Lecture 9: Heat Transfer

Introduction to ANSYS CFX
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.

• Learning Aims:
  – You will be familiar with CFX’s heat transfer modeling capabilities and be able to set up and solve problems involving all modes of heat transfer
Mechanisms

• Convection
  – Heat transfer due to the bulk movement of a fluid
  – Forced: flow is induced by some external means
  – Natural or Free: fluid moves due to buoyancy effects
  – Boiling: phase change (not covered in this course)

• Conduction
  – Heat transfer in a fluid or solid due to differences in temperature
  – Conduction is described by Fourier’s law:
    - heat flux is directly proportional to the negative temperature gradient

\[ q_{\text{conduction}} = -k \nabla T \]
Mechanisms

• Radiation
  – Transfer of thermal energy by electromagnetic waves from 0.1 µm (ultraviolet) to 100 µm (mid-infrared)
  – Radiation intensity is directionally and spatially dependent

• Viscous Dissipation
  – Energy source due to viscous heating
  – Important when viscous shear in fluid is large (e.g., lubrication) and/or in high-velocity, compressible flows
Governing Equation : Fluid domain

\[
\frac{\partial (\rho h_{\text{tot}})}{\partial t} - \frac{\partial p}{\partial x} + \nabla \cdot (\rho \mathbf{U} h_{\text{tot}}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (\mathbf{U} \cdot \tau) + S_E
\]

- Transient
- Advection
- Conduction
- Viscous work

**Sources**

- Viscous work
- Advection
- Transient
- Conduction

\( h_{\text{tot}} \) total enthalpy, is related to static enthalpy, \( h \), as follows:

\[
h_{\text{tot}} = h + \frac{1}{2} \mathbf{U}^2
\]

Radiation is solved with a separate radiative transport equation

- Radiation
- Chemical reactions
- Interphase energy transfer

additional models

external energy source
Heat Transfer Models: Fluid Domains

- Heat Transfer option
  - None:
    - Energy Transport not solved
  - Isothermal:
    - Energy Transport not solved, temperature is set for the evaluation of fluid properties
  - Thermal Energy:
    - Energy Transport is solved; kinetic energy effects neglected
    - For low speed flow
  - Total Energy:
    - Transport of enthalpy and kinetic energy effects
    - For Mach number > 0.3 or compressibility effects,
    - For low speed liquid flow when the specific heat is not constant
Heat Transfer Models: Fluid Domains

• If natural convection is important, then switch on the Buoyancy Model
  – For varying density full buoyancy model used
  – For constant density Boussinesq model used

• Thermal Radiation model
  – Should be accounted for when

\[
Q_{rad} = \sigma (T_{max}^4 - T_{min}^4) \geq Q_{conv} = h(T_{wall} - T_{free})
\]

• Several radiation models are available which provide approximate solutions to the RTE (more information in appendix)
Governing Equation : Solid domains

• Heat transfer in a solid domain is modeled using the following conduction equation:

\[
\frac{\partial (\rho h)}{\partial t} + \nabla \cdot (\rho U_s h) = \nabla \cdot (\lambda \nabla T) + S_E
\]

Transient  Solid motion  Conduction  Source

• \( h \) is the sensible enthalpy:

\[
h = \int_{T_{ref}}^{T} c_p dT
\]
• For conduction set Heat Transfer option to Thermal Energy

• Thermal radiation in solids
  – Required only for transparent or semi-transparent materials, e.g. glass; no radiation in opaque solids
  – Monte Carlo model only
Thermal Wall Boundary Condition

- Thermal boundary conditions come in three types, all available in ANSYS CFX:
  - Neumann
  - Robin/Fourier
  - Dirichlet

- They represent heat transfer phenomena outside the domain

**Neumann Condition** (Specified Flux)

\[ q \text{ (W.m}^{-2}\text{)} \]

**Robin/Fourier Condition** (Specified HTC)

\[ h_{\text{conv}} \text{ (W.m}^{-2}.\text{K}^{-1}); q = h_{\text{conv}}(T_{\text{amb}} - T_{\text{body}}) \]

**Dirichlet Condition** (Specified Temperature)

\[ T \text{ (K)} \]
Conjugate Heat Transfer (CHT) - Domain Interfaces

- GGI connection is only option for Fluid-Solid and Solid-Solid interfaces because...
  - GGI interface
    - No discontinuity in values of temperature across the interface
    - CFX Solver calculates a "surface temperature" based on a flux-conservation equation
  - 1:1 interface
    - May result in temperature discontinuity at the interface
- Radiation conditions are set on the side in which radiation is modelled, e.g. the fluid side of a fluid/solid interface
Conjugate Heat Transfer – Modelling Walls

Wall thickness meshed:
• Fourier’s Law Solved in 3D
• Energy equation solved in solid
• Accurate approach → requires more meshing & computational effort

Wall thickness NOT meshed (thin wall):
• 1-D Fourier’s Law through Wall
  Thermal Resistance
  • Artificially models conduction across wall thickness
  • Limitation: conduction is assumed to be normal to the wall
Conjugate Heat Transfer – Thin walls

