3D CAD SOFTWARE APPLIED TO TEACHING OF MECHANICAL SYSTEMS DESIGN AND FINITE ELEMENT ANALYSIS FOR IFSP CONTROL AND AUTOMATION ENGINEERING

Caio Igor Gonçalves Chinelato [1]

Data de entrega dos originais à redação em: 20/09/2019
e recebido para diagramação em: 14/02/2020

This works presents the didactic experience of using 3D CAD software Autodesk Inventor® applied to control and automation engineering of IFSP Campus São Paulo, in integrated laboratory VIII discipline. The objective was that the students could understand fundamental concepts of mechanical systems design and finite element analysis through a 3D CAD software. Initially, the students performed didactic exercises to assimilate fundamental concepts of drawing with 3D CAD software and posteriorly, applied the finite element analysis. Finally, the students elaborated a final project, showing the design and finite element analysis of a mechanical system. As results, will be presented some projects developed by students and the performance and efficiency of students learning will be discussed. It can be verified that the didactic experience proposed was very effective to assist students to assimilate fundamental concepts of mechanical systems design and finite element analysis with 3D CAD software.

Keywords: Teaching Engineering. Mechanical Systems Design. 3D CAD Software. Finite Element Analysis..

1 INTRODUCTION

CAD (Computer-Aided Design) is a graphic software essential for engineers who develop mechanical systems, mechanisms and machines. The performance improvement and cost reduction of computational systems in recent years, enable CAD software to become more accessible, cheaper and easier to work (AMARAL and FILHO, 2010). As design and analysis of mechanical systems are topics approached in control and automation engineering, it is very important that the students become familiar with CAD software and finite elements analysis.

There are some works in the literature that reports didactic experiences with CAD software and finite elements analysis applied to engineering teaching. Silva (2011) presents an experience related to teaching drawing in engineering courses, showing the importance of using different didactic resources to promote motivation and encourage the students in the course. In Barbosa and Cheng (2007), the application of CAD Freeware for teaching and learning engineering drawing is shown. Silva et al (2015) demonstrate the application of CAD software and finite element analysis to assist students in carrying out mechanical testing of materials, such as torsions and flexions, on solid mechanics discipline. In Mello (2017), the application of CAD software Autodesk Inventor® is shown didactically, using customization tools, design changes and finite element analysis. Neto et al (2007) demonstrate a simple and didactic finite element analysis application in trusses to obtain displacements, stresses and support reaction for the discipline computational mechanics in engineering. Bandeira and Chivante (2006) present results obtained in learning finite element analysis for civil engineering and the interdisciplinarity with other course topics.

In control and automation engineering of IFSP Campus São Paulo, mechanical systems drawing is a topic approached in semiannual disciplines engineering drawing I (*desenho para engenharia I*) and engineering drawing II (*desenho para engenharia II*), where students learn basics concepts of technical drawing, such as views, sections, perspectives and work with two-dimensional (2D) CAD software. Mechanical systems analysis is a topic approached in semiannual disciplines materials resistance (*resistência dos materiais*), general mechanics (*mecânica geral*) and machine elements (*elementos de máquina*), where students calculate internal stresses in mechanisms and general mechanical systems using fundamental mechanics principles. Posteriorly, these topics are approached together in integrated laboratory VIII (*laboratório integrado VIII*) discipline, where the students work with solid modeling in three-dimensional (3D) CAD software. Therefore, with 3D CAD software, the students can produce complex mechanical systems drawings and with finite elements analysis, the students can verify numerically internal mechanical stresses.

This work presents the didactic experience of using 3D CAD Autodesk Inventor® software applied to control and automation engineering of IFSP Campus São Paulo, in integrated laboratory VIII discipline. The objective was that the students could understand fundamental concepts of mechanical systems design and finite element analysis through a 3D CAD software.

1 - Engenharia de Controle e Automação – Instituto Federal de São Paulo – IFSP Câmpus São Paulo < caio.chinelato@ifsp.edu.br >.

2 3D CAD SOFTWARE

The first CAD systems were developed in the late of 1960s and contained basic 2D geometric entities, such as lines, circles, arcs and polygons. The models contained the part or mechanism edges as well as dimensions, texts and other information that could be found in a manual drawing (LIEU and SORBY, 2009). The main 2D CAD software disadvantage was that could not generate visualization and manipulate 3D Models.

In the early 1980s, CAD software with 3D models were developed. These systems were still limited to using entities such as lines, circles, arcs and were denominated wireframe model. The wireframe model presented ambiguity in representations, because the part or mechanism was not represented as a solid (ZEID, 2005).

With the performance improvement and cost reduction of computational systems, surface modeling techniques were developed. Surface modeling was a wireframe model evolution. This modeling describes mathematically and displays the surfaces between the wireframe model edges. This modeling method is also known as boundary representation. The bounding entities can be planes, cylinders or other 3D surfaces (ZEID, 2005).

Solid modeling is very similar to surface modeling, however it is possible to distinguish between inside and outside of the object. Therefore, it is possible to distinguish an empty box and a brick, for example. This modeling is constructed using constructive solid geometry technique, where the models are composed by standard building blocks in the form of simple solids, such as prisms, cylinders and spheres, called primitives. CAD software based in this modeling allows to compute information of part or mechanism, such as volume, mass, center of mass and other properties (LIEU and SORBY, 2009).

An even more accurate and efficient modeling is the feature-based solid modeling, developed in the mid-1990s. This modeling allows to create more complex parts quicky using standard features, such as extrusions, chamfers, holes and cuts. The most recent modeling is parametric solid modeling, which is feature-based solid modeling that allows the designer to change the part or mechanism dimensions quickly and easily (LIEU and SORBY, 2009).

There are several CAD systems, such as AutoCAD®, SolidWorks® and Catia®. The CAD software used in this work was Autodesk Inventor® (INVENTOR, 2019), because it offers a free student version on its website and IFSP Campus São Paulo has licenses to use the 2017 software version in the laboratory. The student version presents the main resources of full version available in the laboratory, so many students also used the student version for study, training and project development outside the laboratory. Furthermore, Autodesk Inventor® software already has finite element analysis resource, therefore there is no need to purchase other specific software for this analysis (WAGUESPACK, 2014).

It is important to highlight that currently there are several cloud based CAD software that can be executed by a local browser or by a web or mobile application, differently from a traditional CAD software installed on a local computer (AUTODESK, 2020). Examples include Fusion 360®, Tinkercad® and Onshape®.

