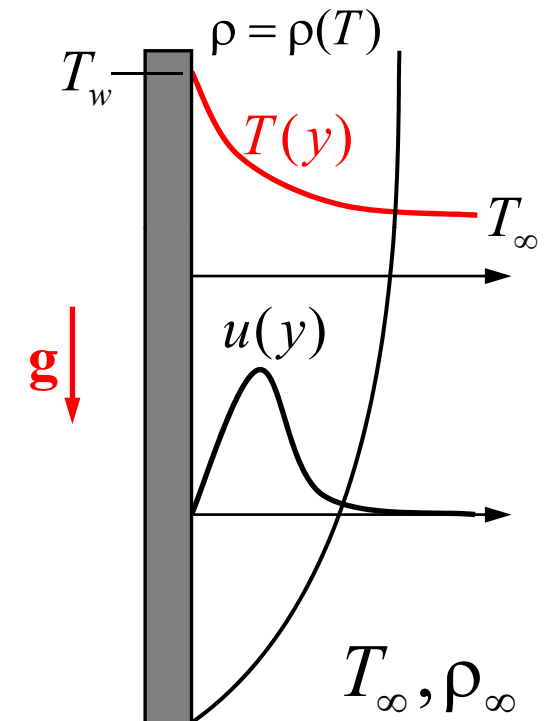
Natural Convection

- In natural convection, fluid motion occurs due to buoyancy effects.
 - As the fluid is heated, its density decreases.
 - This density gradient causes a buoyant force to be generated which induces flow opposite to gravity.
 - The buoyancy force (per unit volume) can be computed by

$$f_B = (\rho - \rho_\infty) g$$

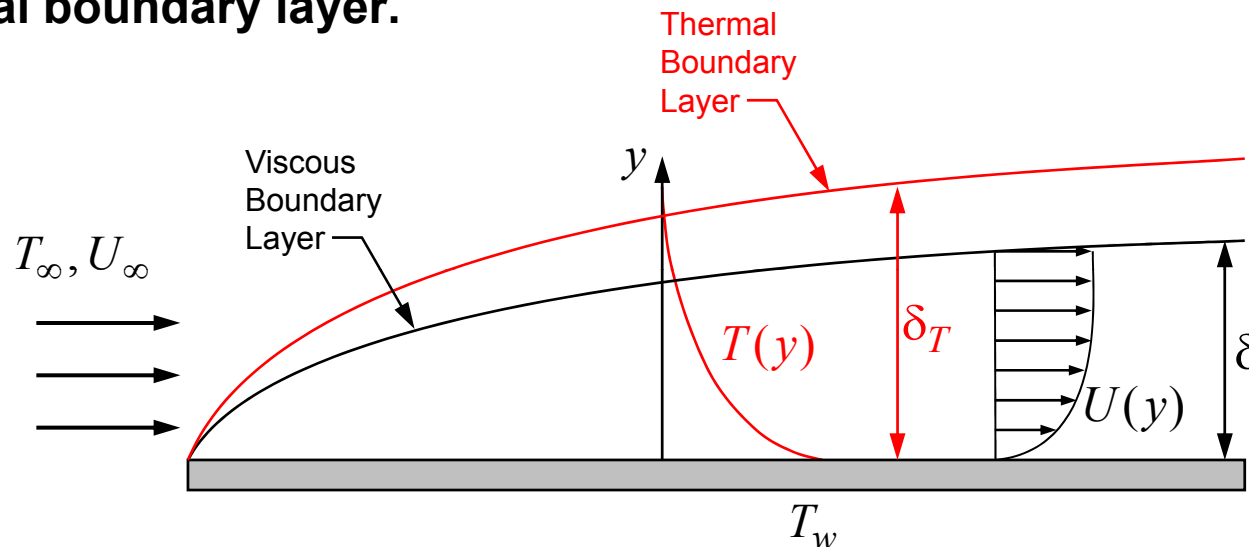
- Natural convection problems are characterized using the Rayleigh number.

$$Ra_L = \frac{\beta g L^3 \Delta T}{\alpha \nu} \begin{cases} Ra_x < 10^8 \text{ laminar flow} \\ Ra_x \approx 10^9 \text{ transition} \\ Ra_x > 10^{10} \text{ turbulent flow} \end{cases}$$



Boundary Layer Flow

- Analogous to the viscous boundary layer that develops, there is also a thermal boundary layer.



- In most industrial applications, free and forced convection occur simultaneously. The relative magnitude of these effects can be determined by using a modified Froude number, Fr .

$$Fr = \frac{Gr}{Re^2} = \frac{\beta g L \Delta T}{U^2}$$

$\ll 1$ Forced convection dominates
 ≈ 1 Natural and Forced convection are important
 $\gg 1$ Natural convection dominates

Nusselt Number

- The Nusselt number (Nu) represents the relative magnitude of “real” heat flux to conduction heat flux.
- Nusselt number derivation
 - Equate the heat conducted from the wall to the same heat transfer in convective terms:

$$k \frac{\partial T}{\partial y} = h (T_w - T_\infty)$$

- Define dimensionless quantities: $\tilde{T} = \frac{T_w - T}{T_w - T_\infty} \quad \tilde{y} = \frac{y}{L}$

- Rearrange:

$$\frac{\partial \tilde{T}}{\partial \tilde{y}} = \frac{h L}{k} = \text{Nu}_L$$

Nusselt number
(dimensionless heat
transfer coefficient)

- **Correlations can be used in FLUENT to set appropriate boundary conditions.**

- **Examples:**

- **Heat transfer resulting from flow around a sphere**

$$\text{Nu}_D = 2 + 0.6 \text{Re}_D^{1/2} \text{Pr}^{1/3}$$

- **Heat transfer from a flat plate in laminar flow (thermal boundary layer)**

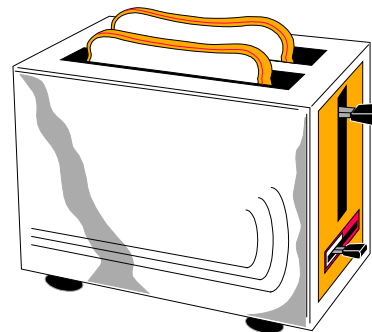
$$\text{Nu}_D = 0.332 \text{Re}_D^{1/2} \text{Pr}^{1/3}$$

- **The above correlations depend on the Prandtl number:**

$$\text{Pr} = \frac{\text{Momentum Diffusivity}}{\text{Thermal Diffusivity}} = \frac{\nu}{\alpha} = \frac{\mu C_p}{k}$$

Typical Prandtl Numbers	
Liquid metals	0.01
Most gases	0.7
Water at ambient conditions	6

- Thermal radiation is an emission of energy via electromagnetic waves
- Intensity depends on body temperature and surface characteristics
- Important mode of heat transfer at high temperatures
- Real-world examples
 - Toaster, grill, broiler
 - Fireplace
 - Sunshine



Radiation – Black Bodies

- A black body is a model of a perfect radiator and has specific characteristics.
 - Absorbs 100% of incident radiation ($\alpha = 1$).
 - Reflects and transmits no incident radiation ($\rho = \tau = 0$)
- The energy emitted by a black body obeys the Stefan-Boltzmann Law.

$$q = \frac{Q}{A} = \sigma T^4$$

↑ Stefan-Boltzmann constant
 $5.6697 \times 10^{-8} \text{ W/m}^2 \cdot \text{K}^4$

- This energy emission (q) represents the theoretical maximum at the temperature of interest (T).

Radiation Heat Transfer – Real (Gray) Bodies

- In general, real bodies emit less radiation than a black body

$$Q_{\text{rad}} = \varepsilon A_s \sigma T^4$$

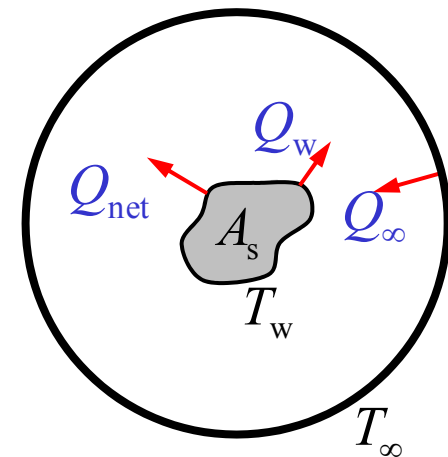
Surface area

Surface emissivity ($0 < \varepsilon < 1$)

- ◆ Consider radiation emitted by a small body which has temperature T_w and surface area A_s to its surroundings which are at temperature T_∞ .

- Both the body and its container emit thermal radiation.
- The net heat transfer is from the hotter body to the colder body.

$$Q_{\text{net}} = \varepsilon A_s \sigma (T_w^4 - T_\infty^4)$$



When Is Radiation Important?



