Modeling Skills-Thermal Analysis
J.E. Akin, Rice University

Introduction
Most finite element analysis tasks involve utilizing commercial software, for which you do not have the source code. Thus, you need to learn how various features are used by experienced engineers. Tutorials are useful insight into where to find various icons, but they often do not include good engineering judgment or the details for formulating the simulation of real world problems. Validation problems and benchmark problems have been adopted by the industry to prove that properly trained engineers can use each software product to properly solve sets of problems that have analytical solutions, and problem sets deemed difficult by various professional organizations.

Real world parts and assemblies are usually attached to other components which interact with the rest of the world. At some point one always terminates the consideration of the surroundings. Then, one introduces assumed essential boundary conditions to approximate the removed surroundings. Those boundary conditions must be sufficient to uniquely define the solution. When the heat flows are self-equilibrating the restraints may not be obvious or unique and the solution is only known to within an arbitrary constant. Then the temperature must be specified at one point and the computed temperatures will be relative to that point. This is like removing rigid body motion in structures. There are several analogies between one-dimensional structural analysis and general thermal analysis. Those are listed in Figure 1. While the temperature is a scalar, keep in mind that the heat flux is a vector quantity and it should be checked with vector plots as well as with its contour values.

<table>
<thead>
<tr>
<th>Thermal Analysis Item, [units], symbol</th>
<th>Structural Analysis Item, [units], symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Unknown: Temperature [K], T</td>
<td>Unknown: Displacements [m], u</td>
</tr>
<tr>
<td>Gradient: Temperature Gradient [K/m], VT</td>
<td>Gradient: Strains [m/m], ε</td>
</tr>
<tr>
<td>Flux: Heat flux [W/m²], q</td>
<td>Flux: Stresses [N/m²], σ</td>
</tr>
<tr>
<td>Source: Heat Source for point, line, surface, volume [W], [W/m], [W/m²], [W/m³], Q</td>
<td>Source: Force for point, line, surface, volume [N], [N/m], [N/m²], [N/m³], g</td>
</tr>
<tr>
<td>Indirect restraint: Convection</td>
<td>Indirect restraint: Elastic support</td>
</tr>
<tr>
<td>Restraint: Prescribed temperature [K], T</td>
<td>Restraint: Prescribed displacement [m], u</td>
</tr>
<tr>
<td>Reaction: Heat flow resultant [W], 11</td>
<td>Reaction: Force component [N], F</td>
</tr>
<tr>
<td>Material Property: Thermal conductivity [W/m-K], k</td>
<td>Material Property: Elastic modulus [N/m²], E</td>
</tr>
<tr>
<td>Material Law: Fourier’s law</td>
<td>Material Law: Hooke’s Law</td>
</tr>
</tbody>
</table>

Figure 1 Analogies between structural analysis and thermal analysis

Validation and benchmark problems omit the above phases of investigation and state the essential boundary conditions as if they are the law of the land. That said, one must be able to properly utilize the software to solve problems after the model has been established. There are many commercial finite element systems, so this discussion of modeling skills will utilize the SolidWorks Simulation (SWS) which has more than a million users worldwide. In addition to these notes the reader should review the very good general overview by V. Adams, Building Better Products with Finite Element Analysis, Onward Press, 1999 and the SWS specific J. Akin, Finite Element Concepts via SolidWorks, World Scientific, 2010.

Local singularities
Analysts need to be aware of regions where the simulated differential equation becomes invalid or non-physical. There are three common situations where the solution of an elliptical differential equation (like thermal analysis) develops singularities (infinite gradients) at points in the domain. A finite element analysis will not give true heat flux results at such points, but the mesh should have moderate refinements at such points. Singularities occur at points where there
are discontinuities in the essential boundary conditions (prescribed temperatures). If two lines have different assigned temperatures, but join at a common point, then a singularity occurs at the junction point.

A singularity also occurs at any sharp re-entrant corner. The singularity strength depends on the interior angle, in the material, from one side to the next. It is strongest for a crack or slit (angle of about 360 degrees). At sharp right angle re-entrant corners the strength of the singularity is less and a moderate mesh refinement can be used. In reality, such corners should have some fillet radius, or the material becomes non-linear, which changes the local differential equation. Figure 2 shows half of a rectangular region with a center slit held at one temperature and the right edge set at a higher temperature. All other boundaries default to being insulated in any finite element analysis. Expecting large gradients near such a point the mesh should have been made much finer near the singularity. Sometimes you miss a singularity until up contour the heat flux levels or plot the heat flux vectors. When that happens, refine the mesh and re-run the problem. Remember the infinite gradient is theoretical. You could refine the mesh there over and over and never get the theoretical results. Usually you would run out of memory first. Use “reasonable” refinements in such regions. The refinement is “reasonable” when the total heat flow from the thermal reactions do not change “much”, say < 2%.

