

DEVELOPMENT OF A DEVICE FOR BENDING METAL SHEETS: CONCEPTION, FINITE ELEMENT ANALYSIS AND CONTROL SYSTEM

Eduardo Perez Liberato

Universidade de Brasília, Brasília, DF
eduardo.perez.liberato@gmail.com

Lucival Malcher

Universidade de Brasília, Brasília, DF
lucival.malcher@gmail.com

Abstract. *The process of bending a metal sheet is commonly used in aerospace, automotive and chemical industries. For this reason, one could easily argue that it is important to have a study about how a bending machine can be made and how to avoid failures while bending a metal sheet. This project is focused on developing a bending machine which collects data from the force used in the bending process by an Arduino system. It is important to mention that the project was validated according to a simulation that used the Finite Element Analysis, supported by Abaqus software. In these simulations, the bending itself, the external structure and the punch will have, each one, a simulation to measure its load applied on each part. After the Finite Element Analysis, the manufacturing process began, assembling the parts used to make the bending machine and to put together the actuator with the punch. Once this was completed, the mold was placed inside the device and an aluminum sheet was on it, the bending process could be done. After that, a strain gauge was glued on one of the punch's lateral and this strain gauge was connected to an Arduino module. At this point, the Arduino was calibrated to measure the correct strain collected by the strain gauge. Once the strain was measured, the stress could also be calculated, according to the measured strain. Finally, the bending process has the results and its variation, controlling the bend and manufacturing metal sheets bend. For the validation of this experiment, some aluminum sheets were bent as well. This device has the objective to serve as a project to help students to understand manufacturing technologies, processes and to improve their knowledge on materials sciences, since the dimensions chosen for the project are small and make it portable by hand.*

Keywords: *Bending Machine; Finite Element Analysis; Manufacturing Technology;*

1. INTRODUCTION

Metal forming is very useful in the aerospace, automotive, metallurgical, mechanical engineering and construction industries, Barrans and Muller (2009), Bisagni *et al.* (2022), Freitas *et al.* (2020), Freitas *et al.* (2022), and in Boukar *et al.* (2022). Some examples can be given: in the manufacture of wire support profiles, for the development of bent sheets in automotive systems from the exterior of automobiles, in water pipes, as in flanges, in buildings, in aircraft wing profiles and testing these, as mentioned in Bouvet *et al.* (2022). Currently, due to the demand for materials that are lighter and how industries (mainly automotive) are looking for energy efficiency and less waste emissions. Once this information is available in Cruz *et al.* (2021), and in Freitas *et al.* (2021) bending processes should receive more study.

The work to be presented will serve as a contribution to studies on bending in sheet metal, since it is of interest to industries and engineering, even though it is small. Hence, this project aims to show a test of bending of metal sheets in small dimensions, contributing to a material's failure test validation, focusing in the area of machine's project development, plasticity, mechanics of materials and control of mechanical equipment. In this way, the project, in addition to being compact and easy to transport, will be a facilitator of understanding for academic subjects that are a part of future engineers' formation.

It is important to mention that the project has simulations made in the Abaqus software (using Abaqus 6.14 version), since the project was simulated and dimensioned on it. Thus, with several simulations, it is possible to perceive that the project has an importance through the field of simulation with the Finite Element Analysis, used in the aerospace, mechanical and civil construction industries, since the simulation of forces being applied in mechanical elements of construction is interesting. to be analyzed by these types of industries, as presented in Bouvet *et al.* (2022), for example.

The bending process can also serve biomedicine as a way to discover new materials with a focus on treatments for diseases related to skeletal fragility (for example, osteoporosis), as presented in Ridha and Thurner (2013). These treatments may come from a new material design that is more resistant to fractures and is easy to implant in a person. Thus, this project covers a vast area of utilities, to also serve as an eventual mechanism to develop research in materials and their applications, as also seen in Söntgen *et al.* (2013). Following with another biomedical application, the process of folding materials can be applied as a test for materials used in the manufacture of catheters, seen in Badrou *et al.* (2022).

The first objective of the project completed is the Finite Element Analysis: the test itself (the actuator pushing the punch against a metal plate that will be deformed according to the mold model), and the chassis (the structure around the project, which will receive the reaction force from the actuator).

Secondly, the device that bends sheet metal was built and certified that bending occurs without failure. At this instance, a device will be developed with an actuator that will push a punch against a metal plate that will deform according to the chosen mold.

Finally, once the mechanical components of the project are ready, the control system will be implemented to make the punch able to measure the force being applied through a sensor attached to it. The sensor will collect data and will be translated by an Arduino device, which will be linked to a computer, where its programming will be implemented to validate the force that will be applied to the bending machine.

2. THE FINITE ELEMENT ANALYSIS FOR THE PROJECT

For this project, the finite element method was used with Abaqus Software. This method is commonly used in engineering (mainly mechanical, mechatronics, aerospace, civil, metallurgical and chemical) since it is possible to analyze quantitative variables along an object, for example: stresses and strains.

This method is applied by a computer to object analysis. Due to this, it is possible to make decisions on which will be the best option for a given object according to the conditions of its material. It is interesting to mention that this project only uses the method, not reporting comparisons about variations of this method. Thus, all the information used throughout this work is implemented following the suggestions of the Abaqus manual (Abaqus/CAE User's Manual, 2011). It is important to mention that all the simulations used a quad element type in the meshes.

