Proceedings of the Annual Symposium of the Institute of Solid Mechanics
and Session of the Commission of Acoustics, SISOM 2016 Bucharest 12-13 May

THERMAL ANALYSIS USING ANSYS SOFTWARE PACKAGE

Doina TĂRĂBUŢĂ (ENE), Mariana PĂTRAŞCU (ANTONESCU), Melania TĂMAŞ

Polytechnic University Bucharest; doinaene@ymail.com

This paper presents a thermal analysis who calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are:

- The temperature distributions
- The amount of heat lost or gained
- Thermal gradients
- Thermal fluxes.

Thermal simulations play an important role in the design of many engineering applications, including internal combustion engines, turbines, heat exchangers, piping systems, and electronic components. In many cases, engineers follow a thermal analysis with a stress analysis to calculate thermal stresses.

The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

ANSYS supports two types of thermal analysis:

1. A steady-state thermal analysis determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.
2. A transient thermal analysis determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

Keywords: ANSYS, conduction, convection, radiation, steady-state, transient.

1. INTRODUCTION

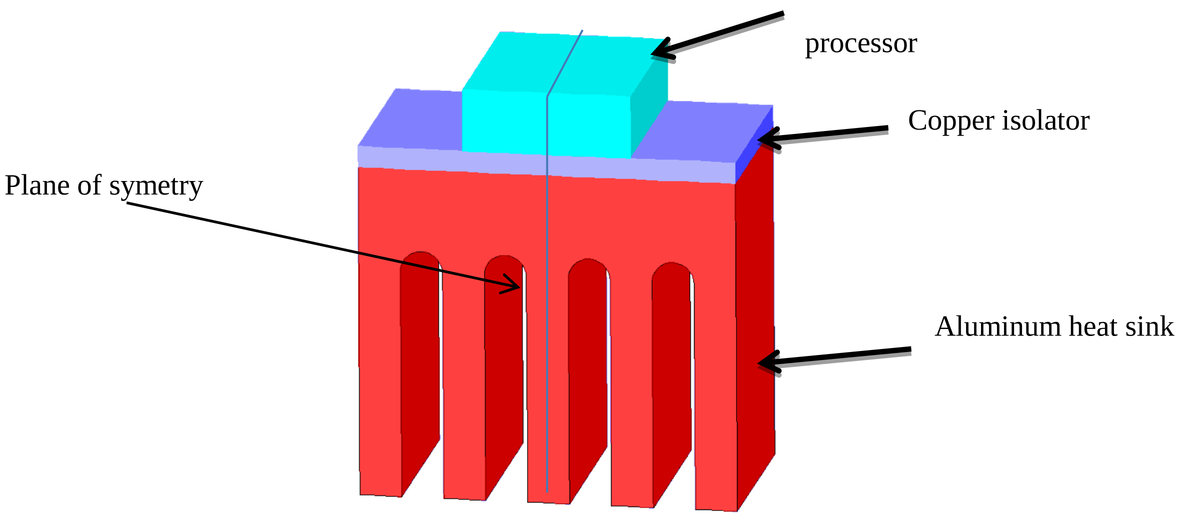


Figure 1 The system

For a stady-state thermal analysis, we consider a system composed of a processor who is mounted on a copper isolator which is in turn mounted on an aluminum heat sink. The processor generates heat, and the system also receives heat radiated by other nearby components. The hole system is fan cooled. The steady-state temperature of the system must be less than 100°C (figure 1).

2. GENERAL ANALYSIS GOALS

For this workshop, assuming steady-state operation at high power (20 Watts), we have to complete the following:

- Determine the temperature distribution in the system during steady-state operation.
- Find the location and the value of the maximum temperature in the system.
- Determine which regions, if any, violate the design requirement of operation below 100 °C.
- Verify results using hand calculations.

<u>Assumptions</u>

- One plane of mirror symmetry.
- Use SI units. (kg, m, s, W, J)
- Radiation from surrounding components is modeled as an equivalent uniform heat flux.
- Neglect contact resistance at component interfaces.
- Homogeneous processor material.
- Uniform processor heat generation.
- No heat transfer on front and back surfaces; (i.e., no axial temp gradient).
- Convection modeled with uniform film coefficient and constant bulk fluid temperature.
- Depth is 0.025m
- All dimensions shown are in meters.