3 FINITE ELEMENT ANALYSIS

Finite element analysis is an advanced design analysis technique applied to 3D CAD models and used to numeric calculation of interest variables in complex systems. Considering, as an example, a simple mechanical system, such as a circular rod subjected to a stress. Using basic knowledge of physics and solid mechanics, an engineer can determine the points where the internal stresses will present higher or lower magnitude values and make design changes to reduce internal stresses.

Unfortunately, most of the mechanical systems found in engineering applications are more complex and modeled by differential equations whose analytical solution are not trivial. Therefore, finite element analysis is a numeric method for solving differential equations, where the analyzed object is divided into elements joined by nodal points and which together constitute a mesh (CHAPRA and CANALE, 2016). The numerical method is applied to each mesh element. The computer compiles the results of all mesh elements and generates the final result of analyzed object. It is important to highlight that, as it is a numerical solution, the result is an approximation. Finite element analysis is typically applied to several engineering problems, such as mechanical, fluidic, thermal and electromagnetic systems.

Finite element analysis consists of the following steps (LIEU and SORBY, 2009; MELLO, 2017):

- Geometric model development: The geometric model of part or mechanism is developed in 3D CAD software;
- Constraints application: if the part or mechanism is subjected to an external force and there are no constraints, it will move in space. Therefore, in finite element analysis, the object must be fixed in space by a constraint;
- Loads application: after constraints have been specified, loads can be applied, such as forces, pressures and/or torques;
- Meshing process: the choice of mesh elements is a very important point in finite element analysis. Generally, small elements generate more accurate results, however, the simulation takes longer.

Besides that, in points where the part or mechanism has curvatures or internal stresses vary abruptly, the elements size must be reduced and in points where internal stresses are relatively static and vary little, the elements size can be increased to make the simulation faster. Fortunately, modern systems that performs finite element analysis makes this meshing process simpler. The user specifies an element nominal size and the software generates the elements and the adjustments in points with curvature, for example. However, meshing process involves experience and practice;

- Results computation: in the case of mechanical systems analysis, the numerical results generated by finite element analysis are stresses, displacements, deformations and safety factor based on the applied boundary conditions, given by the constraints and loads;
- Results investigation: the results are shown numerically and graphically in the geometric model developed;
- Design modification: according to results verified in the previous step, the designer makes design structural changes to improve the mechanical characteristics;

4 METHODOLOGY AND DEVELOPMENT

The work objective was that the students could understand fundamental concepts of mechanical systems design and finite element analysis through a 3D CAD software. The methodology was elaborated so that the objectives were achieved.

The students of integrated laboratory VIII discipline from control and automation engineering of IFSP Campus São Paulo were divided in groups with an average of twelve students per group and each group realized the discipline for five weeks taking six classes of forty five minutes on a single day of the week. Initially, the students performed didactic exercises to assimilate fundamental concepts of drawing with 3D CAD software and posteriorly, they used the finite element analysis from 3D CAD software. Finally, the students elaborated a final project showing the design and finite element analysis of a mechanical system proposed by them. The students evaluation was applied based on the accomplishment of all proposed didactic exercises and based on the final project report.

4.1 Didactic Exercises

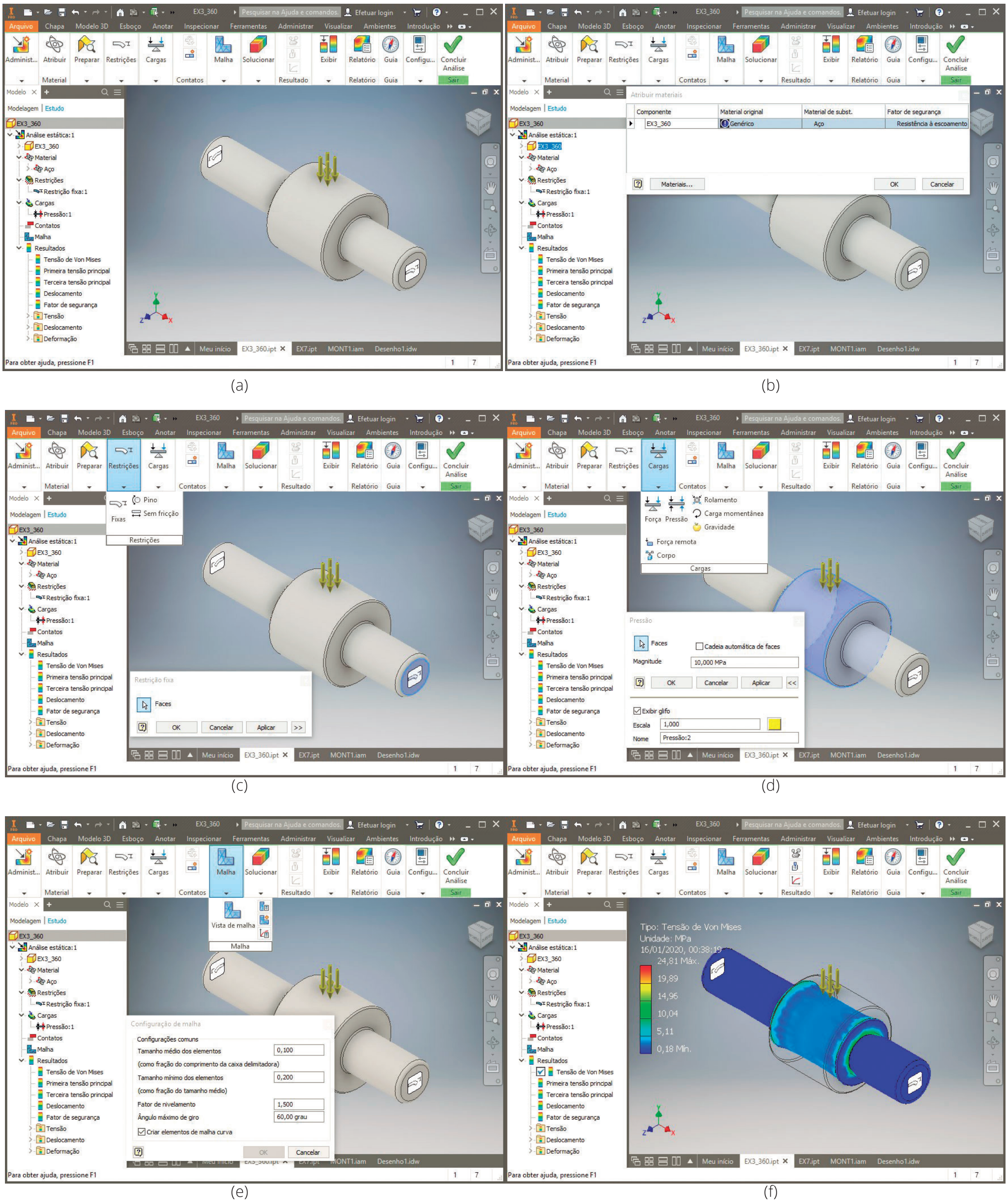
In the first two weeks of the discipline, didactic exercises were proposed to students work with part, assembly and drawing files. The part files consist of solid modeling applied in individual parts of mechanical systems, such as a screw for example. These files require more time to learn and more exercises, because the students need to become familiar with 2D sketching tools and posteriorly with 3D features. The assembly files consist by sets of parts that are assembled using geometric relationships and specific constraints and when integrated form a complete mechanical system. Drawing files consist by drawing sheets of parts and assemblies. In these drawing sheets it is possible to edit the layout, mechanical system views and dimensioning (WAGUESPACK, 2014).

