Introduction to FEMAP for Thermal Model Generation

Thermal and Fluids Analysis Workshop 2006
Hume Peabody
Swales Aerospace

ABSTRACT

FEMAP is a robust Finite Element Model (FEM) pre- and post-processor whose primary strengths are the ability to import and export many geometry and FEM formats and its intuitive and easy to use interface for building models. A primary advantage to Finite Elements for thermal analysis is the automatic generation of the conduction network. Until recently, however, the radiation network generation, was not well supported. With the development of tools like Thermal Desktop (and TMG), the radiation network can also be computed. Thermal Desktop supports the import of a FEMAP neutral file, including nodes, elements, properties, and materials (and most importantly optical properties). Therefore the combination of these two tools allows for the rapid construction of analysis models based on solid models from a mechanical designer and a better interface to the design evolution by the thermal subsystem. This paper serves as an introduction to the thermal engineer as to the basic construction techniques used to build a finite element model from a solid model and provides some suggestions to consider when using the FEM as the basis for a Thermal Desktop model.

1. INTRODUCTION

Traditionally, two methods of thermal modeling have been used: finite difference and finite element. In both cases, a matrix of terms is generated and solved to determine temperatures. While many commercially available thermal codes have supported finite element capabilities, the generation of a mesh has not been particularly easy. Some commercial thermal codes have begun to include automatic element generation capabilities for quick generation of a mesh suitable for thermal analysis; others have provided interfaces to existing meshing/FE programs. Conversely, structural codes rely almost exclusively on the finite element approach, but lack the radiation and orbital capabilities of thermal codes. The FEMAP® program (Finite Element Modeling And Post-processing) includes an intuitive and easy to use interface, the ability to import and export many formats (both geometry and FE), and robust automatic mesh generation capabilities. Using various export capabilities from FEMAP and import capabilities of the destination thermal code, the majority of the effort to build a detailed finite element thermal model can be performed in FEMAP. The FEM can then be made into a thermal model, complete with orbital and electrical heating, radiation exchange, and other features of thermal models using Thermal Desktop®/SindaFluint, IDEAS/TMG, TSS/SINDA, or SINDA/ATM. This paper provides an introduction to basic Finite Element modeling techniques and provides recommendations based on the efforts and lessons learned from generating the Laser Interferometer Space Antenna (LISA) models using FEMAP and Thermal Desktop.

2. BASIC FEM TECHNIQUES

Most finite element codes support a common set of analysis objects (*Material*, *Property*, *Node*, *Element*, *Load*) and geometry objects (*Point*, *Curve*, *Surface*, *Solid* / *Volume*). A *Material* defines the material properties to be used for computing the interaction between nodes. For a thermal model, the relevant material properties are density, specific heat, and thermal conductivity. If radiation is to be included, then front side and reverse side thermo-optical properties should also be included. A *Property* references a particular *Material* but defines additional parameters specific to a set of *Element*s, such as thickness (for 2D elements) or

cross sectional area (for 1D elements). This allows for the same material to be used numerous places where the physical geometry may be different. *Node*s define specific locations in XYZ space where a value (e.g. temperature) should be calculated. The interaction between *Node*s is defined by *Element*s. An *Element* is defined by 2 or more nodes and a *Property* (which in turn defines the *Material* to be used for the *Element*).

A model could be built by specifying the location for each individual node and building each element manually, but this would be extremely tedious. Fortunately, a user can define the number of nodes along a curve or surface or the desired element size and allow FEMAP to automatically generate the corresponding mesh. To do this, it is helpful to understand the geometry hierarchy. The most basic element for geometry is a *Point*, which, like a node, is simply a point in XYZ space. *Point*s are used to define *Curve*s, which may be arcs or straight line segments. Three or four *Curve*s are used to define *Surface*s, which need not necessarily be planar. More complicated surfaces may be defined using boundary surfaces, which allow for cutout sections such as holes. Lastly, *Surface*s are combined to form *Solid*s. A variety of operations may be performed on solids, such as slicing, exploding, and add/subtract Boolean operations.
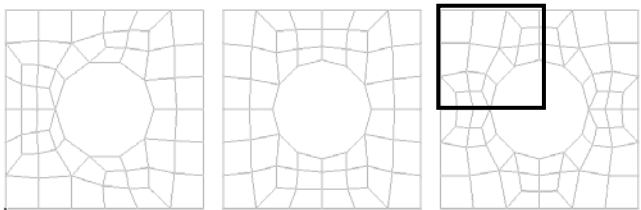