Once the Finite Element method was presented, the Von Mises Criterion for failure prediction was used to measure and compare where the largest stress would be in each part and in the chassis. The von Mises yield criterion can be found in the book Mechanical Behavior of Materials (Hosford, 2005).

The Finite Element Analysis shows where several parts are presented in the project. The simulation in Abaqus Software starts with the concept of applying a force strong enough to deform a metal sheet in the shape of the mold defined through a punch connected to an RC-51 type actuator, RC-51 (Enerpac, 2022), simulating a three-point bending.

2.1 Project's conception

The project structure is presented as a structure formed by two plates, four bars of circular section and a bar above to improve the rigidity of the project – see Fig. 1 f). This structure was designed regarding its easy manufacturing for an academic work. In this way, the project was developed to be small, light and easy to transport.

Describing the parts, it is possible to comment that the bottom plate, see Fig. 1 a), was idealized having 400 mm in length, 20 mm in height and 100 mm in depth. The upper plate is a replica of the lower one, making it easy to manufacture. Also, the circular section bars are identical and solid: they have a radius of 7.5 mm for their base, as shown in Fig. 1 b), and 212 mm in height.

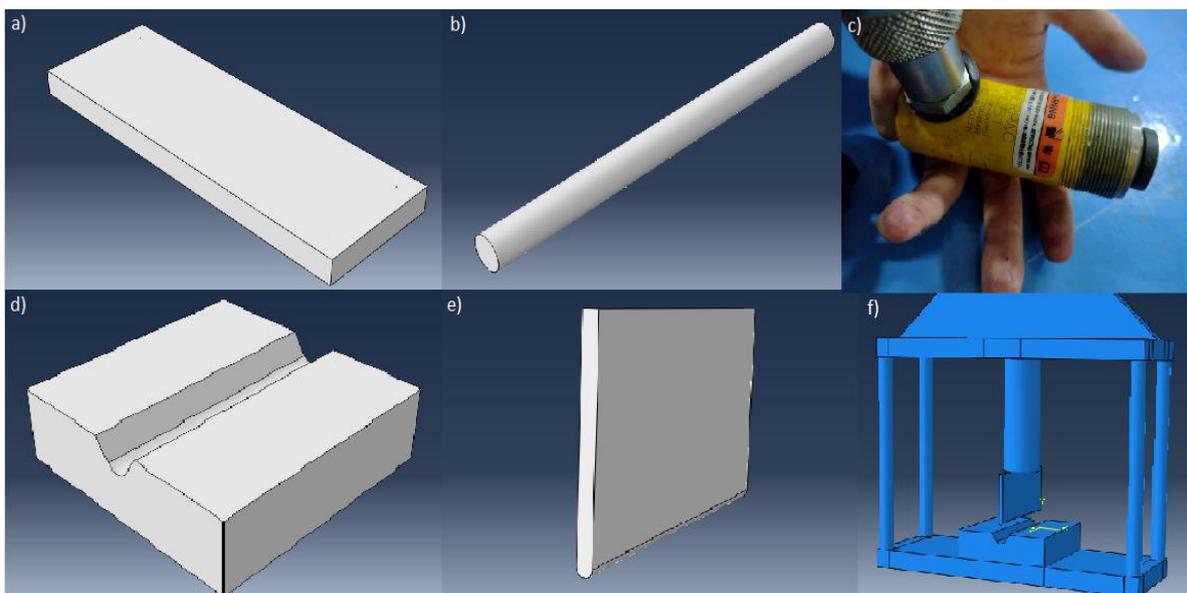


Figure 1. Parts of the project: a) plates design, b) circular bars design, c) RC-51 actuator, d) mold, e) punch and f) bending machine.

Moreover, the mold that will shape the sheet metal bend - Fig. 1 d) - was planned to have the dimensions of base height and depth, respectively: 30 mm, 100 mm and 100 mm. Furthermore, the mold has a "V" recess with height varying according to the size of the thickness of the sheet to be bent in half (where the punch pushes the sheet metal against, deforming it). In such manner, the distances from the minimum point of the recess in "V" were defined, so the punch (Fig. 1 e), pushed by the actuator (presented in Fig. 1 c), would always have a final displacement of 10 mm from its initial position.

The punch, seen in Fig. 1 e), was designed to be a part that has a height of 50 mm, a width of 4 mm and a depth equal to that of the mold, that is, 100 mm. The punch tip, in frontal view, appears to be rounded with a circle of radius 2 mm, see Fig. 2 b), for example. Furthermore, it is important to mention that for the simulation in the Finite Element Method, a reference point was placed on top of the punch to verify the force variation that it got during the bending process.

2.2 Discovering the maximum bending load

The sheets (plates) used in the Abaqus simulation were aluminum plates (say the type of aluminum) with different thicknesses - 1 mm, 2 mm, 3 mm and 4 mm. These plates had the following properties, shown in Table 1 and the stress-strain table presented in Malcher *et al.* (2020):

Table 1. Conditions of the metal used to verify the bending process shown in Malcher *et al.* (2020)

Metal	Young's Modulus [GPa]	Poisson's Ratio	Density [kg/m ³]
Aluminum	70	0.3	2700

As a consequence, once the Abaqus simulation was completed, it was possible to predict forces' measures applied when bending the metal sheet. In order to facilitate the arrangement of the data, Table 2 was created showing the applied force for each sheet thickness.