Basic and intermediate level exercises were elaborated and that used several software resources, so that the students acquired experience for the final project development. It is important to highlight that many students had access to the free student version of Autodesk Inventor®, as mentioned in section 2, and therefore they were also able to study and practice the exercises outside the laboratory.

In the third week of the discipline, the students worked with the finite element analysis resource of Autodesk Inventor® software. Basically, the analysis consists by steps mentioned in section 3: geometric model development, constraints application, loads application, meshing process, results computation and results investigation. Besides that, it is also possible to change the material characteristics of the mechanical system. As finite element analysis results, all stresses, deformations, displacements and safety factor of the mechanical system are generated (HIBBELER, 2019). The analysis was done in some part and assembly files of exercises previously proposed.

Figures 1a, 1b, 1c, 1d, 1e and 1f show the steps for finite element analysis using Autodesk Inventor® software applied in a part proposed as a didactic exercise. Figure 1a shows the part. At the top of the software, all parameters to be determined in finite element analysis are shown. After the parameters have been determined, they appear on the left side of the software. Figure 1b shows the material selection (Atribuir materiais). The software provides a library with several materials with predetermined physical characteristics. However, the user can add new materials that do not exist in the library. Figure 1c shows the constraints application (Restrições). The constraints types are fixed constraint (Fixas), that restrict movement in all direction for the selected geometry, pin constraint (Pino), that restrict movement to the radial, axial and tangential directions according to the defined options and frictionless constraint (Sem fricção), that prevent a face movement in perpendicular direction to a face. Figure 1d shows loads application. It can be applied force (Força), pressure (Pressão), bearing load (Rolamento) in cylindrical faces, moment (Carga momentânea), gravitational force (Gravidade), remote force (Força remota) at any part location and body load (Corpo). Figure 1e shows meshing process, where the main numerical meshes parameters are detailed, such as average element size (Tamanho médio dos elementos), minimum element size (Tamanho mínimo dos elementos), grading factor (Fator de nivelamento) and maximum turn angle (Ângulo máximo de giro). Besides that, it is possible to have a mesh overview in the part, to control the mesh size in each point of the part and to determine numerical convergence parameters. Finally, Figure 1f shows the results. The results are shown numerically and graphically through a color chart for each range of values and the maximum and minimum values of each variable are indicated. The analyzed variables are Von Misses stress (Tensão de Von Misses), first and third main stress (Primeira e Terceira tensão principal), displacement (Deslocamento), deformation (Deformação) and safety factor (Fator de segurança). After the results analysis, a report in .rtf or .html format files can be automatically generated with all parameters and results of finite element analysis.

Figure 1 – Finite element analysis applied in a part proposed as a didactic exercise. (a) Part proposed. (b) Material selection. (c) Constraints application. (d) Loads application. (e) Meshing process. (f) Results



At the end of first three weeks of the discipline, the students showed all the didactic exercises to discipline's teacher so that they could be evaluated.

4.2 Final Project

After performing the didactic exercises to assimilate fundamental concepts of drawing with 3D CAD software and finite element analysis, a final project was proposed. In this project, the students developed the design and the finite element analysis of a mechanical system proposed by them. In the design task, the students elaborated the mechanical system part or assembly file together with the drawing file. In the finite element analysis task, the students evaluated and discussed the influence of different variables on the developed mechanical system, such as different types of stresses, materials and geometries. Therefore, they performed a significant number of numerical simulations to verify the influence of these variables in the mechanical system developed. Finally, they developed a final report describing the entire final project, which was evaluated by discipline's teacher.

The projects were developed in pairs or trios and in the last two weeks of the discipline. It is important to highlight again the fact that students had access to the student version of Autodesk Inventor® software and part of the project development could be done outside the laboratory.

5 RESULTS AND DISCUSSIONS

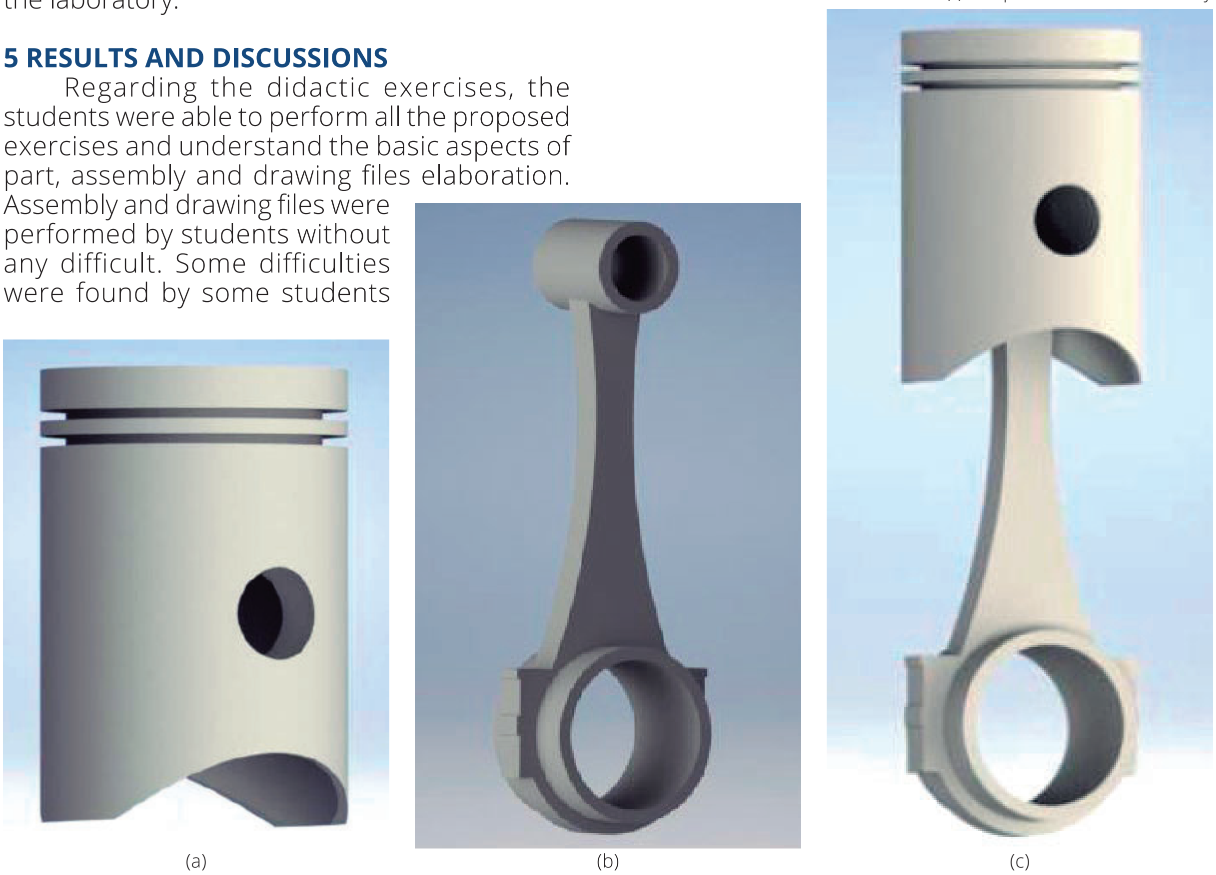
Regarding the didactic exercises, the students were able to perform all the proposed exercises and understand the basic aspects of part, assembly and drawing files elaboration. Assembly and drawing files were performed by students without any difficult. Some difficulties were found by some students in part files due to the greater number of commands and steps for solving the exercises. However, nothing that would significantly compromise the learning and development of the classes. The finite element analysis was also performed efficiently. The students performed all steps mentioned in the previous section, were able to understand the numerical method operation and how to analyze the results. These didactic exercises were fundamental for the students acquire experience for the final project development.