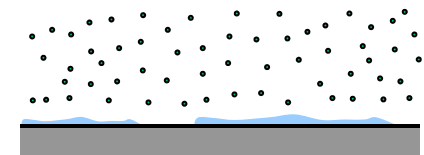
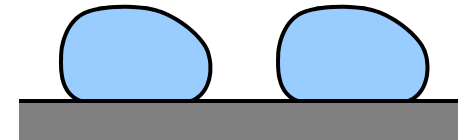
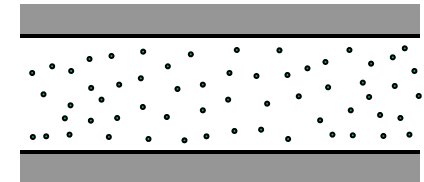
- Radiation heat transfer is significant in high temperature applications such as combustion.
- Radiation properties can be strongly dependent on chemical composition, especially CO₂, H₂O.
- Radiation heat transfer equations are difficult to solve explicitly (except for simple configurations) – we must rely on computational methods.
- Radiation should always be considered when radiation heat transfer is large compared to the heat transfer due to conduction or convection

$$q_{\text{rad}} = \varepsilon \sigma T^4$$

- **Heat transfer due phase change can be present in many forms.**
 - **Condensation**
 - **Evaporation**
 - **Boiling**
 - **Solidification / Melting**
- **In order to adequately model these phenomena, we often must rely on multiphase models and user-defined functions (UDFs).**

Phase Change Heat Transfer – Condensation

- **Condensation is the transformation of a substance from vapor to liquid resulting from energy removal from the vapor phase.**
- **In condensation processes, the vapor temperature is at or below the saturation temperature.**
- **Condensation occurs in various modes.**
 - **Droplet formation in vapor**
 - Droplets may form on particulate matter either in vapor or homogeneously
 - **Liquid droplet formation on a cooled surface**
 - Droplets appear on the surface.
 - Droplets grow until they either run off of the surface under gravity OR the liquid temperature reaches the saturation temperature
 - **Liquid film condensation on a cooled surface**
 - Latent heat of condensation transferred from liquid-vapor interface to the wall by convection and conduction
 - Vapor phase can consist of one or more chemical species.

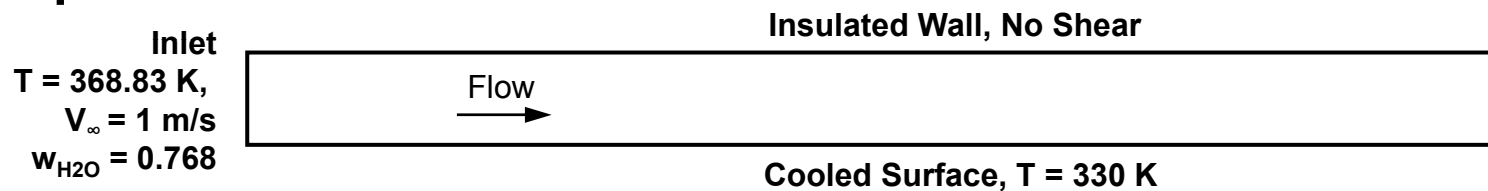


Condensation Example



Customer Training Material

- **CFD simulation of humid air condensation on a cooled surface**
 - Multiple species
 - Turbulence
 - UDF to compute saturation pressure and mass flux
- **Computational domain**

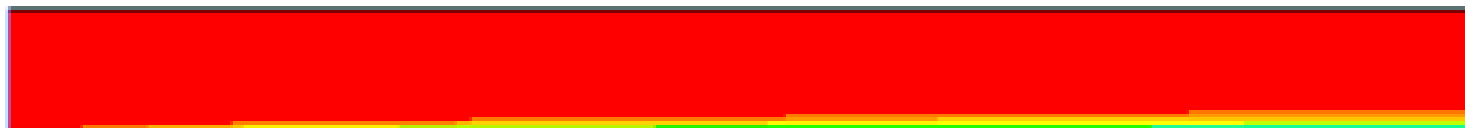
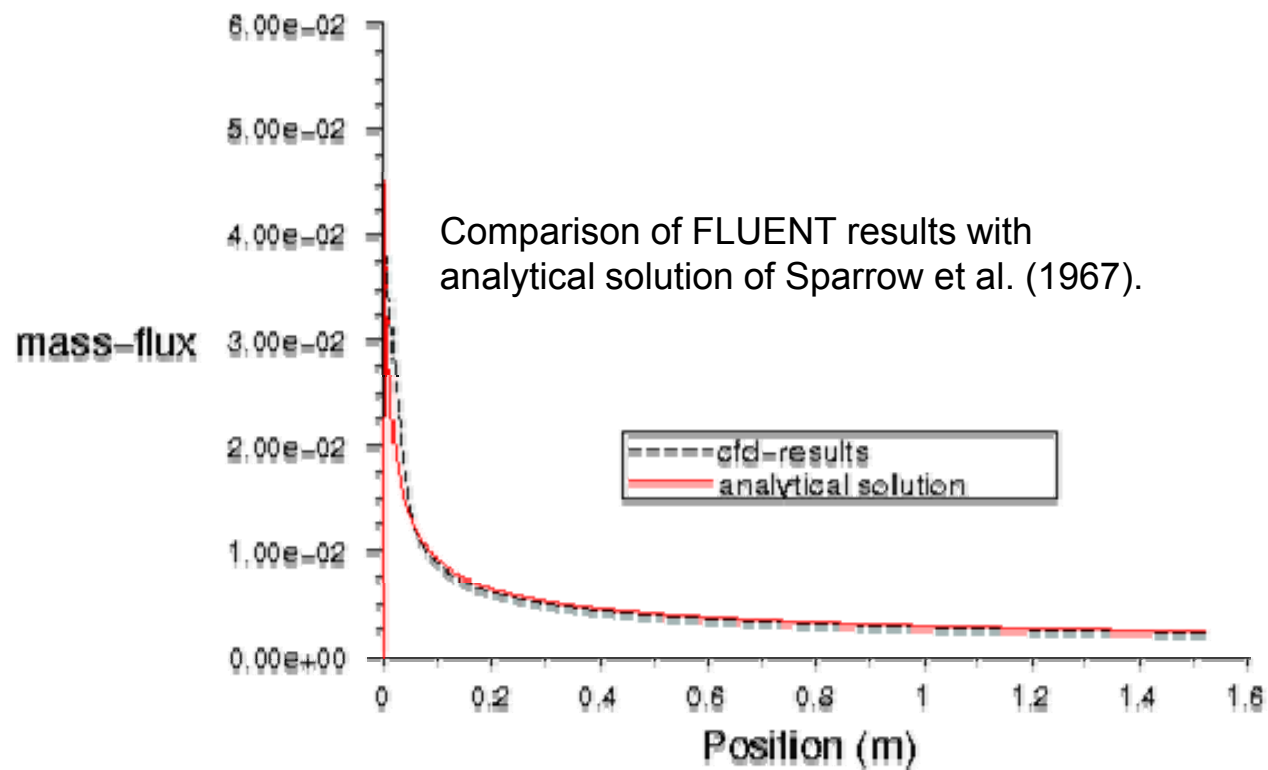


- **Mass transfer rate at cooled surface controlled by diffusion**

$$\dot{m}_{\text{H}_2\text{O}} = \frac{\rho D}{\omega_{\text{H}_2\text{O}} - 1} \frac{\partial \omega_{\text{H}_2\text{O}}}{\partial n}$$

- **This mass flux applied as a source term in cell adjacent to the wall.**
- **No film motion calculated.**

Condensation Example



Predicted Temperature Distribution ($331\text{ K} < T < 370\text{ K}$)

Phase Change Heat Transfer – Evaporation

- **Evaporation is the transformation of a substance from liquid to vapor resulting from energy addition.**
- **FLUENT contains a model to account for mass transfer due to evaporation.**
 - **Liquid interfacial temperature must be equal to or greater than the vapor saturation temperature at the interface.**
 - **Mass transfer rate is proportional to the difference in partial pressure between the interface and the bulk vapor (depends on the pressure gradient)**
 - **The FLUENT evaporation model is based on the standard discrete phase model (DPM).**

- The evaporation of water droplets injected into hot air stream is modeled using the DPM vaporization law.

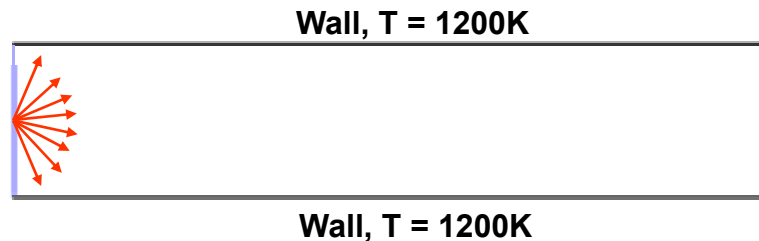
Inlet:

T = 650 K, V = 1 m/s

Water Droplets:

T = 300 K, d = 250 μm.