![Figure 2 Near infinite gradients at the singular point of a slit](image)

**Cyclic Symmetry**

Cyclic symmetry requires parts where the same geometry and loads occurs at constant angle increments. Figure 3 shows a part with the geometry repeating every $360/5 = 72$ degrees. That means that for any given radius, the temperature distribution would repeat itself every 72 degrees. The automation of a cyclic symmetry analysis requires that the software can express the repeated temperatures at selected lines or surfaces. A thermal model with 90 degree cyclic symmetry is shown in Figure 4. Relative to the center point, one can model any 90 degree segment of the part with a cyclic simulation. If the software does not have that feature it may have a “Repeated Freedoms” option. Then you would identify each node pair: Aa, Bb, Cc etc as repeated unknown temperatures. Other programs offer “Rigid Links” for thermal studies. Then you would define the above node pairs as rigidly linked together.
Symmetries and anti-symmetry

You generally do not have a computer with enough memory and/or enough speed to solve all the problems you need to solve. Thus, you need to understand how to use symmetry, anti-symmetry (and how to combine the two) and cyclic symmetry. Generally, any time you can cut a model in half, then you cut memory requirement in half, and you cut the solution time by a factor of eight. Thus, a quarter symmetry (half of half symmetry) problem will run 64 times faster than the full model. Similarity, a one-eighth symmetry model could execute more than 500 times faster than a full model. Some of the validation problems and benchmark problems will be recast below with symmetry or anti-symmetry.

A plane of symmetry is flat and has mirror image geometry, material properties, surface fluxes, and boundary conditions. The heat flux perpendicular to the symmetry plane is zero. In other word, a symmetry plane is insulated. A boundary in a finite element study is insulated unless another condition is stated. Thus, a symmetry condition is a natural boundary condition in thermal studies. Figure 5 sketches the cases for thermal symmetry (left) and anti-symmetry. For symmetry, the temperature is unknown but the same on both sides of the plane. Clearly, that means that the temperature gradient (and the heat flux) is zero on that plane. That is the natural boundary condition in finite element simulation. That is, all surfaces in a finite element thermal model are defaulted to being insulated, with no heat flux, unless the analyst imposes a different boundary condition. For anti-symmetry consider the temperature difference between two sides of the plane. On one side the temperature may increase, while on the other side, at the same distance, it decreases by an equal amount. Therefore, there is an unknown heat flux across the anti-symmetry plane from the warm side to the cooler side. At the plane, the temperature is the average of the two temperatures imposed on the component, away from the symmetry plane. Usually, that average is zero.
Figure 5 Temperatures along a line normal to a plane of symmetry or anti-symmetry

Figure 6 illustrates that a component with quarter symmetry of materials essential boundary conditions and face convection. Employing symmetry can drastically reduce the memory and time required to obtain a solution.

Figure 6 A rectangular component with face convection and with quarter symmetry

In practice, it can be difficult to establish a convection coefficient, \( h \). Heat transfer courses teach ways to calculate it, based on reasonable assumptions. Figure 7 shows some commonly accepted values. Note the wide range of values. If the fluid is moving the higher values are more likely. Either way, you should consider running models with the upper and lower estimates unless you have better data on the convection coefficient.

<table>
<thead>
<tr>
<th>Fluid Medium</th>
<th>( h )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air (natural convection)</td>
<td>5-25</td>
</tr>
<tr>
<td>Air / superheated steam (forced convection)</td>
<td>10-500</td>
</tr>
<tr>
<td>Oil (forced convection)</td>
<td>60-1800</td>
</tr>
<tr>
<td>Steam (condensing)</td>
<td>5000-120,000</td>
</tr>
<tr>
<td>Water (boiling)</td>
<td>2500-60,000</td>
</tr>
<tr>
<td>Water (forced convection)</td>
<td>300-6000</td>
</tr>
</tbody>
</table>

Figure 7 Typical values of the convection coefficient

You should always examine the temperature contours for expected ranges and behavior. Wiggles in the contours show that the mesh is too crude there. If the region is one of high heat flux, then you should re-mesh and re-run the simulation. Figure 8 shows how the temperature contours should be distributed in a homogenous isotropic material. The corresponding heat flux vector behavior is given in Figure 9. The insulated boundary at the lower right implies a slit there (think about why). Thus, the heat flux vector at the point joining it to the constant temperature condition should be very large.
Multipoint Constraints

