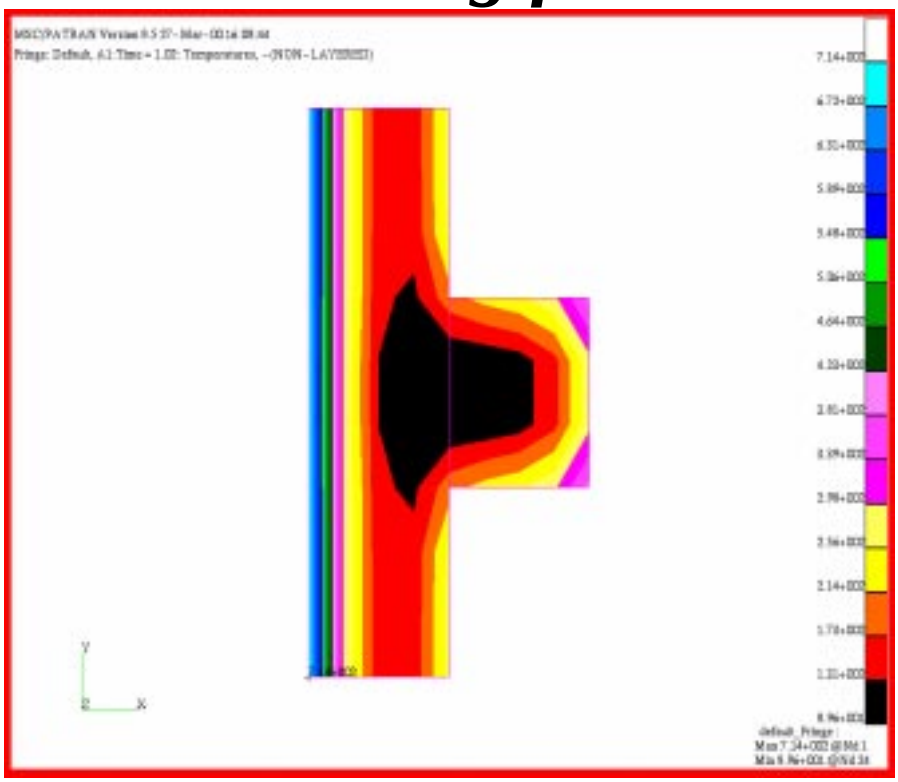
APPENDIX A

Transient Thermal Analysis of a Cooling fin



Objectives:

- Create a new database.
- Create the surface.
- Assign the thermal loads
- Submit the model for analysis

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	**fin**

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. Create the surfaces of the cooling fin

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Reference Coordinate Frame	**Coord 0**
Vector Coordinates List	**[0.5, 2, 0]**
Origin Coordinates List	**[0, 0, 0]**

Apply

Repeat the previous step to create the remaining surface.

Vector Coordinates List	**[0.5, 0.666667, 0]**
Origin Coordinates List	**[0.5, 0.666667, 0]**

Apply

3. Generate the mesh seed for the surfaces created:

Finite Element

Action:	**Create**
Object:	**Mesh Seed**
Method:	**Uniform**
Element Edge Length Data	**Number of Elements**
Number =	**9**
Curve List	**Surface 1.1 1.3**

Apply

Number =	**4**
Curve List	**Surface 1.2 1.4 2.2 2.4**

Apply

Number =	**3**
Curve List	**Surface 2.3**

Apply

Using the mesh seed generated in the previous step, mesh the geometry and create finite elements.