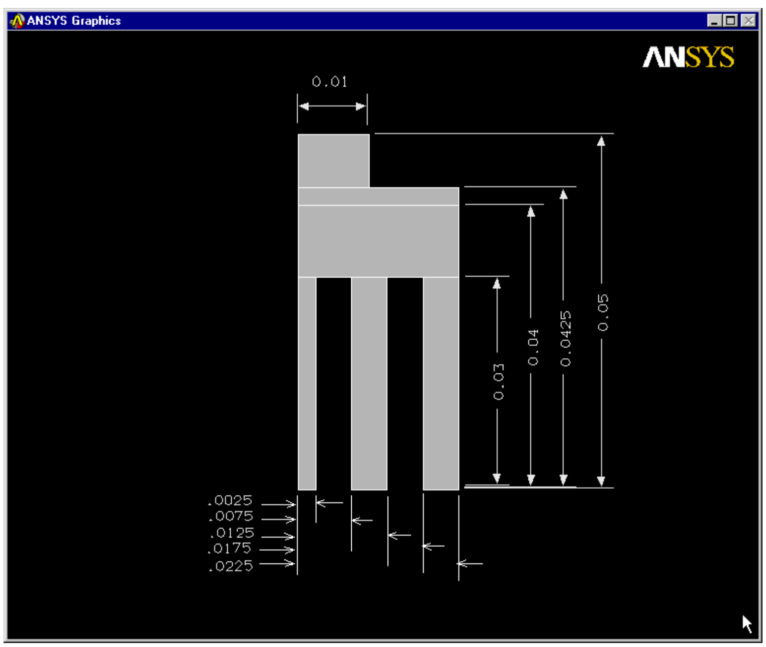
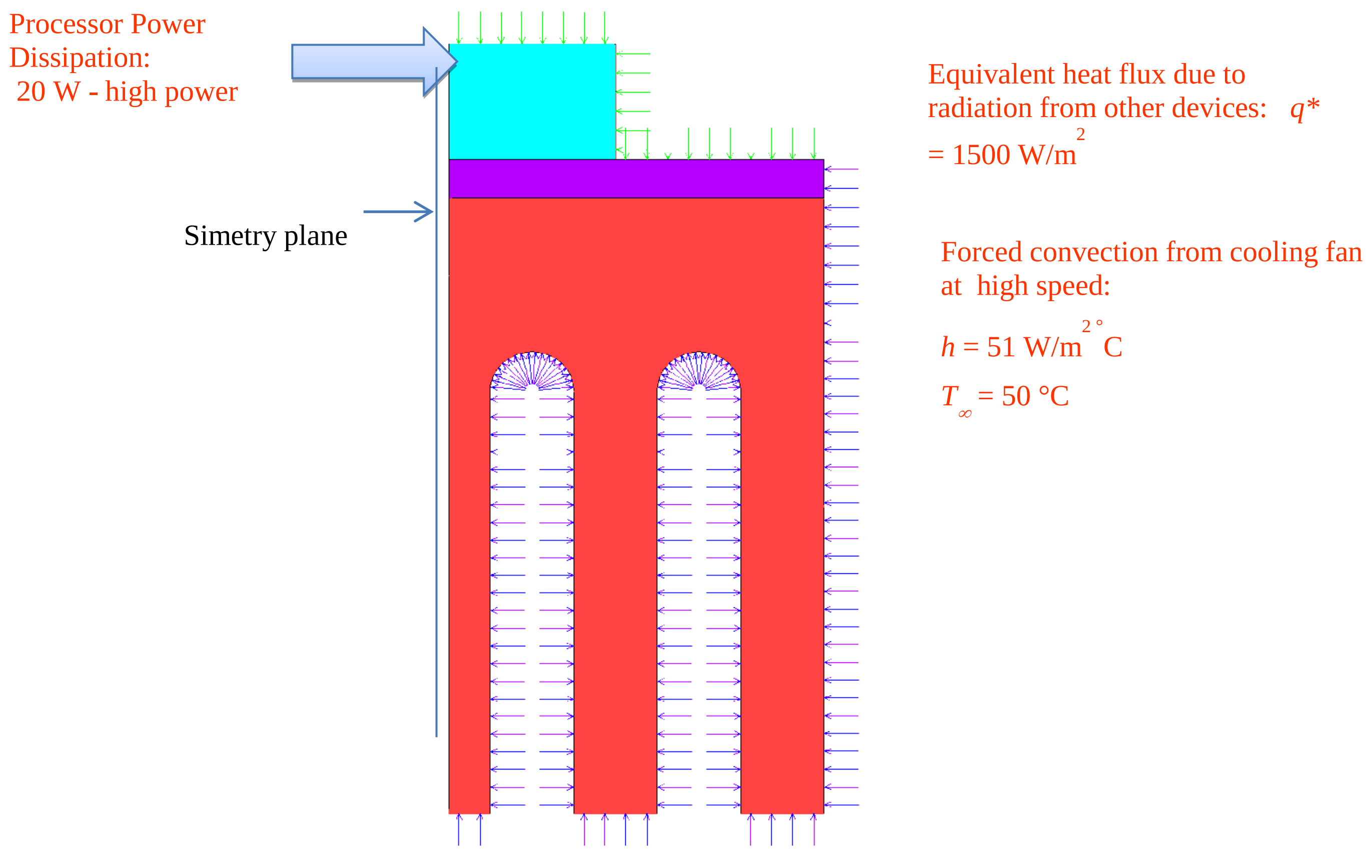


