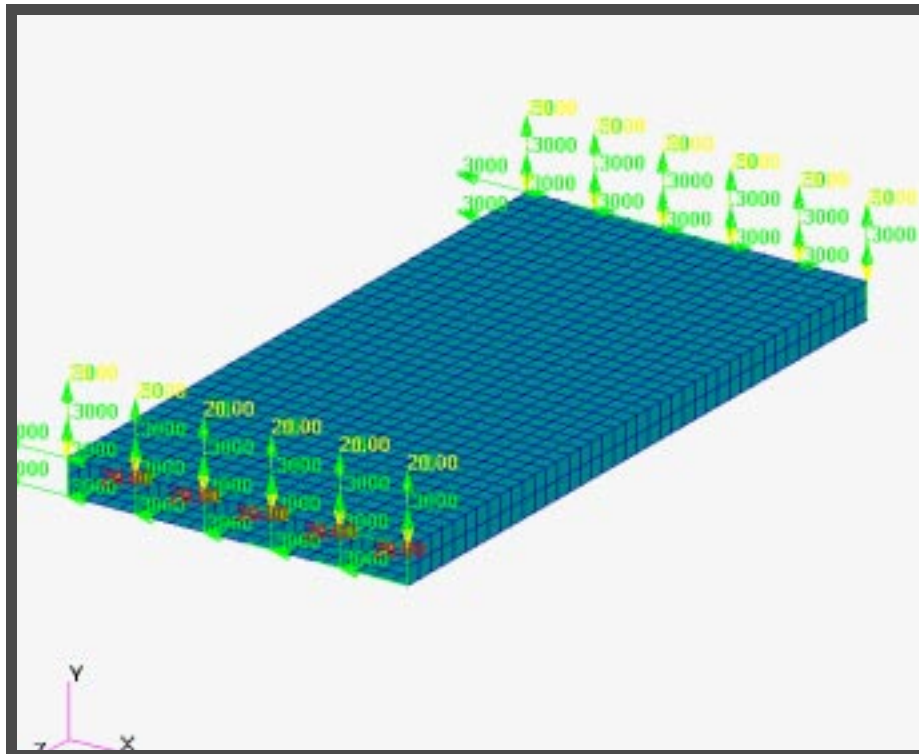

WORKSHOP 5

Typical Avionics Flow



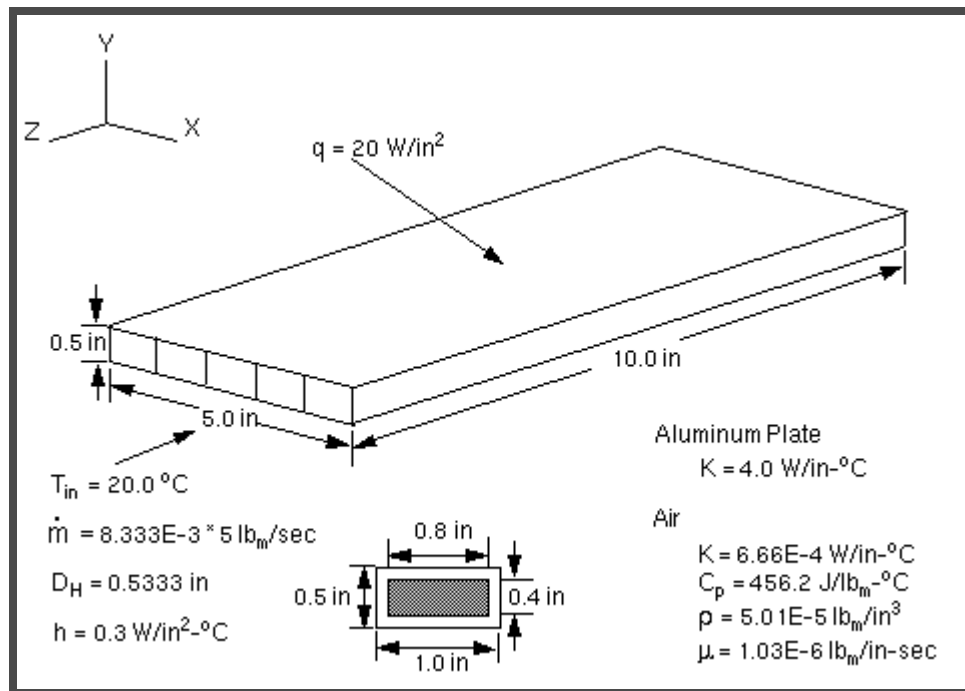
Objective:

- Modeling this problem within the MSC.Patran and MSC.Nastran.
- This method allows the analyst an option to specify non-coincident mesh sizes on the structure and the fluid nodes.



