The theoretical bases and study parameters with FEA application approach on a bending tool design

D C Negrău[1,2], G Grebenişan[1] ,T Vesseleny[1]

[1] University of Oradea, Faculty of Managerial and Technological Engineering, Universității Street No. 1, Oradea, Romania

dan.negrau@yahoo.com

Abstract. This paper presents the theoretical basis of the approach to Finite Element Analysis (FEA) techniques and their application in the case of a bending request for a piece of aluminum sheet. The plate of a bending device deforms elasto-plastically under the action of the drive forces of the device, but also under the action of the reaction forces in the supports, more for a material such as aluminum and a few times less for a structural steel. If the forces acting on the device deform the carrier plate, which is in the operator's attention, in the elastic field, then there is no danger of permanent deformations occurring. Conversely, if deformation exceeds the threshold of elastic deformation, the material undergoes plastic deformation stresses, then the piece deforms plastic, with the risk of permanent deformations occurring. These deformations, in the first place, can generate production errors, or device failure, by modifying the execution rates and the games in the studied ensemble. This is precisely the purpose of this work: to prove, with the results of the FEA, that in the case of a certain material, with the same configuration of working conditions, the system can function without producing errors, obtaining qualitatively conformable parts.

1.Introduction

In the automotive industry, the future direction requires the continuous development of new concepts and the optimization of all business processes. Also, the growing globalization produces a growing distance from traditional business processes, both at the design, manufacturing, sales and logistics level, as well as convergence between information and communication technologies and process optimization:

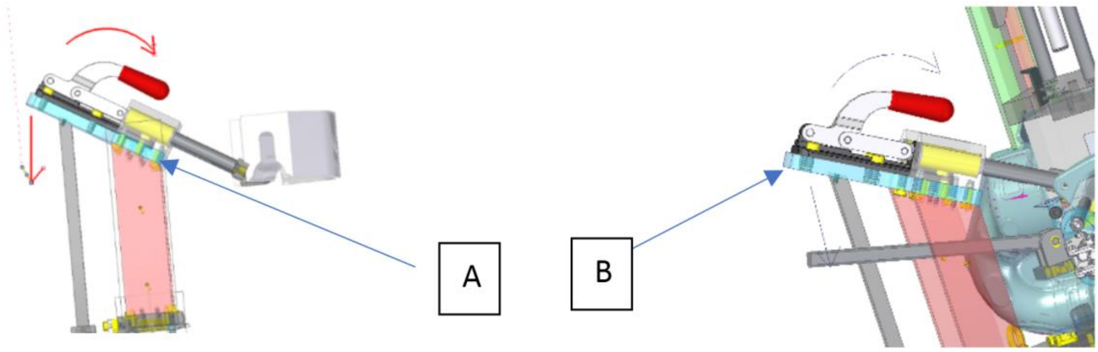
- Market and customer requirements on each product are continually changing, which increases the complexity of production processes and costs together;
- Heterogeneous IT fleets and high maintenance costs for old systems generate unnecessary and growing IT costs;
- Strong and fast sales fluctuations require increasing flexibility in production;
- Planning, deploying, controlling and improving major production processes and resources requires the methodical implementation of a "digital factory", including IT-based equipment for a well-founded management decision;
- Comfort and safety are increasingly traded by automotive customers, and investment in electronics, and therefore in software solutions and applications, is needed.

The car exhaust manufacturers to request the construction of a manual assembly of the car exhaust protector parts. The device contain more much subassemblies, one of these subassemblies being designed to support a clamp . The clamp is attached to a support plate and at the acting o this clamp,I has been observed that when the device is actuated to fixing the subassemblies to the exhaust, the support plate yield elastically [1,2]. The subassembly is presented in figure 1, elastically yields due to their support plate.

