

Getting started with heat transfer simulation

Jackson Hartman,
Application Engineer,
Noran Engineering, Inc.

Finite Element Analysis (FEA) software is most closely associated with structural analysis and linear statics. While this is a common use of FEA — and determining stresses in a part subjected to loads is a basic FEA skill — there are other extremely useful analyses options that should be part of the skilled engineer's portfolio. The level of complexity in each of these optional areas can also lead to further specialization in the software required by considering aspects such as whether the system is linear or nonlinear, the materials are homogenous or orthotropic, and whether transient conditions need analysis.

The most popular types of analyses performed with FEA software and their engineering parameters are:

Linear Statics — used to determine displacements, loads, stresses, strains, and mass properties in structures subjected to static loads.

Normal Modes — used to analyze the vibration characteristics of structures, natural frequency and mode shapes.

Buckling — used to assess stability of a structure under load.

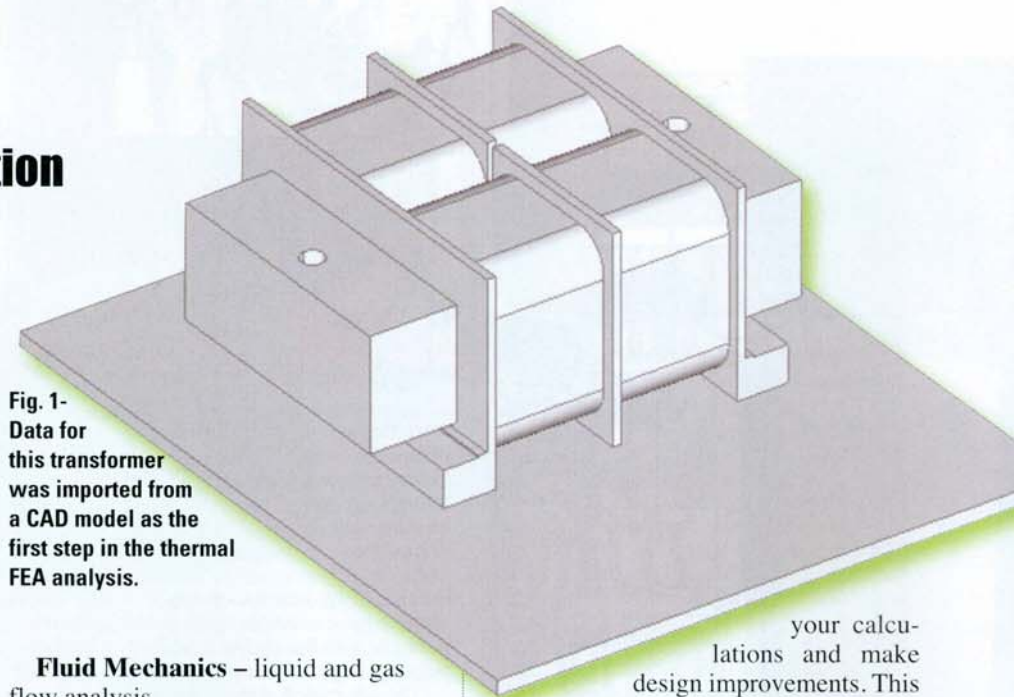
Dynamic Response — used to determine displacements, loads, stresses, and strains in structures subjected to transient or frequency dependent loads. Transient response, frequency response, random response, response shock spectrum, modal summation.

Motion — used to determine displacements, velocities, accelerations, loads, stresses, and strains in moving mechanisms and impacting objects.

Heat Transfer — used to determine temperatures, thermal gradients, heat fluxes, and heat flow due to conduction, convection, and radiation in structures and fluids.

Fatigue — used to determine the life of parts subjected to cyclic or dynamic loads.

Fig. 1-
Data for
this transformer
was imported from
a CAD model as the
first step in the thermal
FEA analysis.



Fluid Mechanics — liquid and gas flow analysis.

Heat transfer analysis is used in an incredibly wide array of products and applications. For example, in electronic products you may need to know the temperature range of certain components in your portable device; in manufacturing you may want to know what the cure cycle may look like in an industrial molding process; or in vehicle design you may want to study how heat dissipates in different brake assemblies.

The starting point is your concept idea. Then, in a traditional design cycle, you gather sufficient detail and information to make some basic heat transfer calculations. In this phase, consider the basic heat transfer modes of conduction, convection, and radiation, and a quantitative approximation of which mode will make a significant contribution. The calculations most likely would involve the classical equations for describing heat transfer and could be simple linear equations or more complex differential equations using a spreadsheet or a math program.