After performing all exercises, students developed the final project. Several interesting and widely applied mechanical systems were proposed. In all cases, the students were able to develop the design and the finite element analysis of the proposed mechanical system.

Some projects developed by students are shown below. The part or assembly files, drawing files and some finite element analysis results are shown.

The first project was the mechanism composed by the piston and connecting rod of an internal combustion engine. The parts piston and connecting rod and the complete mechanism assembly are shown respectively in Figures 2a, 2b and 2c. In Figures 3 and 4, the piston and connecting rod drawing sheets are shown respectively.

Figure 2 – Mechanism composed by the piston and connecting rod of an internal combustion engine. (a) Piston part. (b) Connecting rod part. (c) Complete mechanism assembly



(a) (b) (c)

Figure 3 – Piston drawing sheet

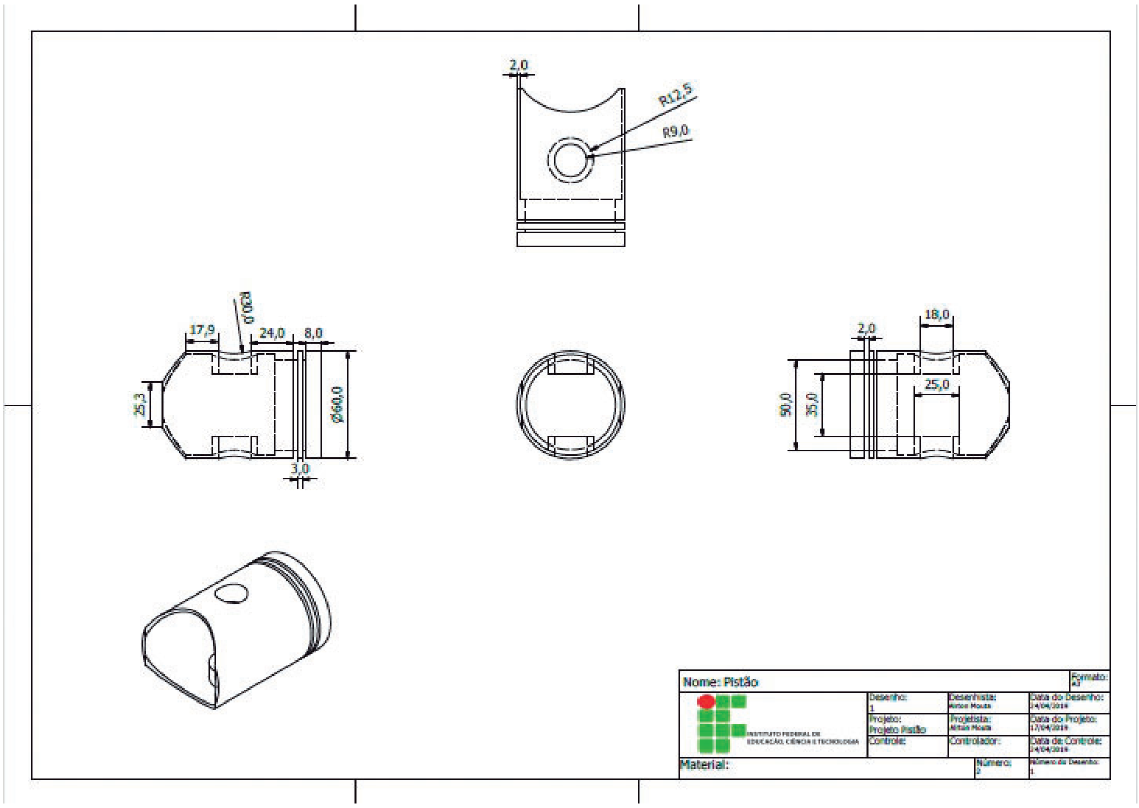
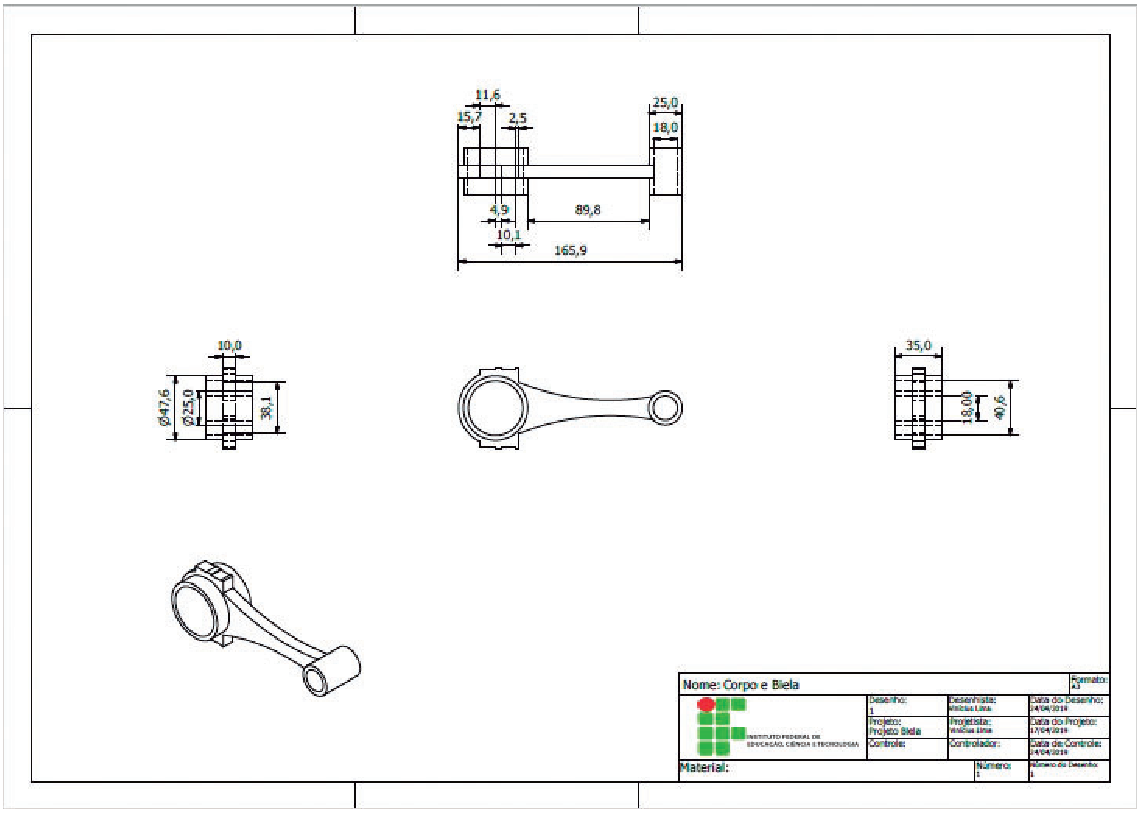
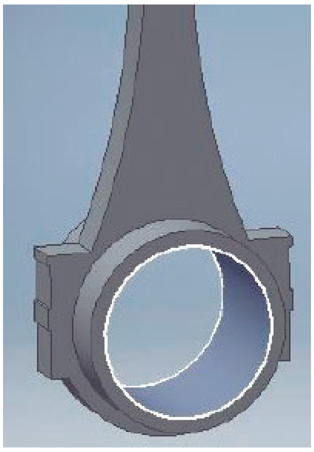