Total mass flow 0.04 kg/s



$$m_p c_p \frac{dT_p}{dt} = h A_p (T_\infty - T_p) + \frac{dm_p}{dt} h_{fg}$$

C_p = Droplet heat capacity

T_p = Droplet temperature

h = Convection heat transfer coefficient

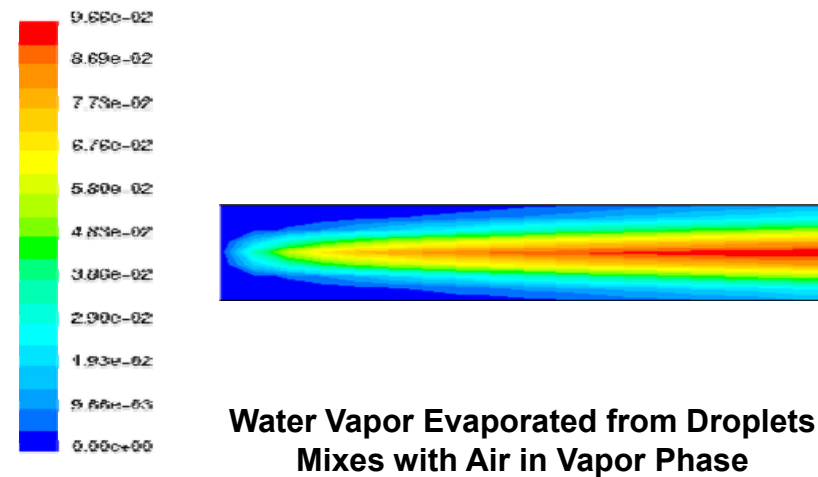
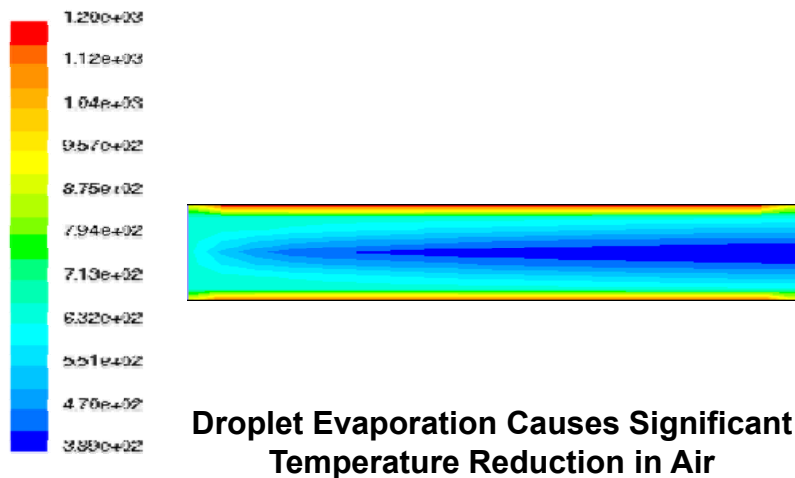
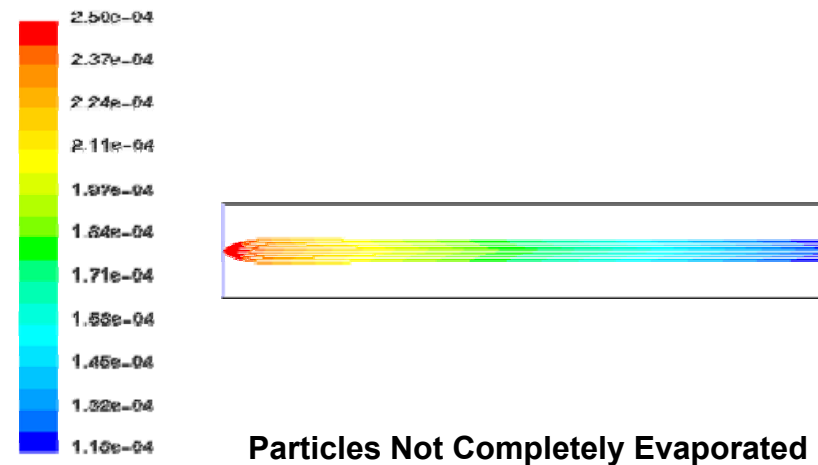
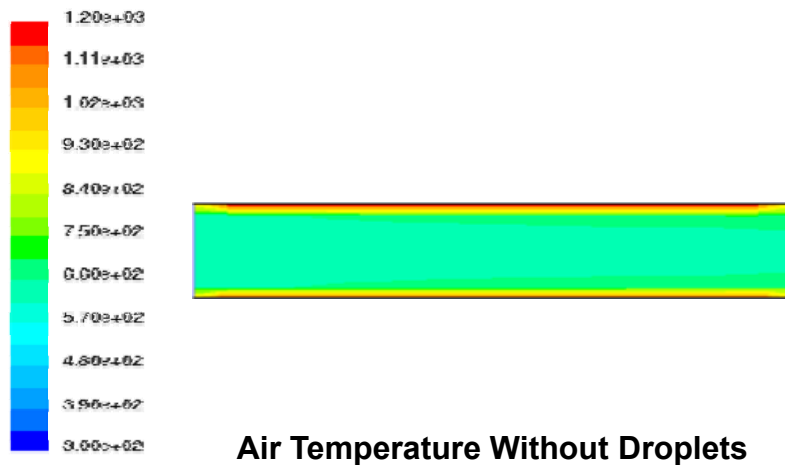
T = Continuous phase temperature

h_{fg} = Latent heat

$\frac{dm_p}{dt}$ = Evaporation rate

Heat Transfer Modeling using ANSYS FLUENT

Evaporation Example



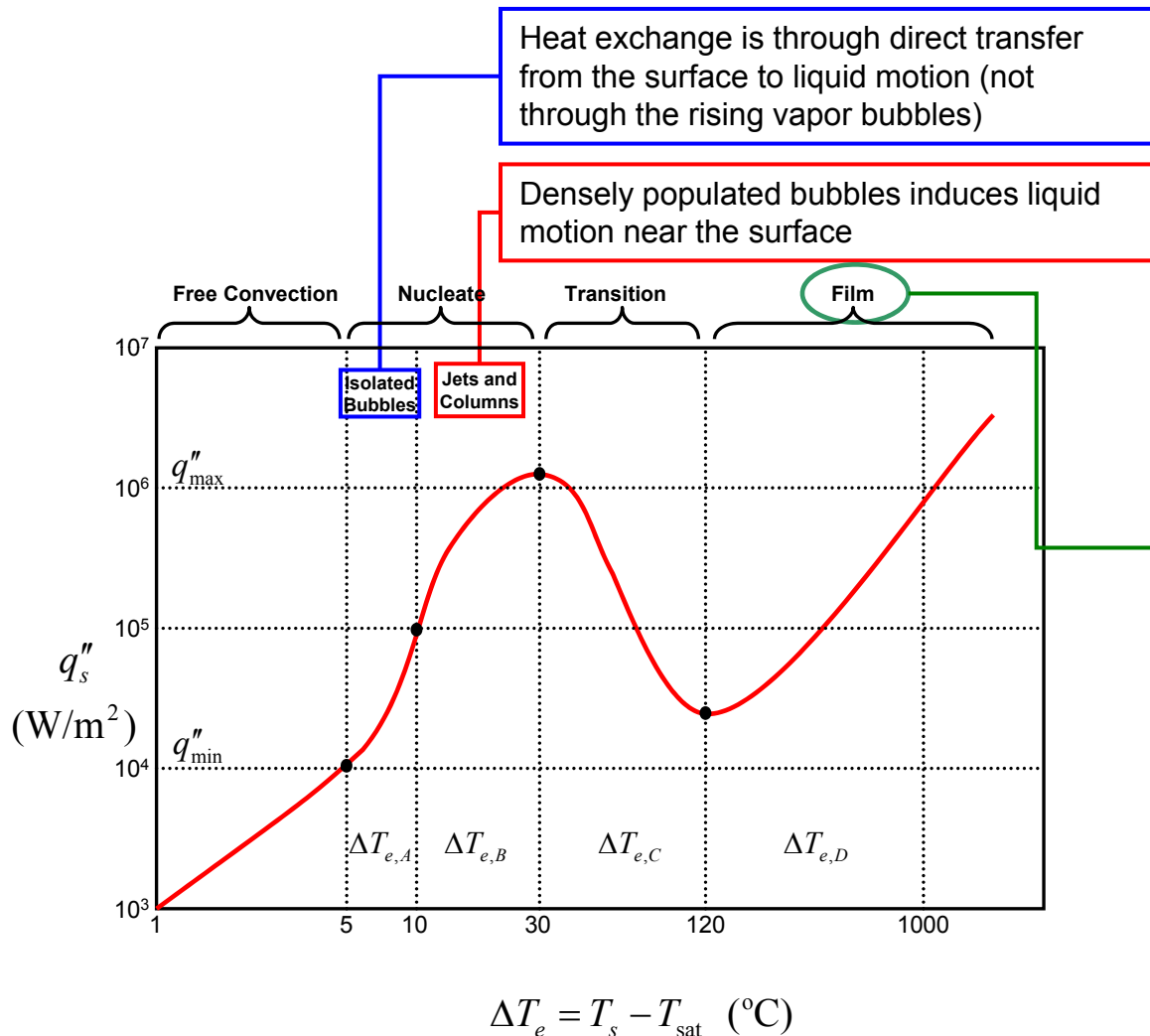
Phase Change Heat Transfer – Boiling

- **Boiling is a fluid phase change that is classified as a convection mode of heat transfer (involves fluid motion).**
- **This process occurs at solid-liquid interfaces.**
 - **Characterized by the formation of vapor bubbles at the solid surface**
 - **Bubbles grow and detach from the surface.**
- **Because of the phase change, large heat flux can be achieved with relatively small temperature differential.**
- **Heat transfer is given by**

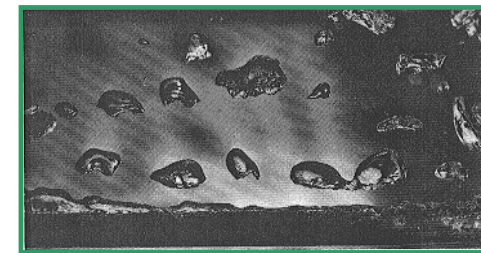
$$q_s = h \Delta T_e$$

$$\Delta T_e = T_{\text{surf}} - T_{\text{sat}}$$

$$h = h(\Delta T, g(\rho_l - \rho_v), h_{fg}, C_p, k, \mu, L, \sigma)$$



Surface completely covered by vapor film. Heat transfer from surface to liquid and conduction through vapor phase.

