ANNALS OF THE UNIVERSITY OF ORADEA
Fascicle of Management and Technological Engineering
ISSUE #2, AUGUST 2014, http://www.imtuoradea.ro/auo.fmte/

FINITE ELEMENTS ANALYSIS OF THERMAL STEADY STATE, USING ANSYS

Gavril GREBENIŞAN[1], Ioan Eugen RADU[2] Ioan ANTON[3]
[1]University of Oradea, grebe@uoradea.ro
[2]University of Oradea, iradu@uoradea.ro
[3]GMAB SA Oradea, ianton65@yahoo.com

Abstract— The Analysis by Finite Element Method (FEM), accomplished using ANSYS software, has the central aims to determine the behavior of a part's material, geometrically well-defined, of a mechanical, electrical, thermal assembly, subjected to any of external actions, such as physics, mechanical, thermal, electrical, and any other, well defined.

The material behavior, under conditions well defined, i.e. geometrically or physically, is characterized by thermal deformation results in each element of the mesh, in every node of the network's element, respectively, subjected to charges due to boundary conditions (constraints and blocking, respectively, thermal or physical agents, external or internal).

Keywords—finite elements method, steady-state, thermal laws, geometric model

I. INTRODUCTION

The laws of heat transfer allow engineers in many branches to design or to the operation of equipment, assembly or simply parts which are working in heat environment. The heat transfer is made by three modes, such as conduction, convection, and radiation, respectively. These modes are governed by specifically laws of heat transfer, [1]:

$$q_x = -k\frac{dT}{dx}; \text{ -heat conduction- the Fourier's law} \quad (1)$$

where q_x is the heat flux in the x direction (W/m^2), and k is the thermal conductivity (W/mK), which should be seen as a property of material, $\frac{dT}{dx}$ is the temperature gradient (K/m)

$$q = h(T_2 - T_1) \text{- heat convection- the Newton's law of cooling} \quad (2)$$

where q represents the convective heat flux, (W/m^2), $(T_2 - T_1)$ is the temperature difference between two surfaces, in (K)) (Kelvin degrees), and h is the convection heat transfer coefficient (W/m^{2}K), (named also the film coefficient)

$$\begin{aligned} a)\ & q = \sigma T^4 \\ b)\ & q = \epsilon\sigma T^4 \\ c)\ & Q = F_\epsilon F_G \sigma A_1\ T_1^4 - T_2^4 \end{aligned} \quad (3)$$

(3), a)- is Stefan-Boltzmann law –the flux that can be emitted by radiation from a black surface ;
(3), b)- the flux that can be emitted by radiation from a non-black surface;
(3), c)- the net radiant energy exchange by radiation, between surface 1 and 2;
where q is the radiative heat flux, (W/m^2); σ is the Stefan-Boltzmann constant (5.669x10^{-8}), in (W/m^{2}K^4), and **T** is the temperature, in (K) (Kelvin degrees); ϵ is the radiative property of the emitting surface, named *emissivity*; F_ϵ is a factor which characterize the nature of the two radiating surfaces; F_G is a factor that takes into account the geometric orientation of the two radiating surfaces; A_1 is the area of surface numbered 1.

The Helmholtz equation, state the general form of the system equation of 2D linear steady state field problems, [2]:

$$D_x\frac{\partial^2\emptyset}{\partial x^2} + D_y\frac{\partial^2\emptyset}{\partial y^2} - g\emptyset + Q = 0 \quad (4)$$

where $\emptyset$ is the field variable, and D_x, D_y, g, Q are given constants.

The weak form of steady-state field problems has the expression, [3]:

$$-\nabla^T k\nabla\emptyset + Q = 0 \quad (5)$$

where $\nabla = \begin{matrix} \frac{\delta}{\delta x} \\ \frac{\delta}{\delta y} \\ \frac{\delta}{\delta z} \end{matrix}$ is the gradient operator.

