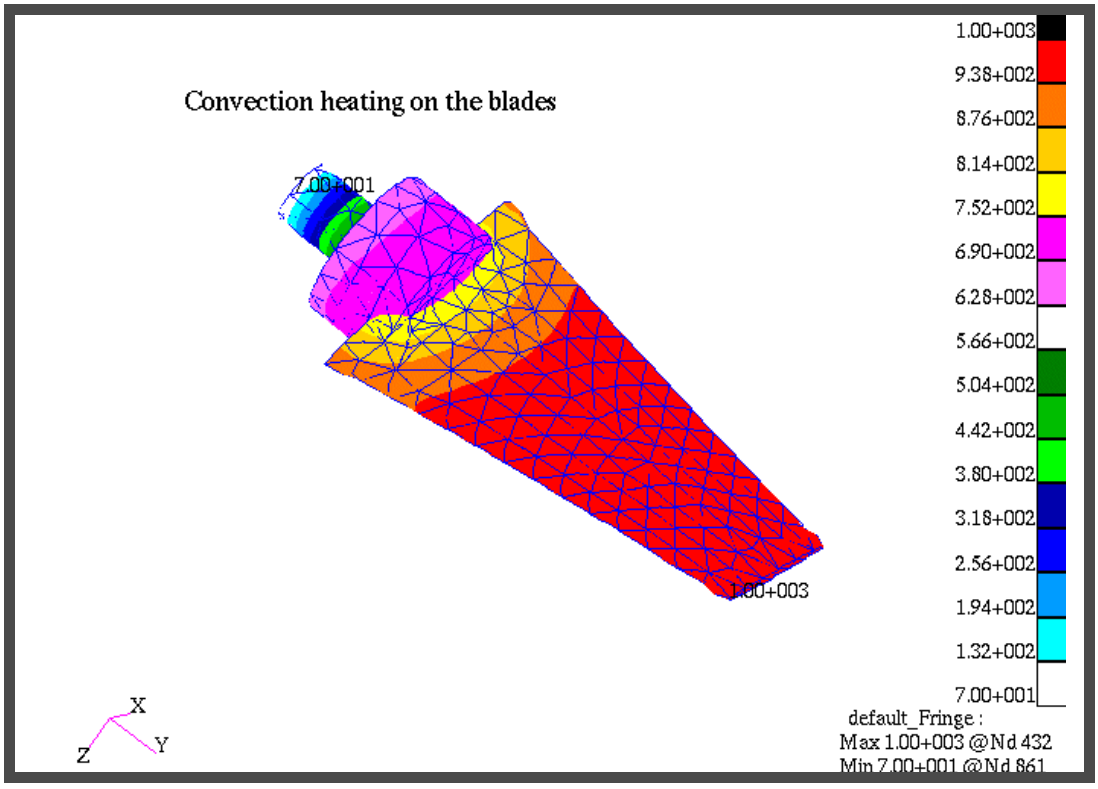
APPENDIX E

Importing IGES file and auto-tet mesh the model



Objectives:

- Importing IGES geometry, and create a B-Rep solid from all the surfaces
- Auto-Tet mesh the model with TETRA10
- Apply convection heating on the blade surfaces with h=10, and hot ambient temperature at 1000 F
- Apply a constant base temperature of 70 degree F
- Obtain a temperature contour

Suggested Exercise Steps:

- Create a new database and name it **fin.db**.
- Create a surface model of the cooling fin
- Generate the finite elements using mesh seeds
- Define material and element properties.
- Apply the convection conditions to the model.
- Submit the model to MSC.Nastran for analysis.
- Review results.

Exercise Procedure:

1. Open a new database called **fin.db**.

   File/New...

   New Database Name **prob16**

   OK

   In the New Model Preferences form set the following:

   New Model Preference

   Tolerance ◆ **Default**

   Analysis Code: **MSC/NASTRAN**

   Analysis Type: **Thermal**

   OK

   Whenever possible click ❑ **Auto Execute** (turn off).

2. Import the IGES geometry

   File/Import...

   Source: **IGES**

   Import File **mblade.igs**

   Apply

   Click **OK** when the IGES Import Summary appears

3. Create a B-Rep solid from all the surfaces

   ◆ **Geometry**

   Action: **Create**

   Object: **Solid**

   Method: **B-rep**

   Surface List **Surface 1:34**
   (Select all the surfaces)

Apply

Switch the viewport to ISO2 view by click on the ISO 2 View icon on the Main Menu.

Iso 2 View

Also view the model in Smooth Shaded view.

Smooth Shaded

4. Mesh the solid using TetMesh

Finite Element

Action:	**Create**
Object:	**Mesh**
Type:	**Solid**
Mesher	**TetMesh**
Element Topology	**Tet10**
Input List	**Solid 1**

Apply

5. Next, define a material using the specified thermal conductivity.

Materials

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**Titan**
Input Properties	
Thermal Conductivity	**10.8859**

OK

Apply

6. Next, reference the material that was created in the previous step. Define the properties of the blade.

Properties

Action:	**Create**
Object:	**3D**
Type:	**Solid**
Property Set Name	**solid**
Input Properties	
Material Name	**m:Titan**

OK

Select Members	**Solid 1**

Add

Apply

7. Apply the constant temperature boundary conditions at the base.

Loads/BCs

Action:	**Create**
Object:	**Temp (Thermal)**

Type: **Nodal**

New Set Name **base_temp**

Input Data

Boundary Temperature **70**

OK

Select Application Region

Geometry Filter **Geometry**

Select Geometry Entities **Surface 3 5**

(Select the bottom 2 arcs)

Add

OK

Apply

Now assign the Convection boundary conditions on the surfaces of the blade.

Loads/BCs

Action: **Create**

Object: **Convection**

Type: **Element Uniform**

New Set Name **convection**

Target Element Type: **3D**

Input Data

Convection Coefficient **10**

Ambient Temperature **1000**

OK

Select Application Region

Geometry Filter **Geometry**

Select Solid Faces **Solid 1.22 1.21**

(Click on the blade to select both faces)

Add

OK

Apply

8. Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here:

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Type: **Analysis Deck**

Translation Parameters

Data Output: **XDB and Print**

OK

Solution Type

Solution Type ◆ **STEADY STATE ANALYSIS**

OK

Apply

An MSC/NASTRAN input file called **prob16.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

Submitting the Input File for Analysis:

9. Submit the input file to MSC/NASTRAN for analysis.

   9a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob16.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

   9b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob16 scr=yes**. Monitor the run using the UNIX **ps** command.

10. When the run is completed, edit the **fin.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

11. MSC/NASTRAN Users have finished this exercise. MSC/PATRAN Users should proceed to the next step.

12. Proceed with the Reverse Translation process, that is, attaching the **prob16.xdb** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

Analysis

Action:	**Attach XDB**
Object:	**Result Entities**
Method:	**Local**

Select Results File

Select Results File	**prob16.xdb**

OK

Apply

13. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

Choose the **Default** result case, and plot the result by selecting **Temperature** in the Select Fringe Result.

Results

Action:	**Create**
Object:	**Quick Plot**
Select Result Cases	**Default, A1:Non-Linear:100.% Load**
Select Fringe Result	**Temperatures**

Apply

Importing IGES file and auto-tet mesh the model