IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 18, Issue 1 Ser. III (Jan. – Feb. 2021), PP 31-41
www.iosrjournals.org

The verification of a redesigned part of a telescopic hydraulic cylinder based on finite element analysis

Zejnelabedin Aziri[1], Elizabeta Hristovska[2], Zore Angelevski[3], Ivo Kuzmanov[4], Roberto Pasic[5], Vangelica Jovanovska[6]

[1]Faculty of Technological Sciences / Mother Teresa University in Skopje, North Macedonia
[2,3,4,5]Faculty of Technical Sciences / St, Climent Ohridski University in Bitola, North Macedonia
[6]Faculty of Biotechnical Sciences/ St, Climent Ohridski University in Bitola, North Macedonia
Corresponding Author: Zejnelabedin Aziri

Abstract: *As engineering designs become more complex, the conventional manufacturing design methods are becoming more and more time consuming and costly. The use of Computer Aided Engineering tools has been able to greatly eliminate the design restrictions previously faced by manufacturers. One of the methods that are used to solve the complex systems of the engineering designs is the Finite Element Analysis (FEA) which is the most well-known method used in Computer Aid Engineering (CAE). This study uses SolidWorks simulation to carry on FEA in order to test and verify the redesigned part of a front end telescopic hydraulic cylinder, which is important for the proper operation of the device when the defined pressure is being applied. The results of the analysis show the stress distribution and whether the proposed design and material of the product will offer resistance to the applied pressure. This type of simulation avoid the multiple prototype building and testing cycles and achieve reliable results through easier, faster and more cost-efficient design cycles.*

Key Word*: Redesign, 3D model, FEA, SolidWorks simulation, CAE.*

Date of Submission: 23-01-2021 Date of Acceptance: 07-02-2021

I. Introduction

Not long ago the computers were out of use in the design offices. Instead, designers worked at drawing boards and they would draw their design with pencils which were afterwards passed to stress analysts. With advances in technology, things have changed. Nowadays, designers generate CAD (Computer Aided Design) files and translate them into analysis-suitable geometries, meshed and input to large-scale finite element analysis (FEA) codes. Engineering designs are becoming more complex with time and this is estimated to take over 80% of the overall analysis time (Cottrell, 2009). Cottrell states that design of sophisticated engineering systems is based on a wide range of computational analysis and simulation methods, such as structural mechanics, fluid dynamics, acoustics, electromagnetics, heat transfer, etc. (Cottrell, 2009). As engineering designs become more complex, the labor hours, parts and weight increases and that makes the analysis a time consuming and expensive endeavor. The analysis of these systems can become possible through the Finite Element Method (FEM) or Finite Element Analysis (FEA) which is the most well-known method used in Computer Aid Engineering (CAE).

The Finite Element Analysis (FEA) method is a powerful computerized technique for resolving a variety of engineering problems with complex domains. In other words, it is a numerical method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow, and other physical effects (FEA Software, 2020). The basic idea of FEM is to divide the structure into finite elements, connected by nodes, and obtain an approximate solution.

The aim of the present study is to test and verify the redesigned component of an existing telescopic hydraulic cylinder – base eye- by using Finite Element Analysis tools. The objectives set were to present a new component that weighted less than the exiting one and that remains strong enough to survive the stress applied.

II. Literature Review

FEA analysis include 3 basic phases: pre-processing phase, solution phase and post processing phase. The pre-processing process begins by determining the subdivision of problems into nodes and elements. Then the selection of the interpolation functions is made in order to provide an approximation of the unknown solution within an element. Finally equations for an element are developed. In the solution phase, a set of linear or nonlinear algebraic equations are solved to obtain nodal results. Finally, in the post processing phase we obtain other important information such as values of principal stresses, heat fluxes etc.

Kurowski (2017) has summarized the most essential characteristics of FEA for design engineers as follows:

- For Design Engineers, the FEA is one of many design tools and is used along CAD, spreadsheets, catalogs, data bases, hand calculations, text books, etc.
- FEA is based on CAD models
- Analysis iterations must be performed fast and because results are used to make design decisions, the results must be reliable even though not enough input data may be available for analysis conducted early in the design process.
- The FEA used in the design environment should meet quite high requirements. It must be executed fast and accurately. Design engineers should handle relatively simple types of analysis in support of design process. More complex types of analyses, too complex and too time consuming to be executed concurrently with design process, are usually either better handled by a dedicated analyst or contracted out to specialized consultants.
- The ultimate objective of using the FEA as a design tool is to change the design process from iterative cycles of "design, prototype, and test" into a streamlined process where prototypes are used only for final design verification. With the use of FEA, design iterations are moved from physical space of prototyping and testing into virtual space of computer-based simulations

Kurowski (2017) on his book wrote that in 1960s when the numerical analysis were first introduced, there were many other methods in use, but FEM became the dominant numerical method over time because of its generality and numerical efficiency. He adds that the FEM can be applied to any type of analysis and this generality and numerical efficiency is a major consideration for programmers when they decide which method to use in commercial analysis program (Kurowski, 2017).

III. Methodology and materials

The proposed methodology for the present study is applying FEA for testing and verification of a redesigned product by using SolidWorks. Software SolidWorks is mainly used for designing the parts and then collecting them for the further assessments. Lately the software is also able to carry out nonlinear processes (Magomedov & Sebaeva, 2020). As any other similar tool for carrying out non-linear processes under distortion, SolidWorks allow to verify the quality of the products quality, their operation and security when they are being developed (Matt. 2018).

The first step in the study, is to examine the product conditions, such as functional conditions, loading conditions and the type of material to be used in FEA. The second step starts with finite element analysis (FEA), which will examine the loads applied on the designed product and predict where will the maximum deviation and stress appear. Next, the evaluation of the proposed design and verify the final model in order to make sure that the final product meets the loading and displacement criteria. All of the steps are explained in detail in the upcoming parts.