Figure 1 – Different Mesh Discretizations

Once the geometry is defined, it can be meshed using automatic or mapped meshing algorithms. Automatic meshes tend to be less ordered than mapped meshes. The mesh on the left of Figure 1 was automatically generated based on an average element size. The mesh in the middle was based on 6 elements along the outer sides and 12 around the cutout. Lastly, the mesh on the right was based on 3 elements along each side of the top left section and reflecting the elements about lines of symmetry. Mapped meshes are best suited for simpler geometries; while automatic mesh generation is better for complicated surfaces/solids.

3. FEMAP INTERFACE

Figure 2 shows the FEMAP v9.0 interface which features components found in many Windows based programs, including a menu and toolbars at the top. The left side includes a Model Info tree, where entities in the model can be selected. The corresponding properties of the selected entity are shown in the Entity Editor window below the object tree. The right side features the main visualization window and command log/response messages. The menus follow a logical progression for model generation: File, Tools, Geometry, Model, Mesh, Modify, List, Delete, Group, View, Window, and Help as shown in Figure 3. Frequently used functions for each menu are described in Table 1.
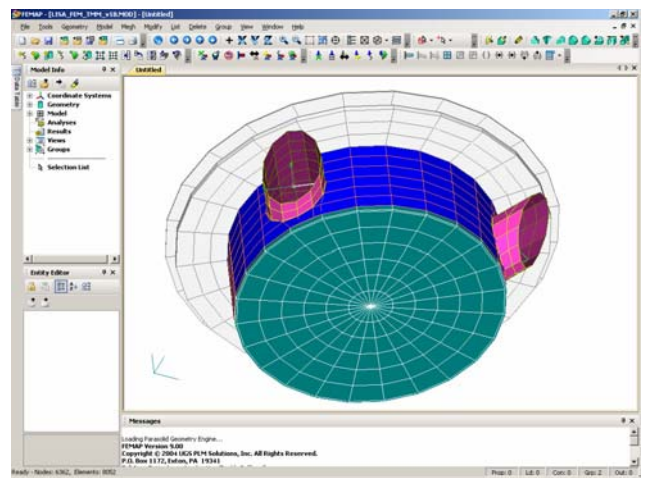

Figure 2– FEMAP Interface

Figure 3– FEMAP Menu

| Menu	Functions
File	load existing models, save model, importing and exporting, save image
Tools	Measure distances, check for coincident entities
Geometry	Generate geometry entities (Points, Curves, Surfaces, etc) anew or from existing versions (rotate, copy, reflect, etc.)
Model	Generate Materials, Properties, Nodes or Elements
Mesh	Element/curve sizing, automatic or mapped mesh generation, generation of new elements from existing ones
Modify	Change features of all existing entities (Nodes, Materials, Elements, Properties, etc)
List	Display information for selected entities
Delete	Remove selected entities from model
Groups	Create Groups for display/selection purposes
View	Set viewpoint, control visibility of entities

Table 1 – Major Menu Functionality

4. MAKING A MESH FROM SOLID MODELS

Unfortunately, a typical solid model from a mechanical designer cannot be used "as is" to generate a mesh suitable for thermal analysis. Often these models include features not needed for thermal analysis, including: bolts, holes, chamfers, fillets, slots, etc. The solid model for the current LISA design is shown in Figure 4.
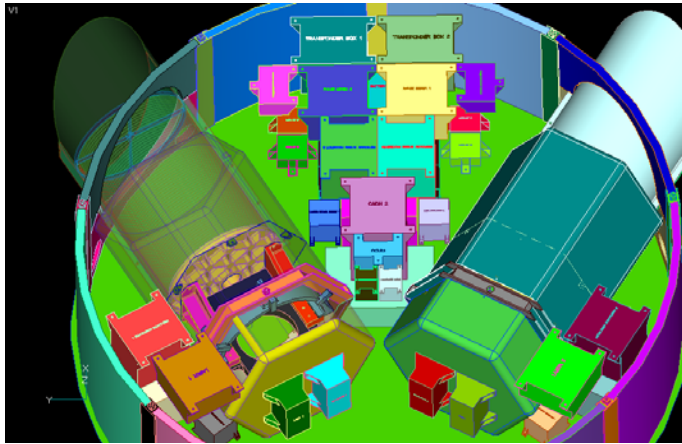