As a conclusion, any of these types of forms respects the general rule, [4]:

$$E_{in} - E_{out} + E_{generation} = E_{stored} \quad (6)$$

where E_{in}, E_{out} represent the amount of energy crossing the surfaces of the system, into and out, respectively; the thermal energy generation rate,

$\boldsymbol{E_{generation}}$, represents the rate of the conversion energy from any type: electrical, chemical, nuclear, electromagnetic forms to thermal energy within the entire volume of the system; the energy stored, $\boldsymbol{E_{stored}}$, is the increase or decrease, in the quantity of thermal internal energy in the closed volume of the system, due to the transient processes.

The aim of this paper is to find and determination of temperature distribution in a medium, as a function of time during a transient state, [5]. For convenient reasons it may use the conduction heat transfer law, namely the Fourier's law. The governing matrix equation is, [5], [6]:

$$\boldsymbol{K} \quad \boldsymbol{T} \; = \; \boldsymbol{f} \qquad (7)$$

where $\boldsymbol{K}$ is the global matrix (generalized matrix of rigidity, obtained by the assemblage of the individual element matrices $K_e = \frac{Ak}{x_j - x_i}\begin{matrix} 1 & -1 \\ -1 & 1 \end{matrix}$); $\boldsymbol{f}$ is the global load vector (which is obtained by the assemblage of the individual element load vectors, $f_e = \begin{matrix} Q_i \\ Q_j \end{matrix}$, for linear finite element, with nodes $\boldsymbol{i}$ and $\boldsymbol{j}$) and $\boldsymbol{T}$ is the global unknown vector (this vector should be determined, $\boldsymbol{T} = \begin{matrix} \boldsymbol{T_i} \\ \boldsymbol{T_j} \end{matrix}$). The transient heat conduction equation, for a stationary system, in ($\boldsymbol{x, y, z}$) coordinates, (depending on time $\boldsymbol{t}$) is (respecting the law, eq. (6): {inlet energy}+{energy generated}={energy stored}+{exit energy}), [7]:

$$\frac{\partial}{\partial x}\; k_x(T)\frac{\partial T}{\partial x} + \frac{\partial}{\partial x}\; k_y(T)\frac{\partial T}{\partial y} + \frac{\partial}{\partial x}\; k_z(T)\frac{\partial T}{\partial z} + G = \rho c_p \frac{\partial T}{\partial t} \qquad (8)$$

where $\boldsymbol{G}$ is the heat generated per unit volume; ρ is the density of material; $\boldsymbol{c_p}$ is the specific heat; $\boldsymbol{k_x}$, $\boldsymbol{k_y}$, $\boldsymbol{k_z}$ are the thermal conductivity, on directions $\boldsymbol{x}$, $\boldsymbol{y}$, $\boldsymbol{z}$ respectively (the equation (8) caught the situation of anisotropic material, and the transient system). Over the space of study, the temperature is discretized as follows, [1], [8], [9]:

$$T \;\; x,y,z,t \; = \; {}_{i=1}^{n} N_i(x,y,z)T_i(t) \qquad (9)$$

where $\boldsymbol{N_i}$ is the shape functions; $\boldsymbol{n}$ is the number of nodes in an element, and $\boldsymbol{T_i(t)}$ is the time-dependent nodal temperature. The solving process of equation (5) by Galerkin method implies an iterative solution, since $\boldsymbol{k_x}$, $\boldsymbol{k_y}$, $\boldsymbol{k_z}$ are temperature-dependent, [1]:

$$_{\Omega}\; N_i \;\; \frac{\partial}{\partial x}\; k_x(T)\frac{\partial T}{\partial x} + \frac{\partial}{\partial x}\; k_y(T)\frac{\partial T}{\partial y} + \frac{\partial}{\partial x}\; k_z(T)\frac{\partial T}{\partial z} + G - \rho c_p \frac{\partial T}{\partial t} \;\; d\Omega = 0 \qquad (10)$$

and the more convenient, [1]:

$$C_{ij} \;\; \frac{\partial T_j}{\partial t} \; + \; K_{ij} \;\; T_j \; = \; f_i \qquad (11)$$

