# Chapter 5: Heat Transfer

This chapter describes the theory behind heat transfer in ANSYS Fluent. Information is provided in the following sections:



For more information about using heat transfer in ANSYS Fluent, see Modeling Heat Transfer in the User's Guide.

## 5.1. Introduction

The flow of thermal energy from matter occupying one region in space to matter occupying a different region in space is known as heat transfer. Heat transfer can occur by three main methods: conduction, convection, and radiation. Physical models involving conduction and/or convection only are the simplest (Modeling Conductive and Convective Heat Transfer (p. 133)), while buoyancy-driven flow or natural convection (Natural Convection and Buoyancy-Driven Flows Theory (p. 137)), and radiation models (Modeling Radiation (p. 138)) are more complex. Depending on your problem, ANSYS Fluent will solve a variation of the energy equation that takes into account the heat transfer methods you have specified. ANSYS Fluent is also able to predict heat transfer in periodically repeating geometries (Modeling Periodic Heat Transfer in the User's Guide), therefore greatly reducing the required computational effort in certain cases.

For more information about using heat transfer models in ANSYS Fluent, see Modeling Conductive and Convective Heat Transfer, Modeling Radiation, and Modeling Periodic Heat Transfer in the User's Guide.

## 5.2. Modeling Conductive and Convective Heat Transfer

ANSYS Fluent allows you to include heat transfer within the fluid and/or solid regions in your model. Problems ranging from thermal mixing within a fluid to conduction in composite solids can therefore be handled by ANSYS Fluent. When your ANSYS Fluent model includes heat transfer you will need to activate the relevant physical models, supply thermal boundary conditions, and input material properties that govern heat transfer and/or vary with temperature as part of the setup. For information about setting up and using heat transfer models in your ANSYS Fluent model, see Modeling Heat Transfer in the User's Guide.

Information about heat transfer theory is presented in the following subsections.



### 5.2.1. Heat Transfer Theory

#### *5.2.1.1. The Energy Equation*

ANSYS Fluent solves the energy equation in the following form:

$$\frac{\partial}{\partial t}(\rho E)+\nabla\cdot(\vec{v}\,(\rho E+p))=\nabla\cdot\left(k_{eff}\nabla T-\sum_j h_j\vec{J}_j+\left(\overline{\overline{\tau}}_{eff}\cdot\vec{v}\right)\right)+S_h \tag{5.1}$$

where $k_{eff}$ is the effective conductivity ($k+k_t$, where $k_t$ is the turbulent thermal conductivity, defined according to the turbulence model being used), and $\vec{J}_j$ is the diffusion flux of species $j$. The first three terms on the right-hand side of Equation 5.1 (p. 134) represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. $S_h$ includes the heat of chemical reaction, and any other volumetric heat sources you have defined.

In Equation 5.1 (p. 134),

$$E=h-\frac{p}{\rho}+\frac{v^2}{2} \tag{5.2}$$

where sensible enthalpy $h$ is defined for ideal gases as

$$h=\sum_j Y_j h_j \tag{5.3}$$

and for incompressible flows as

$$h=\sum_j Y_j h_j+\frac{p}{\rho} \tag{5.4}$$

In Equation 5.3 (p. 134) and Equation 5.4 (p. 134), $Y_j$ is the mass fraction of species $j$ and

$$h_j=\int_{T_{ref}}^{T} c_{p,j}\,dT \tag{5.5}$$

The value used for $T_{ref}$ in the sensible enthalpy calculation depends on the solver and models in use. For the pressure-based solver $T_{ref}$ is 298.15 K except for PDF models in which case $T_{ref}$ is a user input for the species. For the density-based solver $T_{ref}$ is 0 K except when modeling species transport with reactions in which case $T_{ref}$ is a user input for the species.

#### 5.2.1.2. The Energy Equation in Moving Reference Frames

The energy equation is solved in moving (relative) frames of reference. In moving frames of reference, the energy transport equation uses rothalpy as a conservative quantity. See Equation 2.6 (p. 20) for the energy equation in moving frames of reference.