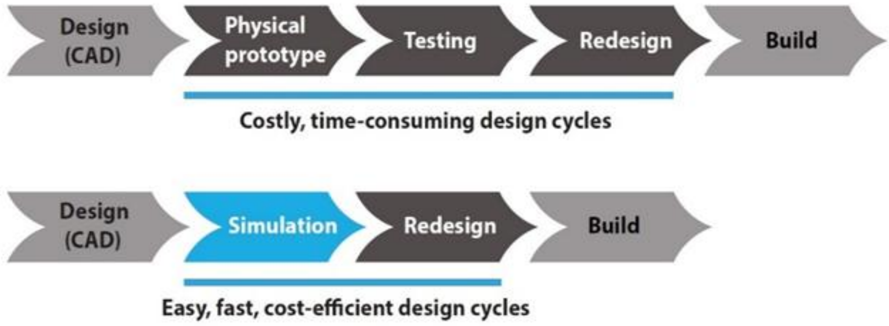
FEA is used to carry out numerous analysis of a product design under different specifications, materials and load applications (Salonitis & Zarban 2015). After the maximum stress and distortions are provided, changes can be maded if it is needed (Salonitis & Zarban 2015).In order to determine the stress the Von Mises law is is applied which will show the stress distribution over the designed model. This is a value that is used to determine whether the material of the product will yield or fracture.

The performed Solidworks simulation help us achieve easier, faster and cost efficient design cycles by replacing the time consuming and costly prototype testing as shown in Graph 1.

The unit system used in the project is shown below in Table 1.

Table 1. Unit system used for the project.

Unit system:	SI (MKS)
Length/Displacement	mm
Temperature	Kelvin
Angular velocity	Rad/sec
Pressure/Stress	N/m^2



Graph 1. Design process with and without simulation

IV. Simulation and Results

Case study

The product chosen for redesign is the base eye. The "base eye" is a part of telescopic hydraulic cylinder that is installed to the body of the cylinder and joins it to the frame. The Figure 1 shows the redesign of a base eye in 3D model created in SolidWorks.

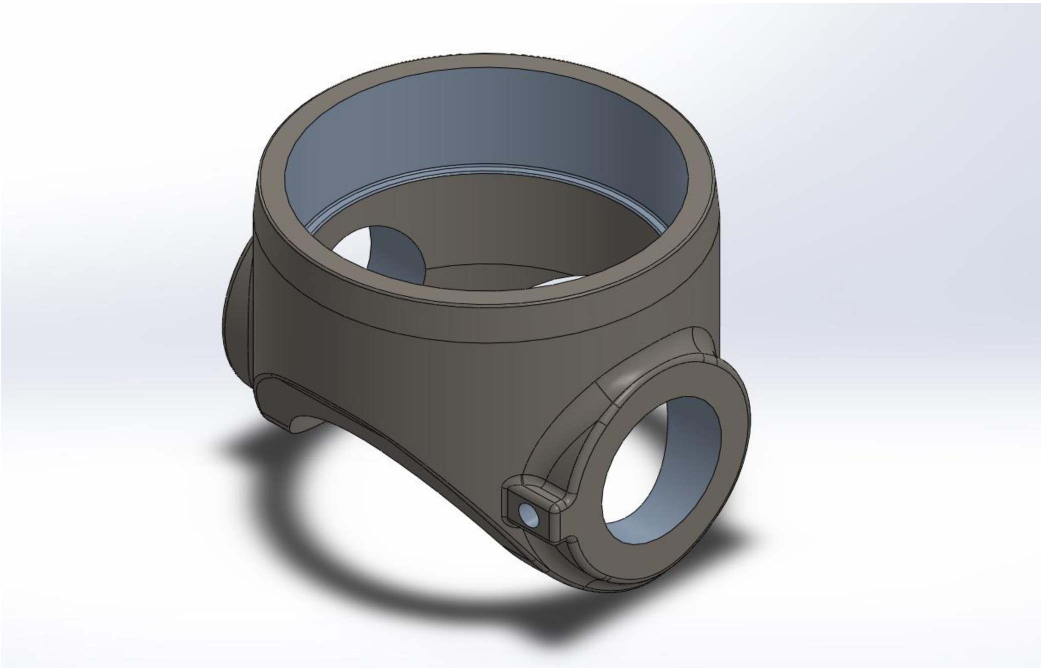


Figure 1. Redesigned "base eye" 3D model created in SolidWorks

In order to make a static stress analysis upon the base eye, a cylinder barrel is created as shown in Figure 2. Additionally we have created an eye trunnion tube where the base eye will be supported (Fig. 3). All three parts of the analysis are inserted and combined together as shown in figure 4 in order to get ready for the simulation process. The chosen material for the base eye is AISI 1035 SAE Carbon Steel the characteristics of which are presented in Table 2.