where

$$C_{ij} \; = \; _{\Omega}\; \rho c_p N_i N_j \, d\Omega \qquad (12)$$

$$K_{ij} \; = \; _{\Omega} \;\; k_x(T)\frac{\partial N_i}{\partial x}\frac{\partial N_j}{\partial x} \; T_j \; + k_y(T)\frac{\partial N_i}{\partial y}\frac{\partial N_j}{\partial y} \; T_j \; + k_z(T)\frac{\partial N_i}{\partial z}\frac{\partial N_j}{\partial z} \; T_j \; + \; d\Omega + \; _{\Gamma} \; hN_iN_j \, d\Gamma \qquad (13)$$

$$f_i \; = \; _{\Omega} \; N_i G \, d\Omega - \; _{\Gamma_q} qN_i \, d\Gamma_q + \; _{\Gamma_q} qN_i \, hT_a d\Gamma_q \qquad (14)$$

II. PROCEDURE AND ANALYSIS

Geometric pattern represented by reference numeral "Bac_mobil" was "loaded" with the heat generated by thermal injection material in a fluid state at operating temperature T1 = 300 ° C (Celsius degrees), and subjected to thermal tempering actions due heat flow generated by the cooling fluid T2 = 50° C. These two agents are acting on part and have the results in a thermal-mechanical deformation of the workpiece material "Bac_mobil", materialized in significant dimensional changes at the structural level (at elements or nodes of the mesh level, respectively). These dimensional changes can lead if they exceed the required limits by the technological and dimensional reasons, through the design of injection mold, dimensional changes of thermal and mechanical parts affected, which in turn can cause dimensional changes of machined parts (injected, in this case) or malfunction of the mold (active parts affected by thermal-mechanical deformation, which are not able to maintain the designed preserves dimensions, i.e. not close dimensional chains in respect of tolerance limits fields, and the precision dimensional and functional respectively). Therefore, the finite element analysis is a convenient tool, but power consumption (consistent computational effort, which requires a properly equipped in terms of computer systems, highly documented operator and well anchored in numeric analysis, adequate technical documentation, consistent with software used, also in agreement with execution drawings) which provides a complete map, up to a structural level, sometimes microscopic level (for structures discretized by mesh refining at the unit cell size of the material used).

Geometric modeling of the studied part, in respect of load conditions, is only a stage in the process of FEM analysis.

To conclude the modeling and framing of part, it should provide, for studied piece, a material in-law behavior so-called "material models".

Material models are the laws of the behavior of a material or groups of materials, structurally related, sometimes even physical-chemically defined conditions of use.

The geometric model analysis, includes consequently, necessarily, also the material analysis, within the geometric dimensions (defined as limits of the analysis domain - "domain edges"- but do not be confused, always within the analytical model, that does not overlap with limitations data loading conditions of the loads-supports external or internal). The domain analysis boundaries are captured, in FEM analysis, by the "boundary conditions", which is imposed and restricted by the behavior, starting with canceled "displacements", (Degrees Of Freedom, i.e. DOF) or imposed certain limits and ending with thermal or chemical loads (TABLE II), whose energy may include influences of different types, which can affects the behavior of a material during use.

TABLE I
MODEL GEOMETRY

Thermal Strain Effects	Yes				
Bounding Box					
Length X	0,196 m	5,e-003 m			
Length Y	5,6e-002 m	2,519e-003 m			
Length Z	4,e-002 m	1,76e-002 m			
Properties					
Volume	3,1958e-004 m³	1,493e-007 m³			
Mass	2,5087 kg	1,172e-003 kg			
Centroid X	9,8064e-002 m	0,123 m	0,163 m	7,3e-002 m	3,3e-002 m
Centroid Y	3,0898e-002 m	4,951e-002 m			
Centroid Z	1,9809e-002 m	2,65e-002 m			
Moment of Inertia Ip1	8,3166e-004 kg·m²	3,2429e-008 kg·m²			
Moment of Inertia Ip2	8,0084e-003 kg·m²	3,3776e-008 kg·m²	3,3777e-008 kg·m²	3,3775e-008 kg·m²	
Moment of Inertia Ip3	8,1963e-003 kg·m²	2,2096e-009 kg·m²	2,2104e-009 kg·m²	2,209e-009 kg·m²	2,2089e-009 kg·m²
Statistics					
Nodes	70067	325	327	325	269
Elements	41924	36	37	36	28
Mesh Metric	None				

Referring to geometric model analysis (TABLE I) , of part named "Bac_mobil", the physical characteristics was provided by the drawings, in a ".iges" (Initial Graphics Exchange Specification (IGES)) format, which is a general format, interchangeable, universally compatible formats supported by ANSYS, [10].

