A Connected Steady-State Thermal with a Structural Analysis using FEA in ANSYS

G. Grebenişan[1], D. C. Negrău[1], N. Salem[2]

[1]University of Oradea, Industrial Engineering Department
[2]Zarqa University, Jordan, Mechanical Engineering Department

ggrebenisan@gmail.com

Abstract. A research topic was launched to solve a current engineering problem: the design of a 3D printing device/equipment for the implementation of additive technology. This device shall be adapted on a numerically controlled machine, CNC TMA-AL- 550, for dual operation under a milling machine or the printing equipment control. During the development of the research topic solution, several errors related to the functionality of the equipment and the control of the advance of the deposition material occurred. The decision adopted was, without a doubt, the most efficient and handy, namely, the application of the finite element analysis method (FEA), in two phases: 1) -analysis of heat transfer; 2) - analysis of the state of stresses and strains, for which the exact discretization in finite element mesh was used. The FEA method was developed using ANSYS® 2021 R1 software. This procedure included solving the mathematical model developed for the FEA method by connecting two analysis systems: the ANSYS® Steady-State Thermal system and the ANSYS®-Static Structural system. Thus we managed to achieve the transfer, very quickly, of the nodal temperatures from the heat transfer analysis system (ANSYS® Steady-State Thermal system) to the analysis system of the stress state and deformations of the solid (ANSYS®-Static Structural system - "Thermal stresses").

1. The first section in your paper

Many of the problems encountered in engineering practice are thermal. Therefore, the knowledge and skills that an engineer must have are of a disarming diversity [1]. When tackling a thermal problem, the engineer must know and understand the flow phenomena, heat or mass transfer phenomena, the influences and connections that thermal phenomena create, and their effects: expansion, contraction, etc.. Devices such as appliances, advanced electronics, motors and heating, ventilation and air conditioning systems must be evaluated for thermal performance during the design process. This idea will discuss thermal analysis using FEA for 3D printing equipment attached to a numerically controlled machine [2]. Under the given conditions, we must operate controlled, stationary, or transient if the parameters of the technological process are so defined. The objective of thermal analysis is to understand the response and behaviour of a thermally charged structure. The temperature distribution resulting from the investigation, the heat flow distribution and the structural response in different thermal loading conditions are essential knowledge in ensuring the successful design of thermal engineering products. They are taken into account and kept under observation to issue claims in control. The thermal problems correspond to the stationary and transient states when the temperature depends on time. Thermal stresses are included and discussed in the same analysis to find the structural response due to temperature changes [3].

Thermal analysis can be performed To study the heat transfer in the mass of a mechanical part or between the components of a mechanical assembly, which shall be determined thermal (physical) quantities such as temperature, thermal gradient and heat flow distributed. There are two types of thermal analysis: steady-state thermal analysis and transient thermal analysis [4], [5], [6]. Equilibrium thermal analysis aims to find the temperature or distribution of heat flux in structures when thermal stability is reached, and transient thermal analysis seeks to determine the time history of how the temperature profile and other thermal quantities change in time. In addition, thermal expansion or contraction of engineering materials often leads to thermal stress in structures, which can be examined by performing thermal stress analysis. The basic equations for thermal and thermal stress analysis are presented as follows.

2. Model, modelling and analysis methodology

To solve the problem, the heat flow distribution and the thermal stresses dispersion, and the construction of the generated deformation state, in any of its locations, the energy conservation law (Fourier law), described by the differential equation, is used [1], [2], [7], [8]:

$$\rho c\frac{\partial T}{\partial t}-\left(\frac{\partial}{\partial x}\left(k_x\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_y\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k_z\frac{\partial T}{\partial z}\right)\right)-\dot{q}=0 \qquad \textbf{(1)}$$

where **ρ** is the density of the studied solid material, **c** is the specific heat, **k** is the coefficient of thermal conductivity, $\dot{q}$ is the internal heat generation rate, per unit volume, per unit time, and **T** is the variable temperature for the coordinates **x**, **y**, **z** and time **t**. For stationary heat transfer, the previous equation becomes:

$$\left(\frac{\partial}{\partial x}\left(k_x\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k_y\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k_z\frac{\partial T}{\partial z}\right)\right)+\dot{q}=0 \qquad \textbf{(2)}$$

Suppose the heat transfer takes place in two dimensions, for example, in the x-y plane, with the coefficient of thermal conductivity, k, constant. In that case, the differential equation is reduced to one with partial derivatives, called the Poisson equation:

$$\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}=-\frac{Q}{k} \qquad \textbf{(3)}$$

If, in addition, no internal heat is generated, the differential equation is reduced to the partial derivative equation, called the Laplace equation:

$$\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}=0 \qquad \textbf{(4)}$$

The solution of Laplace's equation is T (x, y). Although Laplace's equation is straightforward, its exact solution is complicated to determine, especially in the case of more complex geometries.

