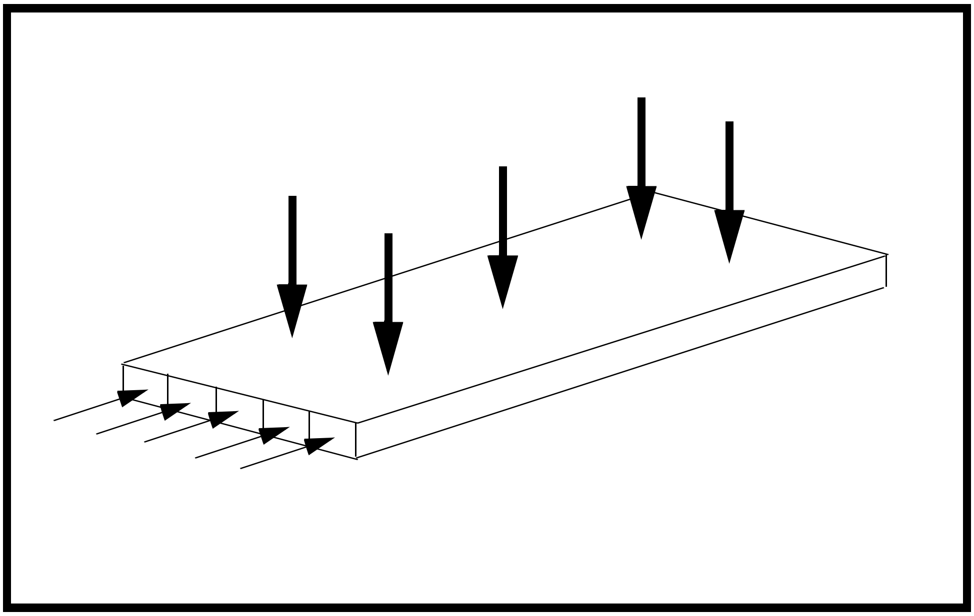
LESSON 2

Fluid Advection Thermal Analysis

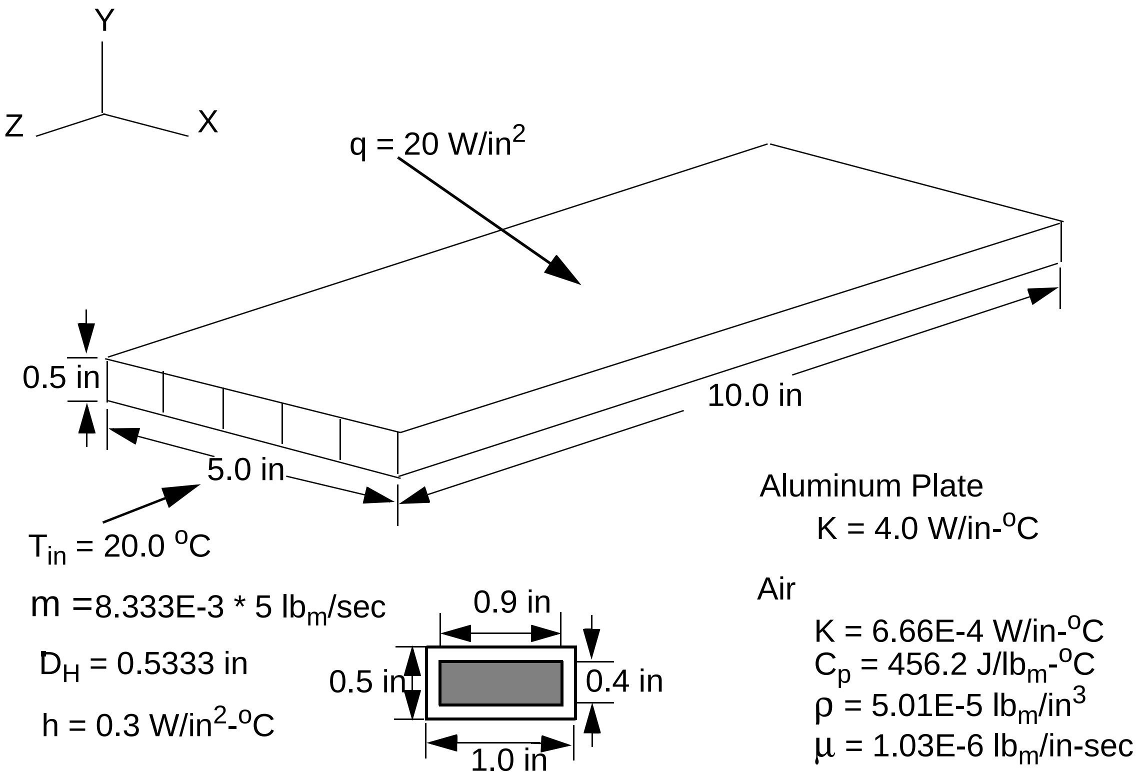


Objectives:

- Create a geometric representation of a rectangular duct.
- Apply appropriate convection and advection loads to model the airflow through the duct of this heat exchanger.
- Add a constant heat flux to one side of the duct..
- Run a steady-state heat transfer analysis of the duct.

Model Description:

Below is shown a compact heat exchanger consisting of 5 small ducts which pass the cooling air through them. The model contains both a constant heat flux and constant air flow properties. In this example you will model the behavior of one of the ducts in a steady-state heat transfer analysis.



Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0.2 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.2.

On the *Open Model File* form, select **New Model.**

Open Model File:	**New Model**

2. Create the NASTRAN geometry for the duct.

First, create a surface for one side of the duct.

Geometry/Surface/Plane...

OK

Width (along Plane X):	**1**
Height (along Plane Y):	**10**

OK

Cancel

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale...

Next, copy the surface to create the opposite side of the duct.

Geometry/Copy/Surface...

Select All

OK

OK

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**0**	**0**	**0.5**

OK

Rotate the model to a dimetric view.

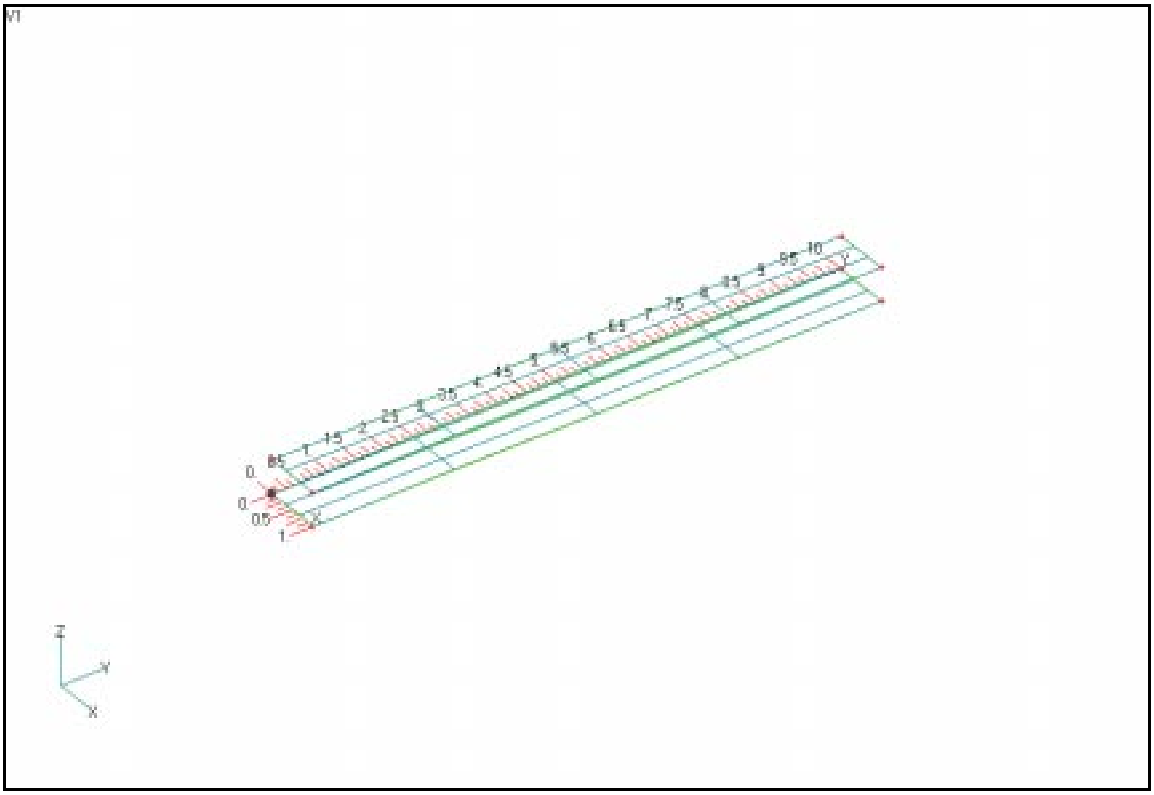
View/Rotate... **(or use <F8>)**

Dimetric

OK

Your model should be like the following:

Figure 2-1: Geometry of the 2 sides of the duct model



Create the third side of the duct by connecting corners.