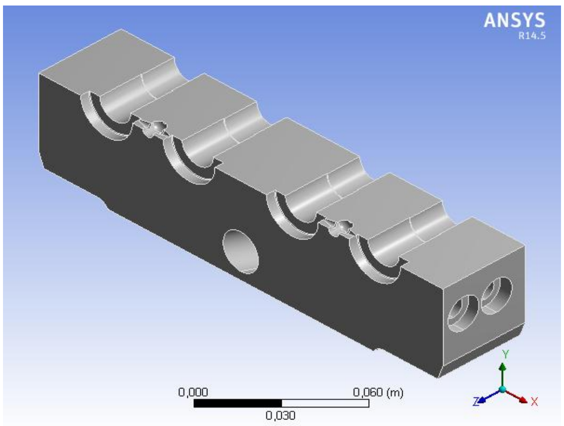

Fig. 1.- Geometric model

TABLE II
ANALYSIS SETTINGS-THERMAL LOADS

Object Name	*Temperature*	*Temperature 2*	*Temperature 3*
State	Ful		
S			
Scoping Method	Geometry Selection		
Geometry	16 Faces	4 Faces	20 Edges
D			
Type	Te		
Magnitude	300, °C	50, °C (ramped)	
Suppressed	N		

Statistics related to the geometric model data, discretized into finite elements is summarized in TABLE III of Mechanical Report. Thus, the model is meshed in a convenient tetrahedral mapped network, with 42,061 tetrahedral finite elements and 71,313 nodes. Analysis settings module, of ANSYS contains work environment, operating temperatures, etc. TABLE II summarize the thermal loads in the work environment: from 22 Celsius degrees up to 300 Celsius degrees, and from 22 Celsius degrees at 50 Celsius degrees, respectively, also from Mechanical Report it can be seen, that the process takes place in two different steps, [10].

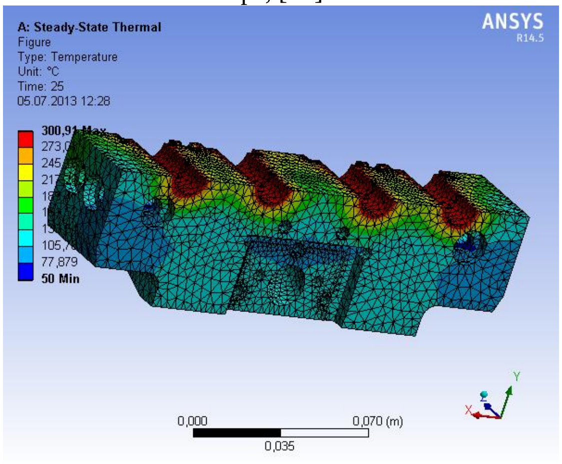

Fig. 2.- The Model Steady-State Thermal Solution-Temperature Distribution

More tables, from Mechanical Report (provided by ANSYS), shows the energy flow, generated per unit area, and consumed during the injection process, for the two sources of thermal energy (T1 and T2), as one stated on first Section of this paper, eq. (4), [1], [4], [10].

