

# Heat Transfer Concepts

Draft 2, 10/01/06

## Part 1

### *Introduction*

There are three different types of **heat transfer**: conduction, convection, and radiation. A temperature difference must exist for heat transfer to occur. Heat is always transferred in the *direction* of decreasing temperature. Temperature is a scalar, but heat flux is a vector quantity.

**Conduction** takes place within the boundaries of a body by the diffusion of its internal energy. The temperature within the body,  $T$ , is given in units of degrees Celsius [C], Fahrenheit [F], Kelvin [K], or Rankin [R]. Its variation in space defines the temperature gradient vector,  $\mathbf{grad} T$ , with units of [K/m] say. The heat flux vector,  $\mathbf{q}$ , is defined by Fourier's Conduction Law, as the thermal conductivity,  $k$ , times the negative of the temperature gradient,  $\mathbf{q} = -k \mathbf{grad} T$ . Thermal conductivity has the units of [W/m-K] while the heat flux has units of [W/m<sup>2</sup>]. The conductivity,  $k$ , is usually only known to two or three significant figures. For solids it ranges from about 417 [W/m-K] for silver down to 0.76 [W/m-K] for glass.

A perfect insulator material ( $k \equiv 0$ ) will not conduct heat; therefore the heat flux vector must be parallel to the insulator surface. A plane of symmetry (where the geometry,  $k$  values, and heat sources are mirror images) acts as a perfect insulator. In finite element analysis, all surfaces default to perfect insulators unless you give a specified temperature, a known heat influx, a convection condition, or a radiation condition.

**Convection** occurs in a fluid by mixing. Here we will consider only *free convection* from the surface of a body to the surrounding fluid. *Forced convection*, which requires a coupled mass transfer, will not be considered. The magnitude of the heat flux normal to a solid surface by free convection is  $q_n = h A_h (T_h - T_f)$  where  $h$  is the convection coefficient,  $A_h$  is the surface area contacting the fluid,  $T_h$  is the convecting surface temperature, and  $T_f$  is the surrounding fluid temperature, respectively. The units of  $h$  are [W/m<sup>2</sup>-K]. Its value varies widely and is usually known only from one to four significant figures. Typical values for convection to air and water are 5-25 and 500-1000 [W/m<sup>2</sup>-K], respectively.

**Radiation** heat transfer occurs by electromagnetic radiation between the surfaces of a body and the surrounding medium. It is a highly nonlinear function of the absolute temperatures of the body and medium. The magnitude of the heat flux normal to a solid surface by radiation is  $q_r = \epsilon \sigma A_r (T_r^4 - T_m^4)$ . Here  $T_r$  is the absolute temperature of the body surface,  $T_m$  is the absolute temperature of the surrounding medium,  $A_r$  is the body surface area subjected to radiation,  $\sigma = 5.67 \times 10^{-8}$  [W/m<sup>2</sup> - K<sup>4</sup>] is the Stefan-Boltzmann constant, and  $\epsilon$  is a surface factor ( $\epsilon = 1$  for a perfect black body).

**Transient**, or unsteady, heat transfer in time also requires the material properties of specific heat at constant pressure,  $c_p$  in [kJ/kg-K], and the mass density,  $\rho$  in [kg/m<sup>3</sup>]. The specific heat is typically known to 2 or 3 significant figures, while the mass density is probably the most accurately known material property with 4 to 5 significant figures.

**Conversions:** Energy, 1 J = 1 N-m, 1 BTU = 1.055 kJ. Power, 1 W = 1 J/s, 1 BTU/s = 1055 W. Temperature, K = 5/9 R, C = K – 273.15, F = 9/5 C + 32. Temperature difference,  $\Delta C = \Delta K$ ,  $\Delta R = 5/9 \Delta K$ ,  $\Delta F = 9/5 \Delta C$ .

**Thermal – Structural Analogy:** (They have the same steady state ODE in one-dimension.)

<b>Thermal Analysis Item, [units], symbol</b>	<b>Structural Analysis Item, [units], symbol</b>
Temperature [K], T	Displacements [m], u
Temperature Gradient [K/m], $\nabla T$	Strains [m/m], $\epsilon$
Heat flux [W/m <sup>2</sup> ], q	Stresses [N/m <sup>2</sup> ], $\sigma$
Heat Source for point, line, surface, volume [W],[W/m],[W/m <sup>2</sup> ], [W/m <sup>3</sup> ], Q	Load for point, line, surface, volume [N],[N/m],[N/m <sup>2</sup> ],[N/m <sup>3</sup> ], Q
<b>Restraint</b> Prescribed temperature value [K], T	<b>Restraint</b> Prescribed displacement component [m], u
<b>Reaction</b> Heat flow resultant [W], Q	<b>Reaction</b> Force component [N], Q
<b>Material Property</b> Thermal conductivity [W/m-K], k	<b>Material Property</b> Elastic modulus [N/m <sup>2</sup> ], E
<b>Material Law</b> Fourier's law	<b>Material Law</b> Hooke's law

### One-dimensional Governing Differential Equation

$$\partial(A(x) k(x) \partial T / \partial x) / \partial x + Q(x) A(x) = \rho c_p A(x) \partial T / \partial t,$$

or for the common case of a constant area, A, and conductivity, k:

$$\partial(k \partial T / \partial x) / \partial x + Q(x) = \rho c_p \partial T / \partial t,$$

for  $0 \leq x \leq L$  and time  $t \geq 0$ . With steady state ( $\partial T / \partial t = 0$ ) boundary conditions of:

1. T prescribed at 0 and L, or
2. T prescribed at one end and a heat source at the other, or
3. T prescribed at one end and a convection condition at the other, or
4. A convection condition at one end and a heat source at the other, or
5. A convection condition at both ends.