Figure 4 – Sample Solid Model for LISA

This model includes many features that will not be preserved when a mesh is built, such as: electronics box mounting feet, baffles in the telescope shield, and fillets on the payload shield. However, the points used to define the solids model's curves and surfaces can be used to create intermediate points, curves and surfaces useful for making simplistic representations of solid model components. Figure 5 shows a section of the solid model with the payload shield "ghosted" to show some of the internal components. Figure 6 shows the simplistic geometry that was later used to define the mesh on the surfaces.
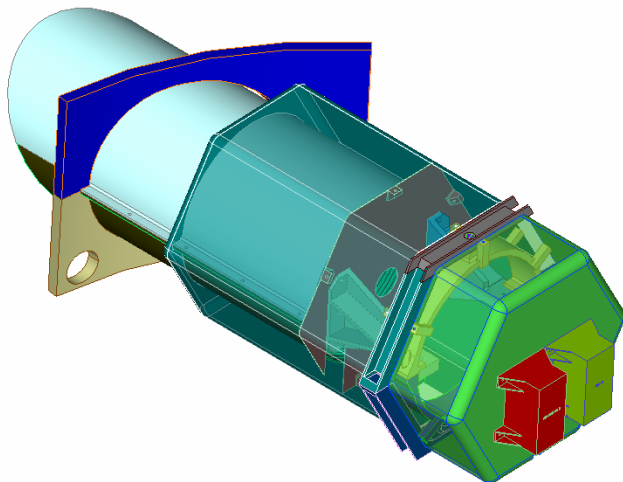

Figure 5 – Solid Model Representation
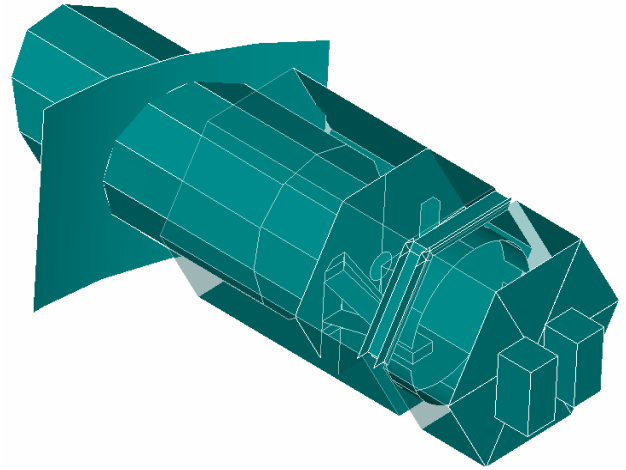
Figure 6 – Simplified Surface Representation

It is up to the analyst to determine the appropriate element type suitable for analysis. While a wide variety of element types are supported by FEMAP, not all of them translate properly into Thermal Desktop. Generally, thermal analysts should use *bar*, *plate*, and *solid* element types when exporting to Thermal Desktop. Typically, thinner sections (e.g. thin walls, panels) are represented with *plate* elements, bolts are represented with *bar* elements, and thicker sections are represented with *solid* elements. It should be noted that typically only *plate* elements are included in radiation. Therefore, to include radiation from solid elements, a shell coat of zero thickness plate elements can be applied to the free faces. One benefit to using solid elements is that they may reference anisotropic materials ($k_x \neq k_y \neq k_z$), such as a honeycomb core. To correctly model this, the material orientation must also be defined to specify the x, y, and z directions. For example, the honeycomb panels in the LISA model included *plate* elements for the outer facesheet, anisotropic *solid* elements for the honeycomb core, and *plate* elements for the inner facesheet. This allowed for the proper simulation of radiation from both sides of the honeycomb panel as well as including the anisotropic conductivity of the honeycomb core. Figures 7-11 show the solid model section and the resulting finite element mesh.
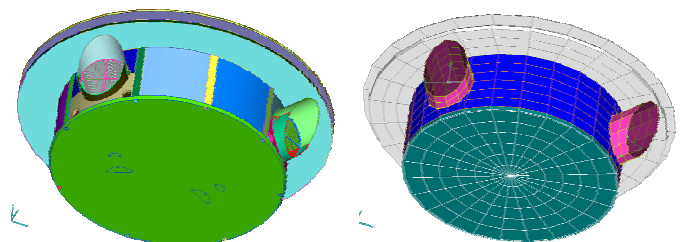
Figure 7 – Solid Model (Left) vs. FEM (Right)