From this series of calculations, you would explore key aspects of the design and vary parameters like material selection or heat transfer coefficients. Then, you would proceed to a fabrication and test phase. Here you would build the prototype and develop a test regime that would validate the device's performance. From the test data, you gain the information needed to judge the correctness of

your calculations and make design improvements. This process is essentially iterative, repeated until time, project budget, and design objectives are reached.

For these traditional design cycles, FEA software can cut out many of the time consuming and expensive iterations requiring physical prototyping and testing. Virtual testing with FEA software can wring out many design issues before the first prototype is built. To illustrate the actual workings of the heat transfer capabilities of FEA software a sample analysis will be performed on a very specific part, in this case an electrical transformer common in most power supply units. For this problem, we want to know where the highest temperature occurs and what that temperature will be under the loading and specifications imposed. For our demonstration, we will be using Femap®, FEA modeling software to construct the model and to display the results. NEi Nastran® FEA software will generate the solution for Femap. Both are from Noran Engineering (www.NEiNastran.com).

We can construct the FEA model in Femap by either importing it from a 3D CAD system and modifying it as needed, or by creating it from scratch using Femap's 3D CAD tools. In this case, we have constructed the 3D model of the transformer in SolidWorks CAD software (figure 1).

Next, we imported the model into Femap and performed the steps of defining the materials, the thermal load, and the boundary conditions. For model

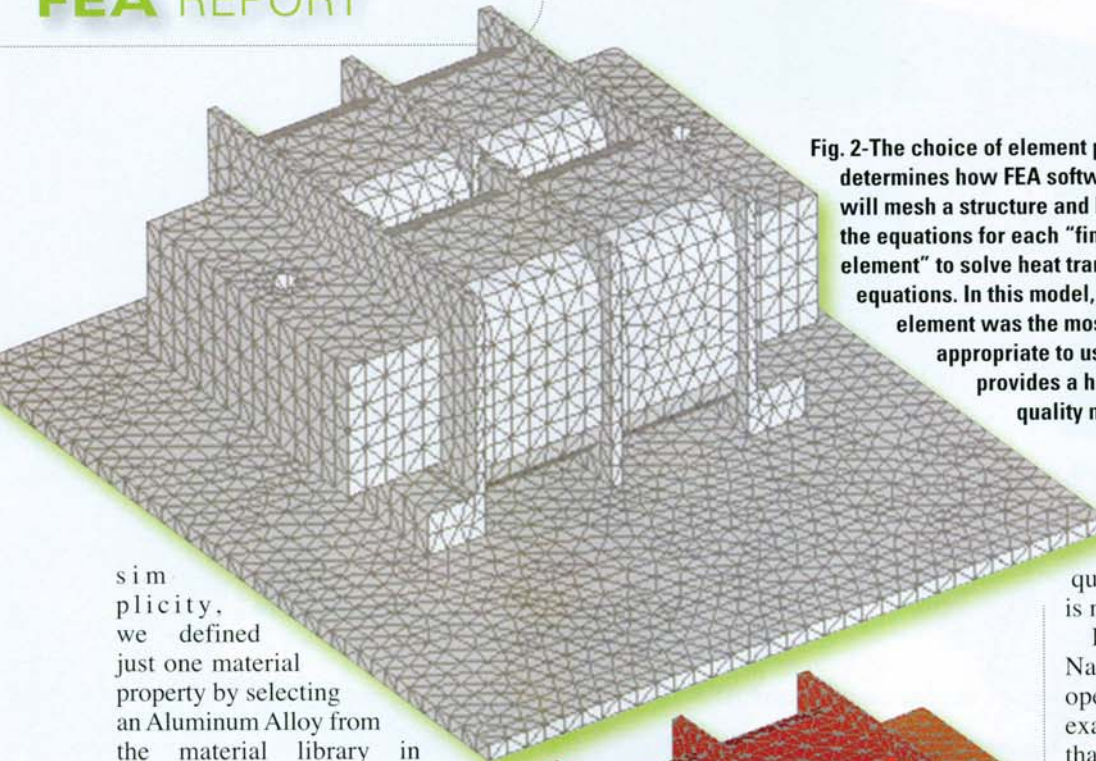


Fig. 2-The choice of element property determines how FEA software will mesh a structure and handle the equations for each "finite element" to solve heat transfer equations. In this model, a solid element was the most appropriate to use and provides a high quality mesh.

simplicity, we defined just one material property by selecting an Aluminum Alloy from the material library in Femap. Here all the material mechanical and thermal properties needed for analysis are already defined.

The thermal load for this model was determined by using a 70% efficiency rating for a 400 W transformer or 120 W (i.e., 30% x 400). The boundary condition would be the ambient air temperature or room temperature of 22°C.

