Experimental approach and finite element analysis of the behavior of a steel bending tool

D C Negrău\textsuperscript{1}, C I Indre\textsuperscript{1}, G Grebenişan\textsuperscript{1}

\textsuperscript{1} University of Oradea, Faculty of Managerial and Technological Engineering, Universităţii Street No. 1, Oradea, Romania

E-mail: dan.negrau@yahoo.com

Abstract. This work contains the 3D design and finite element analysis of a manual device for assembly by bending the sheets around an exhaust muffler. As a result of the design of the device, the requested was executed by the information provided by the mechanical designer; only when the device of bending was put into operation, some faults appeared, namely that its parts during the operation has been elastically deformed. For the construction of the elastically yielding parts, the finite element method was used to study the behavior of the tool. Following the FEA analysis, it was concluded that the dimensions and material from which those pieces had been built had to be changed. By making the new parts based on FEA calculations, they no longer elastically yield (elastic deformation less than 0.1%), so the device is accepted by the customer.

1. Introduction

The standard exhaust pipes are used to reduce the noise produced by exhaust gases in the engine to acceptable limits to our ears by diverting the flow of gas through the baffles and special absorbing materials resistant to high temperatures. The catalytic drums are designed to reduce the emissions emitted into the atmosphere [1]. On the one hand, the exhaust allows control of the vehicle noise level and, on the other hand, the limitation of pollutant emissions according to current legislation. It is known that an exhaust drum is made of several sheets that are welded one over the other and inside them are heat sensors and one or more catalysts [2].

One of the car exhaust manufacturers has proposed that the exhausted drum for newer generation cars need external protection elements to protect the drum against dust, mud and other impurities that are encountered at the time of rolling of the car as well as the role of mitigating the sound emitted by the drum when operating the car, a noise that is due both to the assembly of the drum on the car body (vibrations) but also to those noise from the engine [3]. The exhaust mufflers are made of thin sheets, which are assembled by bending around the exhausted drum [4].

2. Bending device

2.1 Overview

By those mentioned in the last part of section 1, a car manufacturer has requested the construction of a device for the manual assembly of the car exhaust protection parts, a device that is presented in this paper and the problems encountered in its manufacture.
These types of devices can be automated (pneumatically, hydraulically or manually operated by robots) or manual (man-operated).

The assembly device is based on the bending process of the parts (figure 1).

![Figure 1. Bending process](image)

Bending is intended to convert flatware into curved pieces. Bending operations are performed using bending molds. The most representative bending operations are profiling and straightening [5]. The mathematical model for the acting force for deformation is represented by equation 1:

\[
F = \frac{bg^2}{L} \cdot \gamma_r
\]

where \( g \) is the material thickness (sheet); \( L \) is the width between the supports [m]; \( b \) is the thickness of the sheet [mm] and \( \gamma_r \) is the tensile breaking strength [daN/cm²].

2.2. 3D device modeling

At this stage, the prototype of the bending device was designed in the manually operated version. To design this device, one used Solid Edge software, 3D modeling CAD software. It runs on Microsoft Windows and provides solid-state modeling, assembling of the assemblies, and drawing capabilities for mechanical engineers [6].

Starting from the 3D CAD model of the drum containing the protection plates, the parts of the device were modeled to perform the corresponding bending operation.

This tool was designed to bend the edge of the escape shield, which is shown in figure 2 a), b) and figure 2 c) shows the area of assembly sheet.

Constructively, the tool in figure 3 has a lot of constructive elements, elements that are divided into two main categories: a) Standardized Elements, b) Parts under design condition

![Figure 2. Car exhaust and protection table](image)
a) Standardized Elements
Nowadays, a significant advantage in CAD design modeling is that standardized items which can be downloaded directly from manufacturers' sites, items that are encoded and then can be ordered according to that code. The not only advantage that one can order, the most significant advantage is that you can try lots of variants of standardized elements, elements that appear virtually within the assembly you want to build, not need the CAD designer to mold parts up when one finds the optimal option.

b) Parts under design condition
Manufacturing is the process by which a raw material is given a specific shape by modifying its dimensions. As one can see, there may be used very sophisticated manufacturing methods and procedures, namely: classic milling, CNC milling or even the deposition of materials with printers and 3D robots [7].

The pieces of this device were made of several types of material (Aluminum, Steel, Polyamide), some steel parts being heat treated. The most complicated parts were the support and positioning of the exhaust during the bending process, which was made of aluminum, shown in figure 4.

The question is to study the behavior of the material and to use the results for the project data replacement if possible without modifying the constructive data of the device

3. Finite element analysis

3.1 Overview
CAE-allow 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; …[8]. 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):

- In general, engineering problems which are described by known mathematical models of situations and physical phenomena.
- Differential equations derive from the application of fundamental laws and principles of nature, a system or a controlled volume. These fundamental equations are the balance of the masses; the balance of forces or energies.
- When it is possible to obtain an accurate solution, these equations provide details of the behavior of the modeling system under the given conditions:

\[
\frac{d^2Y}{dx^2} = \frac{PX(L-X)}{2EI}
\]

Boundary conditions:
X=0, Y=0
X=L, Y=0

**Figure 5.** The generic scheme of solving the bending problem [10]

Beam deformation in the Y direction, as a function of X is given by:

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

where \(W(F)\) is the force [N/m²]; E is the Young’s modulus; I is the modulus of inertia \(I=M\cdot L^2\) [kg·m²]; L, a and b are characteristic dimensions; \(R_1\) and \(R_2\) are the force reaction at supports

3.2 FEA port assembly and clamp assembly

The Finite Element Analysis was done using ANSYS software, [9]. It was wanted to resolve it, following a problem, namely the support plate of the active plate clamps bent at the end of the working stroke with a value of about 0.2 [mm] as a result of the action of the force applied by the punch one of the bending plate, as shown in figure 6 [10].