Figure 8 – Solid Model (Left) vs. FEM (Right)

Figure 9 – Solid Model (Left) vs. FEM (Right)

Figure 10 – Solid Model (Left) vs. FEM (Right)

Figure 11 – Solid Model (Top) vs. FEM (Bottom)

5. MESH VALIDATION AND MODEL ORGANIZATION

Once a mesh is generated, model checks should be performed to ensure that the resulting thermal model will be correct. First, element normal directions should be verified. Elements generally follow the right hand rule to determine the +Z (or front) side of the element. If front side thermo-optical properties are defined for the material used by the element, then these properties will be used for the +Z side of the element. Conversely, if properties are defined for reverse side, then these properties will be assigned to the –Z side of the element. Unfortunately, reverse side thermo-optical properties cannot be assigned without first assigning front side properties. If only one side of an element is radiating, then the element normal must point in the direction of the element's field of view for proper radiation view computation. FEMAP has the capability to visually display the element normal direction via arrows or backface shading.

Nodes must also be merged to ensure that connections exist between all sections of a model. It is important to realize that heat flows through nodes; therefore, to correctly simulate interfaces, components must share nodes or be explicitly connected by elements. To merge nodes, the user specifies an allowable tolerance for two nodes to be separated; any nodes within this tolerance will be merged together. Elements referencing either of the two nodes are updated to reflect the remaining node number. Coincident nodes connected by a bar element will be listed, but will not be merged. Before merging nodes, the user is advised to check which nodes will be merged to ensure that unintended mergings are not performed. FEMAP provides the option to create two groups of coincident nodes (remaining and replaced nodes) if further investigation is needed. Similar capabilities exist for detecting coincident elements, to ensure that a surface was not meshed twice.