Figure 4 – Connecting rod drawing sheet



Posteriorly, the students performed several finite element analyzes testing different types of materials and stresses for the mechanism. The selected analysis simulates a stress that the mechanical system is subjected during the explosion of gas and fuel mixture inside the engine and considering the parts made of 6061 aluminum. Figure 5a shows the connecting rod constraint on the crankshaft, Figure 5b shows the connecting rod movement constraint related to the piston and Figure 5c shows a load of 3.040 MPa applied to the piston due to combustion inside the engine. Figures 6a, 6b, 6c and 6d show the finite elements analysis results.

Figure 5 – Constraints and loads applied in the mechanism composed by the piston and connecting rod of an internal combustion engine. (a) Connecting rod constraint on the crankshaft. (b) Connecting rod movement constraint related to the piston. (c) Load of 3.040 MPa applied to the piston due to combustion inside the engine



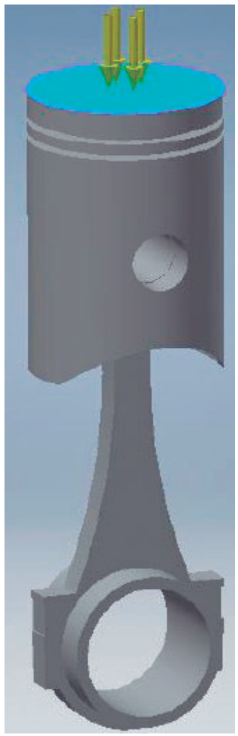
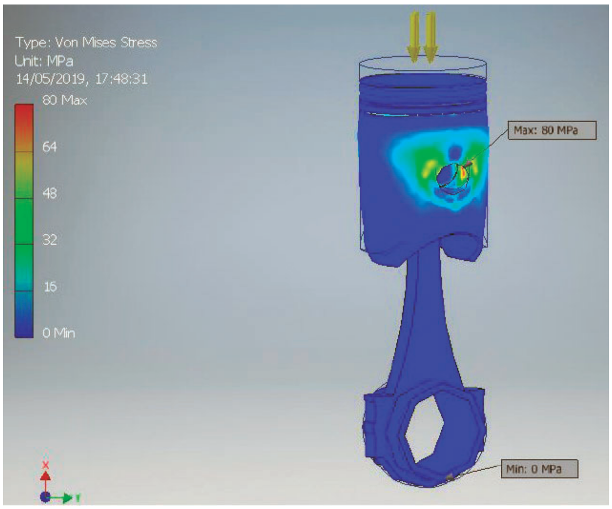
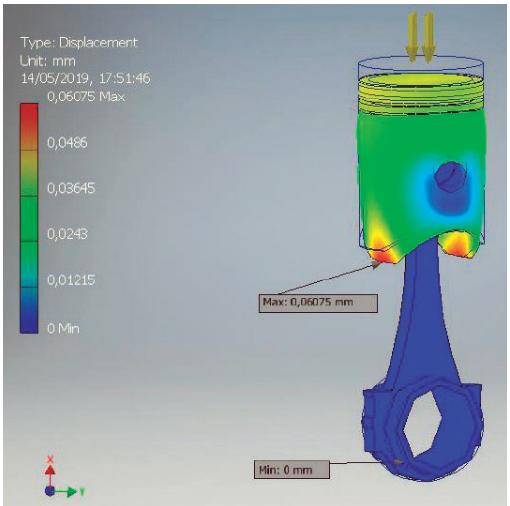