As can be seen, the Fourier law relates the components of the conduction heat flux to the temperature gradient. By definition, the element of the conduction heat flux in the x-direction, for an isotropic material, [9], [10], [11] is:

$$q_x=-k\frac{\partial T}{\partial x} \qquad \textbf{(5)}$$

3. Boundary conditions

The boundary conditions required to complete the input data (in addition to the 3D geometric model and material properties) for heat transfer to the surface area are:

- *-Specified temperature*

$$T = T_s$$

where $\boldsymbol{T_s}$ may vary in the three directions: **x**, **y**, **z** and with time **t**;

- *-Specified heat flux*

$$q = -q_s$$

where $\boldsymbol{q_s}$ is the specified heat flux and must be in equilibrium with the conduction heat flux **q** at the surface area;

- *-Convection heat transfer*

$$q = h(T_s - \boldsymbol{T}_\infty)$$

where h is the surface convection coefficient (surface) and $\boldsymbol{T}_\infty$ is the ambient temperature;

- *-Radiation heat transfer*

$$q = \varepsilon\sigma(T_s^4 - T_\infty^4)$$

where **ε** is the surface emissivity, and **σ** is the Stefan-Boltzman constant.

Unlike the stationary regime analysis system, in the case of the transitive one, an initial condition is imposed:

$$T(x, y, z, 0) = T_0(x, y, z)$$

where $\boldsymbol{T_0}$ is the initial temperature of the solid.

Finite element equations for heat transfer problems can be obtained by applying the weighted residue method to the differential equation that governs the heat transfer problem. According to [1], one can write the matrix form of the finite element equation:

$$[C]\{\dot{T}\} + \left[[K_c] + [K_h] + [K_r]\right]\{T\} = \{Q_c\} + \{Q_Q\} + \{Q_q\} + \{Q_h\} + \{Q_r\}$$

where:

$[C]$ is the capacity matrix
$[K_c]$ is the conduction matrix
$[K_h]$ is the convection matrix
$[K_r]$ is the radiation matrix
$[\dot{T}]$ is the vector that contains the rates of change of nodal temperatures
$[T]$ is the vector containing the nodal temperatures
$\{Q_c\}$ is the vector of the charge conduction
$\{Q_Q\}$is the vector of the heat generated, of the charge
$\{Q_q\}$ is the vector of the specified heat, of the charge
$\{Q_h\}$ is the vector of convection, of charge
$\{Q_r\}$ is the vector of radiation, of charge

Greater attention must be paid to defining these matrices/vectors of the elements because they depend on the type of finite elements.

The unknowns of the stationary heat transfer problem are the nodal temperatures.

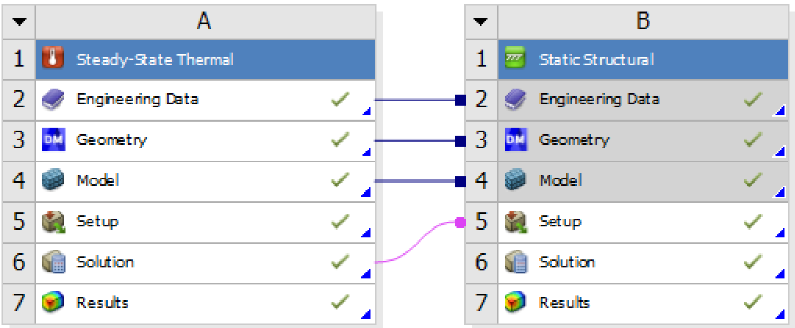
Finite element equations for solid problems can be obtained by applying the weighted residue method to the differential equation that governs this problem [1], [2], [5]. Matrix form of the finite element equation:

$$[K]\{\delta\} = \{F\} + \{F_0\} \quad \textbf{(6)}$$

where [K] is the stiffness matrix of the elements; {δ} is the vector containing the components of the unknowns u, v, w in nodes; {F} is the vector containing the nodal forces, and $\{F_0\}$ is the vector containing the nodal forces introduced due to temperature changes.