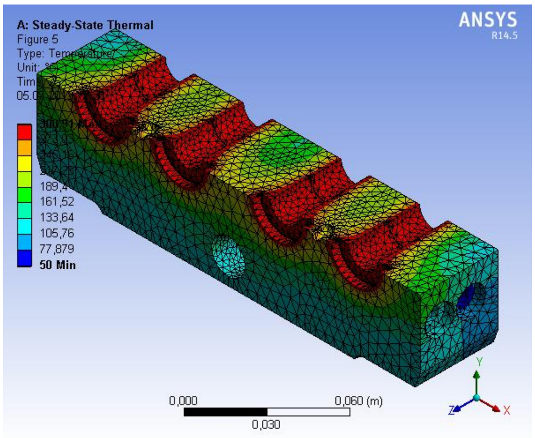

Fig. 3.-Temperature distribution on circular shapes

Figs. 2.-6. shows the thermic distribution into the part "Bac_mobil" loaded to 300 Celsius degrees, in the injection area, and cooled to 50 Celsius degrees, trough the cylindrical channel. The temperature map distribution in the heat affected zone is equivalent to the depth characteristic finite element dimension (2.5 mm), while at a distance of four finite elements (about 10 mm), the temperature is approximated at 77 Celsius degrees, which demonstrates the thermal dissipation energy, according to law behavior of the workpiece material "Bac_mobil".

Fig. 5. shows the spatial distribution of the directional heat flow, into the direction to the x- axis. There is a quasi-uniform distribution of energy flow, the cooling process being correctly designed and distributed into the part body mass named "Bac_mobil".

Fig. 6. shows how properly designed is the part studied.

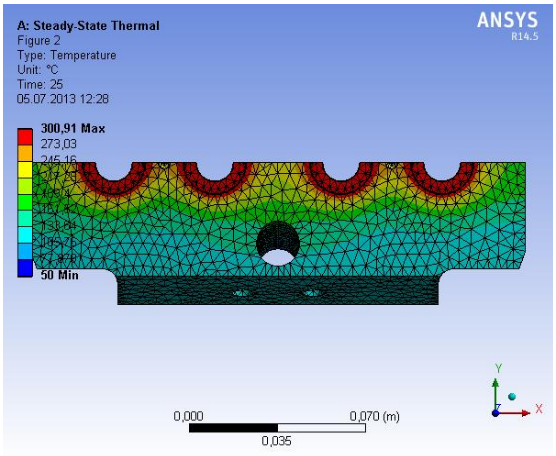

Fig. 4.- Thermal distribution on basis' part

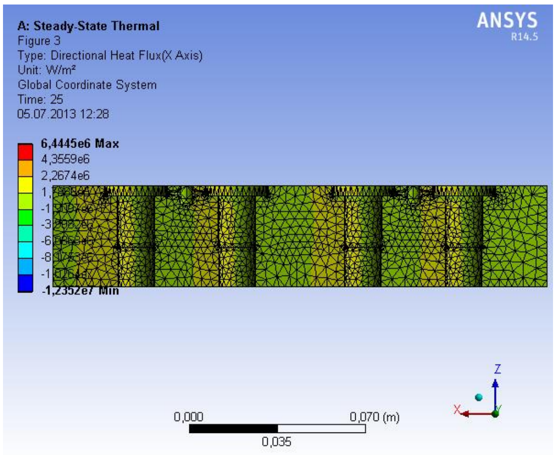

Fig. 5.-Directional Heat Flux-X axis

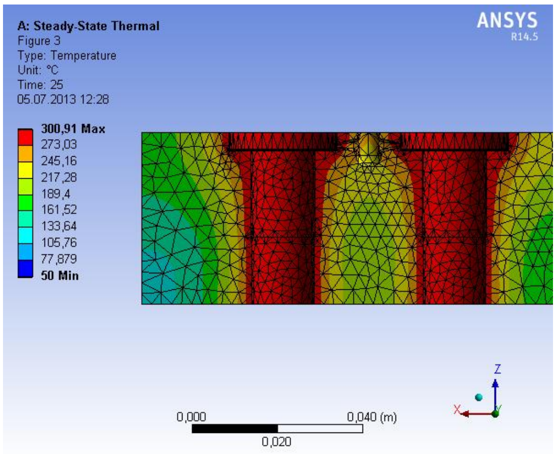

Fig. 6.-Temperature distribution-a detail

The differences (represented by light blue color, the flow is about 100 times smaller than the maximum thermic influenced zone- represented by sharp red color-, which is nonexistent in fact, at the end of the injection process, thus the heat flow is evenly distributed and dissipated), so that, these differences are insignificant in relation to the behavior of the material.