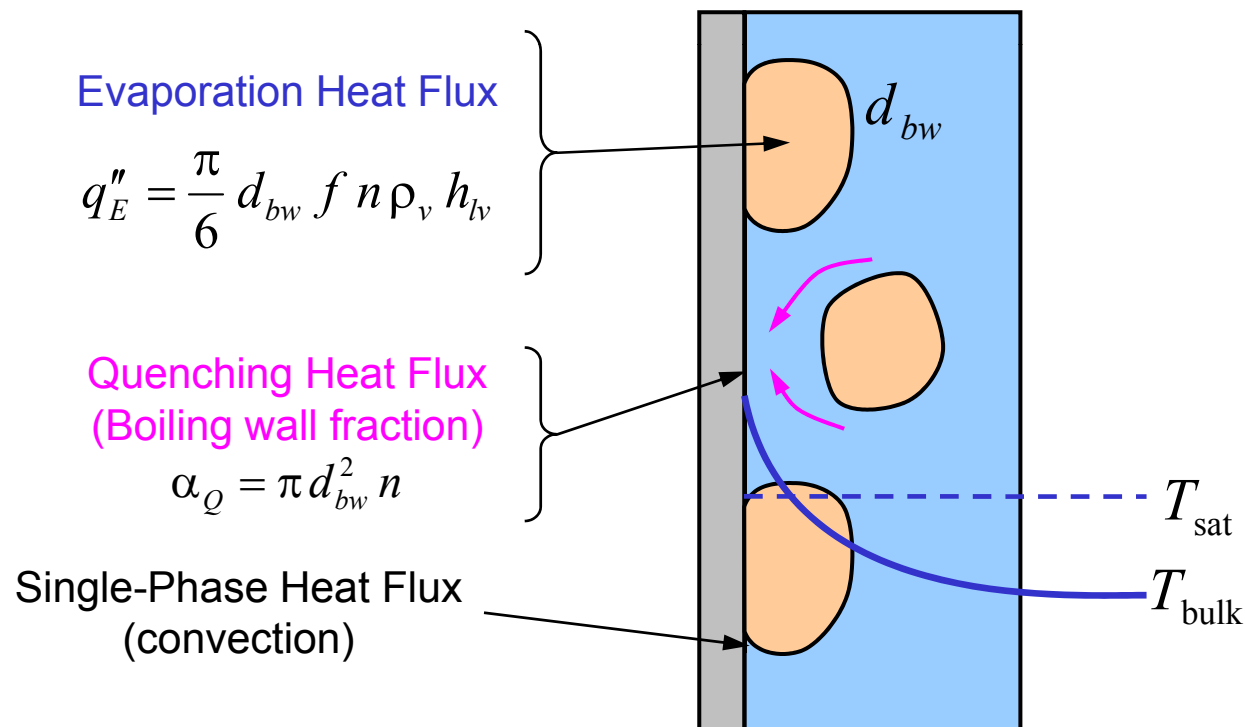


• Modeling Strategy

– Euler-Euler or mixture model

– UDF to handle heat flux calculations

$$q_w'' = \underbrace{q_E''}_{\text{Evaporation}} + \underbrace{\left(h_{sp} \alpha_{sp} + h_Q \alpha_Q \right)}_{\text{Convection}} (T_w - T) \quad \underbrace{}_{\text{Quenching}}$$



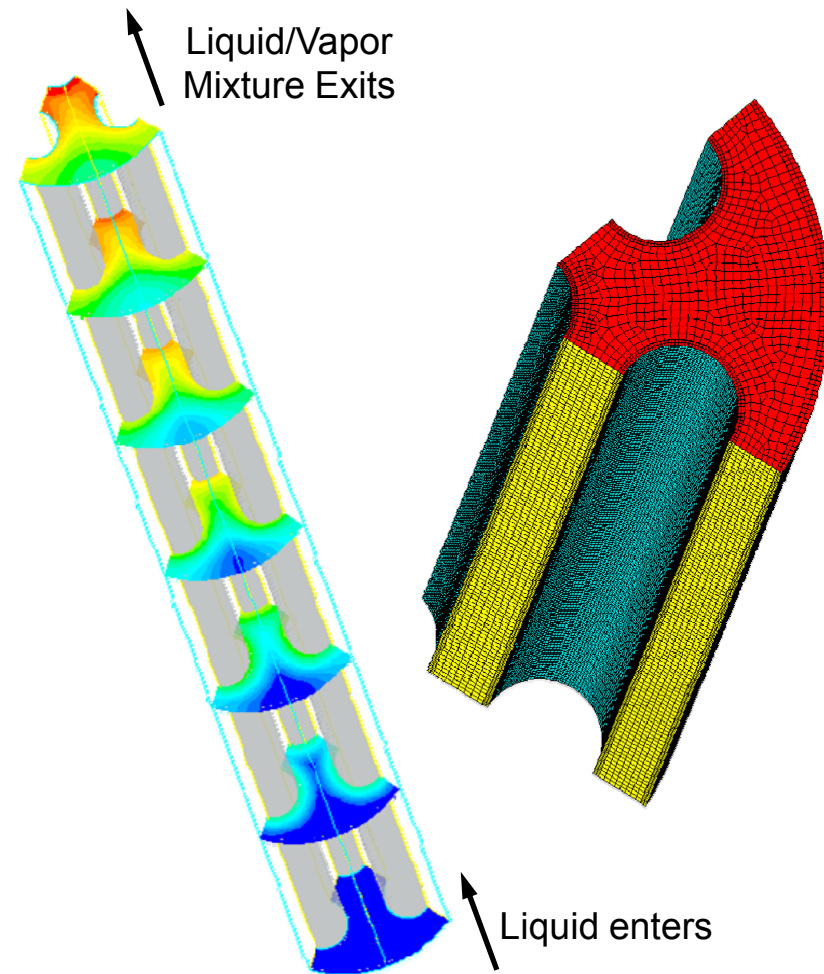
Heat Transfer Modeling using ANSYS FLUENT

Boiling Example – Nuclear Reactor



- **Flow in nuclear fuel assembly**

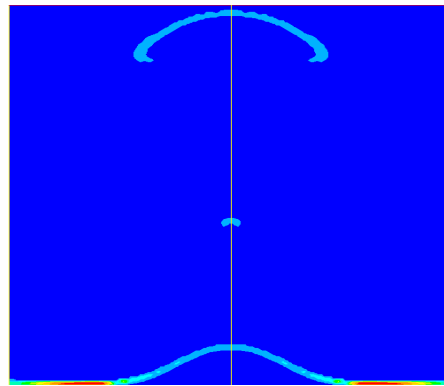
- **Pressure = 50 atm**
- **$Re_{liq} = 300,000$**
- **Heat flux = 0.522 MW/m^2**
- **Inlet sub cooling = 4.5 K**
- **Mesh adapted to $y^+ = 100$**



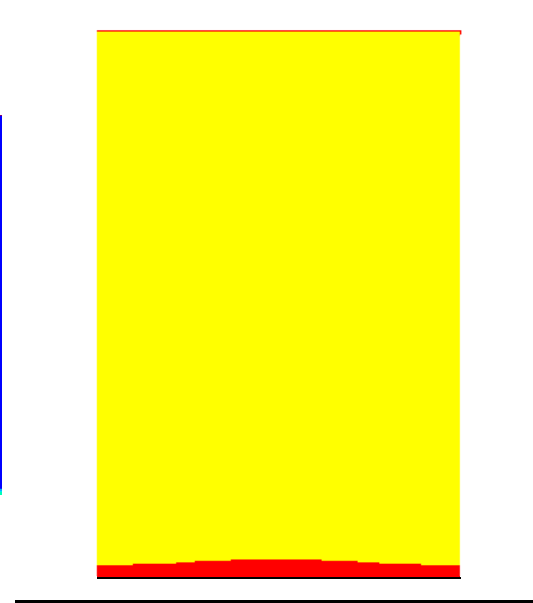
Film Boiling

- Interfacial heat and mass transfer taking into account via a UDF
- Application:
 - Film boiling modeling with VOF
 - UDF and tutorial available on www.learningcfd.com

Contours of Mass
Transfer Rate
(kg/m³/s)



Contours of Vapor
Volume Fraction

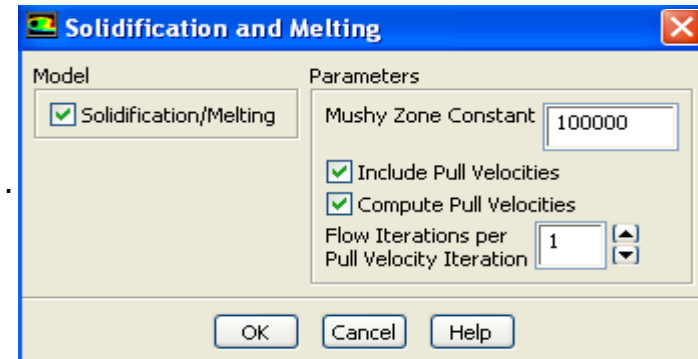


- **Solidification is the transformation of a substance from liquid to solid**
 - Temperature decrease (more frequently encountered) and change of state occurs at the freezing point
 - Pressure increase (in this case temperature remains constant)
 - The solidification process starts with small solid nucleation in the liquid that increases in number with time (until liquid is completely solidified).
 - Application – Casting Process
- **Melting is the transformation of a substance from solid to liquid**
 - Temperature increases, the change of state occurs at the melting point.
 - In general, the melting point is relatively insensitive to pressure.
 - Application – Deicing
- **Freezing & melting point are often equal (certain materials can have different values).**