Geometry/Surface/Corners...

Methods^ *(select **On Point**)*

Point ID:

3	**OK**
4	**OK**
10	**OK**
9	**OK**

Cancel

Copy this surface to create the final side of the duct.

Geometry/Copy/Surface...

ID: **3**

OK

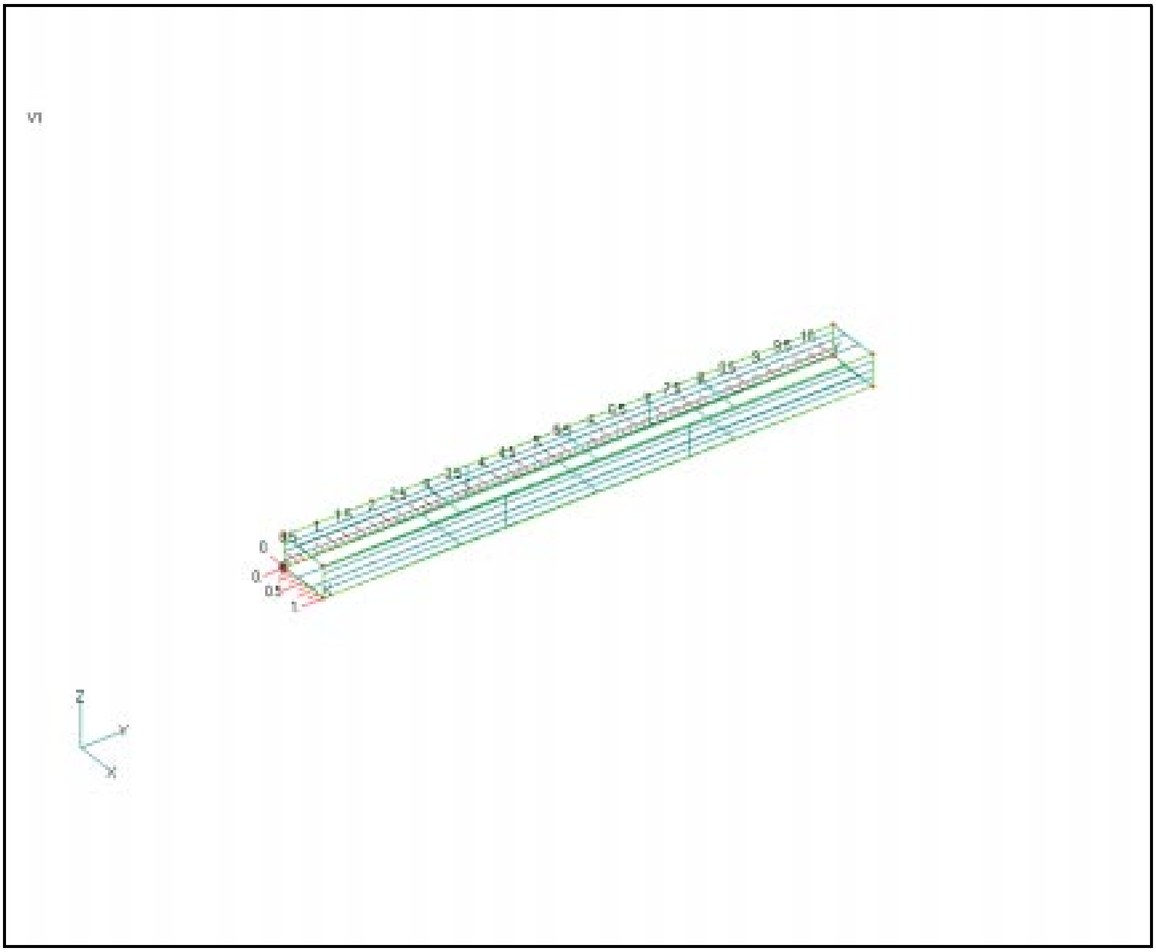
OK

	X:	*Y:*	*Z:*
Base:	**0**	**0**	**0**
Tip:	**-1.0**	**0**	**0**

OK

Your model should be like the following:

Figure 2-2: Geometry of the duct model



3. Set the default size for the mesh.

Mesh/Mesh Geometry/Default Size...