TABLE III
ANALYSIS SETTINGS- MESH

Object Name	*Mesh*
State	Solved
	Defaults
Physics Preference	Mechanical
Relevance	0
	Sizing
Use Advanced Size	Off
Relevance Center	Coarse
Element Size	2,5e-003 m
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	2,8e-004 m
	Inflation
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
	Advanced
Shape Checking	Standard
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For	Yes
Rigid Body Behavior	Dimensionally
Mesh Morphing	Disabled
	Defeaturing
Pinch Tolerance	Please Define
Generate Pinch on	No
Automatic Mesh Based	On
Defeaturing Tolerance	Default
	Statistics
Nodes	71313
Elements	42061
Mesh Metric	None

Knowing the governing deformation law is a linear thermal deformation in relation to the temperature difference, it could be hypothesized that the material withstand thermal loads, without undergoing dimensional changes, the tempering interval being 25 seconds, which is properly chosen in connected considerations of the deformation material due to thermal loads.

The most unfavorable situation is seen on the x- axis in the positive direction thereof, showing the temperature distribution in body part mass. Thus and so, the cooling heat access, through the cylindrical channel, parallel to the direction of the x- axis, generates heat flow direction, the same direction as expected in areas colored green to yellow, to have dimensional changes due to thermal deformation. In terms of technical, engineering point of view, this is the most deprived thermal zone, in dimensional characteristics, that should be considered when one should check the dimensional stability, i.e. the dimensional changes and deviations. This will require verification of deviation from cylindricity or deviation from circularity, accompanied by dimensional verification of the " mating" of the injected part.

III. Conclusion

There were generated two research reports in order to view in two different CAD frameworks, analysis results. The first image of each report shows how graphic presentation below. Thus, if the first image is in "wireframe", the results will be presented as a visible "mesh". This means you can visually analyze the results of finite element analysis, represented and "made visibly". It is a way of analyzing "quantitative-dimensional-active".

If the first image in the research report is "shaded", all results are presented as an "artistic" type, without a somewhat dimensional representation. It is not known how large the influence, analyzing of images, barely the tables and graphs only can deduct these influences. This procedure is less expensive, the report generation is faster as possible can be, the effects are those shown above.

References

[1] Roland N. Lewis, P. Nithiarasu, K. N. Seetharamu-*Fundamentals of the Finite Element Method for Heat and Fluid Flow*, John Wiley & Sons Ltd., Chichester, 2004.
[2] G. R. Liu, S. S. Quek, *The Finite Element Method: A Practical Course*, Butterworth- Heinemann, New York, 2003.
[3] O. C. Zienkiewicz, Taylor R. L., *The Finite Element Method-The Basis, vol. I*, Butterworth- Heinemann, Oxford, 2000.
[4] S. Moaveni- *Finite Element Analysis*, Prentice Hall, New Jersey, 1999.
[5] W. B. J. Zimmerman- *Process Modelling and Simulation with Finite Element Methods*, World Scientific Publishing Co. Pte. Ltd., 2004.
[6] M. A. Bhatti- *Fundamental Finite Element Analysis and Applications, With Mathematica® and MATLAB® Computations*, John Wiley & Sons Ltd., New Jersey, 2005.
[7] N. Nakasone, T. A. Stolarski and S. Yoshimoto-*Engineering Analysis with ANSYS Software*, Elsevier Butterworth-Heinemann, Oxford, 2006.
[8] S. S. Rao- *The Finite Element Method in Engineering, Fifth Edition*, Elsevier Butterworth-Heinemann, Amsterdam, 2011.
[9] A. J. Baker- *Finite elements: computational engineering sciences* John Wiley & Sons Ltd., Chichester, 2012.
[10] G. Grebenişan- *Finite Element Analysis Of A Superplastic Forming Process, Using Ansys Software*, Annals Of The Oradea University, Fascicle Of Management And Technological Engineering, Oradea, 2010.