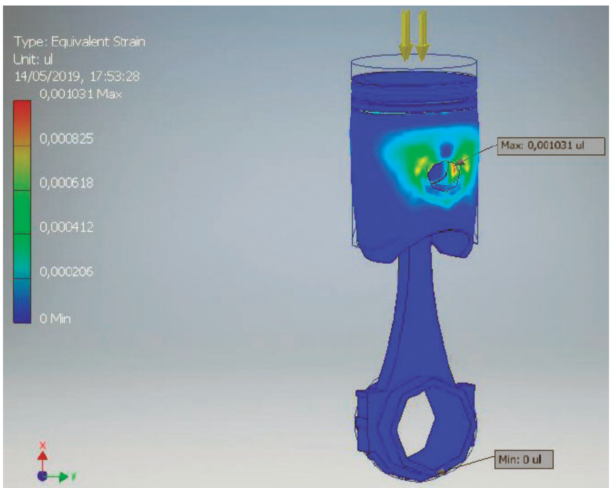
Figure 6 – Finite element analysis results for mechanism composed by the piston and connecting rod of an internal combustion engine. (a) Von Misses stress. (b) Displacement. (c) Equivalent strain. (d) Safety factor



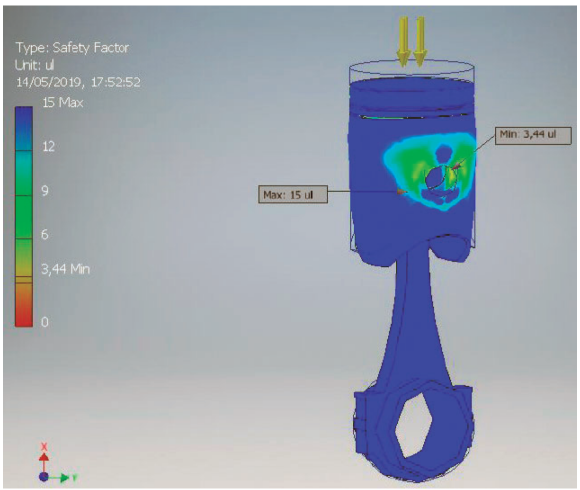
(a)



(b)



(c)



(d)

The second project was a mechanical vise. The mechanical vise assembly is shown in Figure 7 and the mechanical vise drawing sheet is shown in Figure 8.

Figure 7 – Mechanical vise assembly

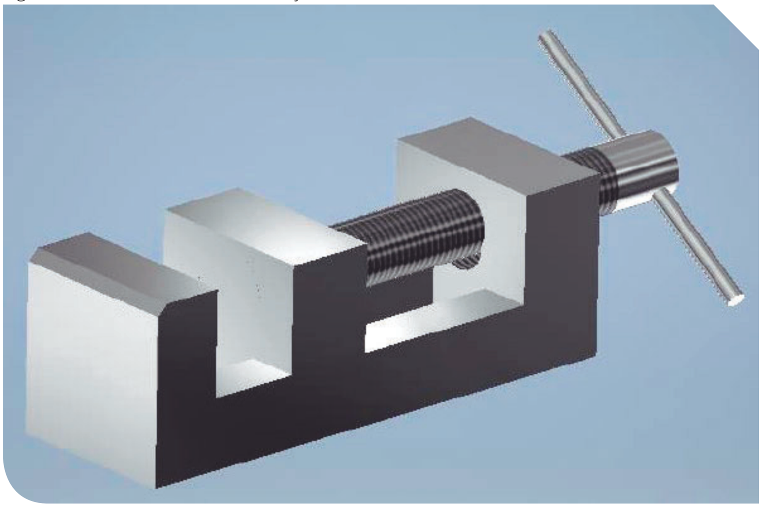
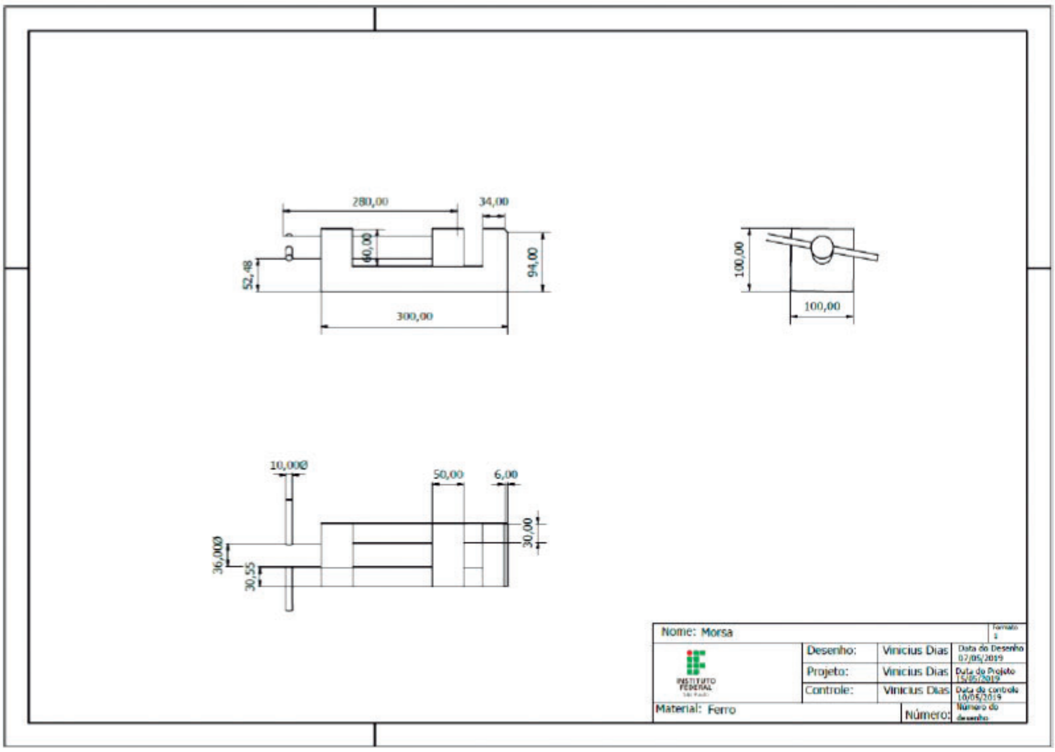


Figure 8 – Mechanical vise drawing sheet



Several finite element analysis were performed again, testing different types of materials and stresses for the mechanism. The selected analysis simulates a torsion stress that the mechanical vise is subjected when the user applies a torque of 5 N·m on its axis and considering the mechanical vise made of steel. In Figure 9, the torsion stress and the constraint applied to one face of the mechanical vise are shown. Figures 10a, 10b, 10c and 10d show the finite element analysis results, as in the previous project.