Model Description:

In this exercise, the compact heat exchanger is being modeled using MSC.Patran. MSC.Patran can associate the structure nodes with the fluid nodes using a technique called the Closet Approach method. This method allows the analyst an option to specify non-coincident mesh sizes on the structure and the fluid nodes. However, it is recommended that you use an identical mesh size for a regular isoparametric rectangular mesh.





Suggested Exercise Steps:

- Create a new database called **avionics_flow.db**.
- Create a new surface that has a total of five rectangular ducts.
- Mesh surfaces and curves with global edge length of 0.25
- Merge nodes by using Equivalence method under Finite Elements.
- Input specify Material Properties
- Define the thickness of four side walls that separate fluid channels.
- Apply loads and boundary conditions to the model.
- Perform analysis and read the analysis results.



Exercise Procedure:

1. Create a **New Database** and name it **avionics_flow.db**.

File/New...

New Database Name

avionics_flow

OK

2. Change the *Tolerance* to **Default** and the *Analysis Code* to **MSC.Nastran** in the *New Model Preferences* form. Verify that the *Analysis Type* is **Thermal**.

New Model Preference

Tolerance

◆ **Default**

Analysis Code:

MSC/NASTRAN

Analysis Type

Thermal

OK

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

3. Create the geometry that represents the the compact heat exchanger with five rectangular ducts.

◆ **Geometry**

Action:

Create

Object:

Curve

Method:

XYZ

Vector Coordinates List:

<1 0 0>

Origin Coordinates List:

[0 0 0]

Apply

You will now use Transformation to create the upper part of the rectangular duct.

◆ **Geometry**

<i>Action:</i>	Transform
<i>Object:</i>	Curve
<i>Method:</i>	Translate
<i>Translation Vector:</i>	<0 0.5 0>
<i>Curve List:</i>	Curve 1
Apply	

Finish the rectangular surface by creating verticals lines connecting the two previous horizontal lines.

◆ **Geometry**

<i>Action:</i>	Create
<i>Object:</i>	Curve
<i>Method:</i>	Point
<i>Starting Point:</i>	Point 1
<i>Ending Point:</i>	Point 3
Apply	
<i>Starting Point:</i>	Point 2
<i>Ending Point:</i>	Point 4
Apply	

Extrude the surface.

◆ **Geometry**

<i>Action:</i>	Create
<i>Object:</i>	Surface
<i>Method:</i>	Extrude
<i>Translation Vector</i>	<0 0 -10>
<i>Curve list</i>	Curve 1:4

Apply

Use Iso 1 View Icon to obtain 3D view



Create another surface and translate it to another surface

◆ **Geometry**

Action:

Create

Object:

Curve

Method:

XYZ

Vector Coordinates List:

<0 0 -10>

Origin Coordinates List:

[0.5 0.25 0]

Apply

◆ **Geometry**

Action:

Transform

Object:

Surface

Method:

Translate

Translation Vector:

<1 0 0>

Repeat Count:

4

Select the Surface Icon



Surface List:

Surface 1 2 4

Apply

Translate the final curve to complete the rectangular duct

◆ **Geometry**

Action:

Transform

Object:

Curve

Method:

Translate

Translation Vector:

<1 0 0>

Repeat Count:

4

Select the Curve Icon

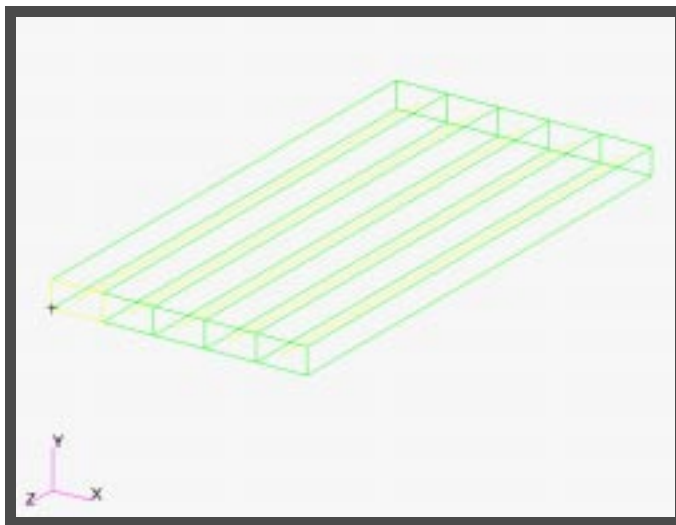


Curve List:

Curve 5

Apply

When you are finished your model should look like the one shown in the figure below.