4. Thermal and structural mapping analysis and results

The analysis results by the finite element method in ANSYS, using the two analysis systems connected (figure 1), confirmed, when displaying the solution, the veracity of the conclusion, according to which the printing equipment does not correctly control the heat transfer problem. The thermal field, with high values of heat and nodal temperatures in the extruder area, generated excessive melting of the filament and, consequently, blocked its advance in the deposition area.



a) Thermal analysis b) Stress and strain analysis

Figure 1. The two ANSYS systems connected

Figure 2. shows the 3D geometric model analyzed simultaneously in two ANSYS systems, connected to the "Solution" cell, from the "Thermal Analysis" system. The boundary conditions (figure 3), in the first system, are the temperature, concentrated in the area of the extruder nozzle, and in the second system are the nodal temperatures (the solution of the primary procedure), taken over by data transfer, by through the connection at the level of the cell "Solution", of the system "Thermal analysis", and the fixed support, materialized by a solid component of the printing equipment, the circular profile from the composition of the equipment.

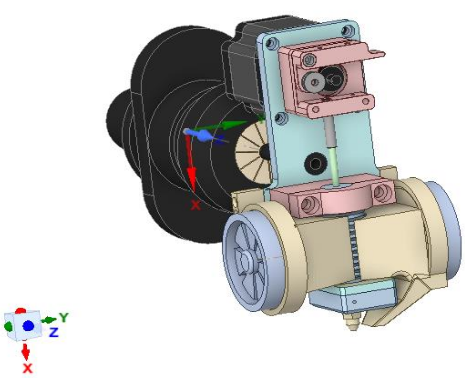


Figure 2. Geometric model of analysis

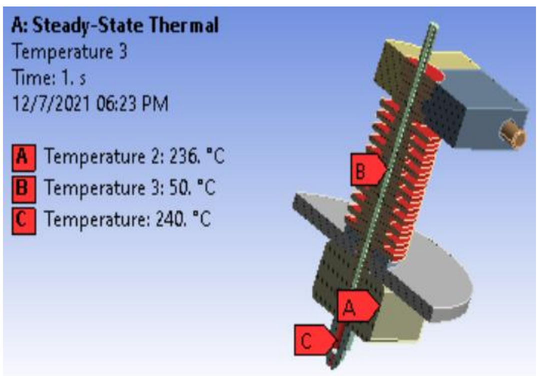


Figure 3. Boundary conditions, in section

The first analysis system, Steady-State System-Thermal Analysis, offered beneficial results in our endeavor to design correctly and find the best constructive variant, respectively, to make the printing equipment more efficient functionally. Thus, in figures 4 and 5, we caught the distribution, visibly outlined, of the temperature, virtually marked, on the whole, geometric model, of the part of the printing equipment, with some "samples" indicating the surface temperature values. Similarly, I noticed, in the axial section of the device, also visibly outlined the temperature distribution in its mass. The graph of the temperature distribution in the device's mass is made for approximately 30,000 nodes in the thermally involved area.