Figure 2 Dimensions



3. PREPROCESSING INSTRUCTIONS

a) Set Preferences for Thermal Filtering.(KEYW)
b) Specify the jobname(/FILNAM) and title (/TITLE).
c) Define two Element Types (ET):

- Type 1: **PLANE77**. Set keyoptions for consistent specific heat matrix, planar behavior (The PLANE77 is capable of modeling a quadratic temperature distribution. It is ideal for curve geometry like the fillets in the heat sink).
- Type 2: **SURF151**. Set keyoptions for planar behavior, include midside node, include extra node, extra node is fluid temp, heat average temp, exclude radiation.

d) Define linear material properties. (MP)
e) The specific heat and density should be input here as well.

Material properties	**Density (kg/m^3)**	**Specific heat(J/kg/C)**	**Thermal Cond. (W/m/C)**
Proccesor (1)	3500	500	50
Isolator(2)	8900	385	393
Heat sink(3)	2700	963	156

4. PREPROCESSING

Build the geometry.

a. Create six independent rectangles, then glue together.

Create>>Rectangle>> By Dimensions and enter coordinates of the corners of each rectangle. (RECT)

To glue areas, *Operate>> Glue>>Areas>>PickAll (AGLUE)*

Rectangle	X1	X2	Y1	Y2
Rectangle 1	0	0,01	0,0425	0,05
Rectangle 2	0	0,0225	0,04	0,0425
Rectangle 3	0	0,0225	0,03	0,04
Rectangle 4	0	0,0225	0	0,03
Rectangle 5	0,0075	0,0125	0	0,03
Rectangle 6	0,0175	0,0225	0	0,03

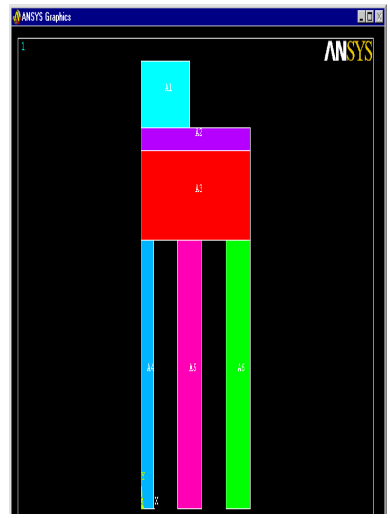


Figure 4 Geometry

b. Create fillets.