4. Mesh Surfaces 1 to 16 to create QUAD4 elements with global edge length 0.25.

◆ **Finite Elements**

Action:

Create

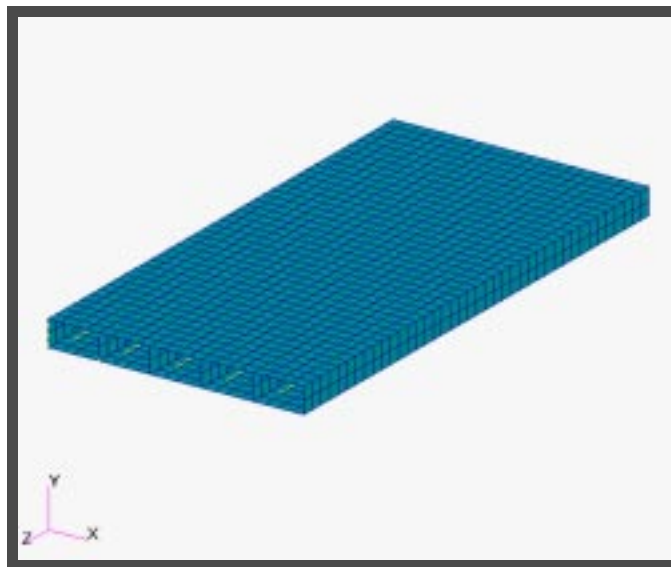
<i>Object:</i>	Mesh
<i>Type:</i>	Surface
<i>Global Edge Length</i>	0.25
<i>Element Topology</i>	Quad 4
<i>Surface List</i>	Surface 1:16
Apply	

5. Similarly, mesh Curves 5 to 9 with Bar2 element using a Global Edge Length of 0.25.

◆ **Finite Elements**

<i>Action:</i>	Create
<i>Object:</i>	Mesh
<i>Type:</i>	Curve
<i>Global Edge Length:</i>	0.25
<i>Element Topology:</i>	Bar2
<i>Curve List:</i>	Curve 5:9
Apply	

Your model should appear like the one shown below.



-
6. Equivalence the Finite Elements to reduce the number of elements by eliminating duplicate nodes.

◆ **Finite Elements**

<i>Action:</i>	Equivalence
<i>Object:</i>	All
<i>Type:</i>	Tolerance Cube
<i>Equivalence Tolerance:</i>	0.005
Apply	

7. Create the isotropic aluminum material properties using the material constants.

◆ **Materials**

<i>Action:</i>	Create
<i>Object:</i>	Isotropic
<i>Method:</i>	Manual Input
<i>Material Name:</i>	alum

Input Properties...

<i>Constitutive Model:</i>	Solid properties
<i>Thermal Conductivity:</i>	4.0

OK

Apply

<i>Material Name:</i>	air
-----------------------	------------

Input Properties...

<i>Constitutive Model:</i>	Fluid properties
<i>Thermal Conductivity:</i>	6.66e-4
<i>Specific Heat:</i>	456.2
<i>Density:</i>	5.01e-5
<i>Dynamic Viscosity-:</i>	1.03e-6

OK

Apply

8. Create the model's element properties assigning the material type and element thickness to the correct region of the model. Use the names of **inner_wall**, and **outside_wall** for the property names. The Thickness of the four side walls that separate fluid channels is 0.1 inch. The other walls have a thikckness of 0.05 inch.

◆ **Properties**

Action:

Dimension:

Type:

Property Set Name:

Input Properties...

Material Name:

Thickness:

OK

Select Front View Icon to choose walls



Select Members:

Add

Apply

Property Set Name:

Input Properties

Material Name:

Thickness:

OK

Select Members

Add

Apply

For the flow tube elements, the equivalent hydraulic diameter is

$$D_h = (4 \cdot 0.32) / (2.4) = 0.5333 \text{ in}$$

◆ **Properties**

Action: **Create**

Dimension: **1D**

Type: **Flow Tube**

Property Set Name: **air_flow**

Input Properties...

Material Name: **m:air**

Diameter at Node 1: **0.5333**

OK

Select Members: **Curve 5:9**

Add

Apply

9. Apply a heat load on the top surface.

◆ **Load/BCs**

Action: **Create**

Object: **Applied Heat**

Type: **Element Uniform**

Options: **Normal Flux**

New Set Name: **flux**

Target Element Type: **2D**

Input Data...

Surface Option: **Top**

Top Surf Heat Flux: **20**

OK

Select Application Region

Geometry Filter: ◆ **Geometry**

Select Surfaces or Edges: **Surface 2 6:15:3**

Add

OK

Apply

10. Define the Inlet Temperature of the Fluid.

◆ **Load/BCs**

Action: **Create**

Object: **Temp (Thermal)**

Type: **Nodal**

New Set Name: **inlet_temp**

Input Data...

