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]:

\[ q_x = -k \frac{dT}{dx} \]  

(1)

where \( q_x \) is the heat flux in the x direction (W/m²), 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)

\[ q = h(T_2 - T_1) \]  

(2)

where \( q \) represents the convective heat flux, (W/m²), \( (T_2 - T_1) \) is the temperature difference between two surfaces, (K) (Kelvin degrees), and \( h \) is the convection heat transfer coefficient (W/m²K), (named also the film coefficient)

\[ a) \quad q = \sigma T^4 \]

\[ b) \quad q = \epsilon \sigma T^4 \]

\[ c) \quad Q = F_c F_e S A_1 T_1^4 - T_2^4 \]  

(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/m²); \( \sigma \) is the Stefan-Boltzmann constant \((5.669 \times 10^{-8})\), in \((W/m^2K^4)\), and \( T \) is the temperature, in \((K)\) (Kelvin degrees); \( \epsilon \) is the radiative property of the emitting surface, named emissivity; \( F_c \) is a factor which characterize the nature of the two radiating surfaces; \( F_e \) 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]:

\[ D_x \frac{\partial^2 \phi}{\partial x^2} + D_y \frac{\partial^2 \phi}{\partial y^2} - g \phi + Q = 0 \]  

(4)

where \( \phi \) 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]:

\[ -\nabla^T k \nabla \phi + Q = 0 \]  

(5)

where \( \nabla = \left[ \begin{array}{c} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{array} \right] \) is the gradient operator.

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

\[ E_{in} - E_{out} + E_{generation} = E_{stored} \]  

(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,
\( E_{\text{generation}} \), 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, \( E_{\text{stored}} \), 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]:

\[
K \ T = f
\]  
(7)

where \( K \) is the global matrix (generalized matrix of rigidity, obtained by the assemblage of the individual element matrices \( K_e = \frac{\alpha E A}{h^3} \)); \( f \) is the global load vector (which is obtained by the assemblage of the individual element load vectors, \( f_e = Q_i \)), for linear finite element, with nodes \( i \) and \( j \) and \( T \) is the global unknown vector (this vector should be determined, \( T = T_0 \)). The transient heat conduction equation, for a stationary system, in \((x, y, z)\) coordinates, (depending on time \( t \)) is (respecting the law, eq. (6): [inlet energy]+[energy generated]=[energy stored]+[exit energy]), [7]:

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

where \( G \) is the heat generated per unit volume; \( \rho \) is the density of material; \( c_p \) is the specific heat; \( k_x, k_y, k_z \) are the thermal conductivity, on directions \( x, y, z \) 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]:

\[
T(x, y, z, t) = \sum_{i=1}^{n} N_i(x, y, z)T_i(t)
\]  
(9)

where \( N_i \) is the shape functions; \( n \) is the number of nodes in an element, and \( T_i(t) \) is the time-dependent nodal temperature. The solving process of equation (5) by Galerkin method implies an iterative solution, since the thermal conductivity, on directions \( x, y, z \) respectively. The transient heat conduction equation is found by process of equation (5) by Galerkin method implies an iterative solution, since the temperature is discretized as follows, [1], [8], [9]:

\[
C_{ij} \ \frac{\partial T_j}{\partial t} + K_{ij} \ T_j = f_i
\]  
(11)

where

\[
C_{ij} = \rho c_p N_i N_j \ d\Omega
\]  
(12)

\[
K_{ij} = \alpha k_x(T) \frac{\partial N_i}{\partial x} \frac{\partial T_j}{\partial x} + k_y(T) \frac{\partial N_i}{\partial y} \frac{\partial T_j}{\partial y} + k_z(T) \frac{\partial N_i}{\partial z} \frac{\partial T_j}{\partial z} + \frac{\partial T_i}{\partial t} + h n N_j \ d\Gamma
\]  
(13)

\[
f_i = \alpha N_i G \ d\Omega - r_q q N_i \ d\Gamma_q + r_q q N_i \ h T_q \ d\Gamma_q
\]  
(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 \( T_1 = 300 \ ^\circ \text{C} \) (Celsius degrees), and subjected to thermal tempering actions due heat flow generated by the cooling fluid \( T_2 = 50^\circ \text{C} \). 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>
<thead>
<tr>
<th>Thermal Strain Effects</th>
<th>Yes</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Bounding Box</strong></td>
<td></td>
</tr>
<tr>
<td>Length X</td>
<td>0.196 m</td>
</tr>
<tr>
<td>Length Y</td>
<td>5.6e-002 m</td>
</tr>
<tr>
<td>Length Z</td>
<td>4.e-002 m</td>
</tr>
<tr>
<td><strong>Properties</strong></td>
<td></td>
</tr>
<tr>
<td>Volume</td>
<td>3,1958e-004 m³</td>
</tr>
<tr>
<td>Mass</td>
<td>2.5087 kg</td>
</tr>
<tr>
<td>Centroid X</td>
<td>9.8064e-002 m</td>
</tr>
<tr>
<td>Centroid Y</td>
<td>3.0898e-002 m</td>
</tr>
<tr>
<td>Centroid Z</td>
<td>1.9809e-002 m</td>
</tr>
<tr>
<td>Moment of Inertia I1</td>
<td>8.3166e-004 kg-m²</td>
</tr>
<tr>
<td>Moment of Inertia I2</td>
<td>8.0084e-003 kg-m²</td>
</tr>
<tr>
<td>Moment of Inertia I3</td>
<td>8.1963e-003 kg-m²</td>
</tr>
</tbody>
</table>

| 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].

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, through 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.-Directional Heat Flux-X axis

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

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.
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 circularity 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 visible". 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