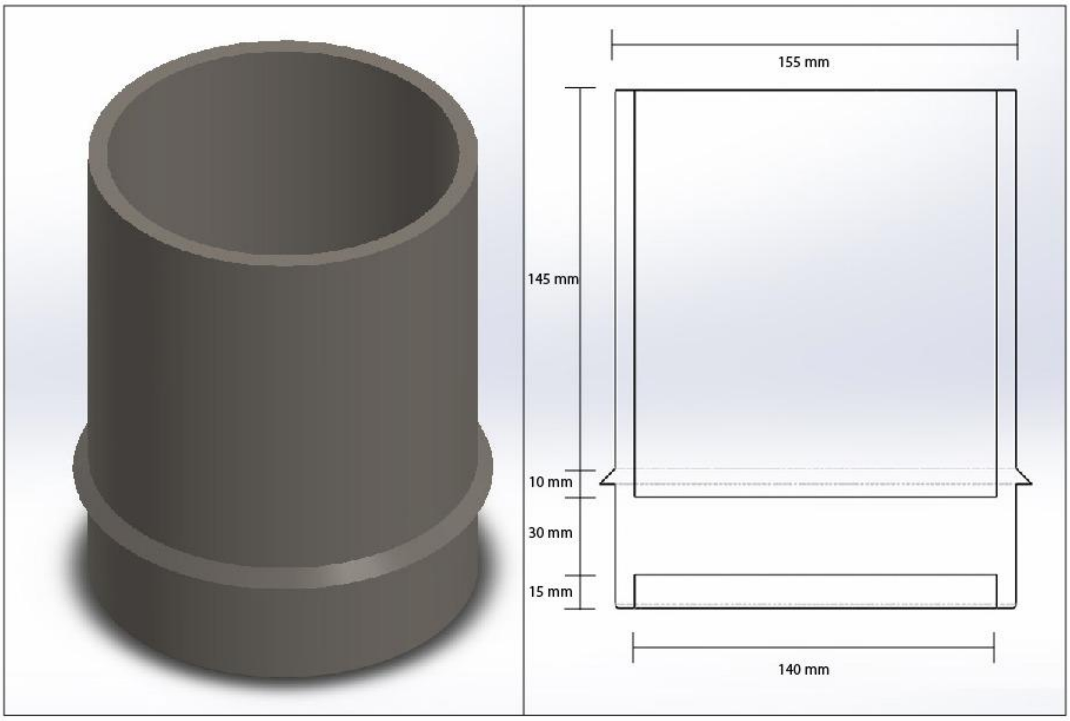


Figure 2. 3D model and dimensions for the cylinder barrel

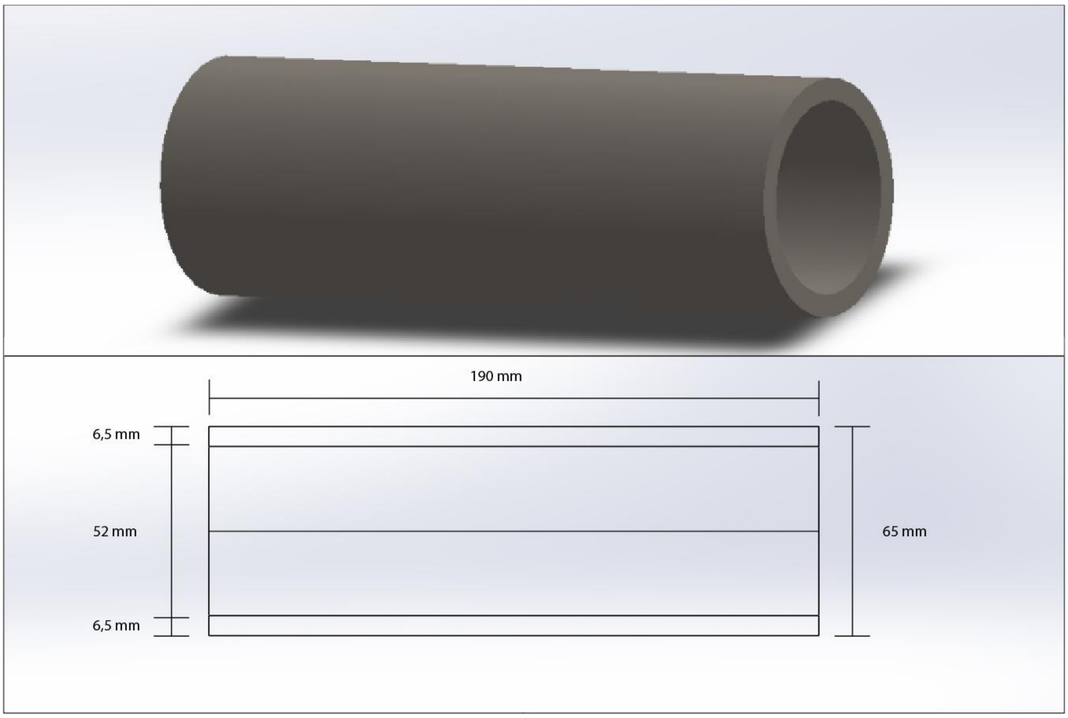


Figure 3. 3D model and dimensions for the trunnion

Table 2. Characteristics of AISI 1035 SAE Carbon Steel

Property	**Value**	**Unit**
Young's modulus	20000	MPa
Tensile strength	585	MPa
Modulus of elasticity	190-210	GPa
Elongation	8-25	%

Fatigue	275	MPa
Yield strength	370	MPa
Specific heat	434	J/kg.K
Density	7850	kg/m^3
Resistivity	0.55	$Ohm.mm^2/m$