Table 2. Applied forces to bend metal sheets in Abaqus simulation

Thickness [mm]	Force [N]	Load [ton]
1	11266	1.15
2	20648	2.12
3	35484	3.62
4	48294	4.93

Figure 2 shows the results collected from Abaqus when the punch bent the metal sheets. The meshes used for the simulation of the aluminum sheets had a minimum of 1600 elements and a maximum of 6400 elements (the first one to the 1 mm thick sheet and the last one for the 4 mm thick sheet). In the same figure, it is possible to see how the simulation was set up as well, using one reference point under the mold and another on punch's top. It is possible to notice that most of the force is concentrated in the bending region of the sheet. This meant that the mesh could be refined only in the central region. Therefore, it is faster for the computer to process the results through the Finite Element Method.

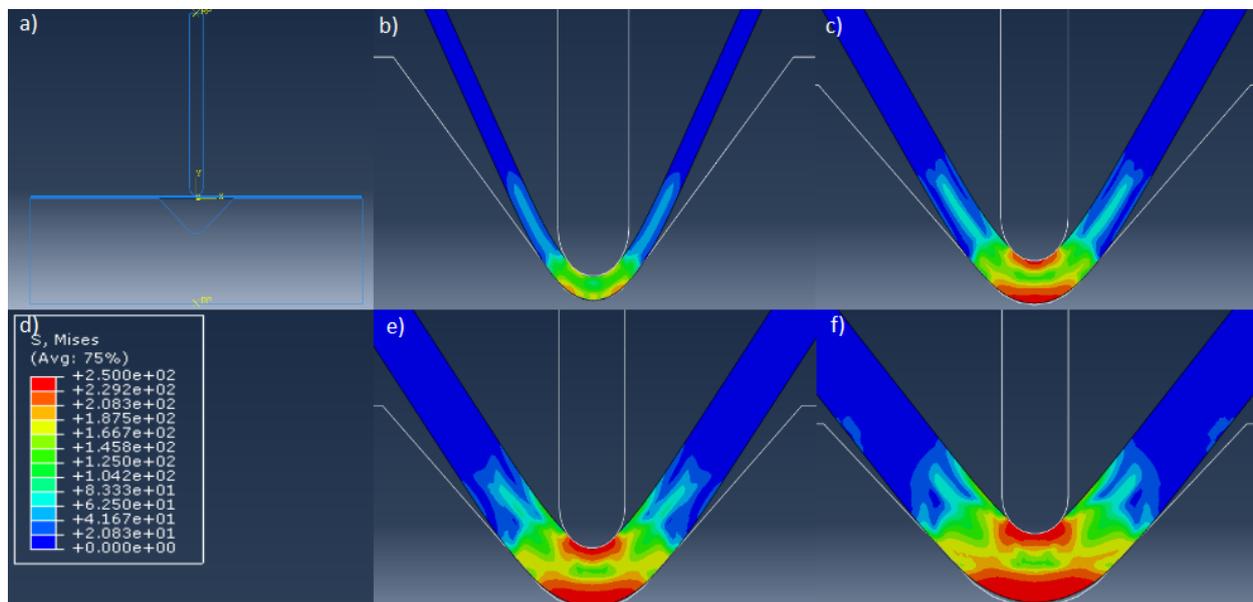


Figure 2. a) Simulation for a bending process, b) bending of a 1 mm aluminum sheet, c) bending of a 2 mm aluminum sheet, d) von Mises stress scale, e) bending of a 3 mm aluminum sheet and f) bending of a 4 mm aluminum sheet.

From the results presented in Fig. 2, it was possible to discover that the maximum load supported by the machine. This load would be a load of approximately 5 tons, the maximum limit of RC-51, (Enerpac, 2022). It is worth mentioning that these tests were implicitly simulated. When the sheet bend simulation was simulated explicitly (with a speed of 1 mm/s), the calculated force was 46876 kN - resulting in a load of 4.78 ton. When the velocity was 10 mm/s, the explicit simulation had the same maximum force being applied.

2.3 Chassis' dimensioning

After the metal sheet's bending simulation in the software and the determination of the forces needed for the bends to be made, project's external structure (chassis) was assembled and simulated in the Abaqus software for the knowledge of how much the material would deform if it were subjected to a maximum load of 5 tons, according to its geometry. The first chassis' simulation was made using only two plates and the four bars, shown in Fig. 3 b) and f).