Figure 9 – Torsion stress of 5 Nm and constraint applied to one face of the mechanical vise

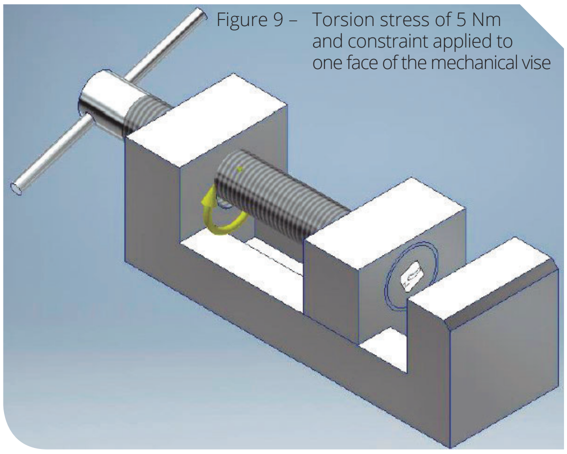
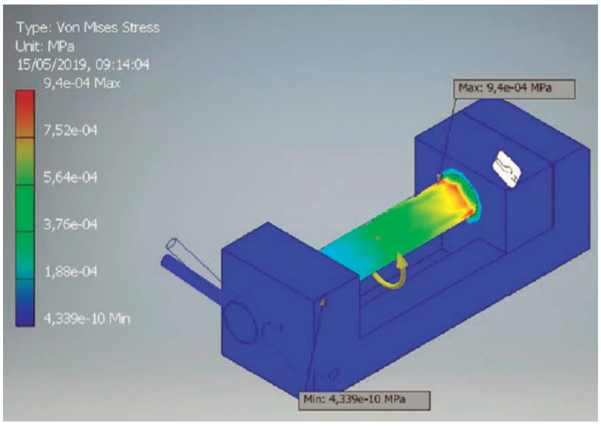
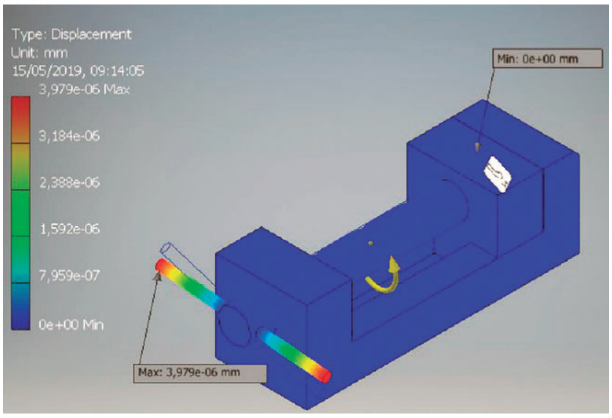


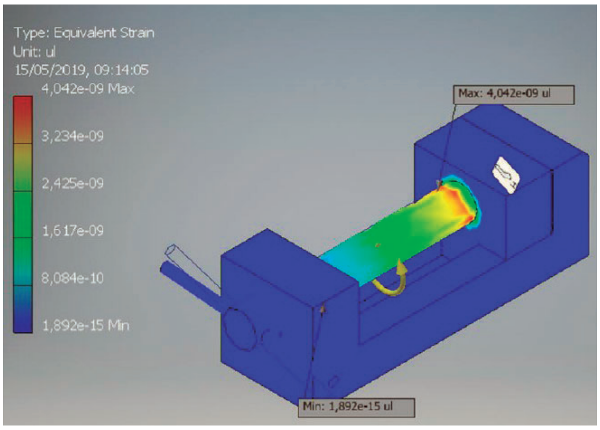
Figure 10 – Finite element analysis results for mechanical vise. (a) Von Misses stress. (b) Displacement. (c) Equivalent strain. (d) Safety factor



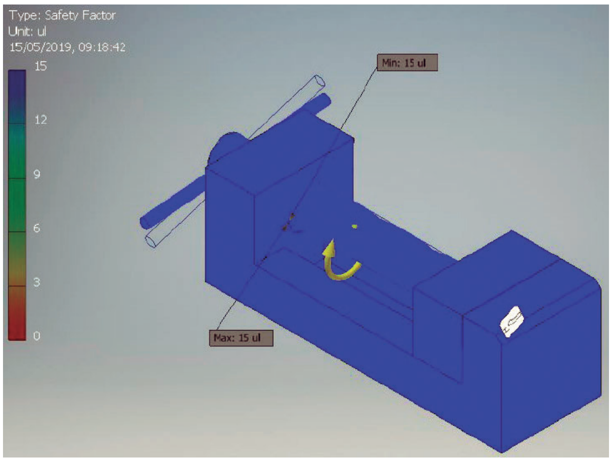
(a)



(b)



(c)



(d)

The third project was an automotive wheel. The automotive wheel is shown in Figure 11 and the automotive wheel drawing sheet is shown in Figure 12.

Figure 11 – Automotive wheel

Several finite element analysis were performed again, testing different types of materials and stresses for the mechanism. The selected analysis simulates a stress of 1.5 MPa applied perpendicularly to the wheel and considering the wheel made of steel. In Figure 13, the stress and the constraint applied to the wheel's holes are shown. Figures 14a, 14b and 14c show the finite element analysis results, as in the previous project.

Figure 12 – Automotive wheel drawing sheet

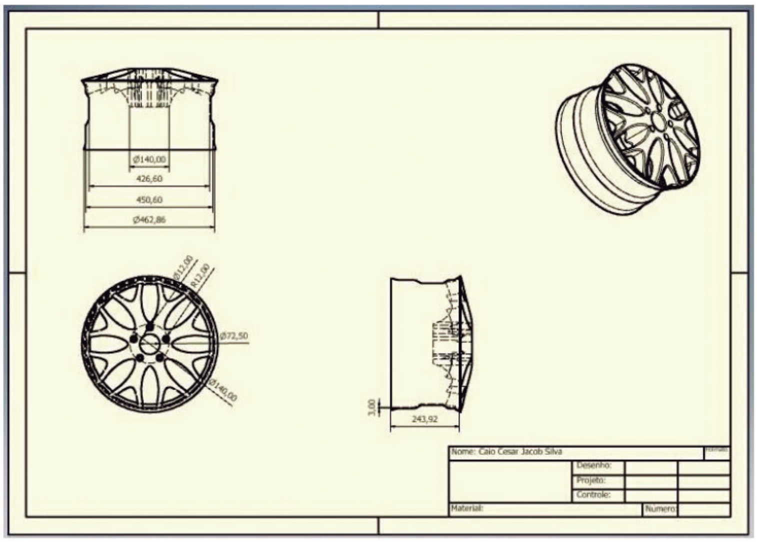


Figure 13 – Stress of 1.5 MPa applied perpendicularly to the wheel and the constraint applied to the wheel's holes