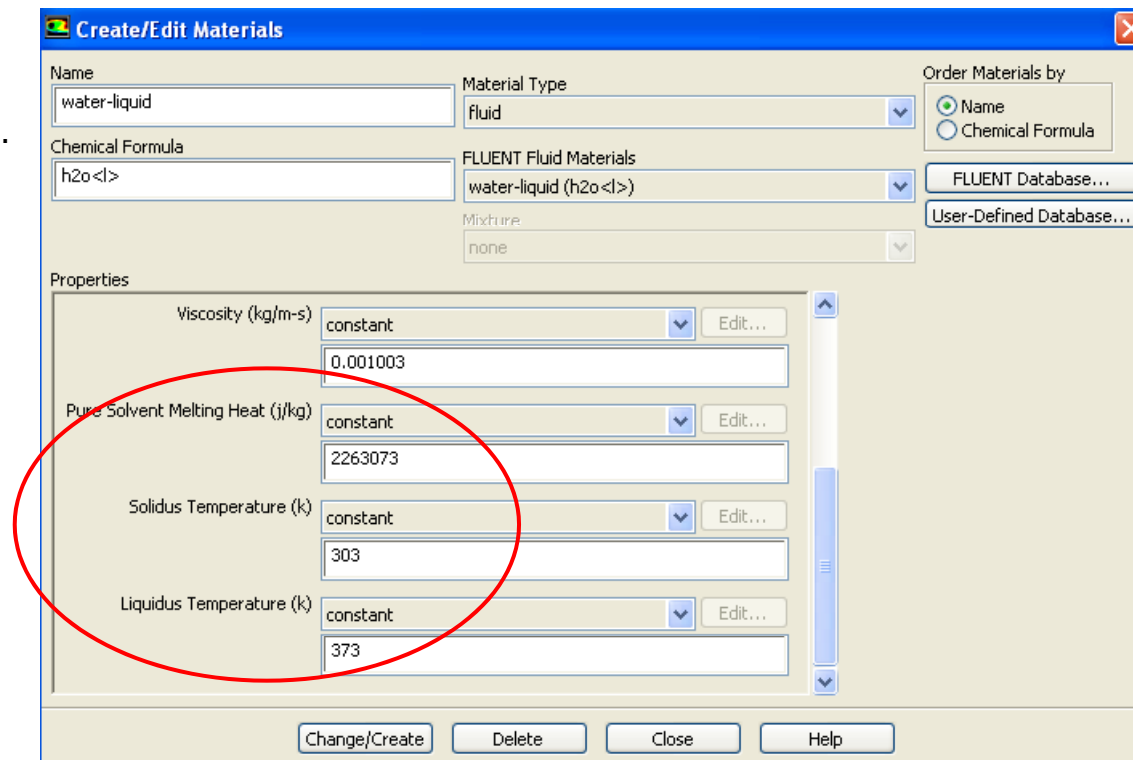
Heat Transfer Modeling using ANSYS FLUENT

Solidification and Melting

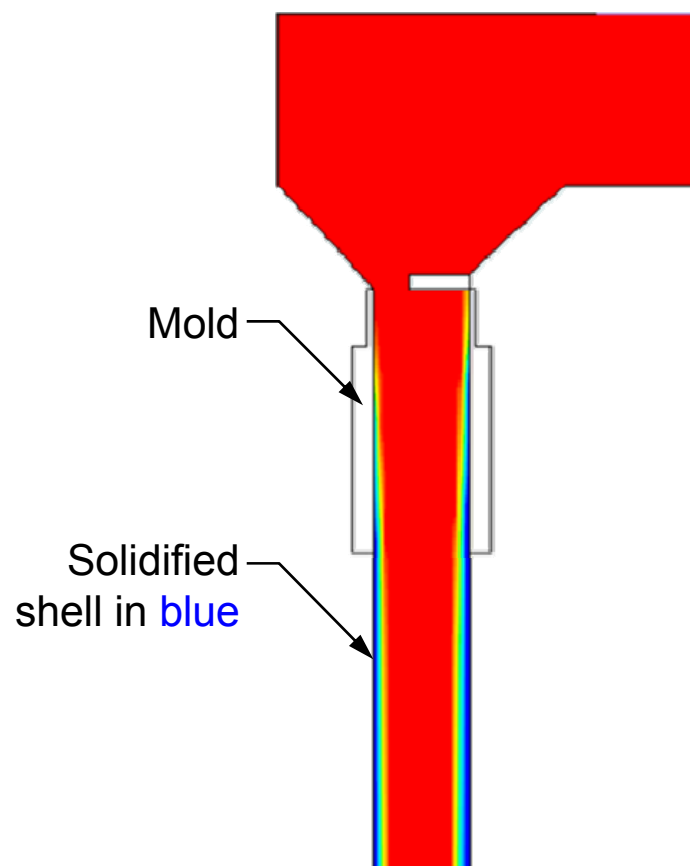
Define → Models → Solidification and Melting...



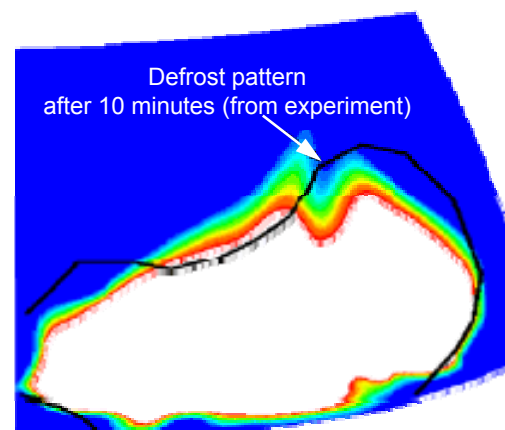
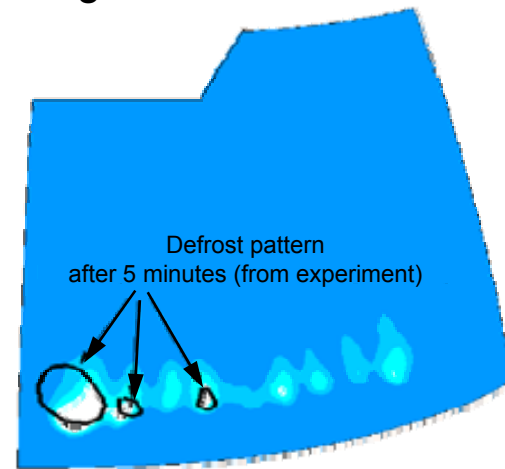
Define → Materials...



Solidification Example Continuous Casting Process



Melting Example Deicing on an Automobile Windshield



- Modes of Heat Transfer
- Basic Heat Transfer Phenomena
 - Conduction
 - Convection
 - Radiation
 - Phase Change
- **The Energy Equation**
- Boundary Conditions

- **General energy transport equation:**

$$\underbrace{\frac{\partial(\rho E)}{\partial t}}_{\text{Unsteady}} + \underbrace{\nabla \cdot [\mathbf{V}(\rho E + P)]}_{\text{Convection}} = \underbrace{\nabla \cdot [k_{\text{eff}} \nabla T]}_{\text{Conduction}} - \underbrace{\sum_j h_j J_j}_{\text{Species diffusion}} + \underbrace{(\bar{\bar{\tau}}_{\text{eff}} \cdot \mathbf{V})}_{\text{Viscous heating}} + \underbrace{S_h}_{\text{Enthalpy source}}$$

- **Energy sources resulting from endothermic/exothermic chemical reactions is included for reacting flows.**
- **Energy source due to species diffusion included for multiple species flows.**
- **Energy source due to viscous heating:**
 - Describes thermal energy created by viscous shear in the flow.
 - Important when the shear stress in fluid is large (e.g., lubrication) and/or in high-velocity compressible flows.
 - Criterion is based on the Brinkman number: $\text{Br} = \frac{\mu U^2}{k \Delta T} \geq 1$
 - Viscous heating is often negligible.
 - Not included by default for the pressure-based solver; always enabled in the density-based solver.

- **Modes of Heat Transfer**
- **Basic Heat Transfer Phenomena**
 - **Conduction**
 - **Convection**
 - **Radiation**
 - **Phase Change**
- **The Energy Equation**
- **Boundary Conditions**

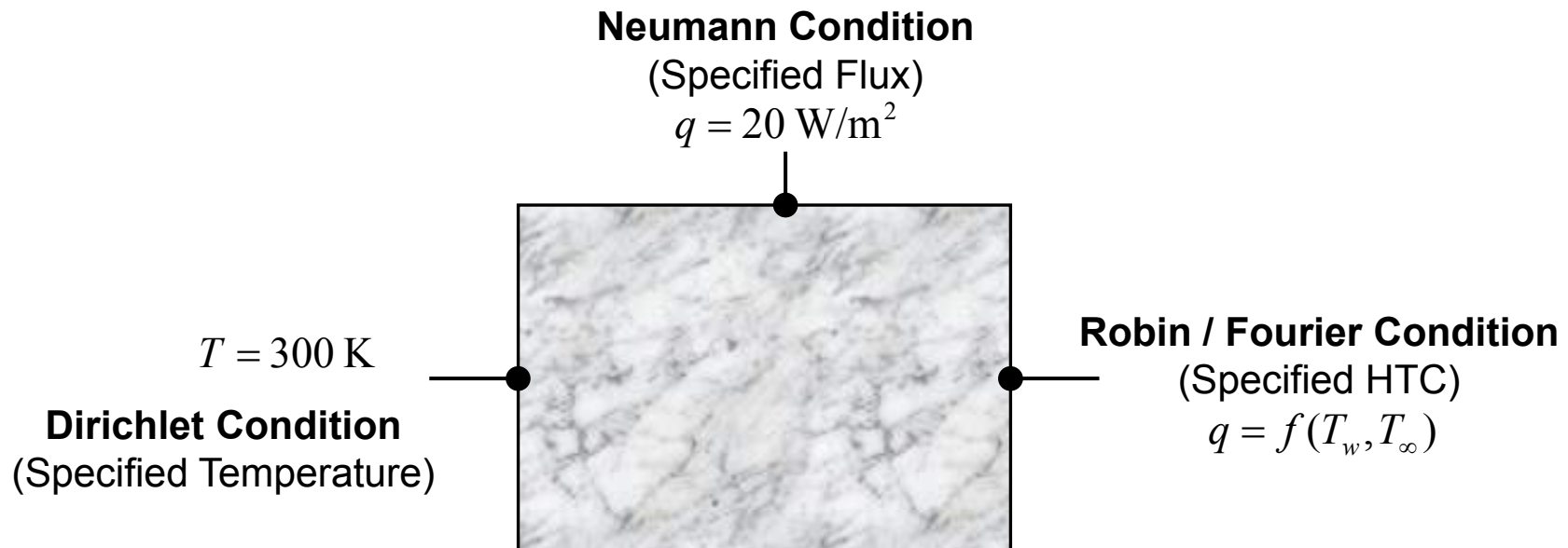
- **Inlets and outlets**
 - Temperature is specified when fluid enters the computational domain.
 - Heat flux includes both convective and diffusive components.
 - Diffusive component can be turned off using the TUI.