**Figure 6.** Clamp assembly and acting clamp.

The Finite Element Analysis study was performed with Ansys software, and for the analysis, a single clamp assembly (figure 6) was taken into consideration, the elastic clamping problem of the clamp plate being present for all assemblies port clamp.

**Figure 7.** Geometrical properties
3.3 Ansys Results - port-clamp plate analysis
For finite element analysis, the limit conditions were specified, namely Force, fixed support:
- The acting force $F = 500$ [N]
  Force $= 500$ [N] acts in the center of the clamp rod (the rod on which the bending pin is fixed).
- Fixed Support-- limit conditions
  The fixed support is considered to be the end of the stem and at the same time the clamping areas of the clamp plate on the base legs, present in figure 8 and figure 9.

![Figure 8. Inserting the acting force.](image8.png)
![Figure 9. Inserting the fixed support (boundary conditions).](image9.png)

3.4 Preprocessing FEA
The FEA post-processing took into account the results presented in figure 10, results that had a significant impact on device construction.

![Figure 10. The FEA post-processing and results review](image10.png)

In figure 10 a) are shown the number of nodes: 155.252 and the number of finite elements: 85.290, in figure 10 b) is present the value of deformation which is equal 0.219[mm], and figure 10 c) represent the Safety Factor=0.89 but the optimal value should be at least 2.

In conclusion, this type of construction (material type - aluminum, and thickness of the board = 20 mm) does not withstand the demands during the operation of the device for the bending process, as shown in fig. 11 (zones A, B, and C) that the joint is not appropriately performed.

![Figure 11. The issues on the tool assembly (bending)](image11.png)
3.5 Final version FEA analysis

After several attempts to change the material and the thickness of the sheet, it was concluded that the best material for the construction of the support plate of the clamp-type assembly is steel-1.0503 and the thickness = 16 mm.

In figure 12 and figure 13 is present the final post-processing results about the analysis of the support plate and shows the total deformation which is = 0.09 [mm].

![Figure 12. Total deformation plate 16 [mm] steel](image)

![Figure 13. Safety factor = 4.439](image)

4. Conclusions

In conclusion, the optimization of a project, using Ansys, based on the experiments above, was accomplished.

The set of experiments has shown the reaction of the active plate with an aluminum plate and steel plate thickness of 20 mm and respectively 16 mm.

1. Plate deformation values Aluminum / Steel thickness = on two cases respectively 20 and 16 [mm]
   - The total deformation, for the aluminum plate, is 0.4 [mm]
   - The total deformation for steel plate deformation = 0.09 [mm]

2. The Safety Factor:
   - For Aluminum plate = 0.89
   - For steel plate = 4.4

It is easy to see that the deformation of the active plate made of steel falls within the elastic range and the safety factor has the optimal value (4.4).

In conclusion, the FEA analysis is necessary because, by the finite element analysis, the exposed areas can be determined from the deformation resistance, but also of the total deformation.

In this way, the possibility of losses due to non-conforming design, as well as of the rebut parts that will be wrongly manufactured, is eliminated.

Based on this analysis, the new device has been built, alongside a more robust construction that works mechanically at the desired parameters and a 40% lower cost of the semi-finished product.

To demonstrate the validity of the results, an optimization analysis was performed using the dedicated Matlab module: Design of Experiments. This optimization gives the result for a 16 mm thick steel piece.

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 ing. Binselean Mihail.
References

[1] Nicolitov V 2011 Replacing the exhaust drums (in Romanian) https://www.autoexpert.ro/inlocuirea-tobelor-de-esapament/, accessed at 2019.01.24

[2] Sen Q Z and Yuan R 2019 Applied Acoustics Volume 145 176-183

[3] Jianping L, Hongwei Z, Xiaoli H, Lin Z, Fengliang H, Cong L, Ning L, Yuexi Z Changyi L 2016 Ins. and Exp. Tech. 59 5 762–767

[4] Josef D, Johannes D, Otto H 2018 Int. Jou.of Fat. 109 49-59

[5] Color Metal Direct, Bending of sheets (in Romanian) https://color-metal.ro/ro/%C3%AEndoirea-tablelor-descrierea-procedurii, accessed at 2019.01.24

[6] Negrău D C, Finite Element Analysis and Solution Optimization for a bending device, 2018, master thesis, University of Oradea.

[7] Tin-C, Toly C, Yu-Cheng L, 2019 A three-dimensional-printing-based agile and ubiquitous additive manufacturing system 55 Part A 88-95

[8] Huei-H L 2017 Finite Element Simulations with ANSYS Workbench 17- Theory, Applications, Case Studies (SDC Publications)

[9] Jankovics D, Gohari H, Tayefeh M, Barari A 2018 Developing Topology Optimization with Additive Manufacturing Constraints in ANSYS, The 16th IFAC Symp. On Information Control Problems in Manufacturing 51/11 p 1359-1364

[10] Grebenișan G, Bogdan S 2017 Parameterized Finite Element Analysis of a Superplastic Forming Process Using ANSYS MATEC Web of Conferences 126 12603003.