Finite Element

Action:	**Create**
Object:	**Mesh**
Method:	**Surface**
Surface List	**Surface 1 2**

Apply

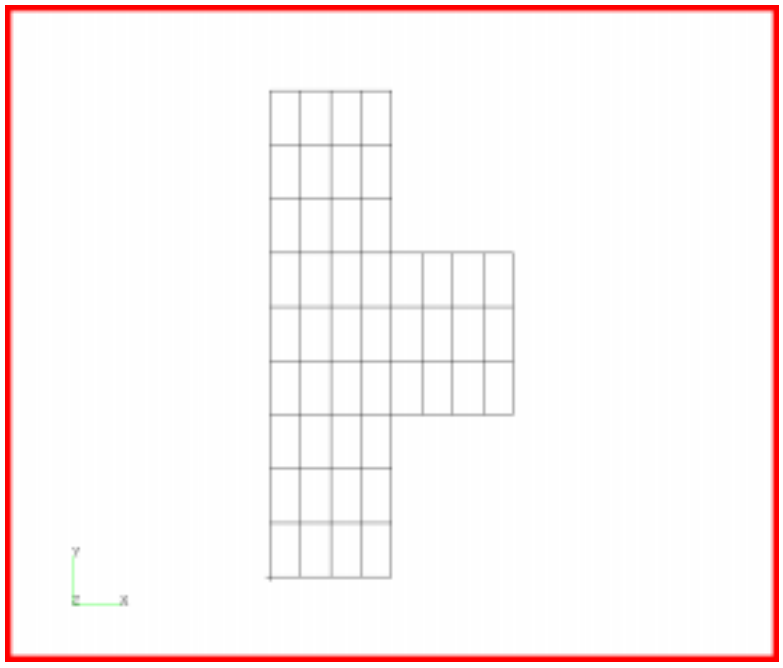
Use equivalence function to make sure all the overlapping nodes are connected.

Finite Element

Action:	**Equivalence**
Object:	**All**

Method: **Tolerance Cube**

Apply



4. Next, define a material using the specified thermal conductivity, specific heat, and density.

Materials

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Material Name: **mat_1**

Input Properties

Thermal Conductivity **6e-4**

Specific Heat **0.146**

Density **0.283**

OK

Apply

5. Next, reference the material that was created in the previous step. Define the properties of the cooling fin.

Properties

Action:	**Create**
Object:	**2D**
Type:	**Shell**
Property Set Name	**fin**

Input Properties

Material Name	**m:mat_1**
Thickness	**1**

OK

Select Members	**Surface 1 2**

Add

Apply

6. Since this is a transient analysis problem, a transient load case needs to be defined before loads and boundary conditions are applied.

Load Cases

Action:	**Create**
Load Case Name	**transient**
Load Case Type:	**Time Dependent**

Apply

7. Assign the convection properties to the cooling fin.

7a. The convection on the left edge is defined as follows:

Loads/BCs

Action:	**Create**
Object:	**Convection**
Type:	**Element Uniform**
New Set Name	**conv**

Target Element Type: **2D**

Input Data

Surface Option: **Edge**

Edge Convection Coef **0.001543**

Ambient Temperature **2500**

OK

Select Application Region

Geometry Filter **Geometry**

Select Surfaces or Edges **Surface 1.1**

Add

OK

Apply

7b. The right hand side of the fin undergoes a different type of convection.

Loads/BCs

Action: **Create**

Object: **Convection**

Type: **Element Uniform**

New Set Name **conv_right**

Target Element Type: **2D**

Input Data

Surface Option: **Edge**

Edge Convection Coef **0.001157**

Ambient Temperature **1000**

OK

Select Application Region

Geometry Filter **FEM**

Select 2D Elements or Edge **Element 37:40.1.1 4:12:4.1.2 28:48:4.1.2 45:48.1.3**

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 ◆ **TRANSIENT ANALYSIS**

Solution Parameters

Default Init Temperature **70**

OK

Subcase Create

Available Subcases **transient**

Subcase Parameter

Initial Time Step = **0.1**

Number of Time Steps = **20**

OK

Apply

Cancel

Apply

An MSC.Nastran input file called **fin.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 fin.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 fin 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 **fin.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	**fin.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:Time = 1.02**
Select Fringe Result	**Temperature**

Apply