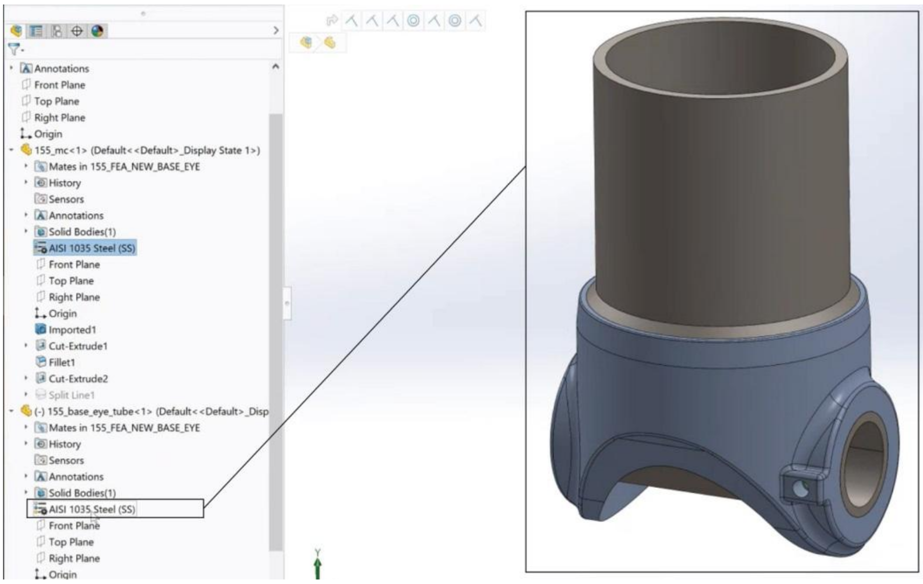


Figure 4. The combination of the parts in SolidWorks and the material selection

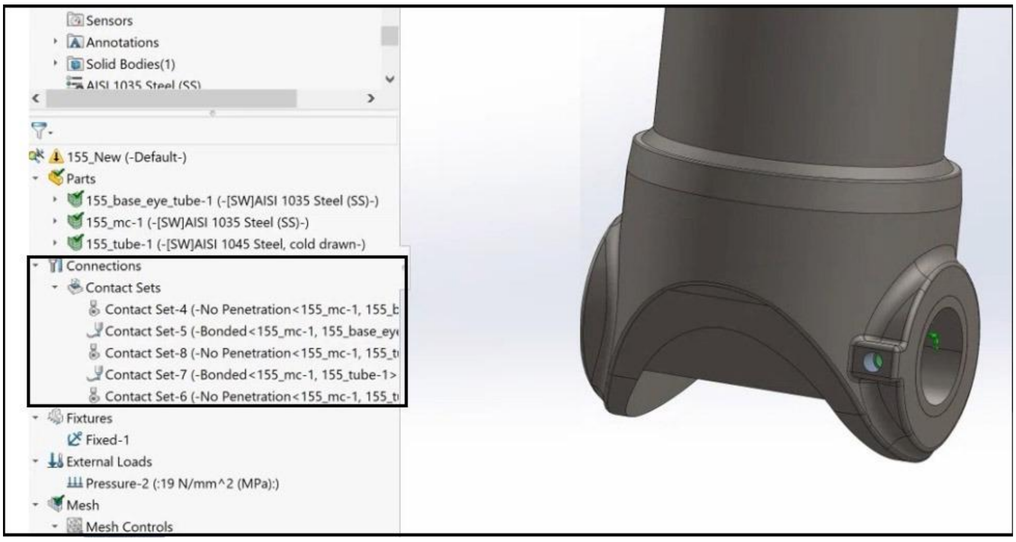


Figure 5. Connection panel in SolidWorks

SolidWorks Simulation

One of the main benefits of the SolidWorks Simulation is the ability to give a better perception of how different parts behave during different loading applications in a product (Visser, 2016). After identifying the parts that we need for analysis, the next step is to connect them together in a proper way. This is done by using the connection panel in SolidWorks as it is shown in Figure 5. The first contact type is done between the outer surface of the trunnion and the base eye by choosing the option of no penetration (Fig 6). This contact type prevents interference between the two parts, but allows gaps to be formed. Then the left part of the trunnion is bonded with the base eye (Fig. 7) and in this case the two parts will function as if they are bonded, which means that they will behave as one part. Additionally, a contact type with no penetration is given between the bottom end of the cylinder barrel and the inner part of the base eye (Fig 8). After that the upper part of the base eye is bonded with the cylinder barrel at the welding point (Fig 9). Finally the inner surface of the base eye is contacted with the outer surface of the barrel with the option of no penetration in order to form a gap between the two parts (Fig 10).

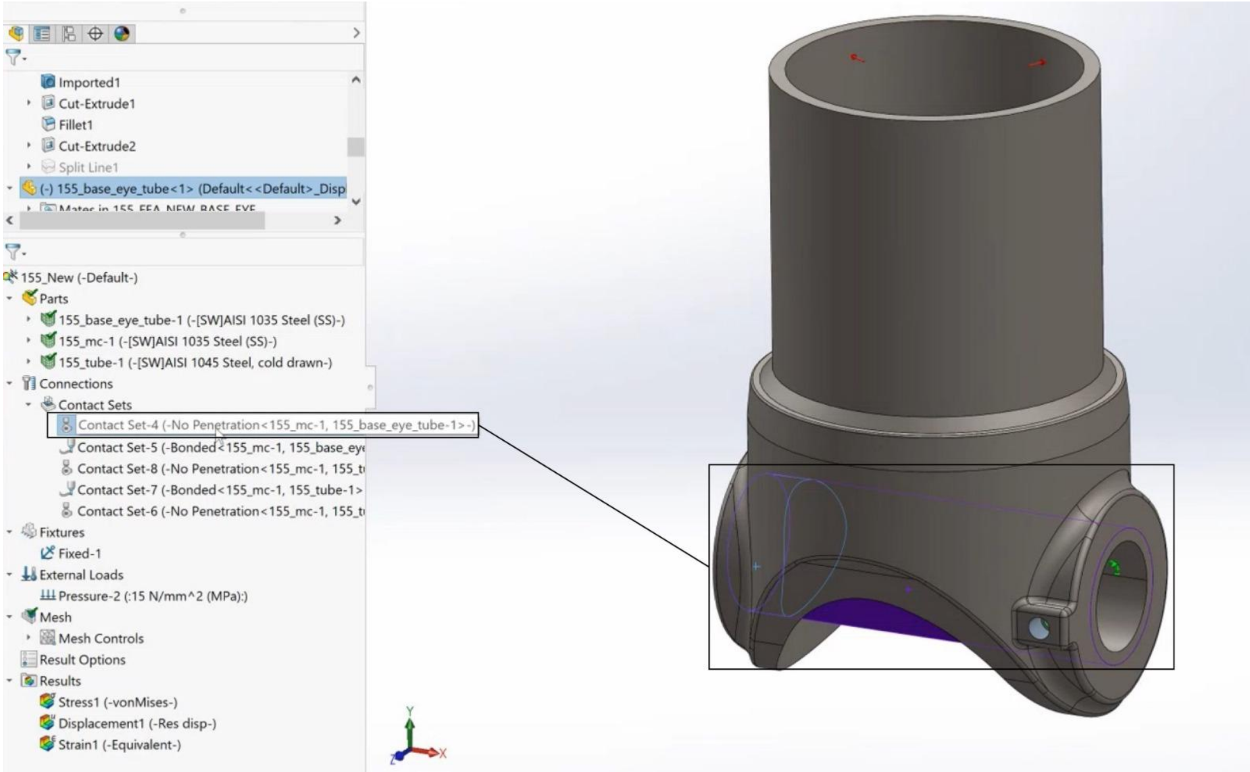


Figure 6. Contact type “no penetration” between the outer surface of the trunnion and the base eye.

