Introduction: In this example you will model the velocity and temperature distribution over a heatsink.

Problem Description:

- Units: Use S.I. units ... meters ONLY
- Geometry: See figure.
- Boundary conditions: There is a two meter square and 1 meter tall space around the fin assembly
- Objective:
 - To determine the flow around the object, temperature and velocity plots!.
 - To determine the difference between constant wall temperature and constant wall flux conditions
- Figure:





• Problem repeated with:

- <u>Constant wall flux</u>
 - o Dimpled Fin's

Basic Outline of the Problem:

Preprocessing:

- 1. Start ANSYS.
- 2. Create volumes.
- 3. Copy volumes to form heat sink fins.
- 4. Define the fluid properties
- 5. Define element type. (Flotran CFD 3D Flotran 142)
- 6. Specify meshing controls / Mesh the areas to create nodes and elements.

Solution:

- 7. Specify boundary conditions.
- 8. Solve.

Postprocessing:

Examine results!

Exit:

11. Exit the ANSYS program, saving all data.

STARTING ANSYS

- Click on ANSYS 6.1 in the programs menu. Select Interactive.
- The following menu that comes up. Enter the working directory. All your files will be stored in this directory. Also enter 64 for Database. (more if you are working with extra memory)
- Click on Run.

	ANSYS/University Low	2
Enable ANSYS Parallel Po	erformance 🛛 🗖 Use ANSYS Drop	Test Module
Working directory	C:\Documents and Settings	s\dwj
Graphics device name	win32 💌	
Initial jobname	file	_
MEMORY REQUESTED (megal	oytes)	
for Total Workspace	64	
for Database	32	_
Read START.ANS file at star	t-up? Yes 💌	
Parameters to be defined (-par1 val1 -par2 val2	, [_
Language Selection	[english]	<u> </u>
Execute a customized ANSY	S executable	
Run Close	Beset Cancel	About

MODELING THE STRUCTURE

- Go to the ANSYS Main Menu
- Preprocessor>Modeling>Create>Volumes>Block>By 2Corners and Z.
 - The following window comes up:

🛃 Block by 2 Corners & Z 🛛 🔯		
• Pick	🔿 Unpick	
WPX =		
¥ =		
Global X =		
¥ =		
Z =		
WP X	0	
WP Y	0	
Width	0.056	
Height	0.0043	
Depth	0.129	
ок	Apply	
Reset	Cancel	
Help		

- Enter the following data values to create the thin base of the heat generating volume. •
- Next, we want to create the fins that will constitute the assembly. If you had hit APPLY instead of OK, then the window w • use Create Volume by 2 Corner and Z again to create the first fin.

A Block by 2 C	orners & Z 🛛 🔀
• Pick	🔿 Unpick
WPX =	
¥ =	
Global X =	
¥ =	
Z =	
WP X	0
WP Y	0.0043
Width	0.002
Height	0.0402
Depth	0.129
ОК	Apply
Reset	Cancel
Help	

- Now we want to copy the fin volume and paste it offset so that it is correctly positioned along the bottom of the fin asser
 Preprocessor>Modeling>Copy>Volumes Select the volume to be copied. Next, the following window appears:

Copy Volumes	$\overline{\mathbf{X}}$
[VGEN] Copy Volumes	
ITIME Number of copies -	2
- including original	
DX X-offset in active CS	0.009
DY Y-offset in active CS	
DZ Z-offset in active CS	
KINC Keypoint increment	
NOELEM Items to be copied	Volumes and mesh
OK Apply	Cancel Help

- Enter the value for the offset, which is 0.009m and then click Apply. Note that the copy volumes dialog box on the left is new volume and hit **apply** again. Hit **apply** in the window that appears because the offset is still 0.009. Continue until all didn't hit apply just start from **Modeling>Copy>Volumes**.
- The model should appear as below so far.



- Now, make the aluminum fin assembly one volume by using Preprocessor>Modeling>Operate>Booleans>Add>V the Dialog Box.
- Create one more volume, the control volume through which flow will pass. Modeling>Create>Volumes>Block>By D

Create Block by Dimensions		
[BLOCK] Create Block by Dimensions		
X1,X2 X-coordinates	-0.2	0.256
Y1,Y2 Y-coordinates	0	0.2445
Z1,Z2 Z-coordinates	-0.2	0.329
OK Apply	Cancel	Help

- After this step it may prove useful to plot by lines. This will enable you to see everything more easily.
- Finally, subtract the aluminum fin assembly from the big box. Use Modeling>Operate>Booleans>Subtract>Volume Select the box volume, hit OK, then select aluminum (this should only take one click!). Hit OK again and now you're done
- The modeling is now finished. If you plot lines, and using **Pan-Zoom-Rotate**, select **Obliq** and hit the **Fit** button near the figure should appear as the following. Also, if you plot volumes, rotate the model so you can see the underside, and then using **WinZOOM**, zoom in on the heat sink area, the second image is what you should see.. a HOLE!!