#### 5.2.1.3. The Energy Equation for the Non-Premixed Combustion Model

When the non-adiabatic non-premixed combustion model is enabled, ANSYS Fluent solves the total enthalpy form of the energy equation:

$$\frac{\partial}{\partial t}(\rho H)+\nabla\cdot(\rho\vec{v}\,H)=\nabla\cdot\left(\frac{k_t}{c_p}\nabla H\right)+S_h \tag{5.6}$$

Under the assumption that the Lewis number (Le) = 1, the conduction and species diffusion terms combine to give the first term on the right-hand side of the above equation while the contribution from viscous dissipation appears in the non-conservative form as the second term. The total enthalpy $H$ is defined as

$$H = \sum_j Y_j H_j \tag{5.7}$$

where $Y_j$ is the mass fraction of species $j$ and

$$H_j = \int_{T_{ref,j}}^{T} c_{p,j} dT + h_j^0\left(T_{ref,j}\right) \tag{5.8}$$

$h_j^0\left(T_{ref,j}\right)$ is the formation enthalpy of species $j$ at the reference temperature $T_{ref,j}$.

### 5.2.1.4. Inclusion of Pressure Work and Kinetic Energy Terms

Equation 5.1 (p. 134) includes pressure work and kinetic energy terms, which are often negligible in incompressible flows. For this reason, the pressure-based solver by default does not include the pressure work or kinetic energy when you are solving incompressible flow. If you want to include these terms, use the `define/models/energy?` text command. When asked to `include pressure work in energy equation?` and `include kinetic energy in energy equation?`, respond by entering `yes` in the console window.

Pressure work and kinetic energy are *always* accounted for when you are modeling compressible flow or using the density-based solver.

### 5.2.1.5. Inclusion of the Viscous Dissipation Terms

Equation 5.1 (p. 134) and Equation 5.6 (p. 134) describe the thermal energy created by viscous shear in the flow.

When the pressure-based solver is used, ANSYS Fluent's default form of the energy equation does not include them (because viscous heating is often negligible). Viscous heating will be important when the Brinkman number, $Br$, approaches or exceeds unity, where

$$Br = \frac{\mu U_e^2}{k \Delta T} \tag{5.9}$$

and $\Delta T$ represents the temperature difference in the system.

When your problem requires inclusion of the viscous dissipation terms and you are using the pressure-based solver, you should activate the terms using the **Viscous Heating** option in the Viscous Model Dialog Box. Compressible flows typically have $Br \geq 1$. Note, however, that when the pressure-based solver is used, ANSYS Fluent does not automatically activate the viscous dissipation if you have defined a compressible flow model.

When the density-based solver is used, the viscous dissipation terms are *always* included when the energy equation is solved.

### 5.2.1.6. Inclusion of the Species Diffusion Term

Equation 5.1 (p. 134) and Equation 5.6 (p. 134) both include the effect of enthalpy transport due to species diffusion.

When the pressure-based solver is used, the term

$$\nabla \cdot \left( \sum_j h_j \vec{J}_j \right)$$

is included in Equation 5.1 (p. 134) by default. If you do not want to include it, you can disable the **Diffusion Energy Source** option in the Species Model Dialog Box.

When the non-adiabatic non-premixed combustion model is being used, this term does not explicitly appear in the energy equation, because it is included in the first term on the right-hand side of Equation 5.6 (p. 134).

When the density-based solver is used, this term is *always* included in the energy equation.

#### 5.2.1.7. Energy Sources Due to Reaction

Sources of energy, $S_h$, in Equation 5.1 (p. 134) include the source of energy due to chemical reaction:

$$S_{h,rxn} = -\sum_j \frac{h_j^0}{M_j} \mathcal{R}_j \tag{5.10}$$

where $h_j^0$ is the enthalpy of formation of species $j$ and $\mathcal{R}_j$ is the volumetric rate of creation of species $j$.