`define/models/energy/include diffusion at inlets?` No

- **Walls**
 - Heat Flux
 - Temperature
 - Convection (prescribed heat transfer coefficient)
 - Radiation
 - Mixed (combination of Convection and Radiation)
 - Coupled (only available for zero-thickness internal walls).
- **Thermal resistance**
 - Solid at the boundaries
 - Electrical resistance
 - Tips and Tricks
- **Periodic conditions**

Boundary Conditions

- Thermal boundary conditions generally come in three types.
 - Neumann
 - Robin/Fourier
 - Dirichlet
- Boundary conditions generally represent heat transfer phenomena for the region outside the computational domain.



Wall Boundary Conditions

- Heat Flux (Neumann) boundary condition is specified when heat flux profile or value is known.

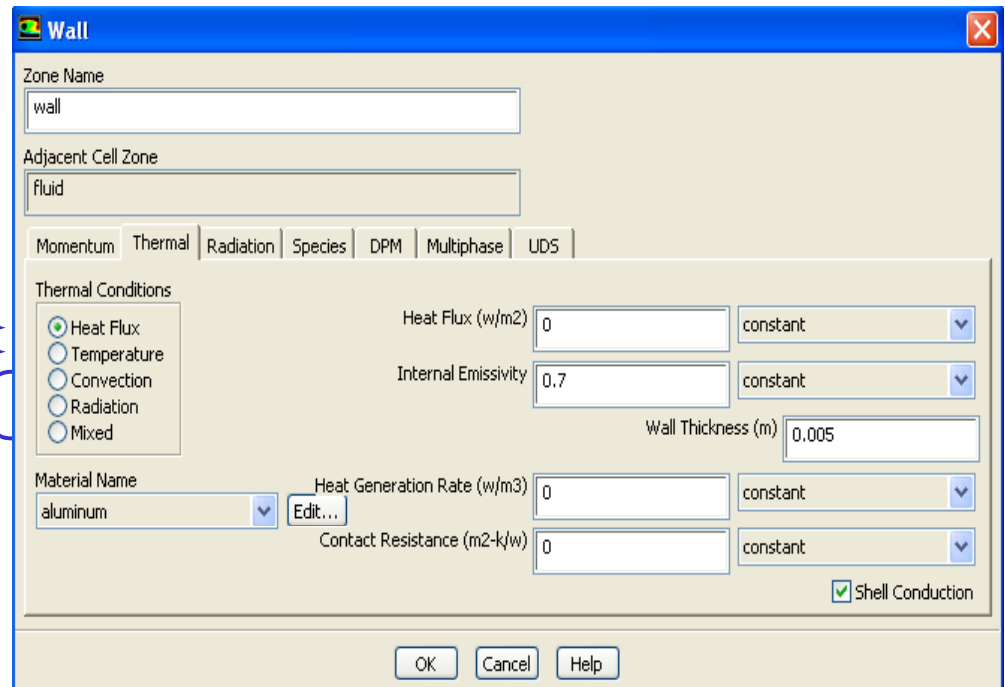
$$q_{\text{conv}} = h(T_{\text{free}} - T_w)$$

- Temperature profile or value can be specified (Dirichlet BC).

$$q_{\text{rad}} = \varepsilon \sigma (T_{\infty}^4 - T_w^4)$$

- Convection, Radiation and Mixed boundary conditions are used to represent convection or radiation exchange with the exterior of the domain.

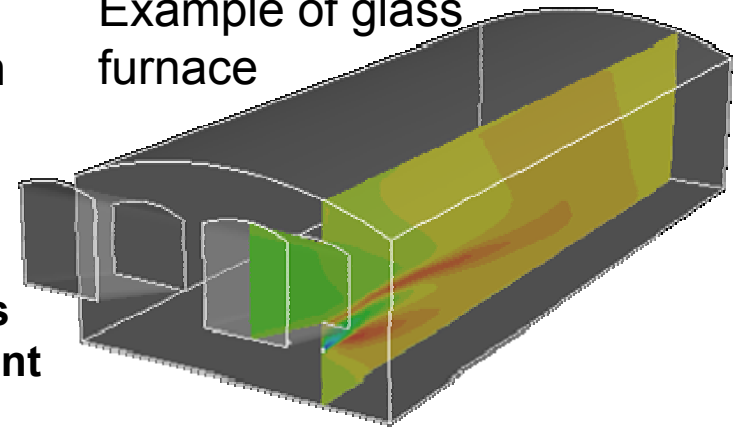
$$q_{\text{mixed}} = q_{\text{conv}} + q_{\text{rad}}$$



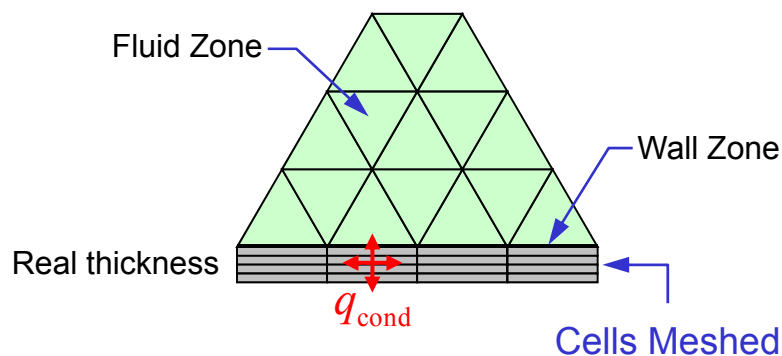
The image shows the 'Wall' dialog box in ANSYS FLUENT. The 'Zone Name' is 'wall' and the 'Adjacent Cell Zone' is 'fluid'. The 'Thermal Conditions' tab is selected, showing options for Heat Flux, Temperature, Convection, Radiation, and Mixed. The 'Heat Flux (w/m2)' is set to 0 with a 'constant' dropdown. The 'Internal Emissivity' is set to 0.7 with a 'constant' dropdown. The 'Wall Thickness (m)' is set to 0.005. The 'Material Name' is 'aluminum' with an 'Edit...' button. The 'Heat Generation Rate (w/m3)' is set to 0 with a 'constant' dropdown. The 'Contact Resistance (m2-k/w)' is set to 0 with a 'constant' dropdown. The 'Shell Conduction' checkbox is checked. The 'OK', 'Cancel', and 'Help' buttons are at the bottom.

- Typically the computational domain of an industrial furnace ends at the boundary between gas and refractories.
 - Option #1: Refractory layers are meshed.
 - Option #2: Thermal resistance approach is used with a zero-thickness wall to represent refractory layers.

Example of glass furnace

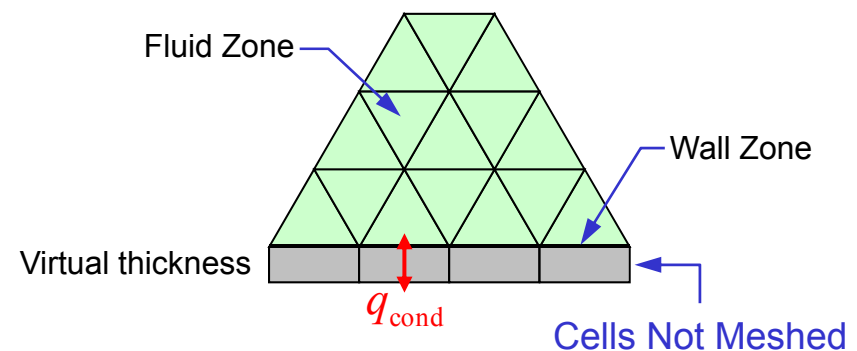


Option #1



Fourier's Law Solved in 3D

Option #2



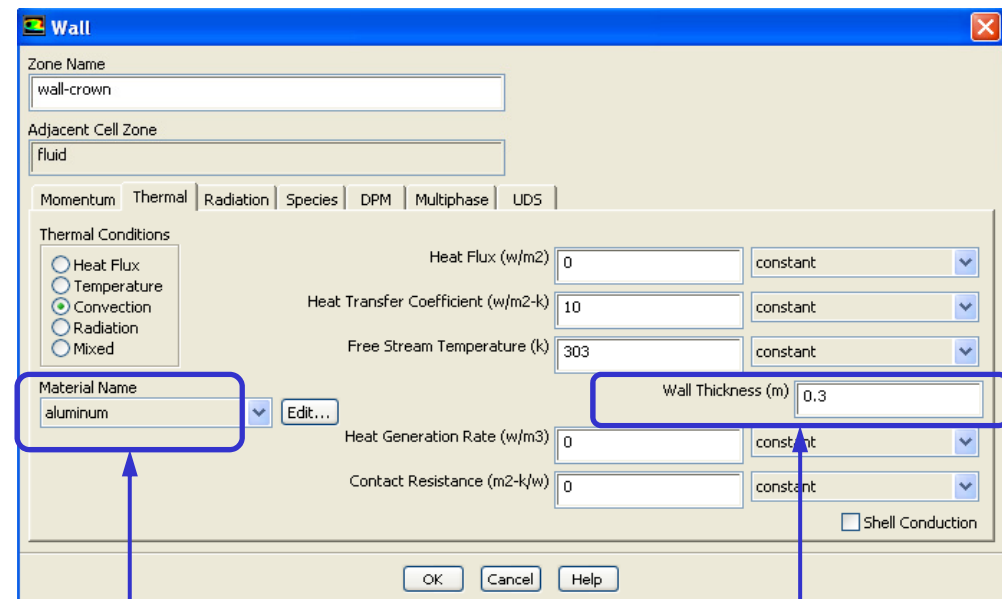
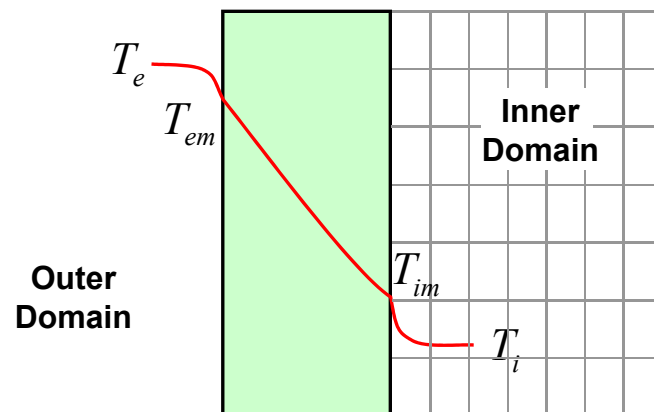
1-D Fourier's Law Introduced Through Thermal Resistance (Assumed Normal Flux Only)