Once all nodes have been merged, it is advised to logically renumber nodes and elements so that associations with particular components or regions of the model can be identified. For example, the Solar Array nodes and elements may be numbered starting with 10000, the Top Panel starting with 20000, the Structural Walls with 30000, etc. This organization will make it easier to interpret and evaluate results later.
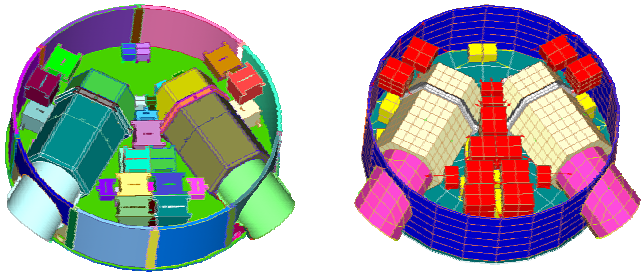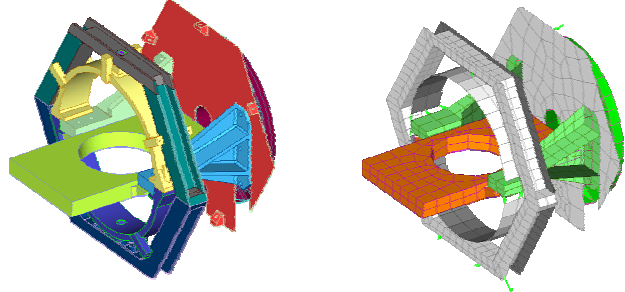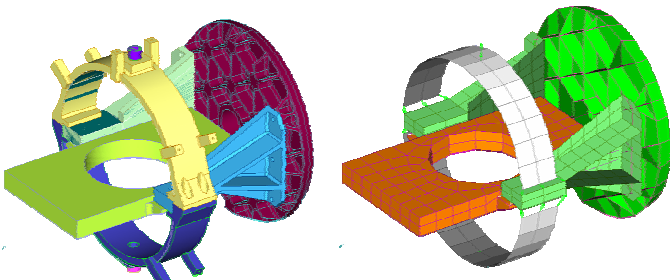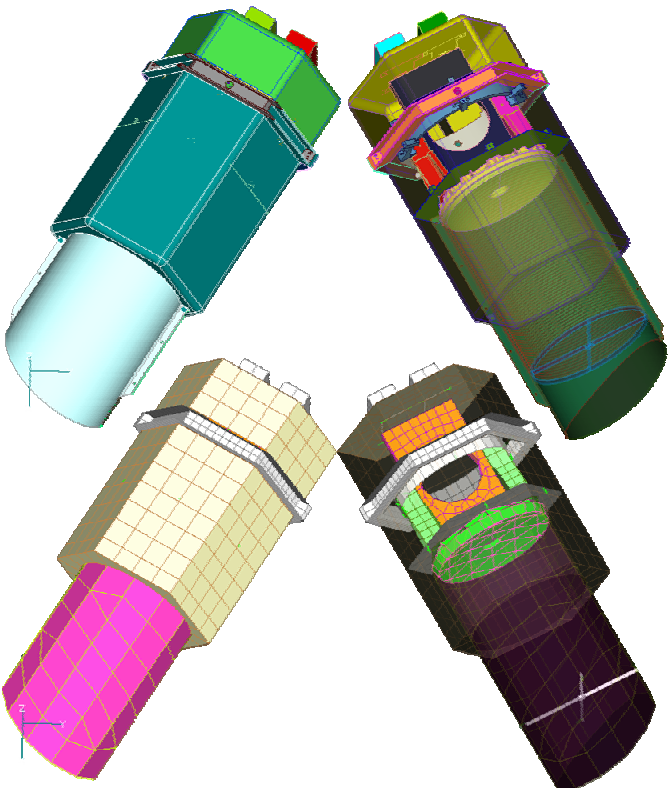
Lastly, for easier view manipulation, the model should be organized to quickly allow for portions to be displayed or hidden. Two methods exist in FEMAP to help a user to control what portions of a model are visible at a time: groups and layers. Nodes and elements may exist in multiple groups, but only on one layer. Multiple layers may be displayed simultaneously, but only one group may be displayed at a time. While layers may seem advantageous, in that many views can be created by displaying only the layers needed, groups do offer an advantage that they may be used to select entities on which to operate (renumber, change color, etc). However, upon import into Thermal Desktop, only layers are preserved; therefore, a combination of groups and layers is recommended. Groups should be used to define collections of elements on which to operate (or perhaps be viewed) and layers should be used for controlling visibility.

6. MAKING A THERMAL MODEL FROM A MESH

Once the mesh has been defined, it is exported as a FEMAP Neutral File v8.0 for further use in Thermal Desktop. It could also be exported as an IDEAS Universal file for use in TMG or used with SINDA/ATM, but this paper focuses on the use of Thermal Desktop. The neutral file is then imported into Thermal Desktop, where it is converted from a simple FE mesh into a thermal model. This includes the need to define thermal boundary conditions and constraints (applied heat loads, environmental heating, radiation exchange, etc.), radiation enclosures, and solution parameters.

The imported mesh includes all node locations, element definitions, material properties, and optical properties. For element heat loads and radiation exchange factors, Thermal Desktop uses shape functions to apply energy to the corner nodes of an element. During a ray trace, when a ray intersects an element the intersection point is first determined. This intersection point is then used along with shape functions for the element to apportion more of the ray's energy to nodes closer to the intersection point and less energy to nodes further away. This approach prevents all the energy from being applied to a single location (sub-element) as done in other codes. A similar process is used to apportion element heat loads to the corner nodes.