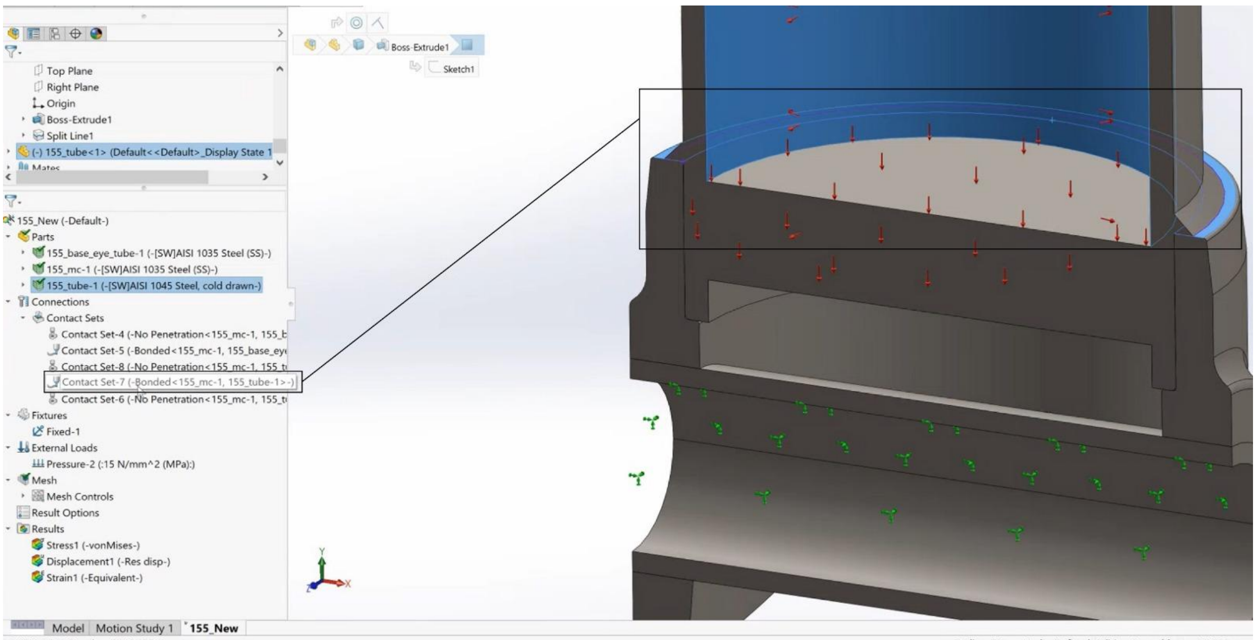


Figure 9. Bonding contact between the base eye and the cylinder barrel at the welding point

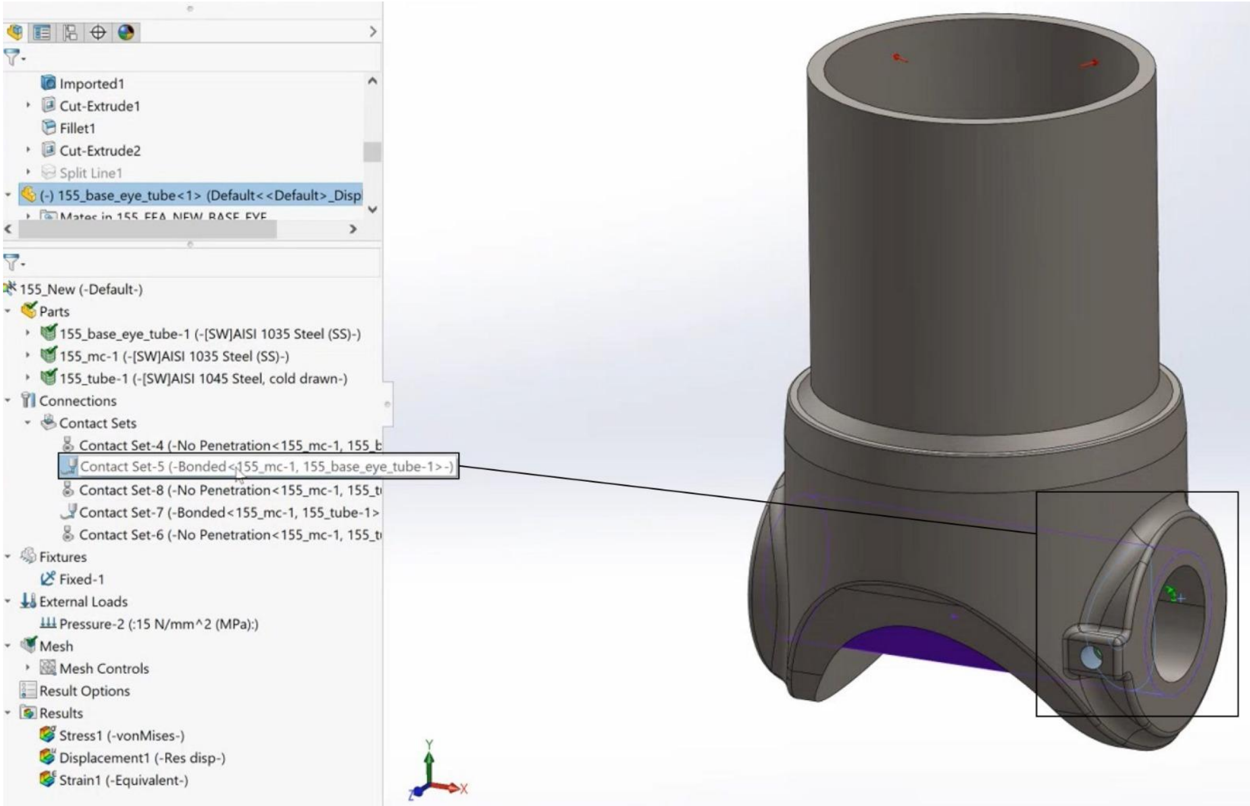


Figure 7. The bonding contact between the left part of the trunnion and the base eye