(The structural version is:  $\partial(A(x) E(x) \partial u / \partial x) / \partial x + Q(x) A(x) = \rho A(x) \partial^2 u / \partial t^2$ , E = elastic modulus, or  $\partial(E \partial u / \partial x) / \partial x + Q(x) = \rho \partial^2 u / \partial t^2$  for constant properties.)

## Part 2

### ***Finite Element Thermal Analysis***

#### **Algebraic equations**

The finite element method creates a set of algebraic equations by using an equivalent governing integral form that is integrated over a mesh that approximates the volume and surface of the body of interest. The mesh consists of elements connected to nodes. In a thermal analysis, there will be one simultaneous equation for each node. The unknown at each node is the temperature. Today, a typical thermal mesh involves 20,000 to 100,000 nodes and thus temperature equations. The restraints are specified temperatures (or a convection condition since it includes a specified fluid temperature). All other conditions add load or source terms. The default surface condition is an insulated boundary, which results in a zero source (load) term.

#### **Post-processing**

The temperature often depends only on geometry. The heat flux, and the thermal reaction, always depends on the material thermal conductivity. Therefore, it is always necessary to examine both the temperatures and heat flux to assure a correct solution.

The heat flux is determined by the gradient (derivative) of the approximated temperatures. Therefore, it is less accurate than the temperatures. The user must make the mesh finer (see Apply control) in regions where the heat flux vector is expected to rapidly change its value or direction.

The temperatures should be plotted as discrete color bands or as contour lines (see Settings). The temperature contours should be perpendicular to insulated boundaries. Near surfaces with specified temperatures, the contours should be nearly parallel to the surfaces. Pretty continuous color contours (frequently the default) tend to prevent these important engineering checks.

The heat flux should be plotted both as magnitude contours, and as vectors (see Arrows under Settings). The heat flux vectors should be parallel to insulated surfaces. They should be nearly perpendicular to surfaces with a specified constant temperature.

The thermal reaction at a node is the total heat flow [W] at the node necessary to maintain the given temperature. The reaction heat flow total is often an important physical consideration.

In SolidWorks (see List selected) it is possible to list, sum, average, and graph results along selected edges, lines, or curves (or surfaces, but that tends to be less helpful). Thus, you should plan ahead and add "split lines" to the mesh where you expect to find such graphs informative.

Written results should not be given with more significant figures than the material input data. For heat transfer problems that is typically two or three significant figures.

## Mesh, element type, accuracy

You should always examine the mesh before starting an analysis run. The size of each element indicates a region where the temperature is approximated (piecewise) by a polynomial. Most finite element systems, including CosmosWorks, use linear or quadratic complete polynomials in each element. You can tell by inspection which is being used by looking at an element edge. If that line has two nodes the polynomial is linear. If it has three nodes then the polynomial is quadratic.

That is, the total number of nodes on an element is:

1. Linear type: line – 2 nodes,  $T = a + b x$ ; straight triangle – 3 nodes,  $T = a + b x + c y$ ; straight edged, flat faced, tetrahedron – 4 nodes,  $T = a + b x + c y + d z$ . Therefore the heat flux in this type of element is constant (since their gradient is constant). Many elements are required to get good heat flux results. In CosmosWorks this is called a "Draft Mesh". These are also called *complete linear* elements.
2. Quadratic type: line – 3 nodes,  $T = a + b x + c x^2$ ; curved triangle – 6 nodes,  $T = a + b x + c y + d x^2 + e xy + f y^2$ ; general curved tetrahedron – 10 nodes,  $T = a + b x + c y + d z + e x^2 + f xy + g y^2 + h xz + i yz + j z^2$ . These are called *complete quadratic* elements and their gradients are *complete linear* polynomials like those above. Therefore, the heat flux in these elements varies linearly in space (e.g.,  $q = -k(b + 2c x)$ , etc.) and fewer elements are required for a good solution. In CosmosWorks this is called a "Quality Mesh".

These polynomial interpolations within an element mean that the temperature is continuous within the element ( $C^\infty$ ) and across element interfaces ( $C^0$ ). But, the heat flux is discontinuous across elements, whereas the exact value is continuous in a homogeneous material. The amount of discontinuity between elements is reduced as the element size is reduced. Of course, the exact heat flux is discontinuous at an interface between different materials because their thermal conductivities will be different.

CosmosWorks also offers p-adaptive elements (p is for polynomial). Keeping the mesh unchanged, it can automatically run a series of cases where it uses complete second, third, fourth, and finally fifth order polynomial interpolations. It allows the user to specify the allowable amount of error. That is, it can solve a given problem quite accurately. However, you still must define the geometry, materials, load and restraint locations, and load and restraint values as well as interpret the results properly. You still have the age old problem of garbage-in garbage-out, so avoid computer aided stupidity.