Size:	**0.5**
OK	

4. Create a material called **heatsink**.

From the pulldown menu, select **Model/Material.**

Model/Material...

Title: **heatsink**

Conductivity, k: **4**

OK

Cancel

5. Create a property called **heatsink** to apply to the members of the duct itself.

From the pulldown menu, select **Model/Property**.

Model/Property...

Title: **heatsink**

To select the material, click on the list icon next to the databox and select **heatsink**.

Material: **1..heatsink**

Thickness, Tavg or T1: **0.05**

OK

Cancel

6. Create the mesh for the duct.

Mesh/Geometry/Surface...

Select All

OK

Property: **1..heatsink**

OK

7. Remove the labels from the screen.

View/Options... **(or use <F6>)**

Quick Options...

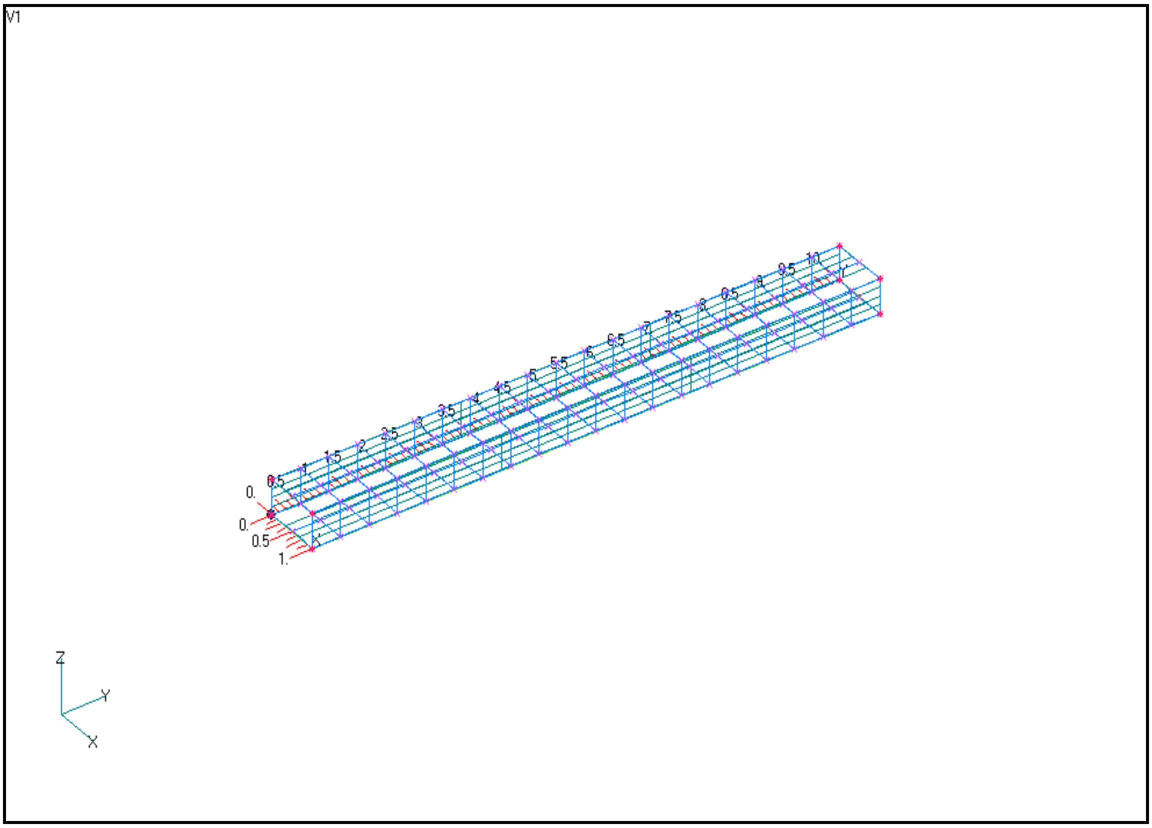
Labels Off

Done

OK

Your model should be like the following:

Figure 2-3: Meshed model



8. Create a curve to apply the airflow representation to.

First, create two points to define the line.

Geometry/Point...

Methods^ *(select **Locate**)*

X:	*Y:*	*Z:*	
0	**0**	**1.5**	**OK**

X: *Y:* *Z:*

0 **10** **1.5** **OK**

Cancel

Next, connect the two points with a line.