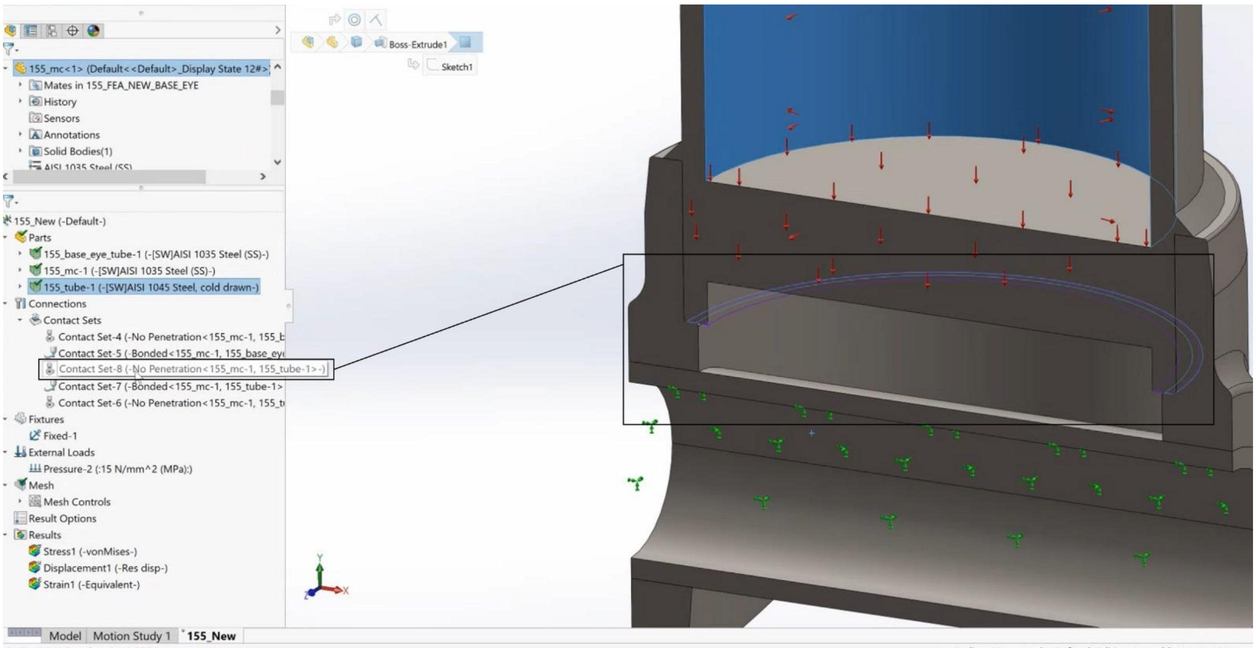


Figure 8. The contact type “no penetration” between the bottom end of the cylinder barrel and the inner part of the base eye

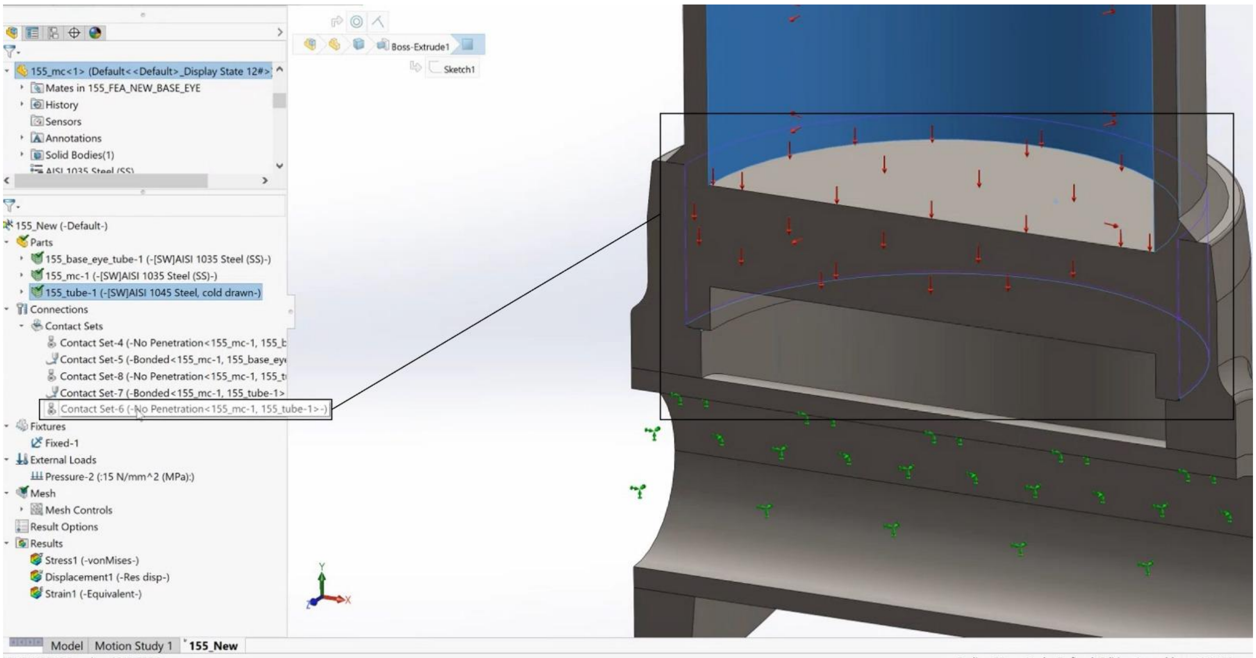


Figure 10. Contact type with no penetration between the inner surface of the base eye and the outer surface of the barrel

After the model is developed we start to determine fixtures and loads on the base eye which are responsible for its distortion. The boundary conditions imposed on the base eye are a fixture on the bottom end (Fig 11) and a pressure of 18MPa applied to the inner face of the hole on the horizontal plane of the solid and on the upper side directed towards –y direction as shown in Fig 12.