Create>>LineFillet, then pick the intersecting lines and enter the radius value (0.0025). (LFILLT)

*Create>>Areas>> Arbitrary>>ByLines*and pick the bounding lines to create each of the four areas. (AL)

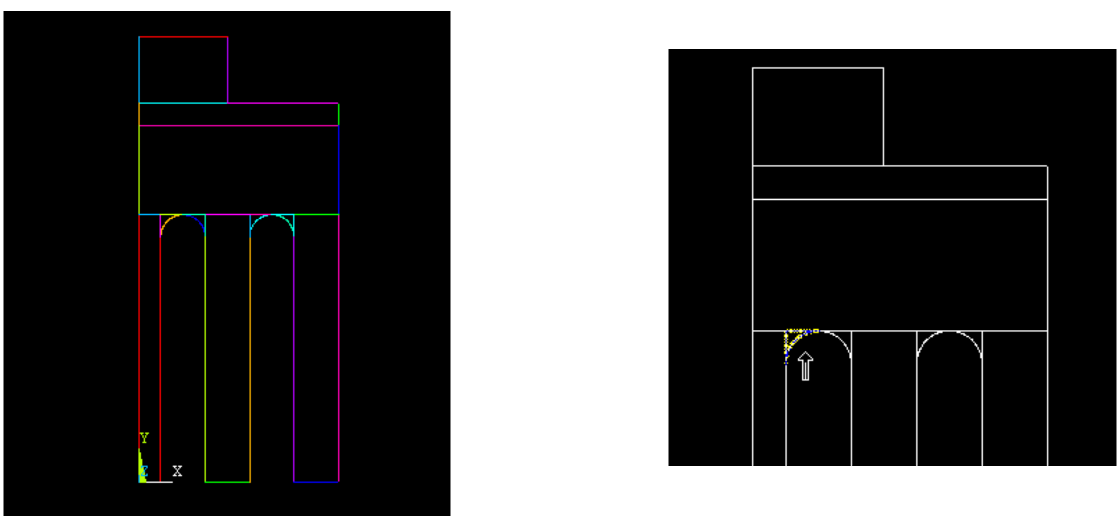
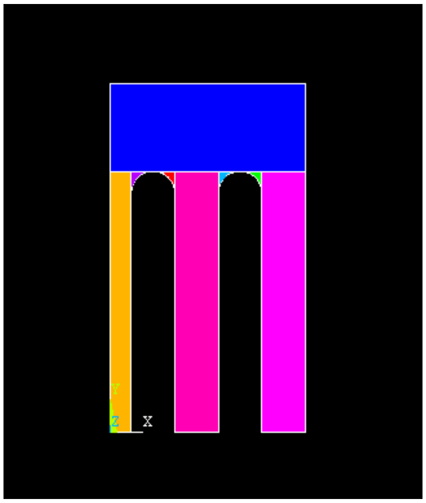
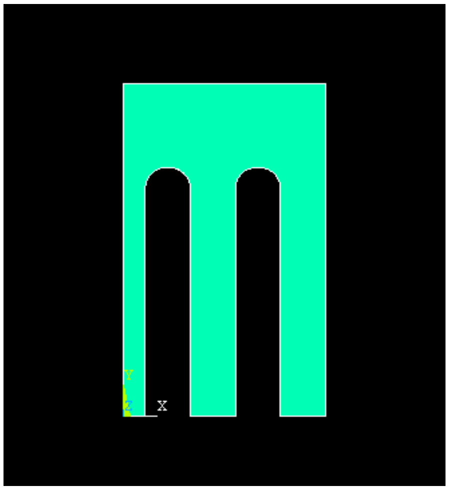


Figure 5 Fillets

c.Add all heat sink areas, including fillet areas, to create one large area (AADD).Adding areas together will produce more uniform element size in the fillet region and better element shapes during meshing.