Heat Transfer Modeling using ANSYS FLUENT

Wall Boundary Conditions



- Option #2 – Thermal resistance of the solid material can be applied in FLUENT

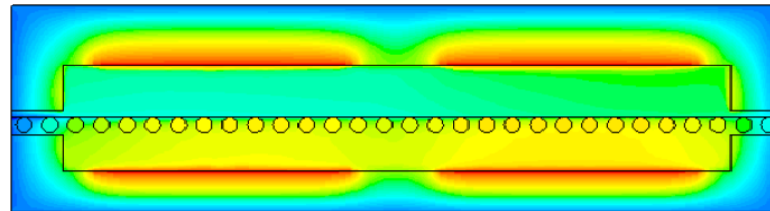


$$R = \frac{t}{kA}$$

Mesh

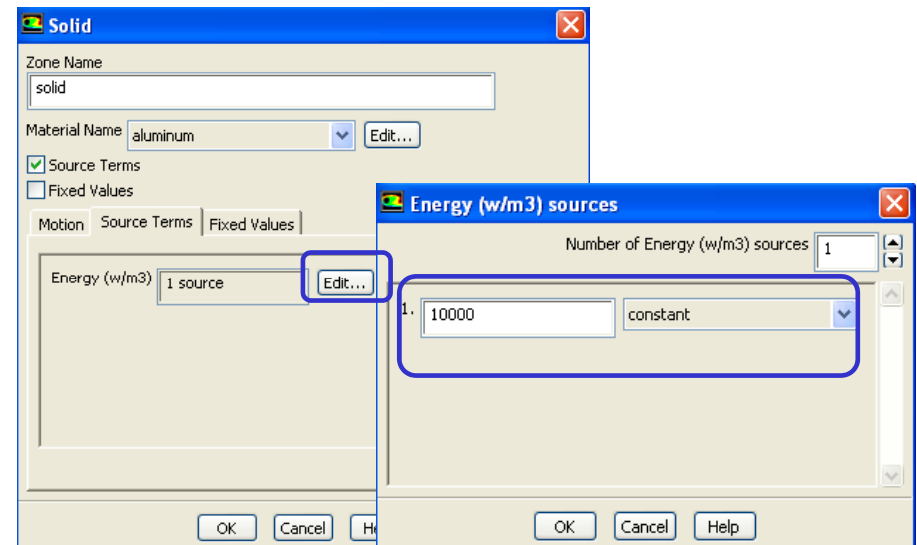
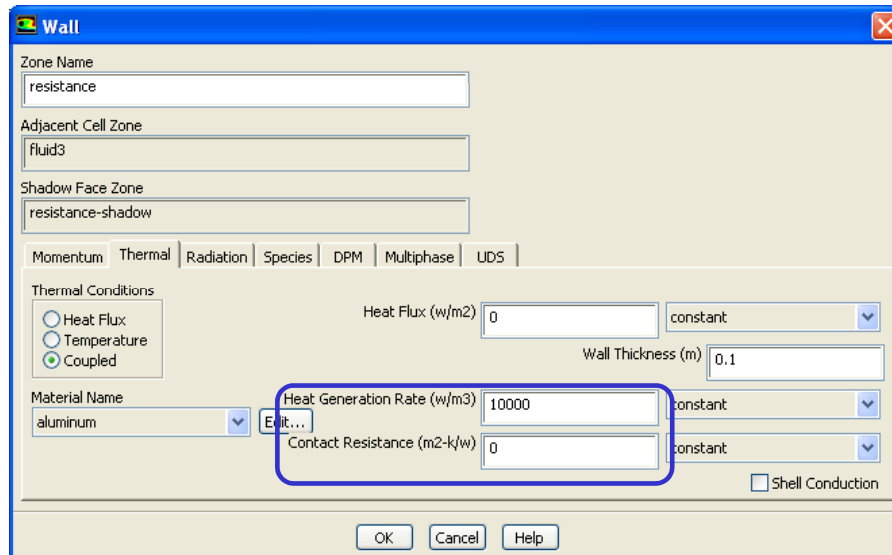
Example – Electrical Resistance in a Furnace

- **Example:** How to represent electrical coils at the top and bottom of the furnace?



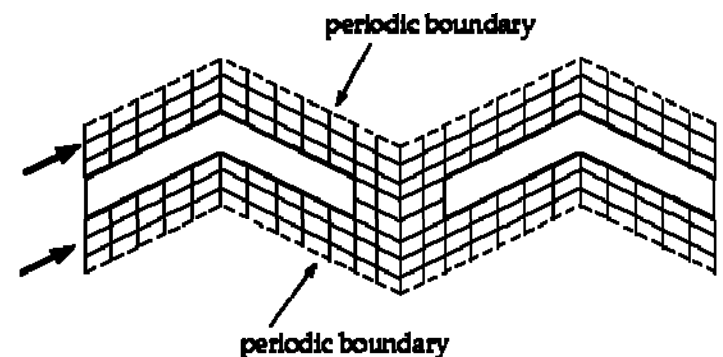
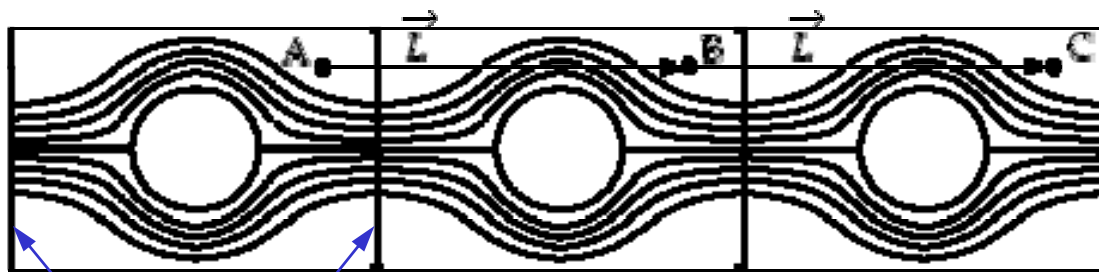
Temperature contours of a continuous glass sheet in a furnace with two heating zones.

(Courtesy PPG Industries Inc)



Periodic Heat Transfer

- Periodic boundary conditions are used when flow and heat transfer patterns are repeated.
- Types of periodic conditions
 - Streamwise
 - Geometry and boundary conditions repeat in the streamwise direction.
 - Constant temperature
 - Uniform heat flux
 - Zero pressure drop
 - No special requirements; thermal conditions at two periods are identical.



Periodic Heat Transfer

- In streamwise periodic problems, temperature is not a periodic function.
- If the temperature is scaled properly, then the scaled temperature does exhibit periodic behavior.

Scaled temperature $\theta = \frac{T - T_{\text{wall}}}{T_{\text{bulk inlet}} - T_{\text{wall}}}$

$T_{\text{bulk inlet}} = \frac{\int_A T \rho \mathbf{V} \cdot d\mathbf{A}}{\int_A \rho \mathbf{V} \cdot d\mathbf{A}}$

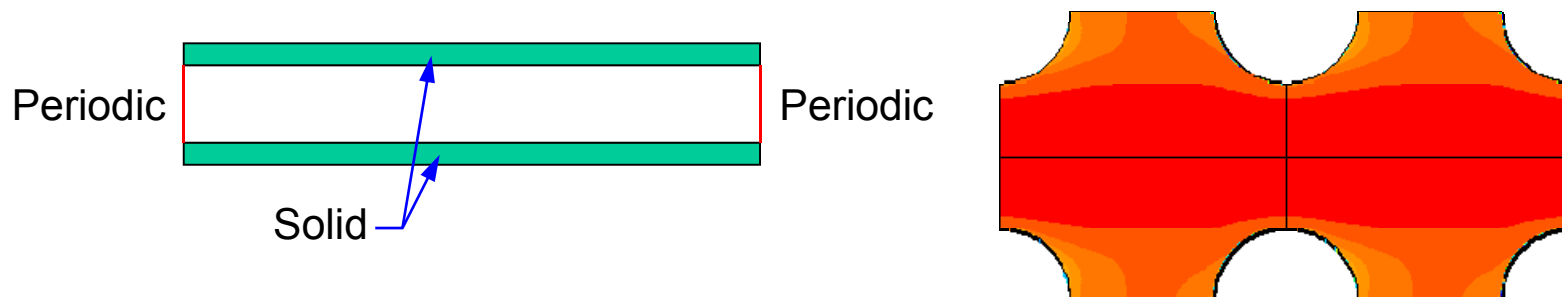
- Streamwise periodic
 - Specified heat flux condition
 - Boundary matching :

$$\frac{T(\mathbf{r} + \mathbf{L}) - T(\mathbf{r})}{L} = \frac{T(\mathbf{r} + 2\mathbf{L}) - T(\mathbf{r} + \mathbf{L})}{L} = \sigma$$

$$\sigma = \frac{Q}{\dot{m} C_p L} = \frac{T_{\text{bulk exit}} - T_{\text{bulk inlet}}}{L}$$

Periodic Heat Transfer Limitations

- **Streamwise periodic heat transfer is subject to the following constraints:**
 - The pressure-based solver must be used.
 - All fixed temperature walls must have the same value; however, varying heat flux on walls is permissible.
 - When constant temperature wall boundaries are used, you cannot include viscous heating effects or any volumetric heat sources.
 - In cases that involve solid regions, the regions cannot straddle the periodic plane (because this may violate one of the above rules).
 - Only constant thermal properties
 - You cannot model species or reacting flows; however they can vary spatially in a periodic manner.
 - Thermodynamic and transport properties cannot be functions of temperature.