Radiation analysis groups are a means for defining enclosures, where elements in the enclosure can only "see" other elements in the enclosure. Careful preparation and verification of the element normal direction and optical properties during the meshing process makes this much easier. However, a user should still view the active sides and assigned optical properties within Thermal Desktop for each radiation group to ensure that the desired elements are participating in radiation exchange as expected.

7. BASIC STEPS FOR MODEL GENERATION

1. Import Solid Model Geometry
   a. *File → Import → Geometry*
2. Generate Materials
   a. *Model → Material*
3. Generate Properties
   a. *Model → Property*
4. Determine Section of Model to Be Meshed
   a. Create Simplistic Surface representations where needed
      i. *Geometry → Point*
      ii. *Geometry → Curve → Multiple Commands*
      iii. *Geometry → Surface → Multiple Commands*
   b. Define Nodalization along Curves/Surfaces
      i. *Mesh → Mesh Control → Size Along Curve*
      ii. *Mesh → Mesh Control → Default Size*
   c. Generate Mesh and Assign Property
      i. *Mesh → Geometry → Surface*
      ii. *Mesh → Geometry → HexMesh Solids*
      iii. *Mesh → Extrude*
      iv. *Mesh → Revolve*
   d. Repeat for other sections of model
5. Connect isolated mesh sections
6. Check mesh
   a. Verify element normal directions
      i. *View → Options → Element - Directions → Show Directions*
   b. Merge nodes
      i. *Tools → Check → Coincident Nodes*
   c. Renumber to logical node/element groupings
      i. *Modify → Renumber → Node*
      ii. *Modify → Renumber → Element*
   d. Logically group section of the model together using Layers and Groups
      i. *Modify → Layer → Node*
      ii. *Modify → Layer → Element*
      iii. *Group → Set*
      iv. *Group → Node → ID → Select Nodes*
      v. *Group → Element → ID → Select Elements*
7. Export FEMAP Neutral File v8.0 from FEMAP
   a. *File → Export → FEMAP Neutral*
8. Import FEMAP Neutral File v8.0 into Thermal Desktop
   a. *Thermal → Import → FEMAP ascii neutral (v8.0)*
9. Re-define orienters/anisotropic properties
10. Define radiation analysis groups

11. Define orbital parameters
12. Define applied heat loads
13. Define analysis cases in Case Set Manager
14. Run Cases
15. Post Process

In summary, the materials, properties, nodes, elements (including orientation), and layers are generated in FEMAP. The material orienters and anisotropic properties are re-defined in Thermal Desktop, since they are not preserved in the export/ import process. Furthermore, heat loads, radiation analysis groups, and orbital parameters are also generated in Thermal Desktop

8. CONCLUSIONS

The use of FEMAP to build a finite element model allows the thermal analyst to have better access to the design evolution as the solid model matures. FEMAP allows the user to make updates quickly without the need to regenerate a conduction network. Recent advances in tools like Thermal Desktop and TMG allow for the quick generation of the radiation network. Given the relatively simple remaining effort to add boundary conditions (applied power, heaters, etc.) and define the radiation environment (radiation groups, orbit parameters), the overall task of generating a thermal model can be mostly automated once a representative mesh is constructed.

9. ACRONYMS

FEM   Finite Element Model
FE   Finite Element
FEMAP   Finite Element Modeling and PostProcessing. FEMAP is a product of UGS PLM Solutions, Inc.
LISA   Laser Interferometer Space Antenna
TSS   Thermal Synthesizer System. TSS is a product of SpaceDesign
SINDA   Systems Improved Numerical Difference Analyzer. SINDA/FLUINT is a product of Cullimore and Ring Technologies. SINDA/G is a product of Network Analysis
TMG   Thermal Model Generator. TMG is a product of MAYA Heat Transfer Technologies
ATM   Advanced Thermal Modeler. SINDA/ATM is a product of Network Analysis.

10. ACKNOWLEDGEMENTS

11. REFERENCES

"Low-Frequency Thermal Performance of the LISA Sciencecraft", H. Peabody, 6th LISA Symposium, June 2006