1) Before



2) After

Figure 6 Adding areas

d)Assign attributes to each area.
MeshTool>>Element Attributes>>Areas>>Set (AATT)
Set SmartSize level = 3
MeshTool>>SmartSize(SMRT)
Mesh the areas.
*MeshTool*to specify Mesh:Areas, Shape: Quad, Mesher: Free (AMESH)

e)Though not strictly required, surface effect elements with the extra node option are used for easy calculation of convective heat loss. To create them: Select nodes on convection surfaces, and attached elements.
Select exterior lines. Then use re-select to select lines for convection loading (LSEL).
Select Nodes>> Attached to>> Lines, all (NSLL).
Select Elements>>Attached to>>Nodes (ESLN).

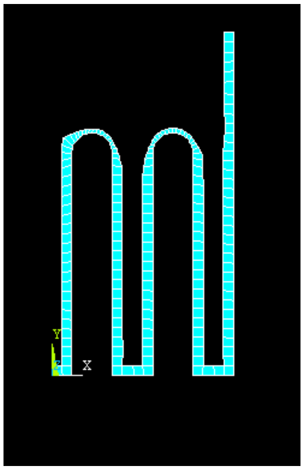


Figure 7 Element plot

f) Create the "extra" node.

*Create>>Node>>In Active Csys>>*and assign node number 5000(N). Location is arbitrary; we can try x=0, y= -0.01, z=0.

g) Define default meshing attributes to be SURF151.

Attributes>>Define>>Default Attributes. (TYPE,MAT,REAL).

h) Create surface effect elements.(ESURF)

Create>>Elements>>SurfEffect>>ExtraNode(ESURF)

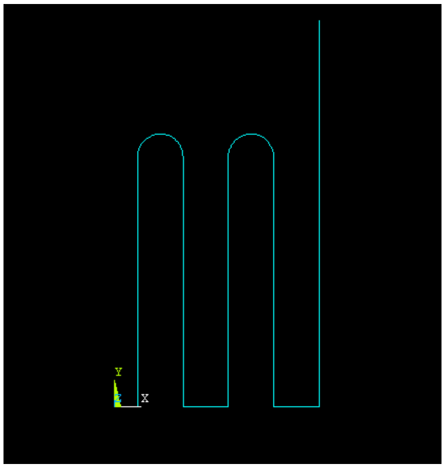


Figure 8. Surface Effect Element Plot

i) Select all model entities, and save the database as processor.db. (SAVE)

File>>SaveAs>>processor1.db to create another copy of the database so it can be resumed for other use.

5. IN THE SOLUTION PROCESSOR

a) Enter the Solution processor and indicate that what we want to perform a new steady-state analysis (ANTYPE).

b) Apply full power heat generation rate directly to processor area (BFA):

$$q=\frac{20W}{(0{,}02m\ x\ 0{,}0075m\ x\ 0{,}025m)}=5.333.333{,}34\mathrm{W/m^3}$$

c) Apply equivalent radiative heat flux to LINES shown (SFL).

d) Apply convection load directly to surface effect elements. Leave bulk fluid temperature field empty (SFE).

e) Fix extra node temperature at 50 °C (D).

f) Use symbols to visually verify all applied loads (/PBC, /PSF, /PBF).

g) Select everything and save the database again as processor2.db

h) Execute the solution (SOLVE). Verify completion and check error file for errors.

6. POSTPROCESSING

Enter the General Postprocessor and plot the temperature distribution in the system. (/POST1, PLNSOL)

Plot Results>>NodalSolution>>DOF solution>> Temperature.