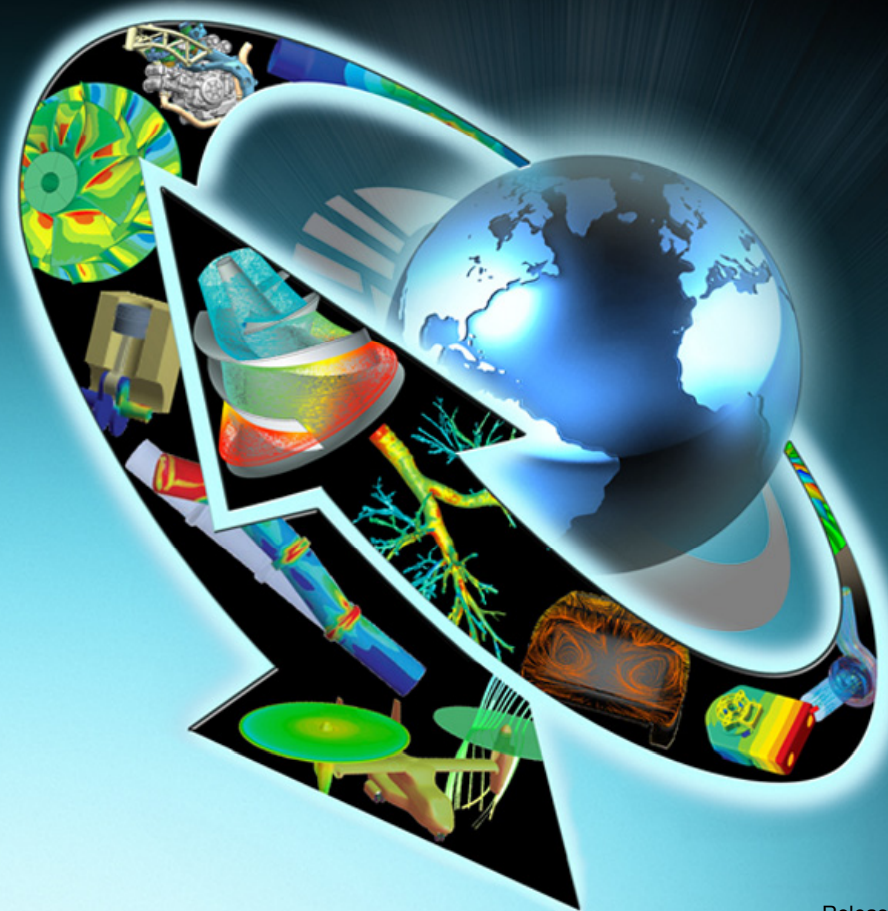


Customer Training Material

Appendix

Lecture 1

Introduction



Tips & Tricks – Thermal Resistance

- Thermal resistance formula in FLUENT is valid for very thin walls or planar surface.

- To model thick walls, typically the effective thickness is specified.

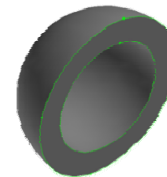
- For composite walls, typically the combined (total) thermal conductivity is specified.



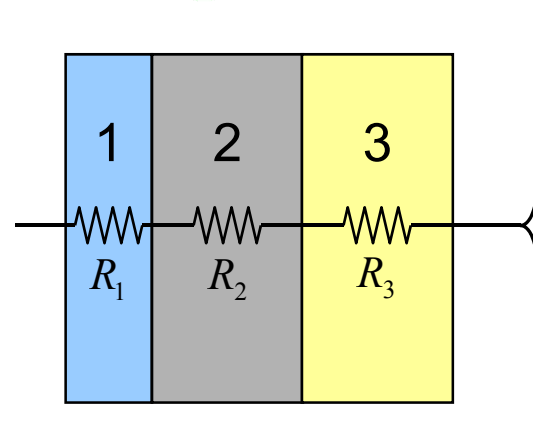
$$R = \frac{e}{k A}$$



$$e_{\text{cylinder}} = R_{\text{inner}} \ln \left(1 - \frac{e}{R_{\text{inner}}} \right)$$



$$e_{\text{sphere}} = R_{\text{inner}} - \frac{R_{\text{inner}}}{R_{\text{inner}} + e}$$



Series Thermal Circuit

$$R_{\text{total}} = R_1 + R_2 + R_3$$

$$e_{\text{total}} = e_1 + e_2 + e_3$$

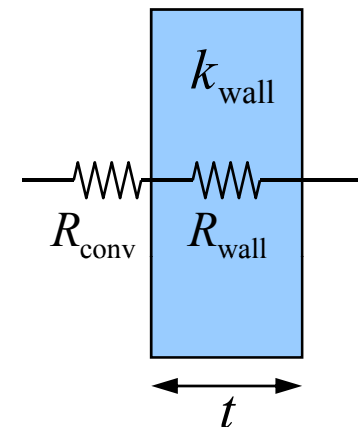
$$k_{\text{total}} = \frac{e_1 + e_2 + e_3}{\frac{e_1}{k_1} + \frac{e_2}{k_2} + \frac{e_3}{k_3}}$$

- Thermal resistance for convection can be defined as

$$R_{\text{conv}} = \frac{1}{h A}$$

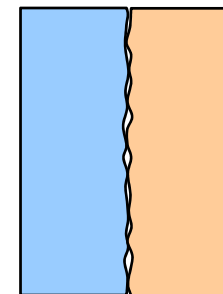
- Thermal resistance can be contained in the heat transfer coefficient

$$\frac{1}{h_{\text{eq}}} = \frac{1}{h} + \frac{t}{k}$$



- Contact thermal resistance

- **IMPORTANT NOTE:** For transient problems, thermal resistance treatment may not be appropriate. Shell conduction or meshed solid may be required.



Order of magnitude for 2 pieces of aluminum (10 μm surface roughness, 10^5 N/m^2) in air
 $R = 2.75 \cdot 10^{-4} \text{ m}^2 \cdot \text{K} / \text{W}$

Tips & Tricks – Free Convection Heat Losses

- Modeling external heat loss for natural convection in ambient air
 - McAdams proposes the following formula (valid for air with $Pr = 0.7$)

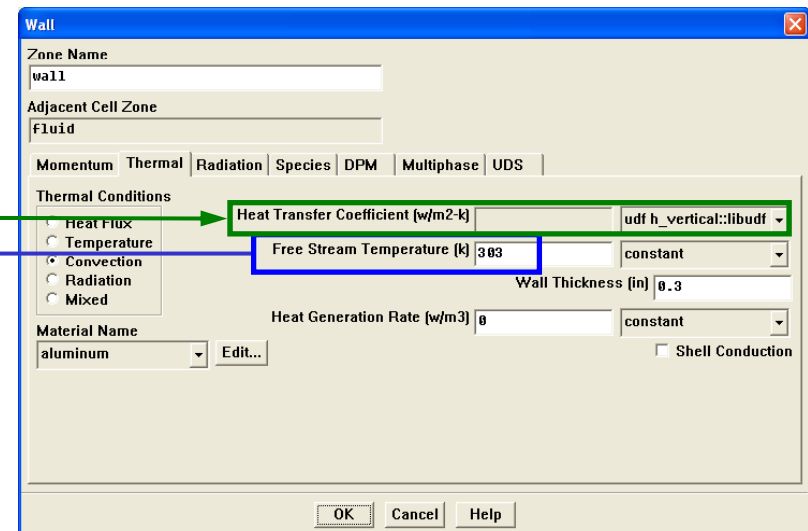
- Horizontal top wall: $C = 2.25\text{--}2.5$
- Horizontal bottom wall: $C = 1.26\text{--}1.36$
- Horizontal or vertical wall ($> 30\text{ cm}$): $C = 1.78\text{--}1.94$

$$h = C (T_{\text{wall}} - T_{\text{free}})^{1/4} \quad h \sim 5\text{--}10 \text{ W/m}^2\cdot\text{K}$$

```
#include "udf.h"

DEFINE_PROFILE(h_vertical,tf,nv)
{
    face_t f;
    real Tfree, Twall;

    begin_f_loop(f,tf)
    {
        Tfree = F_VAR(f,tf,THREAD_VAR(tf).wall.Tinf);
        Twall = fabs(WALL_TEMP_INNER(f,tf));
        F_PROFILE(f,tf,nv) = 1.94*pow((Twall - Tfree),0.25);
    }
    end_f_loop(f,tf)
}
```



Tips & Tricks – Radiation Heat Losses

- **Basic relationship in FLUENT is valid for small convex gray wall in an infinite surrounding.**
- **Radiation heat losses between a black wall and a finite black surrounding surface**
 - External emissivity can be substituted by view factor
- **Radiation heat losses between an infinite plane and an infinite planar surrounding**
- **Heat transfer coefficient for radiation exchange**

$$q_{\text{rad}} = \varepsilon \sigma (T_{\infty}^4 - T_w^4)$$

$$q = F_{w,\text{surr}} \sigma (T_{\text{surr}}^4 - T_w^4)$$

$$q = \frac{\sigma (T_{\text{surr}}^4 - T_w^4)}{\frac{1}{\varepsilon_w} + \frac{1}{\varepsilon_{\text{surr}}} - 1}$$

$$h_{\text{rad}} = \varepsilon \sigma (T_{\infty} + T_w) (T_{\infty}^2 + T_w^2)$$