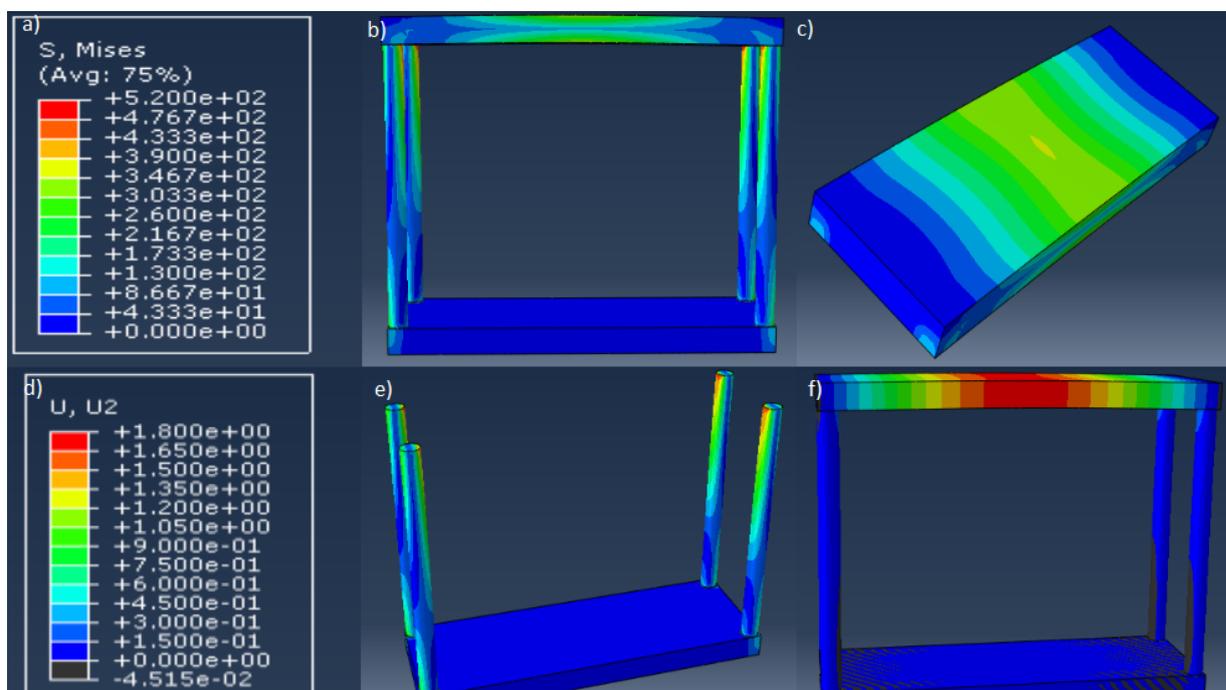


Figure 3. a) Chassis' von Mises scale, b) chassis' simulation, c) upper plate's stress from the chassis, d) strain scale, e) bar's stress and f) total strain in the vertical axis due to the bending process when a force of 5 ton is applied.

While performing the simulations, initially, it was noticed that there was a large deformation (a displacement of 1.73 mm in the vertical axis) resulting in this simple system of the plates, which caused a large deformation for the project, taking into account that the displacement of the punch was of 10 mm. Furthermore, the highest stress of all was located in the bars close to the upper plates, at 518.9 MPa, see Fig. 3 e). Moreover, it was possible to perceive that the deformation of the chassis was too great for the project, as presented in Fig. 3 f).

To mitigate the problem, it was proposed to place a bar perpendicular to the top support in order to improve the stiffness of the top plate, as shown in Fig. 4. To improve the manufacturing process, the design was improved when the bar above the upper plate was configured to have the same height as the upper support length, which would facilitate the fabrication of the material, as shown in Fig. 4.

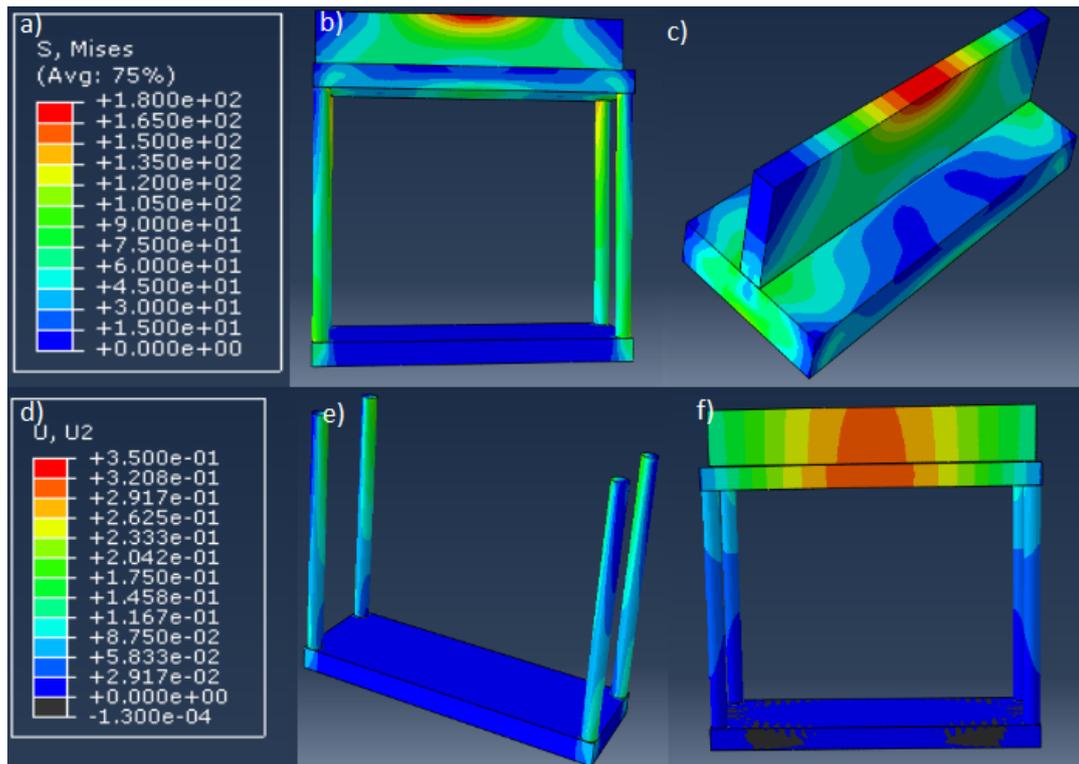


Figure 4. a) Chassis' von Mises scale, b) 2.0 chassis' simulation, c) upper plate's stress from the 2.0 chassis, d) strain scale, e) bar's stress and f) total strain in the vertical axis due to the bending process when a force of 5 ton is applied.