Boundary Temperature: **20**

OK

Select Application Region

Geometry Filter: ◆ **Geometry**

Select Geometry Entities **Point 9 27:33:2**

Add

OK

Apply

11. Apply Coupled Advection. Five load sets, one for each channel, are defined for the fluid-structure coupling.

◆ **Load/BCs**

Action: **Create**

Object:
Type:
Option:
New Set Name:
Target Element Type:
Region 2:

Surface Option:
Top Surf Heat Convection Coef:
Mass Flow Rate:

Geometry Filter:

◆ **Geometry**



Change the view to Front View

Select Surfaces or Edges:

Active list

For the Companion Region

Select Curves:

Do the same for the remaining four(4) channels.

New Set Name:

Select Application Region

Geometry Filter: **Geometry**

Active list

Select Surfaces or edges:

Active list

For the Companion
Region

Select Curves:

New Set Name

Select Application Region

Geometry Filter: **Geometry**

Active list

For the Companion
Region

Select Surfaces or edges:

Active list

For the Companion
Region

Select Curves:

New Set Name

conv4

Select Application Region

Geometry Filter:

◆ **Geometry**

For the Companion
Region

Active list

Select Surfaces or edges:

Surface 10:13

Add

For the Companion
Region

Active list

Select Curves:

Curve 8

Add

OK

Apply

New Set Name

conv5

Select Application Region

Geometry Filter:

◆ **Geometry**

For the Companion
Region

Active list

Select Surfaces or edges:

Surface 13:16

Add

For the Companion
Region

Active list

Select Curves:

Curve 9

Add

OK

Apply

12. Analyze the model.

◆ **Analysis**

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name:

ex5

Apply

An MSC.Nastran input file called **ex5.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:

13. Submit the input file to MSC.Nastran for analysis.
 - 13a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran ex5.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
 - 13b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran ex5 scr=yes**. Monitor the run using the UNIX **ps** command.
14. When the run is completed, edit the **ex5.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.

15. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**
16. Proceed with the Reverse Translation process, that is, attaching the **ex5.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	<input type="text" value="Attach XDB"/>
<i>Object:</i>	<input type="text" value="Result Entities"/>
<i>Method:</i>	<input type="text" value="Local"/>
<input type="text" value="Select Results File..."/>	
<i>Select Results File</i>	<input type="text" value="ex5.xdb"/>
<input type="text" value="OK"/>	
<input type="text" value="Apply"/>	

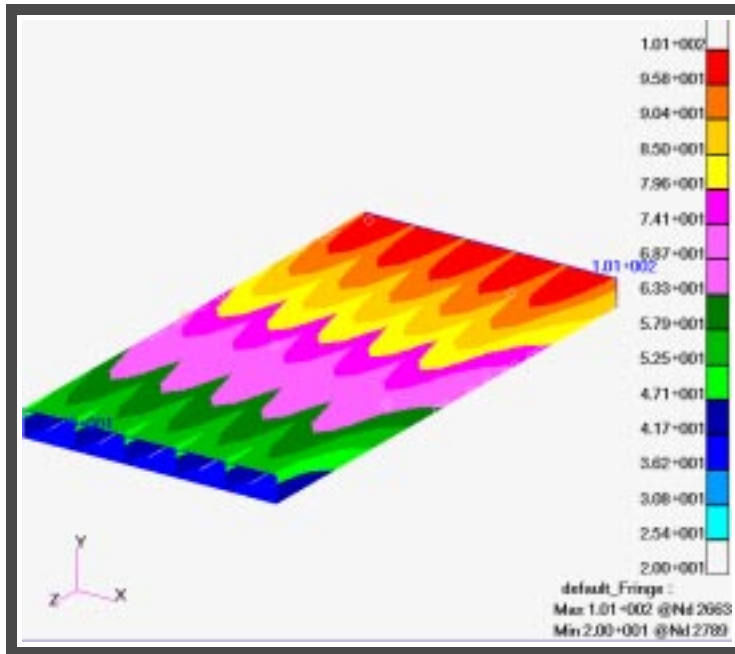
17. Display the Results.

◆ **Results**

<i>Form Type:</i>	<input type="text" value="Basic"/>
<i>Select Results Cases</i>	<input type="text" value="1.1 Default, PW Linear: 100. % of Load"/>
<i>Select Fringe Result</i>	<input type="text" value="1.1 Temperatures"/>



Change to Iso 1 View



The viewport may now be reset by clicking on the broom icon in the main window.



File/Quit...