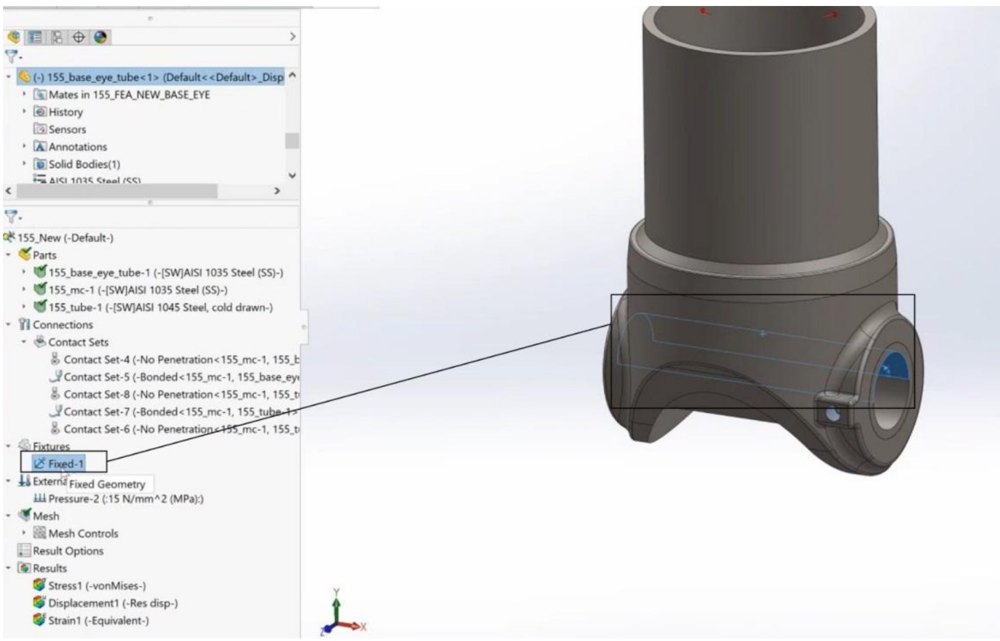


Figure 11. Fixed support of the base eye

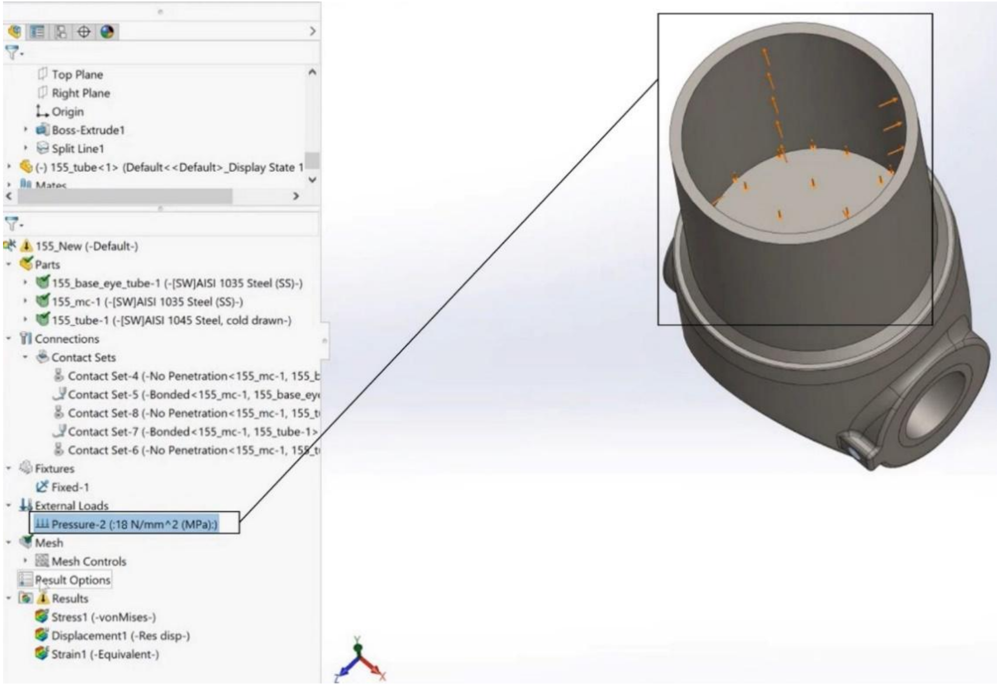


Figure 12. 18 MPa pressure applied on the base eye

The next step is the meshing process, which means splitting the geometry into smaller entities known as elements or nodes. The Table 3 shows the mesh information about the model.

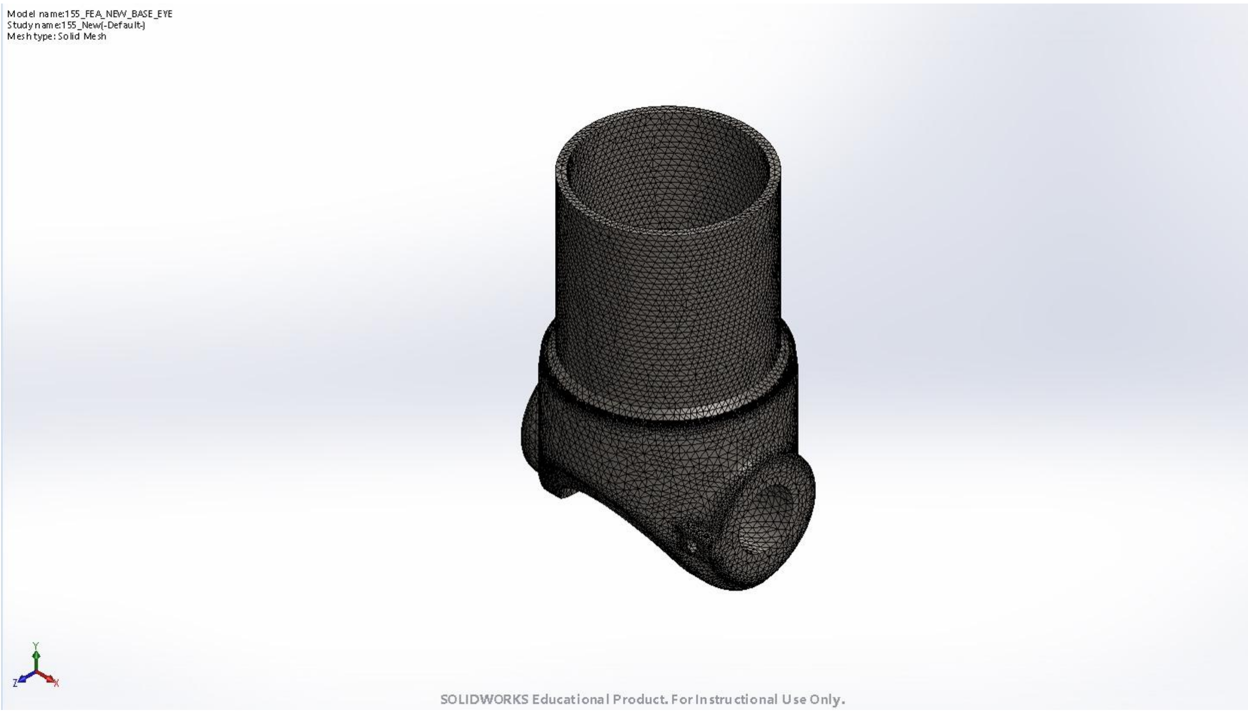


Figure 13. The mesh model of the base eye

Table 3. Mesh information

Mesh type	Solid Mesh
Mesher Used:	Curvature-based mesh
Jacobian points	4 Points
Maximum element size	6 mm
Minimum element size	1.2 mm
Mesh Quality Plot	High
Remesh failed parts with incompatible mesh	On
Total Nodes	378799
Total Elements	239777
Maximum Aspect Ratio	122.68
% of elements with Aspect Ratio < 3	98
% of elements with Aspect Ratio > 10	0.0484
% of distorted elements(Jacobian)	0
Time to complete mesh(hh;mm;ss):	00:00:15

Interpretation of results

The Figure 14. presents the results that have been achieved with the simulation which show the relationship between the loads applied to the system, the geometrical characteristics of the materials, the constraints, and deformations. The results are shown with colors ranging from blue to red that show the minimum and maximum values respectively. As we can see from the figure the proposed form and material offers perfect resistance to the applied pressure and a well distribution of the stress.