One can notice that, in this second simulation, not only the device itself has a lower von Mises tension at all, since the first simulation had a limit of 518.9 MPa of equivalent tension and the new chassis (2.0 chassis) 178.0 MPa, but the bars that were between the supports began to have a lower von Mises tension than the upper support (as it is possible to compare Fig. 3 lower central with Fig. 4 lower central). Hence, the upper section of the 2.0 chassis project has the biggest von Mises tension. Thus, the project already presents a considerable improvement when compared to the initial structure.

However, the project weight could be optimized cutting the upper edges, making the project lighter and not making an important result difference, which resulted in 3.0 chassis. It is important to mention that the scale used in Fig. 5 for von Mises had the maximum of 190 MPa, because the maximum value observed was 180.8 MPa, that's why it is not so important this difference.

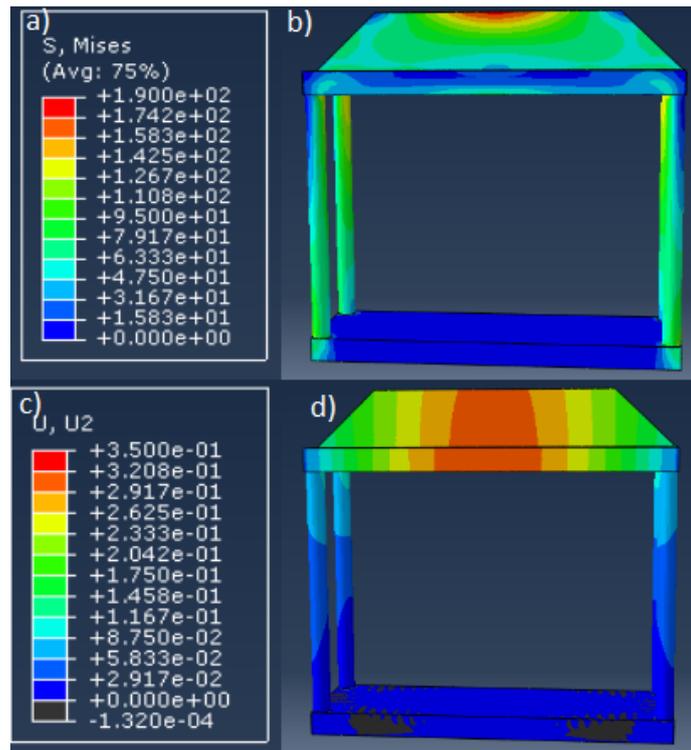


Figure 5. a) von Mises Scale, b) von Mises tensions on 3.0 chassis project, c) displacement scale in millimeters and d) displacement on 3.0 chassis.

Finally, after optimizing the design with the Von Mises maximum final stress being 180.8 MPa, we have that AISI 1020 steel was the steel chosen as the best (in relation to the cost-benefit of the design). During these simulations, it is important to mention that the upper and the lower plates had a mesh with 4800 elements each one, each circular section bar had a mesh with 6572 elements and the perpendicular bar at the top of the upper plate had a mesh with the total of 4464 elements.

2.4 Punch's dimensioning

When a final chassis (3.0 chassis) was ready to be manufactured, a new simulation was carried out with the intention of verifying which would be the best punch size, since it has the risk of buckling when a very strong load is placed on it. This failure that would occur in the punch could, in addition to spoiling the sheet bend, damage the aforementioned external structure, since the actuator would receive a reaction force perpendicular to its main axis.

A simulation was done for a 30 mm punch, as presented in Fig. 6, as the risk of buckling is even lower, and less material is used to build the punch. With that, it was decided to manufacture a 30 mm punch. In this punch, the maximum horizontal displacement measured was 0.0045 mm. It is valid that there was no great difference in the mean Mises stress along the punch, and the mean Mises stress for the 30 mm punch was around 105 MPa, also presented in Fig. 6.