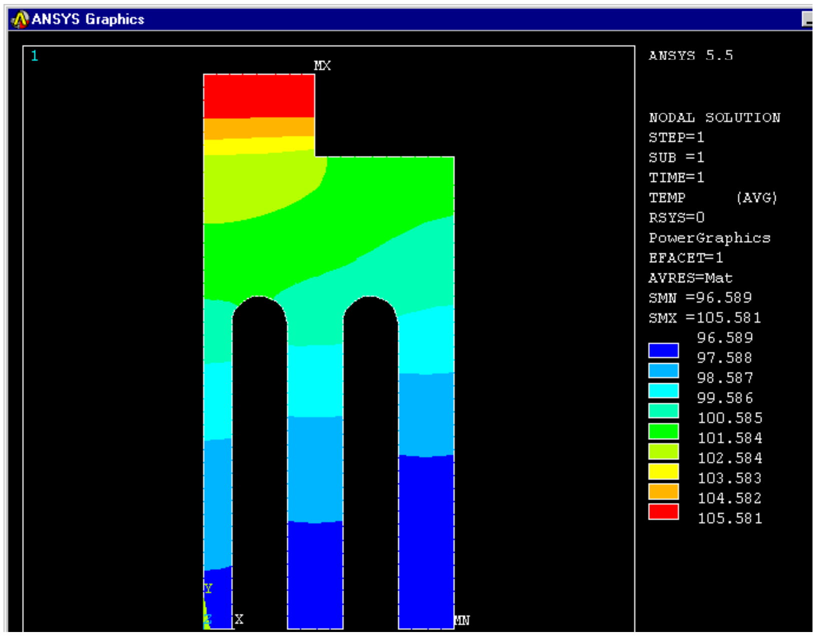


Figure 9 Plot results

Find the value and location of the maximum temperature in the system.

First, we have to sort nodes based on temperature to find node with the highest temperature. Then list results again. List Results>>SortNodes>>DOF Solution>>Temperature. (NSORT)

List Results>>Nodal Solution>>Temperature. (PRNSOL)

Unsort Results. (NUSORT)

Next, it can use plotting to locate the node and plot nodal temperature values on the screen:

(Turn off Power Graphics (/GRAPHICS,FULL) *Query Results>>Nodal Solution>>DOF solution*

Find the convective heat loss for the system. Since heat loss occurs only by convection, we can use the reaction heat flow rate.

(Select Everything (ALLSEL) *List Results>>Reaction Solution>>Heat Flow* (PRRSOL)

Show which regions of the model have a temperature at or above 100 °C.

Select Nodes>> By Results>> DOF Solution>> Temp and input a range of values. (NSEL)

(Select elements attached to nodes, then plot the temperature solution again. (PLNSOL))

Create a vector plot of thermal flux. (PLVECT)

Plot Results>>Vector Plot>>Predefined>>Thermal Flux (it can be used element centroid as the location for vector results, due to mixed materials in model).

Create a vector plot of thermal gradient. (PLVECT)

Plot Results>>Vector Plot>>Predefined>> Thermal Flux

Produce expanded results plot: temperature contours. (select all PLANE77 elements).

Plot Controls>> Style>> Symmetry Expansion>> Periodic/Cyclic (/EXPAND) and enter the plane of symmetry.Then, replot results desired. (PLNSOL)

7. CONCLUSIONS

It can perform hand calculations for use in results verification

$Q_{ss}= (-q x V_P + q^* x A_H)$ where V_p = processor volume

A_H= area exposed to the heat flux

To estimate average temperature it can be used the following equations. Use mass averaged material properties to calculate the approximate average steady-state temperature T_{ss} for the model:

$$\tau = \frac{\rho V c}{h A c}$$

$$b = \frac{q * AH + q}{\rho V c}$$

$$T_{ss} = T_{\infty} + \tau b$$

where : q^*=heat flux

A_H= area exposed to the heat flux

A_C=suprafaţa de convecţie

h= convection film coefficient

T_{∞}= bulk fluid temperature

q= heat generation

Therefore, ANSYS is a general purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers.

So ANSYS, which enables to simulate tests or working conditions, enables to test in virtual environment before manufacturing prototypes of products. Furthermore, determining and improving weak points, computing life and foreseeing probable problems are possible by 3D simulations in virtual environment.

BIBLIOGRAPHY

[1] ANSYS®, Multiphysics™, Release 11.0, Help System, Operations Guide, ANSYS Inc.
[2] Bathe,K.J., Finite Element Procedures, Prentice Hall, Englewood Cliffs, New Jersey, 1996
[3] Champion,E.R.Jr., Finite Element Analysis in Manufacturing Engineering, McGraw-Hill, New York, 1992
[4] Dill,E., The Finite Element Method for Mechanics of Solids with ANSYS Applications
[5] Lee, H.,H., Finite Element Simulations with ANSYS Workbench 14