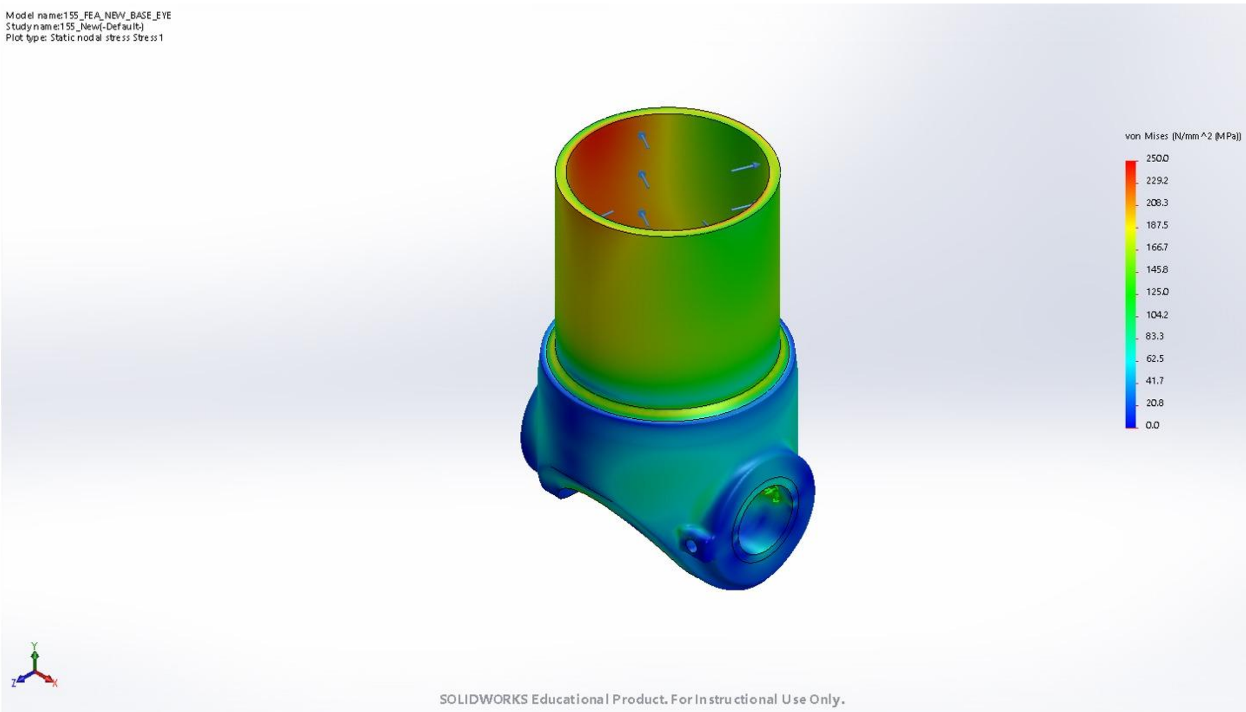


Figure 14. Results of FEA simulation in SolidWorks

V. Conclusion

The product development and redesigning a product is a complex process, with many difficulties during the process. Traditional design methodologies are used for manufacturing constrain the designer's imagination based on the manufacturing processes capabilities and limitations. If we would redesign the product in the present study by relying on physical testing, the process would have been much longer and more expensive. The proposed methodology of FEA simulation removes most of the limitations of conventional product design methodologies by creating the most complex products and in 3D CAD software and transform them into the final product. Traditional manufacturing design restrictions, such as preventing sharp angles, minimizing joint lines, and steady wall thickness should no more be taken into consideration. Using the FEA method the metal capacity of the existing model was reduced by 30.77% where the mass of the product was reduced from 13.0468 kg to 9.07446 kg. FEA method ensured that the obtained model operated normally under the pressure of 18 MPa, thus satisfying the loading constraints and the intended design purposes.

To sum up, simulation enables engineers and designers with little understanding of mathematical analysis, to make decisions development process faster and easier. In this way, more reliable and long-lasting products can be developed.

References

[1]. Churazova A. (2018). Designing More Durable Consumer Products with FEA. https://www.simscale.com/blog/2017/06/durable-consumer-products/
[2]. Cottrell J. A., Hughes T. J. R & Bazilevs Y. (2009). Isogeometric analysis toward integration ofcad and FEA, John Wiley & Sons, Ltd pg
[3]. Finite Element Analysis Software (2020). https://www.autodesk.com/solutions/finite-element-analysis Retrieved on: 19.18.2020.
[4]. Hague R, Mansour S & Saleh N. (2013). Design opportunities with rapid manufacturing. Assembly Automation, 23/4: 346-356
[5]. Kurowski P. M. (2017). Teaching Finite Element Analysis for Design Engineers, SAE International
[6]. Magomedov, I & Sebaeva, Z. (2020). Comparative study of finite element analysis software packages. Journal of Physics: Conference Series. 1515. 032073. 10.1088/1742-6596/1515/3/032073
[7]. Matt L (2018). Mastering SolidWorks. John Wiley & Sons. Pp. 13-57.
[8]. Salonitis K. & Zarban A. S. (2015). Redesign Optimization for Manufacturing Using Additive Layer Techniques. Elsevier B. V. pp. 193-198.
[9]. Visser M. (2016).Solidworks Simulation: Get connected with Contacts. Retrieed on:19.18.2020 https://blogs.solidworks.com/tech/2016/11/solidworks-simulation-get-connected-contacts.html.
[10]. Hristovska E**.**, Correlation of deformed-stressed features and designed properties of the clamp dogs with the carrying structure of the working organ on the excavators, doctoral dissertation, Skopje, June 2000.

Zejnelabedin Aziri, et. al. “The verification of a redesigned part of a telescopic hydraulic cylinder based on finite element analysis.” *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, 18(1), 2021, pp. 31-41.