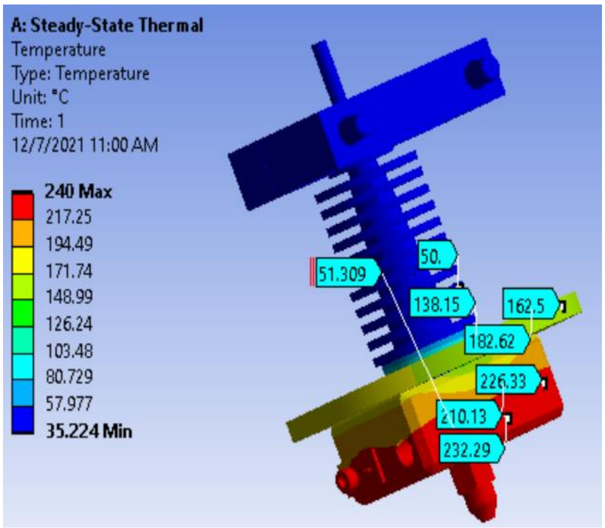


Figure 4. Temperature distribution - samples

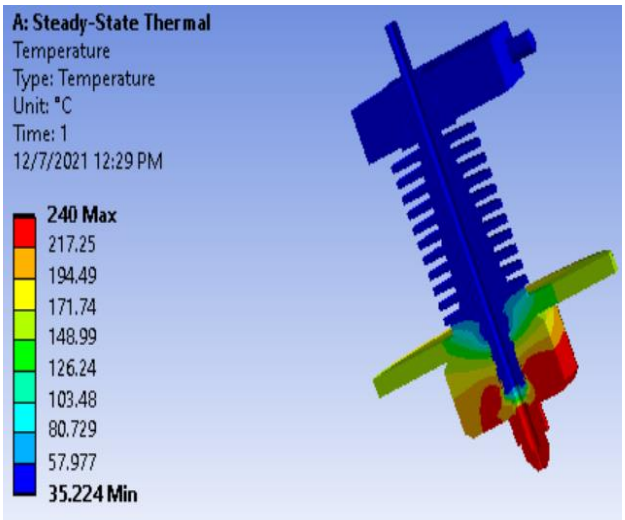


Figure 5. Temperature distribution – in section

The analysis of the heat flow and the temperature distribution generated by the airflow, flow coordinated by the two fans, is presented in figure 6 19. It is easy to see the symmetry of the temperature distribution, which demonstrates the type of Newtonian (non-turbulent) hot fluid flow through the 3D printing equipment.

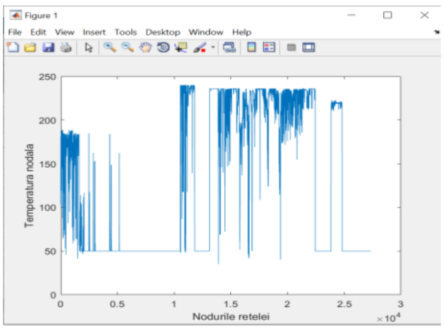


Figure 6. Graph of temperature evolution

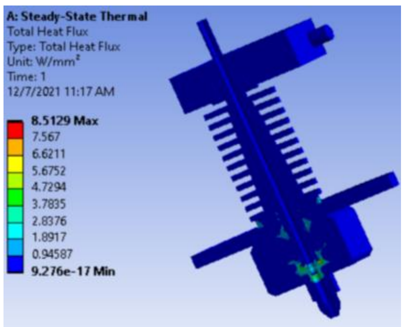


Figure 7. Heat flow distribution-in section

In the second part of the thermal analysis, the study of the thermal stresses generated by the nodal temperatures was followed by how these induced stresses affect dimensionally and functionally the dosing and advanced device of the equipment. To avoid any errors, the thermal analysis system was connected at cell level with a structural, mechanical analysis system, figure 8. This procedure made it possible for the nodal temperatures, the solution of the thermal analysis system, to be imported into the limit condition setting subsystem (the temperatures found by solving the first system became the loads, the loads, in the second analysis system), [12].

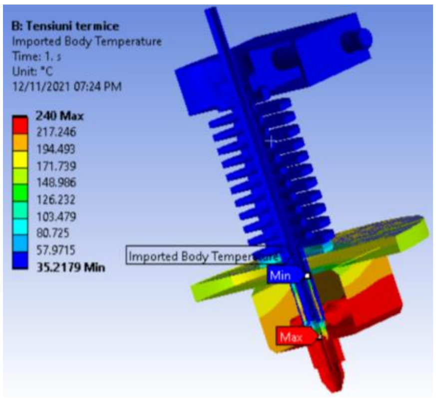


Figure 8. Boundary conditions imported from the first analysis system - nodal temperatures

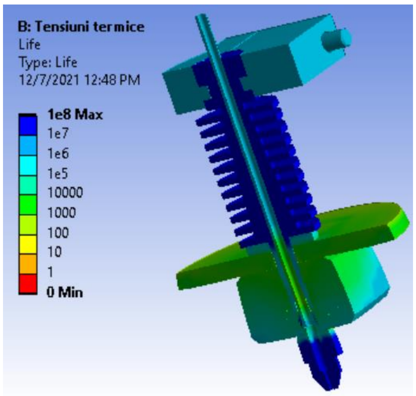


Figure 9. Equipment life under system operating conditions

The solution of the second analysis system was directed towards the determination of the lifetime of the device (figure 9.) towards the analytical determination of the total deformation of the areas affected by the thermal stresses (figure 10.), respectively towards the calculation and evaluation of the von Mises equivalent stress (figure 11).