• Set-up
  – Create a Fluid-Fluid Domain Interface
  – On Additional Interface Models tab set Mass and Momentum = No Slip Wall
  – Enable Heat Transfer toggle and pick:
    • Thin Material and specify a Material & Thickness or Thermal Contact Resistance
    • Note : Other domain interface types (Fluid-Solid etc.) can use these options to represent coatings etc.
Post-Processing Heat Transfer

• Temperature:
  – Local fluid temperature, plotted on a wall it is the temperature at the wall, $T_{wall}$

• Wall Adjacent Temperature:
  – Average temperature in control volume next to wall

• Wall Heat Transfer Coefficient, $h_c$:
  – Based on $T_{wall}$ and the Wall Adjacent Temperature by default
  – To base it on some far-field value instead of the Wall Adjacent Temperature, use the Expert Parameter “tbulk for htc”

• Wall Heat Flux, $q_w$:
  – Total heat flux into the domain by all modes
  – Available on all boundaries

$$q_w = h_c (T_{wall} - T_{ref})$$

Where $T_{ref}$ is the Wall Adjacent Temperature or the tbulk for htc temperature if specified
Post-Processing Heat Transfer

- **Heat Flux:**
  - Total convective and radiative heat flux into the domain
  - Available on all boundaries
  - On flow boundaries, it represents the energy carried with the fluid relative to its Reference Temperature

- **Wall Radiative Heat Flux:**
  - Net radiative energy leaving the boundary
  - Wall Convective Heat Flux + Wall Radiative Heat Flux = Wall Heat Flux
  - Only applicable when radiation is modeled

- **Wall Irradiation Flux:**
  - Incoming radiative flux
  - Only applicable when radiation is modeled
Solution Convergence

• Allow sufficient solution time
  – Heat imbalances in all domains have to approach zero

• Monitor
  – Create Solver Monitors showing IMBALANCE levels for fluid and solid domains
  – View the imbalance information printed at the end of the solver output file
  – Use a Conservation Target when defining Solver Control in CFX-Pre
Summary

• When modelling heat transfer, you must provide:
  – Thermal conditions at walls and flow boundaries
  – Thermal properties for materials

• Available heat transfer modeling options include:
  – Species diffusion heat source
  – Combustion heat source
  – Conjugate heat transfer
  – Natural convection
  – Radiation
  – Periodic heat transfer
Dissipation of heat from a hot electronics component

- Conjugate Heat Transfer (CHT)
- Two runs
  - First run includes convection and conduction
  - Second run adds thermal radiation

The entire calculation takes a long time to run. So results are provided for post-processing.
Natural Convection

• The significance of natural convection can be assessed through the Richardson number, $R_i$:

$$R_i = \frac{g\beta\Delta TL}{U_0^2} = \frac{\text{Natural}}{\text{Forced}}$$

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

• In buoyancy-driven flows, the Rayleigh number, $R_a$, indicates the relative importance of convection and conduction:

$$R_a = \frac{g\beta\Delta T x^3}{\nu \alpha} = \frac{\text{buoyancy force}}{\text{viscous losses & thermal diffusion}}$$

- The size of $R_a$ is a measure of whether a natural convection boundary layer is laminar or turbulent. For a vertical surface the critical value is around $10^9$ but the transition zone ranges from $10^6$ to $10^9$. 
Radiation: Mechanism

- Transfer of thermal energy by **electromagnetic waves** from 0.1 µm (ultraviolet) to 100 µm (mid-infrared).

- Radiation intensity is directionally and spatially dependent

- Transport mechanisms for radiation intensity along one given direction:

  - Scattering occurs when particles are present in the fluid - often neglected.
Radiation: Choice of Model

• The optical thickness should be estimated before choosing a radiation model
  – The optical thickness $\tau = aL$
    • $a$ is the absorption coefficient ($m^{-1}$) (Note: ≠ absorptivity of a surface)
    • $L$ is the mean beam path length (m)
    • $a = 0.25$ to $0.3$ $m^{-1}$ for combustion product gases, $= 0.01$ $m^{-1}$ for air and is proportional to absolute pressure
    • $L$ is a typical distance between opposing walls

• Optically thin ($\tau < 1$) means that the fluid is partially transparent to thermal radiation

• Optically thick ($\tau > 1$) means that the fluid absorbs or scatters the radiation many times before it can interact with the surfaces
Radiation: Choice of Model

- For optically thick media ($\tau > 1$) the P1 model is a good choice
  - Assumes radiative intensity is independent of direction
  - The P1 model gives reasonable accuracy without too much computational effort

- The Monte Carlo and Discrete Transfer models for any optical thickness
  - Both are ray tracing models
  - Discrete Transfer is much quicker as it pre-calculates rays in fixed directions but can be less accurate in models with long/thin geometries due to ray effects
  - Monte Carlo more expensive to run but recommended for complex geometries and multiband spectral modelling

- Surface to Surface Model
  - Available for Monte Carlo and Discrete Transfer models
  - Neglects the influence of the fluid on the radiation field
  - Can significantly reduce the solution time