Common physical conditions impose constraints on a thermal solution. The most common, and difficult, is a non-ideal material interface between to conducting materials in contact. As shown in Figure 10, that causes a temperature difference on each side of the interface. For a perfect interface the mesh generated lets both materials share common nodes on the interface. For imperfect interfaces there need to be two parallel sets of nodes. The common practice (before the development of FEA), has been to have those node pairs are joined by a special “thermal resistance” material. It is difficult to assign a value to such resistance, since it depends on several things such as the surface roughness and the pressure holding the materials together. Some codes accept the resistance and others accept its inverse which is called the interface conductance. Typical values of both are listed in Figure 11.
Figure 10 Imperfect contact faces affect conduction simulations

Almost all mesh generators have a distance tolerance used to combine two near by nodes into a single one. Usually, that is good. But for interface resistance it is not, so you may have to build two different parts and assemble them, or separate the regions by a space larger that the tolerance but one that “looks like” they are in contact.

This condition could be input in modern FE codes an alternate way as a multipoint constraint:

\[ T_A - T_b = \Delta T = R_I q_n. \]

Note that the temperature discontinuity depends on the heat flux which is recovered in post-processing. Thus, imperfect interface effects should involve several simulations. Doing that would allow the analyst can get a good feel about how the interface condition could effect a range of solutions.

<table>
<thead>
<tr>
<th>Contact Pressure</th>
<th>Contacting Faces (pressure unknown)</th>
<th>Conductance</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum/aluminum/air</td>
<td>Aluminum / aluminum / air</td>
<td>2200 - 12000</td>
</tr>
<tr>
<td>Copper/copper/air</td>
<td>Ceramic / ceramic / air</td>
<td>500 - 3000</td>
</tr>
<tr>
<td>Magnesium/magnesium/air</td>
<td>Copper / copper / air</td>
<td>10,000 - 25,000</td>
</tr>
<tr>
<td>Stainless steel/stainless steel/air</td>
<td>Iron / aluminum / air</td>
<td>45,000</td>
</tr>
<tr>
<td>Stainless steel / stainless steel/air</td>
<td>Stainless steel / stainless steel / vacuum</td>
<td>200 - 1100</td>
</tr>
</tbody>
</table>

Figure 11 Typical interface resistances and conductances

Also note that the coupled node pairs in the cyclic symmetry Figure 4 could be treated via multipoint constraints.

NAFEMS Thermal example 9

This problem is re-run with half symmetry and mesh refinement. In addition to demonstrating the use of symmetry, it is shown that the non-physical corner EBC is affected by mesh generation and the order in which the boundary conditions are specified. The mathematically described boundary conditions can not be physically achieved. The original data file used the left line to set the lower corner point temperature to 1,000 C and then overroad it with the lower line at 0 C.
To reduce the sharp gradient there, I applied an average value of 500 C at the single vertex point (see Figure 1, right). The last of the three specifications is what the program applies, so consider the order of application when specifying the boundary conditions.

If you are using linear triangular elements be aware that how a corner is connected to elements in the mesh can change the shape of the boundary. That is illustrated in Figure 2. However, that effect is less important when a very fine mesh is employed. The problem does not apply to higher order interpolation elements. The original validation file used a very fine mesh in the whole region (and is shown in Figure 2). Clearly the elements on the right half were wasted refinements since most of the action is at the lower left corner. I looked at the fine mesh size, used a medium refinement in the full region, and specified the mesh at the corner point to be a tenth of the uniform fine mesh, and for its neighbors to grow at a ratio of 1.2 (rather than the default ratio of 1.5). Half the refined mesh and a zoom on the corner mesh is shown in Figure 3.
The temperature contours for the transitional mesh are given in Figure 4, for the left half of the domain. The gradients on the far right are so small the original uniform mesh wasted about three fourths of its uniform fine mesh. Here symmetry was employed and the mesh was concentrated in the corner. The temperature graph along the full top symmetry line is in Figure 5.
The resulting heat flux vector magnitude in the left portion of the domain is in Figure 6. Note that the color bar scale was manually set to a upper limit that was one percent of the false peak value.