2. Support plate

2.1 General presentation

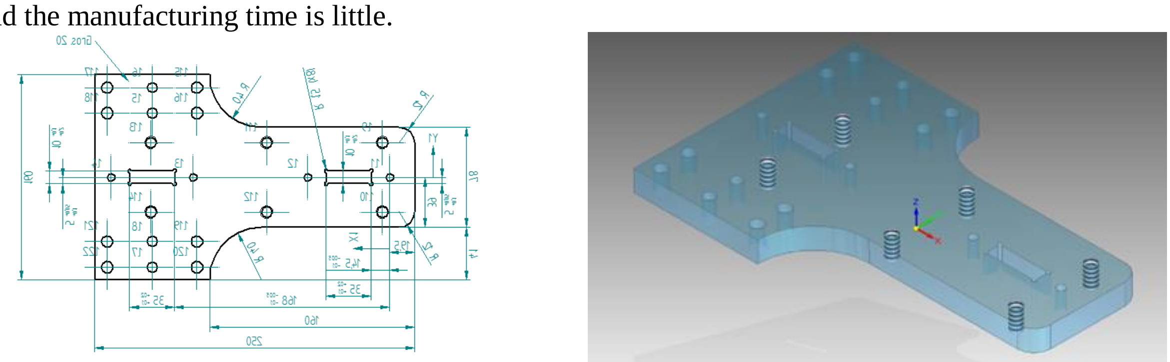
The support plate of a clamp is a component part of a subassembly (figure 1). When the clamp is acting, the support plate elastically (figure 1 A and B) deforms at the end of the work stroke by approximately 0.33 [mm], measured, as a result of the action of the force applied by the punch on the active plate, as shown in Figure 1 [4]. The elastic deflection of this component of the fixing assembly, generates incorrect assembly of the exhaust protect sheet.



a) Subassembly acting clamp b) Assembly device

Figure 1. Clamp port-drive assembly

The support plate, as highlighted in figure 1, was built on the customer's technical prescriptions so as to have the dimensions present in figure 2 a). The material used is duraluminium, a material that meets the technological requirements, namely: it is easy to manufacturing, low specific weight, insignificant cutting tool wear compared to the machining of a harder material (soft steel, hard steel, stainless steel) and the manufacturing time is little.



a) 2D model support plate b) 3D support plate

Figure 2 Geometrical properties

For the drawings has present in figure 2, was used the Cad software Solid Edge ST7, which offer facilities 3D drawing, and after on the base the 3D model results very easy 2D model [6]. Withal the 3D model is used at the simulation FEA.

2.2 Limit condition and mesh support plate – Ansys [5]

Computer Aided Engineering procedures allows the following types of analysis: Structural Analysis; Thermal analysis; Analysis of kinematics and dynamics of fluids; electromagnetism; mechanisms; Plastic deformation; Injection molding, etc .; analyzes that can be acquired helping of the following software: ANSYS; ABAQUS (simulated); ALGOR; COSMOS; I-DEAS NX; LS-DYNA; MSC.MARC; MSC.NASTRAN; …[3]. The analysis was solved by using the default finite elements (tetrahedral and hexahedral elements type). The percentage of needed bending requires the load type, namely the acting force, loading scheme, and the force are perpendicular on the surface.

Few examples of engineering problems where one use the Computer Aided Engineering (CAE):

When it is possible to obtain an accurate solution, a set of equations provides details of the behavior of the modeling system under the given conditions:

$$EI\frac{d^2\cdot Y}{dX^2}=\frac{PX(L-X)}{2} \tag{1}$$

Boundary conditions:
X=0, Y=0
X=L, Y=0
Beam deformation in the Y direction, as a function of X

$$Y=\frac{P}{24EI}(-X^4+2LX^3-L^3X) \tag{2}$$

W(F)-Force [N/m^2]
E- Young Modulus $E=\frac{\sigma}{\varepsilon}$
I-Modulus of Inertia I=M·L^2 [kg·m^2]
L,a,b- characteristic dimensions
R_1, R_2 –Force reaction at supports

2.3 General Finite Element Analysis formulation, [6]

Generally, at any point in a mechanical structure, the stresses and strains are defined by the axes of a global coordinate system as six components:

- for stresses

$$\sigma_x\,,\sigma_y\,,\sigma_z\,,\tau_{xy}\,,\tau_{xz}\,,\tau_{yz}\,, \tag{3}$$

- for strains

$$\varepsilon_x\,,\varepsilon_y\,,\varepsilon_z\,,\gamma_{xy}\,,\gamma_{xz}\,,\gamma_{yz}\,, \tag{3}$$

The displacements field in a 3D element is defined using the N_i shape functions given by the relationships:

$$u=\sum_{i=1}^{N}N_iu_i\,;v=\sum_{i=1}^{N}N_iv_i\,;\;w=\sum_{i=1}^{N}N_iw_i\,; \tag{4}$$

where u_i, v_i, w_i are node values of the displacement on the element, and N is the number of nodes of an element. Using these data, we can determine the stifness matrix with the formula:

$$k=\int_V B^TEBdV \tag{5}$$

in which B is the nodal displacement matrix, on the d direction d (ε = Bd). In coordinates (x, y, z) the displacements matrix is given as:

$$\varepsilon=\begin{Bmatrix}\varepsilon_x\\ \varepsilon_x\\ \varepsilon_x\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{zx}\end{Bmatrix}=\begin{Bmatrix}\frac{\partial u}{\partial x}\\ \frac{\partial v}{\partial y}\\ \frac{\partial w}{\partial z}\\ \frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\\ \frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\\ \frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\end{Bmatrix}=\text{Bd} \tag{6}$$