SELECTING ELEMENT TYPE:

• Click Preprocessor>Element Type>Add/Edit/Delete... In the 'Element Types' window that opens click on Add... Th

Library of Element Types	
Library of Element Types	Thermal Mass Link Solid Shell ANSYS Fluid FLOTRAN CFD Magnetic Vector Scalar Magnetic Vector Scalar Magnetic Vector Scalar Magnetic Vector Scalar Magnetic Vector Scalar
Element type reference number	1
OK Apply	Cancel Help

- Type **1** in the Element type reference number.
- Click on Flotran CFD and select 3D Flotran 142. Click OK. Close the 'Element types' window.
- So now we have selected Element type 1 to be a Flotran element. The component will now be modeled using the principle finishes the selection of element type.

DEFINE THE FLUID PROPERTIES:

- Go to Preprocessor>Flotran Set Up>Fluid Properties.
 On the box, shown below, make sure all the input fields read Constant and then click on OK.

Fluid Properties	
[FLDATA12],PROP,DENS	
Density	Constant 🗸
[FLDATA13], VARY, DENS	
Allow density variations?	I No
[FLDATA12],PROP,VISC	
Viscosity	Constant
[FLDATA13],VARY,VISC	
Allow viscosity variations?	∏ No
[FLDATA12],PROP,COND	
Conductivity	Constant
[FLDATA13],VARY,COND	
Allow conductivity variations?	∏ No
[FLDATA12],PROP,SPHT	
Specific heat	Constant
[FLDATA13],VARY,SPHT	
	Cancel Help

Another box will appear, entitled CFD Flow Properties. Enter the values for Density, Viscocity, Conductivity, and Specific temperature (Tw+Tinfin)/2 = (315+293)/2 = 304 K.

CFD Flow Properties	$\overline{\mathbf{X}}$
Density property type CONSTANT	
Constant value	1.161
Viscosity property type CONSTANT	
Constant value	1.873e-5
Conductivity property type CONSTANT	
Constant value	0.0264
Specific Heat Property Type CONSTANT	
Constant value	1006.5
OK Cancel	Help

• These values correspond to the following constant properties:

Density (p)	1.161 Kg/m^3	
Dynamic Viscocity (µ)	1.873E-5 Kg/m*s	
Conductivity (K)	0.0264 W/m*K	
Specific Heat	1.0065E3 J/Kg*K	

<!--[if !vml]--><!--[endif]-->

MESHING

 Go to Preprocessor>Meshing>Size Controls>Manual Size>Lines>All Lines. In the menu that comes up type .01 Length'.

Element Sizes on All Selected Lines	
[LESIZE] Element sizes on all selected lines	
SIZE Element edge length	.01
NDIV No. of element divisions	
(NDIV is used only if SIZE is blank or zero)	
KYNDIV SIZE,NDIV can be changed	Ves
SPACE Spacing ratio	
Show more options	☐ No
OK	Help

- Next, use Preprocessor>Meshing>Size Controls>Manual Size>Lines>Picked Lines. With the dialog window ope constitute the control volume (all 12 edges). Hit OK. Enter 0.1 for 'element edge length'
- The line segments should look something like the following:



• Finally, use Preprocessor>Meshing>Mesh>Volumes>Free and mesh the volume. The result will look like the follow



BOUNDARY CONDITIONS AND CONSTRAINTS

Hydrostatic:

 Go to Preprocessor>Loads>Define Loads>Apply>Fluid CFD>Velocity>On Areas. Notice the direction of the pos apply a z-velocity so that the air will travel over the heatsink. In this case, we will apply a negative z-velocity to the positive z-direction is towards the front face.

[DA] Apply Velo
Apply VX load If Constant val VX Load value
Apply VY load If Constant val VY a Load value
Apply VZ load If Constant val VZ Load value
Apply to b
Movin
NOTE: Blank v

Apply VELO load on areas	
[DA] Apply Velocity Constraints on areas	
Apply VX load as a	1
If Constant value then:	
VX Load value	ſ
Apply VY load as a	I
If Constant value then:	
VY a Load value	ſ
Apply VZ load as a	Ţ
If Constant value then:	
VZ Load value	ſ
Apply to boundary lines?	F
Moving wall?	ſ
NOTE: Blank values not interpreted as 0's !!!	
OK Cancel	