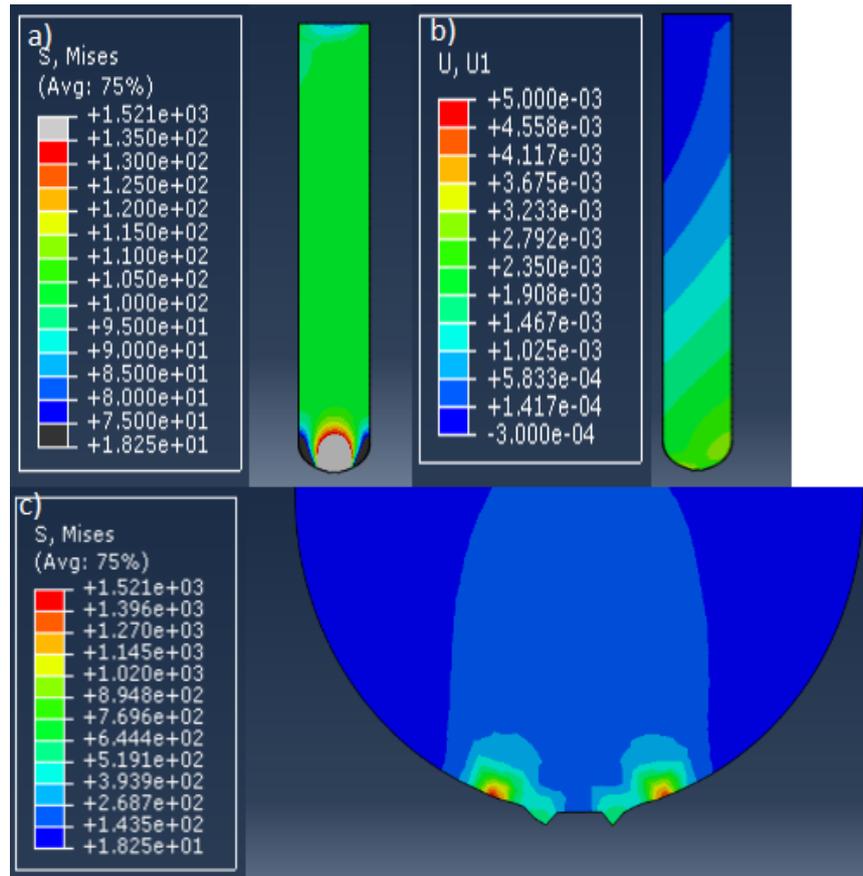


Figure 6. a) Average von Mises tension in the punch and its scale, b) displacement in horizontal axis of the punch and its scale and c) final von Mises tension in the punch's tip and its scale.

It is important to mention that a heat treatment will be carried out on the punch tip to improve its rigidity and reduce the incidence of failures. For the punch's simulation, the mesh used had a total of 1246 elements, which they were assembled to be smaller the closer they were to its tip. Moreover, the mold was also simulated with a mesh using 1184 elements.

2.5 Manufacturing Process for the Project

At this part, the project was built initially buying the equipment parts. After this, the circular bars were cut with a mechanical lathe measuring 212 mm. In addition, the plates (inferior and superior) were milled to have four 2 mm in depth circular recesses with a diameter of 15.5 mm, which fit the bars (presented in Fig. 1 b) in their recesses. The lower plate was designed to have square recess of 2 mm in depth, of 100 mm in length and 100 mm in width, so the mold, seen in Fig 1 d), could be placed and exchanged when needed. The upper plate has a circular recess in its middle, to make it easier to place the actuator (this recess is also 2 mm in depth and 38.5 mm in diameter).

After the cutting process, the welding process began. In it, the bars were welded on the plates and the support bar was also welded on the upper bar. After it, the molds were milled regarding its specificities that require the displacement of 10 mm of the punch. Once this part was successfully completed, a heat thermal treatment was done, to make the point of the punch stiff.

Finally, the mold with an aluminum sheet on it was placed in the assembled project and some tests were done, making some bent aluminum sheets.

3. EQUIPMENT CONTROL SYSTEM

The control system design can be described regarding its components: an Arduino device – seen in Fig. 7 c) - the Wheatstone bridge (component HX711), the strain gauge (sensor that will measure the force according to the load that will be applied when varying the resistance of the internal wires), Fig. 7 b), and some wires to connect the components previously mentioned in this paragraph.

It is interesting to mention that these electronic devices operate in direct current mode and maximum voltage of 5 V, as seen in Arduino's manual (Arduino, 2022). Thus, the equipment is not at risk of being burned or causing electric shocks.

3.1 Electronic device assembly

Initially, the strain gauge (shown in Fig. 7 b) was coupled to the punch, since it is under conditions of an elastic regime. Thus, the force that is applied to the punch generates an elastic deformation in itself, consequently varying the strain gauge measurement, since the strain gauge resistance is variable according to how much it is stretched, as seen in Principles of Measurement Systems (Bentley, 2005).

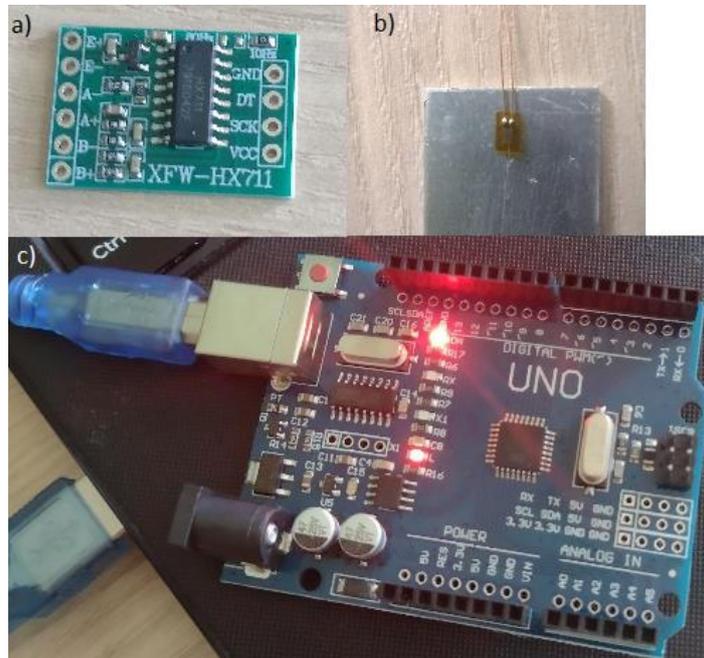


Figure 7. a) HX711 Wheatstone bridge used in project, b) strain gauge, c) Arduino device connected to a computer.

Thus, the strain gauge is connected to the HX711 board (seen in Fig. 7 a), as shown in Fig. 8, such that A+ and A- are connected to the strain gauge wires. The HX711 board is connected to the Arduino with a voltage of 5 V and grounded at the "GND" pin (ground pin). Outputs A1 and A2 present, respectively, the DT ("Digital Output", digital output) and SCK ("Serial Clock") components, as shown in the datasheet in HX711 24-Bit Analog-to-Digital Converter (ADC) for Weigh Scales (Avia Semiconductors, 2022).