The deformation energy is determined with the relationship:

$$U = \frac{1}{2}\int_{V} \sigma^{T} \varepsilon dV = \frac{1}{2}\int_{V} (E\varepsilon)^{T} \varepsilon dV = \int_{V} \varepsilon^{T} E\varepsilon dV = \frac{1}{2} d^{T}\left[\int_{V} B^{T} EBdV\right] d \qquad (7)$$

In order to analyze a part using FEA in Ansys, it can be projected directly into ANSYS using drawing tools or as in the present case, the 3D model of the part to be analyzed will be imported from a system external design [7].

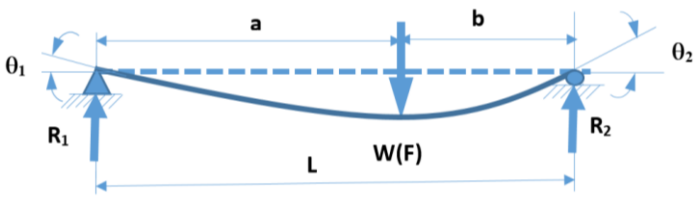


Figure 3. The generic scheme of solving the bending problem, [3]

The first step for FEA analysis, for the support plate, will be to define the boundary conditions. The boundary conditions represent the fixing elements (fixed support) and the force acting on the analyzed part, and at the same time their position [8].

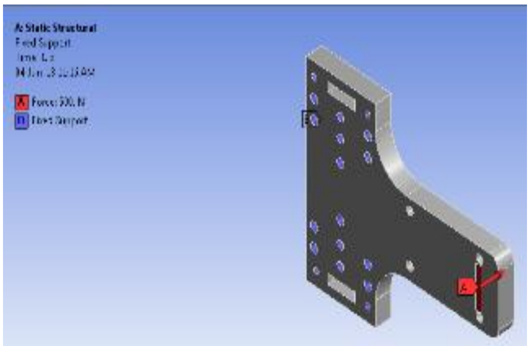


Figure 4 Boundary condition

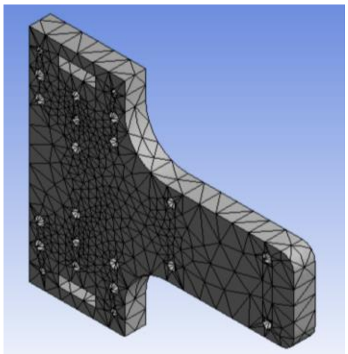


Figure 5 Mesh plate

In figure 4 the boundary condition, marking with A(Fixed Support) and B(Force=500 [N])is presented and in figure 5 the meshing of support plate is shown.

- Number of nodes:9994
- Number of finite elements:5403

After define the boundary condition, will be define the input parameters,variables of optimization from the set of constructive parameters of the support plate, parameters that will generate the output parameters of the best optimal plate, so that on the basis of the final results of the analysis we can manufacture a suitable support plate geometric, mechanical and technological.