- Enter -5 in the VZ value field and click OK. The value corresponds to a velocity of 5 meters per second of air flowing thro
- Then, set the Velocity to ZERO (Vx=Vy=Vz) along the bottom area (the ground) and on all faces of the fin assembly. ALL areas using the box command and then unselect the faces that constitute the ceiling and sides of the control volu
- Go to Main Menu>Preprocessor>Loads>Define Loads>Apply>Fluid CFD>Pressure DOF>On Areas. Pick all the allowed to escape. This includes the end oppostie the applied velocity, both sides and the ceiling. (4 faces out of 6).
- Enter **O** as the pressure value. (This sets the pressure as atmospheric allowing the air to pass out of the volume without

Apply PRES on areas	
[DA] Apply PRES on areas as a	0
If Constant value then:	
PRES Pressure value	
Apply to boundary lines?	v
ОК Арріу	Cancel

Thermal:

- NOTE: applying temperature boundary conditions can be difficult since the outside box is blocking the inside area the areas that make up the large box get selected instead of, or in addition to the areas that make up the fin a doesn't happen. Use the pan/zoom/rotate tool to check which areas are selected before applying boundary co can always unpick areas by either clicking the right mouse button and then selected areas to be unpicked, OR button on the 'Apply TEMP on Areas' box, and then selecting areas to unpick.
 - Go to Preprocessor>Loads>Define Loads>Apply>Thermal>Temperature>On Areas Since we can assume all tl base) are at room temperature (base acts like an insulator), we want to apply 293K to all areas except those that ma all areas except the heatsink use the box command and draw a box around the entire volume. Then, select the 'Unpi around the heatsink:



• Before you click OK, make sure your screen looks like this. Notice 5 areas are selected. These areas are the top and 4 sid



• If your screen looks like the picture above, click OK and enter 293 in the screen that appears:

Apply TEMP on are	as		
[DA] Apply TEMP on areas	as a	Constant va	alue 💽
If Constant value then: VALUE Load TEMP value		293	
Apply LEMP to boundary in	ies?	I ♥ Yes	
ок	Apply	Cancel	Help

- Now you must apply a temperature to the surface of the heatsink. Looking at the results of the 3d heat sink conduction p tutorial, anywhere between 300 and 315K is a good surface temperature. For this example 315K is choosen arbitraril areas on the heatsink, use a similar method to the one described above.
- Loadings are now set, your model should look like this:



NOTE: IGNORE THE DIRECTIONS OF THE ARROWS. THEY ARE CORRECTLY DEFINED, JUST MISLEADING

SOLUTION

• Go to ANSYS **Preprocessor > Flotran Set Up > Solution Options** and change the TEMP field to 'Thermal as shown:

Steady State
l⊽ Yes
Thermal
Laminar
Incompressible
□ No
□ No
□ No
I No
□ No
□ No

- Go to ANSYS Main Menu>Solution>Flotran Set Up>Execution Ctrl.
 The following window appears. Change the first input field value to 1000, as shown. Also, change the velocity termi (0.005). Click OK.

Steady State Control Settings	
[FLDATA2],ITER Iteration Control	
EXEC Global iterations	1000
OVER .rfl file overwrite freq	0
APPE .rfl file append freq	0
[FLDATA3],TERM Termination Criteria	
VX Velocity component	0.005
VY Velocity component	0.005
VZ Velocity component	0.005
PRES Pressure	1e-008
TEMP Temperature	1e-008
ENKE Turbulent kinetic energy	0.01
ENDS Turbulent dissipation	0.01
Note: Termination check is ignored for a DOF	
if its termination criterion is negative	
[FLDATA5],OUTP Output Options	
SUMF Output summary frequency	10
OK Cancel	Help

- Go to Solution>Run FLOTRAN.
- Wait for ANSYS to solve the problem.

POST-PROCESSING

- Plotting the velocity distribution...
- Go to General Postproc>Read Results>Last Set.
- Then, go to the ANSYS Main Menu (the Top Bar) and Click Plot>Lines.
- After the volumes have been plotted, go to the ANSYS Main Menu>WorkPlane and select WP Settings. Input .2
- Now, select WorkPlane > Display Working Plane.
- Now that the working plane is selected, go to ANSYS Main Menu>WorkPlane >Offset WP by Increments and snap increment in the positive x axis and then rotate the working plane about the -Y axis 90 Degrees
- Once the plane is in line, select ANSYS Main Menu>PlotCntrls>Style...>Hidden Line Options.. a pop up window v

Hidden-Line Options	
[/TYPE] [/SHADE] Hidden-Line Options	
WN Window number	Window 1
[/TYPE] Type of Plot	Q-Slice Z-buffer
[/CPLANE] Cutting plane is	Working plane
(for section and capped displays only)	
[/SHADE] Type of shading	Gouraud
[/GRAPHICS] Used to control the way a model is displayed	
Graphic display method is	PowerGraphics 🗨
[/REPLOT] Replot upon OK/Apply?	Replot
OK Apply	Cancel Help