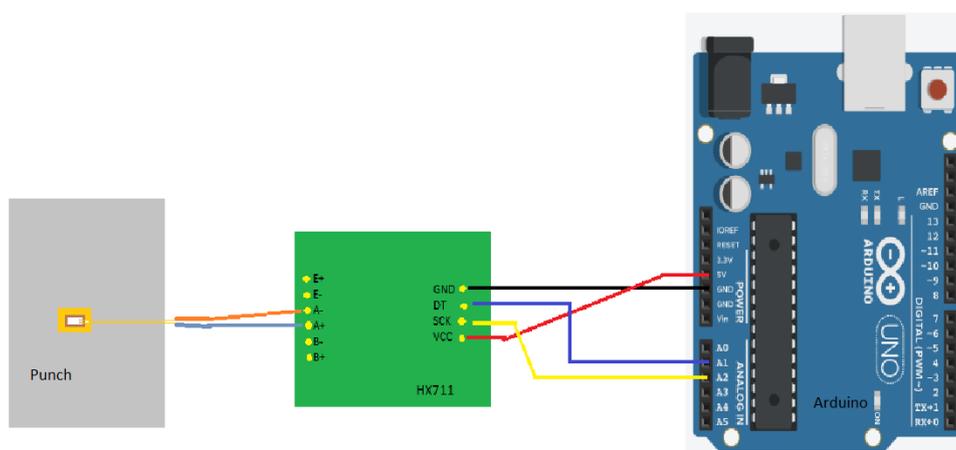


Figure 8. Schematic drawing showing how the Arduino device is mounted on the punch.

Once this schematic, in Fig. 9, was assembled, the calibration part began. Fig 9. shows, from another point of view, how the strain gauge is attached to the punch, how the HX711 board is connected with the strain gauge and how the HX711 is linked to the Arduino Uno.

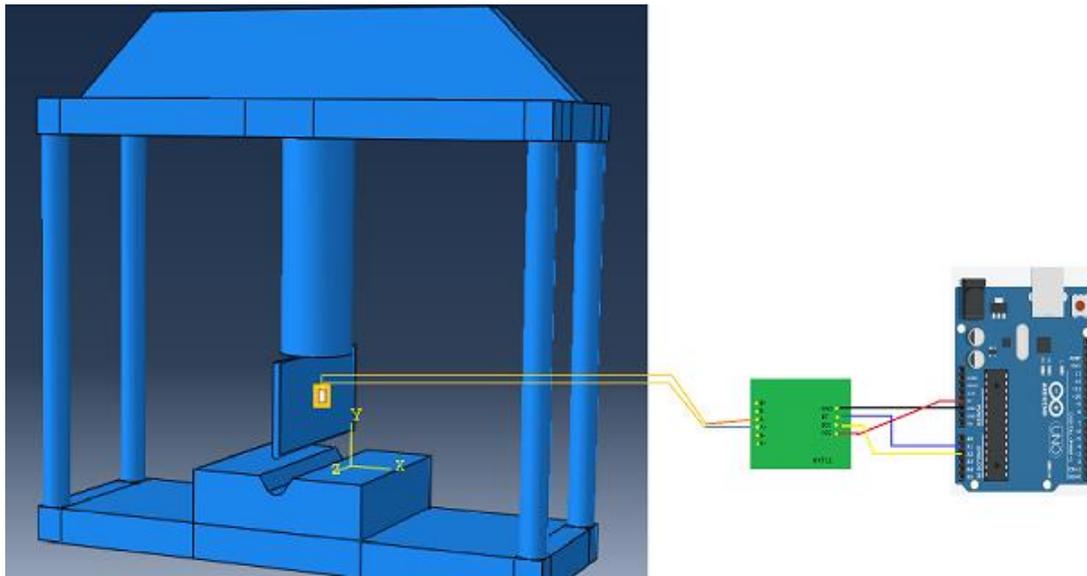


Figure 9. Schematic drawing showing how the Arduino device is assembled with the bending machine.

3.2 Force monitoring

Since the actuator will provide the force on the punch and the punch will undergo elastic deformation, the strain gauge will record how much the punch has been deformed. Thus, the change in force in the system can be measured. According to Measuring Strain with Strain Gages (National Instruments, 2022), the force can be measured by checking the tension between the strain gauge in its initial and final state, as shown in the equation below, where ϵ represents the strain, V the voltage, V_{cc} is the voltage of the Arduino (5 V) and R the resistance (which was varied).

$$\epsilon = \frac{V_{deformed} - V_{not-deformed}}{GF \cdot V_{cc}} \quad (1)$$

$$GF = \frac{\Delta R/R}{\epsilon} \quad (2)$$

Once the entire sensory system is ready, it is interesting to mention how the Arduino device will be calibrated. Thereby, the code used was made in the Arduino IDE virtual platform. The code, at the beginning, connects the A1 and A2 inputs at the Arduino to the outputs DT and SCK, respectively, as shown in Fig. 8. At first, the code starts by calibrating the strain gauge so that it has a mode where the strain gauge has a zero for force, creating an offset value. After that, the control system starts a loop and shows the results of force applied on the computer screen. To turn off the control system, the letter “s” on the keyboard should be pressed.