Figure 5 Graph of temperature along the top symmetry line

Figure 6 Heat flux is theoretically infinite near corner is smoother here
The reaction to a specified temperature boundary condition is a heat flow across the boundary. It is the integral of the normal heat flux crossing that portion of the boundary. From Figure 1 left, the heat flow (or heat power) should all be inward on the left line with a total equal amount all going out along the bottom and right lines. Those reactions were displayed in the Results section with a Heat Power summary along those lines. The three edge summaries are given in Figures 7, 8 and 9. Note in Figure 7 there is some heat flow out, which should not happen. Some times minor non-physical answers like that (here 0.01% error) are from slightly less accuracy from the fast iterative solver and could vanish with a direct solver. Similar errors occur due to choice of the element polynomial and the order in which the EBCs are applied.

The default planar element in SolidWorks is a quadratic six node triangle. Thus, each of its edges is a three node parametric line. The temperatures enforced by the EBCs force non-physical negative temperatures to occur over half of the side of that element. For fine meshes this is not a major problem, but the user needs to be aware of what the program and finite element theory does with your specified inputs.

![Figure 7](image7.png)

**Figure 7** Heat flow in and out on the left edge (note out value)

![Figure 8](image8.png)

**Figure 8** Heat flow in and out on the lower edge (note in value)
Transient Thermal Analysis

The time history solution of the matrix ODE \[ [M] \left( \frac{dT}{dt} \right) + [K][T] = \{F(t)\} \] with essential boundary conditions and initial conditions can be carried out in many ways. The generalized trapezoidal time marching algorithm is one of the more popular ones. The time step selected for the method must not exceed the critical value for the solution to remain bounded and stable. The critical time step size is

\[ \Delta t_{\text{critical}} = \frac{2}{1 - 2\theta} \frac{1}{\lambda_{\text{max}}} \quad \text{for} \quad 0 < \theta \leq \frac{1}{2} \]

where \( \theta \) is the free parameter in the generalized trapezoidal time marching algorithm and \( \lambda_{\text{max}} \) is the maximum eigenvalue of \( |K - \lambda M| = 0 \). Since that eigenvalue is expensive to calculate, it is common to approximate its value. A common approximation is to use the average of the ratio of all of the diagonal terms in the square matrices:

\[ \lambda_{\text{max}} \approx \frac{1}{n} \sum_{j} K(j,j)/M(j,j) \]

If the essential boundary condition of a sub-set of nodes differs from the initial condition at those nodes then a better estimate is the maximum ratio of \( K(j,j)/M(j,j) \) for the nodes in that set (or their average value). Likewise, if a surface is
subjected to a thermal shock (rapid change in normal flux $q$, as from a laser pulse) that same sub-set estimate should be employed to estimate that eigenvalue.

Another way of saying the above is that for a transient analysis you want the non-dimension Fourier number, $Fo$, to be near unity:

$$Fo = \Delta t \frac{1}{n} \sum_{j} \frac{K_{j,j}}{M_{j,j}} \approx 1.$$ 

Transient solutions also depend on the non-dimension Biot number, $Bi$. Figure 12 sketches the temperature time changes in a slab where its surfaces are suddenly subjected to a convecting fluid at temperature $T_{\infty}$. Note that when the Biot number is high you need a finer mesh near the surface to catch the large gradients. Very high Biot numbers suggest a thermal shock condition.

![Figure 12 Transient conduction-convection for different Biot number ranges [Incropera]](image)

**Transient Thermal Shock**

When the initial conditions do not match an essential boundary condition, or if a region is hit with a high strength laser, a very large heat flux must develop in almost an instant. That is almost physically impossible to produce such a state. (If you can, say with a laser, then you probably need non-Fourier heat transfer which becomes a hyperbolic differential equation that is second order in time and the elliptical model used here is invalid.) Thermal shock used with a poor finite element mesh can appear to cause physically impossible answers. The reasons are illustrated in Figure 13.

This numerical behavior is caused by the time step is too small in relation to the physical properties ($K$ and $M$) and/or the element length normal to the thermal shock surface is too large. In Figure 13 the (linear element) mesh near the surface is on the top left and the exact solution along lines normal to the surface are at the top right. After a very small time step the heat flow has not had time to cross the element. Mathematical analysis shows that finite elements spatially approximate internal solutions in a least square way. So if you try to fit the exact first time step distribution inside the first element layer then the straight line goes negative, and the first interior node shows a physically impossible drop in temperature when a heat flow is input.
This can be avoided, in theory, by picking element sizes and time steps that make the local Fourier number at the shock face be near unity. Alternatively, you can ignore the early history, or you can use diagonal capacity matrices, $M$, and hide the fact that you have a poor simulation model (What you don’t know won’t hurt you, maybe.).

Figure 13 Large elements and FE least square matching of gradients cause negative temperatures during the initial time steps