We decided to omit the radiation component and consider only conduction and convection because radiation will be negligibly small. The appropriate components are factored in by specifying the coefficients. In this case, the convection coefficients were chosen as 125 W/m² K and 20 W/m² K and the conduction coefficient was loaded with the material properties. NeNastran does not support forced convection, thus, two convection properties were used to roughly model forced convection over the model. The forced convection is needed in this case to simulate a fan that is used to cool this component in a power supply. In the model, surfaces that would be most effected by the fan were selected to have the higher convection coefficient. The other faces were assigned the lower coefficient to model natural convection.

The next step was to assign FEA

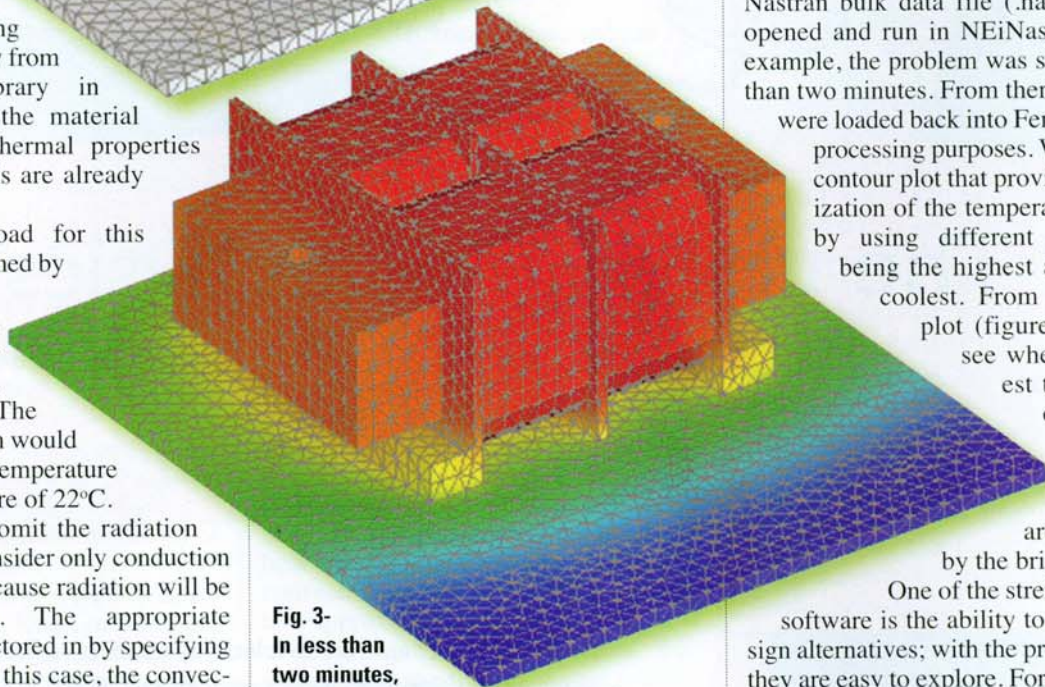


Fig. 3- In less than two minutes, the FEA program solves the thermal equations. The results can be plotted, in a contour plot for example, to show the temperatures ranges. In this plot, red indicates the highest temperatures and blue the coolest.

element properties to this transformer. The choice of element property determines how the software will mesh the structure and hence handle the equations for each of these "finite elements" and solve the heat transfer equations. This is the fundamental nature of the finite element

method. In essence, it is necessary to get uniform, well-shaped and behaved elements in the meshing process to get accurate results. Transitions should be smooth and gradual without sliver or distorted elements. For our model, a

solid element was the most appropriate to use and provides a high quality mesh (figure 2). After the model is meshed, an analysis can be set up.

For the actual analysis, Femap writes a Nastran bulk data file (.nas), which is opened and run in NEiNastran. In this example, the problem was solved in less than two minutes. From there, the results were loaded back into Femap for post-processing purposes. We selected a contour plot that provides a visualization of the temperatures ranges by using different colors—red being the highest and blue the coolest. From the contour plot (figure 3), we can see where the highest temperatures occur (in this plot, the hottest portions are designated by the bright red).

One of the strengths of FEA software is the ability to examine design alternatives; with the problem set up, they are easy to explore. For example, we might want to try providing openings in the mounting board of the transformer to improve convection cooling, or consider different materials. You can view further steps in this example analysis and learn more about the products mentioned in this article at www.nenastran.com/heat-transfer.

NEiNastran
www.NEiNastran.com

Digg This Story! at www.designworldonline.com

del.icio.us store your web bookmarks online