2.3 Graphic representation results

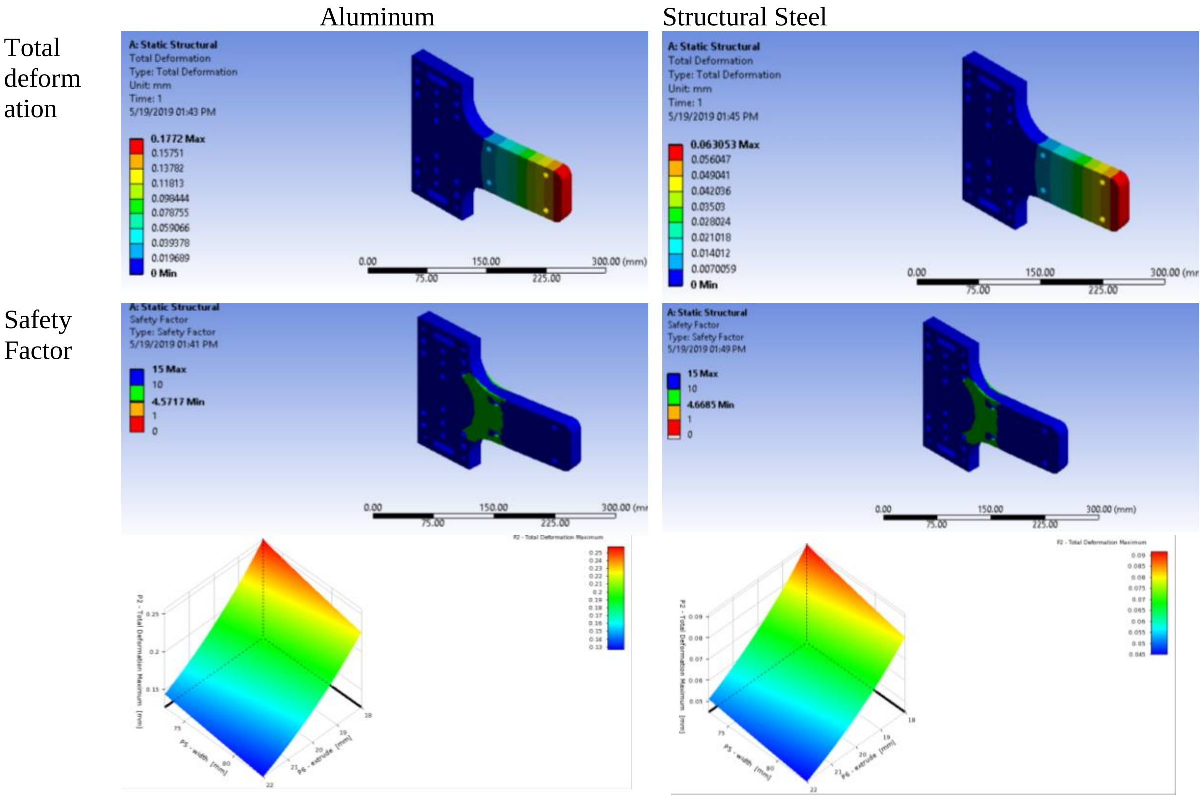
Following the definition of boundary conditions, the software provides the 3D image of the support plate (Figure 5) bending request, which is submitted to the subassembly of which it is part.

The Total deformation value is equal to 0.17 mm, value that influences sub-assembly operation, and the Safety Factor 4.5, for aluminum, and 0.06 mm Total deformation, respectively 4.6 the Safety Factor, for Structural Steel, see figure 6.

3. Conclusion

In conclusion, today's graphical tools (3D design software and graphics simulation) can solve constructive and functional trouble of some components with some assemblies or assemblies without the need to build layouts that later or try tests and attempts to determine which is the best version.

To solve these issues, as in the present case, it is necessary to perform some steps as well as define the conditions at the limit, and once defined as input parameters, the software will provide details on the number of nodes and finite elements and at the same time of the deformation value of the support plate.



a) b)

Figure 6 Total deformation & Safety Factor
a)-for Aluminium; b)-for Structural Steel

References

[1] Exhaustion https://www.autoexpert.ro/inlocuirea-tobelor-de-esapament/(accessed at 2019.03.04)
[2] Jianping L, Hongwei Z, Xiaoli H , Lin Z, 2016 Pengliang H , Cong L , Ning L , Yuexi Za , and Changyi L 2016, Ins. and Exp. Tech. Volume 59, Issue 5 P762–767
[3] Grebenişan G., Bogdan S. 2017 Parameterized Finite Element Analysis of a Superplastic Forming Process, Using ANSYS, MATEC Web of Conferences, **126**, 2017, https://doi.org/10.1051/matecconf/201712603001
[4] Negrău D C, *Finite Element Analysis and Solution Optimization for a bending device*, 2018, University of Oradea.
[5] Huei-H L, 2017 Finite Element Simulations with ANSYS Workbench 17- Theory, Applications, Case Studies, SDC Publications
[6] Xiaolin C, Yijun L 2018 Finite Element Modeling and Simulation with ANSYS Workbench, Second Edition CRC Press DOI: 10.1201/9781351045872
[7] ANSYS 19, Workbench User's Guide http://www.ansys.com (accessed on April 2019)
[8] Negrau D. C., Grebenisan G., Indre C. 2019 Experimental approach and finite element analysis of the behavior of a steel bending machine, IManE&E 2019

Acknowledgments

This research was developed under financing of S.C. Isrom Impex S.R.L Oradea, The Agreement # 17250/2017. Thanks to advisors and all support of the company Manager Eng. Binselean Mihai