- In this window change "Type of Plot" to Q-Slice Z-buffer, and "Cutting Plane is" to Working Plane and click OK. ANSY of the analysis with the working plane as the cutting plane.
 Then go to General Postproc>Plot Results>Contour Plot>Nodal Solution. Select DOF Solution and Velocity an
 The solution now looks like this:



• Now goto General Postproc>Plot Results>Contour Plot>Nodal Solution and select Temperature distribution:



• Go to Main Menu>General Postproc>Plot Results>Vector Plot>Predefined. The following window will appea

Vector Plot of Predefined Vectors	
[PLVECT] Vector Plot of Predefined Vectors	
Item Vector item to be plotted	DOF solution Velocity V Velocity V
Mode Vector or raster display	
	Vector Mode
	C Raster Mode
Loc Vector location for results	
	Elem Centroid
	C Elem Nodes
Edge Element edges	Hidden
[/VSCALE] Scaling of Vector Arrows	
WN Window Number	Window 1
VRATIO Scale factor multiplier	1
KEY Vector scaling will be	Magnitude based
OPTION Vector plot based on	Undeformed Mesh
OK Apply	Cancel Help

• Select **OK** to accept the defaults. This will display the vector plot of the velocity gradient. Remember to rotate the vie best angle.

Here is the vector plot from a few angles.







CONSTANT HEAT FLUX:

• If we repeat the same problem with a constant heat flux of 500 W/m^2 on the walls of the heatsink we get the following



 To get a clearer plot of the temperature distribution you can change the range of temperatures that are plotted. Go to Ut Menu>PlotCtrls>Style>Contours>Uniform Contours. In the window that appears select 'User specified' and then ε

Uniform Contours	$\overline{\mathbf{X}}$	
[/CONT] Uniform Contours		
WN Window number	ndow number All active wind	
NCONT Number of contours	9	
Contour intervals	,	
	C Auto calculated	
	C Freeze previous	
	User specified	
User specified intervals		
VMIN Min contour value	200	
VMAX Max contour value	620	
VINC Contour value incr	10	
[/REPLOT] Replot Upon OK/Apply?	Replot	
OK Analy	Cancel	
	Help	

• Your plot will now look like this:



DIMPLED FIN HEAT SINK:

- If we repeat the problem with constant wall temperature of 315K, but this time use fins with small dimples on them we ge increased surface area from the dimples. The dimples in the example are 2.5mm in diameter, and .5mm deep. There is a each side of the fin. To reduce the memory needed to compute the problem on 2 fins are used. There are a few different repeat this solution
- To ease the modeling of the dimpled fin, an IGS model of 1 dimpled fin is provided: <u>dimpledfin.IGS</u>
- Go to Utility Menu>File>Import>IGES. Unclick SMALL Delete small areas. Click OK. In the next screen select bro the directory you saved it to and click OK.

Import IGES File	$\overline{\mathbf{X}}$
[/AUX15] [IOPTN] Options for IGES Import	
Iges Import Option	
	No defeaturing
	C Defeature model
MERGE Merge coincident keypts?	Ves
SOLID Create solid if applicable	Ves
SMALL Delete small areas?	☐ No
OK Cancel	Help

• When the file finishes importing your screen will look like this:



• Create the rest of the heatsink model the same way as before (create the base, copy the fin, add all the volumes to creat looks like this:



• The surrounding area can be smaller than before, I choose to make it 100mm larger than each side. Go to Modeling>Cr Dimensions and enter the following numbers:

Create Block by Dimensions		
[BLOCK] Create Block by Dimensions		
X1,X2 X-coordinates	1	.229
Y1,Y2 Y-coordinates	0	.105
Z1,Z2 Z-coordinates	-,1	.111
OK Apply	Cancel	Help

• Subtract the fin model from the large volume and the modeling is done:



- To mesh this problem you need at least 500mb of RAM on your computer. Use an element size of 0.003 on all lines of the element size of 0.1 on the 12 lines of the outer volume. You can use an element size less than 0.003 if you have more RA of memory.
- Apply loads the same way as before.
- Flotran setup is the same as before.
- You should get the following results:

Temperature distribution on a fin:



Temperature distribution 0.5 mm from a fin:



Temperature distribution in half way between the fins:



Top view of temperature distribution low down on fin:



closeup:



Top view high up:



Velocity distribution 0.5mm from a fin:



Saving Projects

Simply go to Utility Menu>File>Save As... and save the project using the desired filename. To open the file later, run I explained in this tutorial) as usual, and when that is done, go to Utility Menu>File>Resume From... and choose the save saved in.