4. CONCLUSIONS

This work is just a small perspective of how manufacturing process of machines are not easy to make. Even though, this project was completed as a graduation project, and it can be better used after it. Some improvements can be mentioned in this section, such as: the development of a better control system (making a digital and controllable pneumatic actuator), other types of bending materials (iron, copper or plastic) and perform a flow of a material when it is applied a load during a long period of time.

Also, as presented in the introduction of this article, this device can be used to test materials that might be used in biomedical applications, as shown in Ridha and Thurner (2013), Söntgen *et al.* (2013) and Badrou *et al.* (2022). Finally, since the actuator will be assembled in the chassis, it is possible to make other forming metal processes, for example: stamping or a 4-point bending test.

5. REFERENCES

Arduino, 2022. Arduino UNO R3 - Product Reference Manual. 7 Sep. 2022
 < <https://docs.arduino.cc/hardware/uno-rev3> >

- Avia Semiconductors, 2022. HX711 24-Bit Analog-to-Digital Converter (ADC) for Weigh Scales
- Badrou, A., Tardif, N., Even, A., Chaudet, P., Lescanne, N., Szewczyk, J., Gravouil, A., Hamila, N. and Bel-Brunon, A., 2022. Characterization of Surgical Tools for Specific Endovascular Navigation. *Cardiovascular Engineering and Technology*. Lyon, France.
- Barrans, S. M., Muller, M., 2009. Finite element prediction of the ultimate axial load capacity of V-section band clamps. *Journal of Physics: Conference Series*. Huddersfield, United Kingdom.
- Bentley, J. P., 2005 *Principles of Measurement Systems*. University of Teesside, Middlesbrough.
- Bisagni, C., Raimondo, A., Atanasovska, I., Milic, M., Troian, R., Frulla, G., Polla, A., Cora, O.N., Bekci, M.L., Henriques, B., de Moura, M.F.S.F., Almudaihesh, F. and S. Grigg, 2022. S Comparison of numerical analyses of a composite wing component subjected to 4-point bending. *Composites Part C: Open Access*. Delft, Netherlands.
- Boukar, A., Corn, S., Slangen, P. and Jenny, P., 2022. Finite element modelling of low velocity impact test applied to biaxial glass fiber reinforced laminate composites. *International Journal of Impact Engineering*. Alès, France.
- Bouvet, C., Vieille, B. and Pujols-Gonzalez, J., 2022. Experimental characterization and numerical modelling of the translamellar fracture of woven-ply hybrid fibers reinforced thermoplastic laminates. *Engineering Fracture Mechanics*. Rouen, France.
- Cruz, D., Barbosa, M., Santos, A., Miranda, S. and Amaral, R., 2021 Application of Machine Learning to Bending Processes and Material Identification. *Metals*. Porto, Portugal.
- Dassault Systèmes Simulia Corp, 2011. Abaqus/CAE User's Manual. 18 Oct. 2022
<http://130.149.89.49:2080/v6.11/pdf_books/CAE.pdf>
- Enerpac, 2022. RC-51. 18 Oct. 2022
<<https://www.enerpac.com/en-us/rc-series-cylinders/general-purpose-cylinder/RC51>>
- Freitas, A.G., Santos, R.B., Pereira, L.M., Lima, Y.O. and Riascos, L.A.M., 2020. Parametrization of a new solids feeder based on energy efficiency. *14th IEEE International Conference on Industry Applications (INDUSCON)*. Santo André, Brazil.
- Freitas, A.G., Lima, Y.O., Oliveira, V.F., Santos, R.B. and Riascos, L.A.M., 2021. Batchpump: an alternative to conventional blow tanks. *Latin American Applied Research*. Santo André, Brazil.
- Freitas, A.G., Oliveira, V.F., Santos, R.B., Riascos, L.A.M. and Zou, R., 2022. Optimization method for pneumatic conveying parameters and energy consumption performance analysis of a compact blow tank. *Journal of Pressure Vessel Technology*. Santo André, Brazil.
- Groche, P., Calmano, S., Felber, T. and Schmit, S. O., 2015. Statistical analysis of a model based product property control for sheet bending. *German Academic Society for Production Engineering*. Darmstadt, Germany.
- Hosford, W.F., 2005. *Mechanical Behavior of Materials*. University of Michigan, Ann Harbor.
- Malcher, L., Morales, L.L.D., Rodrigues, V.A.M., Silva, V.R.M., Araújo, L.M., Ferreira, G.V. and Neves, R.S., 2020. Experimental program and numerical assessment for determination of stress triaxiality and J3 effects on AA6101-T4. *Theoretical and Applied Fracture Mechanics*. Brasília, Brazil.
- National Instruments, 2022. Measuring Strain with Strain Gages. 7 Sep. 2022
<<https://www.ni.com/pt-br/innovations/white-papers/07/measuring-strain-with-strain-gages.html>>
- Pradeau, A., Thuillier, S. and Yoon, J.W., 2016. Prediction of failure in bending of an aluminium sheet alloy. *International Journal of Mechanical Sciences*, Lorient, France.
- Ridha, H. and Thurner, P., 2013. Finite element prediction with experimental validation of damage distribution in single trabeculae during three-point bending tests. *Journal of the Mechanical Behavior in Biomedical Materials*. Orleans, France.
- Söntgen, S., Keilig, L., Kabir, K., Weber, A., Reimann, S., Welle, K. and Bourauel, C., 2022. Mechanical and numerical investigations of biodegradable magnesium alloy screws for fracture treatment. *Journal of Biomedical Materials Research Part B: Applied Biomaterials*. Bonn, Germany.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.