Geometry/Curve-Line/Points...

From Point: **29**

To Point: **30**

OK

Cancel

9. Create the mesh for the curve.

Mesh/Geometry/Curve...

(select newly created curve)

OK

New Prop...

Elem/Property Type...

● **Tube**

OK

Title: **flow tube**

Material: **1..heatsink**

OK

OK

10. Check for coincident nodes and merge them.

Tools/Check/Coincident Nodes...

Select All

OK

When asked if you wish to specify an additional range of nodes to merge, select **No**.

No

☒ **Merge Coincident Entities**

OK

11. Create a body load defining the properties of the fluid flow.

First, a load set must first be created before creating the appropriate model loading.

Model/Load/Set...

Title: **heatsink**

OK

Next, create the following body load.

Model/Load/Body...

(next to Thermal options) ☒ **Active**

Default Temperature: **98**

OK

Model/Load/Heat Transfer...

Forced Convection **Alternate Formulation** ☒

Constant Coefficient: **0.023**

Reynolds Exponent: **0.8**

Prandtl Exponent (into fluid): **0.4**

Prandtl Exponent (out of fluid): **0.4**

Fluid Conductivity: **6.66e-4**

Fluid Specific Heat: **456.2**

Fluid Viscosity: **1.03e-6**

Fluid Density: **5.01e-5**

OK

12. Create the inlet temperature for the fluid flow.

Model/Load/Nodal...

*(select left node on curve - **Node 211**)*

OK

Type: ● **Temperature**

Temperature: **20**

OK

Cancel

13. Create the convection loading for the model.

First, create the loading for the duct elements.

Model/Load/Elemental...

ID: **1**

to: **120**

More

OK

Type: ● **Convection**

☒ **Forced Convection**

☒ **Disable Advection**

Flow Rate: **.008333**

Diameter: **.5333**

Area Factor: **3**

OK

Next, create the loading for the flow elements.

ID:	**121**
to:	**140**
More	
OK	
Type:	● **Convection**
	☒ **Forced Convection**
	☒ **Disable Convection**
Flow Rate:	**.008333**
Diameter:	**.5333**
Temperature:	**0**
OK	

14. Create the heat flux load on one side of the duct.

ID:	**1**
to:	**40**
More	
OK	
Type:	● **Heat Flux**
Flux:	**20**
OK	
Face:	**1**
OK	
Cancel	

15. Before running the analysis, check to make sure that all element normals are defined correctly.

Tools/Check/Normals...

Select All
OK
OK

The message box now lists all elements with reversed normals (in our case, they should be elements 81 through 120).

This is unacceptable, since the fluid flow definitions need to be defined with a consistent direction in order to run the analysis.

16. Modify the element normals so the analysis will run correctly.

Modify/Update Elements/Reverse...

ID:	**81**
to:	**120**
More	
OK	
	● **Align First Edge to Vector**
OK	
Methods^	*(select* ***Nodes****)*
Base Node ID:	**168**
Tip Node ID:	**127**
OK	

All of the convection arrows should now point in the same direction. Zoom into the model to check the flow direction if it is necessary. All flow should be in the **Y** direction.

17. Create the input file and run the analysis.

File/Analyze...

Analysis Type:	**20..Steady-State Heat Transfer**

☒ **Run Analysis**

OK

When asked if you wish to save the model, respond **Yes**.

Yes

File Name: **fluid**

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

When asked if it is "OK to Begin Reading File C:\TEMP\fluid.xdb", respond **Yes**.

Yes

18. Remove the thermal loading markers from the screen.

View/Options...

Quick Options...

☐ **Load - Thermal**

☐ **Load - Heat Flux**

☐ **Load - Convection**

Done

OK

19. Create a final temperature distribution contour plot.

View/Select...

Contour Style: ● **Contour**

Deformed and Contour Data...

Output Set: **1..MSC/NASTRAN Case 1**

Deformation: **31..Temperature**

Contour: **31..Temperature**

OK

OK

In Figure 2-4, notice how the temperature profile of the duct increases as you traverse the duct, repesenting the fact that the air removes the most heat at the duct inlet (where the temperature difference is greatest).

Also notice that the elements on the bottom (where the heat flux is applied) have a higher temperature as well.

When done, exit MSC/NASTRAN for Windows.

File/Exit

This concludes this exercise.

Figure 2-4: Fluid Advection Thermal Analysis on a Duct Model