In the energy equation used for non-adiabatic non-premixed combustion (Equation 5.6 (p. 134)), the heat of formation is included in the definition of enthalpy (see Equation 5.7 (p. 135)), so reaction sources of energy are not included in $S_h$.

#### 5.2.1.8. Energy Sources Due To Radiation

When one of the radiation models is being used, $S_h$ in Equation 5.1 (p. 134) or Equation 5.6 (p. 134) also includes radiation source terms. For details, see Modeling Radiation (p. 138).

#### 5.2.1.9. Interphase Energy Sources

It should be noted that the energy sources, $S_h$, also include heat transfer between the continuous and the discrete phase. This is further discussed in Coupling Between the Discrete and Continuous Phases (p. 459).

#### 5.2.1.10. Energy Equation in Solid Regions

In solid regions, the energy transport equation used by ANSYS Fluent has the following form:

$$\frac{\partial}{\partial t}(\rho h) + \nabla \cdot (\vec{v} \rho h) = \nabla \cdot (k \nabla T) + S_h \tag{5.11}$$

where

$\rho$ = density

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

$k$ = conductivity

$T$ = temperature

$S_h$ = volumetric heat source

The second term on the left-hand side of Equation 5.11 (p. 136) represents convective energy transfer due to rotational or translational motion of the solids. The velocity field $\vec{v}$ is computed from the motion specified for the solid zone. (For details, see Solid Conditions in the User's Guide). The terms on the right-hand side of Equation 5.11 (p. 136) are the heat flux due to conduction and volumetric heat sources within the solid, respectively.

### 5.2.1.11. Anisotropic Conductivity in Solids

When you use the pressure-based solver, ANSYS Fluent allows you to specify anisotropic conductivity for solid materials. The conduction term for an anisotropic solid has the form

$$\nabla \cdot \left(k_{ij} \nabla T\right) \tag{5.12}$$

where $k_{ij}$ is the conductivity matrix. See Anisotropic Thermal Conductivity for Solids in the User's Guide for details on specifying anisotropic conductivity for solid materials.

### 5.2.1.12. Diffusion at Inlets

The net transport of energy at inlets consists of both the convection and diffusion components. The convection component is fixed by the inlet temperature specified by you. The diffusion component, however, depends on the gradient of the computed temperature field. Thus the diffusion component (and therefore the net inlet transport) is not specified a priori.

In some cases, you may want to specify the net inlet transport of energy rather than the inlet temperature. If you are using the pressure-based solver, you can do this by disabling inlet energy diffusion. By default, ANSYS Fluent includes the diffusion flux of energy at inlets. To turn off inlet diffusion, use the `define/models/energy?` text command and respond `no` when asked to `Include diffusion at inlets?`

Inlet diffusion cannot be turned off if you are using the density-based solver.

## 5.2.2. Natural Convection and Buoyancy-Driven Flows Theory

When heat is added to a fluid and the fluid density varies with temperature, a flow can be induced due to the force of gravity acting on the density variations. Such buoyancy-driven flows are termed natural-convection (or mixed-convection) flows and can be modeled by ANSYS Fluent.

The importance of buoyancy forces in a mixed convection flow can be measured by the ratio of the Grashof and Reynolds numbers:

$$\frac{Gr}{Re^2} = \frac{g\beta \Delta TL}{v^2} \tag{5.13}$$

When this number approaches or exceeds unity, you should expect strong buoyancy contributions to the flow. Conversely, if it is very small, buoyancy forces may be ignored in your simulation. In pure natural convection, the strength of the buoyancy-induced flow is measured by the Rayleigh number:

$$Ra = \frac{g\beta \Delta T L^3 \rho}{\mu\alpha} \tag{5.14}$$

where $\beta$ is the thermal expansion coefficient:

$$\beta = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_p \tag{5.15}$$

and $\alpha$ is the thermal diffusivity:

$$\alpha = \frac{k}{\rho c_p} \tag{5.16}$$

Rayleigh numbers less than $10^8$ indicate a buoyancy-induced laminar flow, with transition to turbulence occurring over the range of $10^8 < Ra < 10^{10}$.