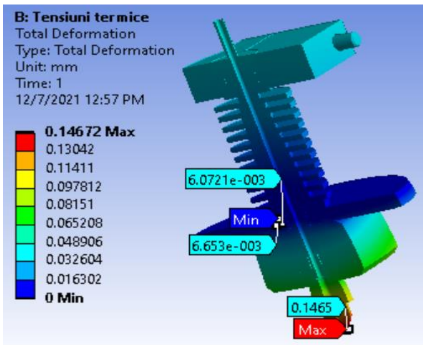


Figure 10. Total deformation due to thermal stresses

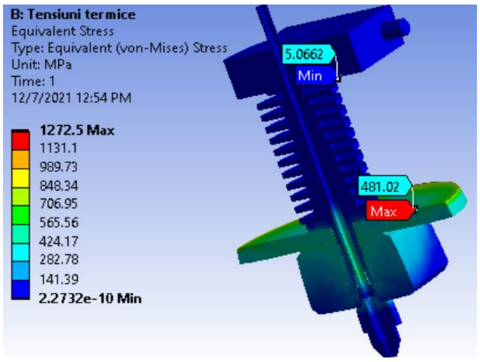


Figure 11. Equivalent von Mises voltage due to thermal stresses

These last two actions can allow continuous monitoring of the influence of induced thermal stresses, respectively, can create a comprehensive framework of the decision management system, in full knowledge of the facts.

5. Conclusion

It is evident and straightforward to note that this decision management worked within the limits of the first thermal analysis system of the equipment. When the system crashed, the filament did not advance in the advance mechanism. For reasons strictly related to controlled efficiency and functionality, we decided on finite element analysis, the behaviour of the advanced device of the printing equipment. The distribution of heat, with the concentration of high-temperature values, in a small area of the analysis range, distribution highlighted by the solution of the thermal analysis system, led to the correct decision of intervention at the level of materials and endowment with cooling elements. These decisions were confirmed by the first finite element analysis results, namely the cooling fluid flow, using the ANSYS software.

References

[1] Rao Singiresu S- *The finite element method in engineering*, Butterworth-Heinemann- an imprint of Elsevier, ISBN: 978-0-12-811768-2, 2018.

[2] Pramote Dechaumphai, S. Sucharitpwatskul, - *Finite Element Analysis with ANSYS Workbench*, Alpha Science International Ltd., Oxford, U.K, 2018.

[3] https://www.mathworks.com, *Solving a Heat Transfer Problem With Temperature-Dependent Properties*

[4] Xiaolin Chen, Yijun Liu- *Finite Element Modeling and Simulation with ANSYS Workbench*, 2nd Edition, 2019, CRC PRESS, Taylor & Francis

[5] Nam H. Kim, Bhavani V Sankar, Ashok V Kumar,- *Introduction to Finite Element Analysis and Design*, John Wiley & Sons Ltd, 2018

[6] D. C. Negrău, G. Grebenişan, and C. Gherghea, "*An optimization approach by Finite Element Analysis, using Design of Experiments and Response Surfaces- a survey*," in *IOP Conference Series: Materials Science and Engineering*, 2019, vol. 568, no. 1, doi: 10.1088/1757-899X/568/1/012065.

[7] D. C. Negrau, G. Grebenisan, T. Vesselenyi, D. M. Anton, and C. I. Indre, "*Modeling and building a 3D print head*," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 1169, no. 1, p. 012028, 2021, doi: 10.1088/1757-899x/1169/1/012028.

[8] Z. Chen, "Finite Element Methods and Their Applications," 2005, [Online]. Available: ISBN-10 3-540-24078-0 Springer Berlin Heidelberg New York.

[9] A. V. Manzhirov and S. A. Lychev, "Mathematical modeling of additive manufacturing technologies," *Lect. Notes Eng. Comput. Sci.*, vol. 2, no. July 2014, pp. 1404–1409, 2014.

[10] G. Ravichandran- Finite Element Analysis of Weld Thermal Cycles Using ANSYS, CRC Press, 2021.

[11] Nitin S Gokhale, Sanjay S Deshpande, Sanjeev V Bedekar, Anand N Thite- *Practical Finite Element Analysis*, HyperWorks Technology Conference, 2008.

[12] Lee Huei-Huang- *Finite Element Simulations with ANSYS Workbench*, SDC Publications, 2021.