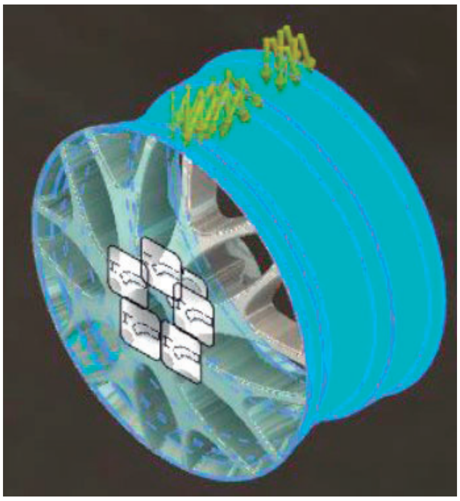
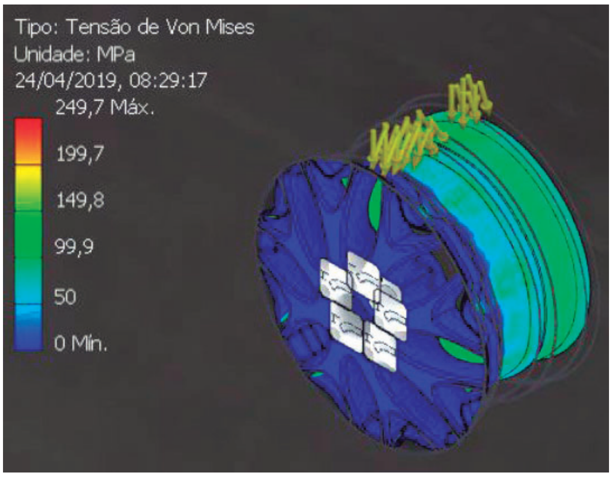
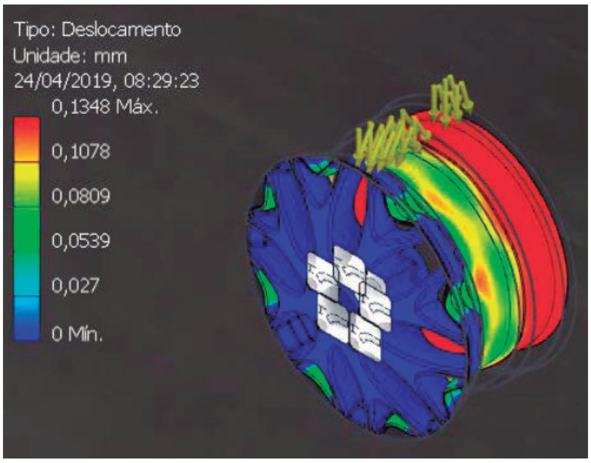


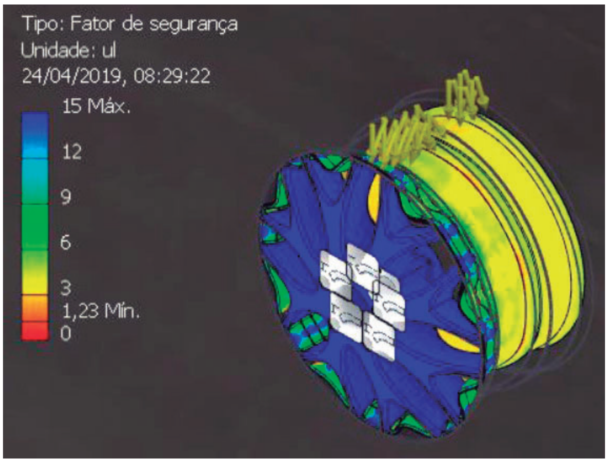
Figure 14 – Finite element analysis results for automotive wheel. (a) Von Misses stress (*Tensão de Von Misses*). (b) Displacement (*Deslocamento*). (c) Safety factor (*Fator de segurança*).



(a)



(b)



(c)

6 CONCLUSIONS

This work presents the didactic experience of using 3D CAD Autodesk Inventor® software applied to control and automation engineering of IFSP Campus São Paulo, in integrated laboratory VIII discipline. After performing the didactic exercises and the final project, the students could understand fundamental concepts of mechanical systems design and finite element analysis through a 3D CAD software.

Regarding the didactic exercises, the students were able to perform all the proposed exercises and could understand the basic aspects of part, assembly and drawing files elaboration and finite element analysis. It is important to highlight that the choice of basic and intermediate level exercises and exercises that uses several software resources were very important to make the learning process faster. Advanced exercises would take a long time to be done and exercises that uses few software resources would not allow students to explore the software with more details.

Regarding the final projects, the students developed several interesting and widely applied mechanical systems. In all cases, the students were able to develop the design and the finite element analysis of the proposed mechanical system. A very important aspect is that many students reported that the project was an essential complementary tool in learning, because only with the didactic exercises they still had some doubts and questions that were explained during the project development.

Finally, it is important to highlight the importance of 3D CAD software and finite element analysis as a didactic complement to disciplines related to technical drawing and mechanical systems analysis.

7 ACKNOWLEDGMENT

The author acknowledges IFSP Campus São Paulo for supporting this work.

REFERENCES

AMARAL, R. D. C., FILHO, A. C. P. A evolução do CAD e sua aplicação em projetos de engenharia. In: IX Simpósio de Mecânica Computacional (SIMMEC), São João Del Rei, MG, 2010.

AUTODESK. Software CAD online e CAD em nuvem. Available in: < https://www.autodesk.com.br/solutions/cloud-based-online-cad-software>. Access in: 15 jan. 2020.

BANDEIRA, A. A., CHIVANTE, M. R. P. A interdisciplinaridade do ensino de elementos finitos no curso de engenharia civil: aprendizagem e aspectos pedagógicos utilizando a ferramenta computacional ANSYS. In: XXXIV Congresso Brasileiro de Educação em Engenharia (COBENGE), Passo Fundo, RS, 2006.

BARBOSA, C. C., CHENG, L. Y. Uso de CAD freeware no ensino de engenharia. In: VII International Conference on Graphics Engineering for Arts and Design (GRAPHICA), Curitiba, PR, 2007.

CHAPRA, S. C., CANALE, R. P. Numerical methods for engineers. New York: McGraw-Hill, 2016.

HIBBELER, R. C. Mechanics of Materials. Pearson, 2019.

INVENTOR. Autodesk Inventor. Available in: < https://www.autodesk.com.br/products/inventor/overview>. Access in: 17 set. 2019.

LIEU, D. K., SORBY, S. Visualization, modeling and graphics for engineering design. New York: Delmar Cengage Learning, 2009.

MELLO, V. S. Utilização das ferramentas iLogic e iPart do Autodesk Inventor na confecção de conjuntos mecânicos. Guaratinguetá, SP, 2017. Undergraduate final project.

NETO, G. C. S., LOPES, R. C., LOPES, A. P. O método dos elementos finitos em treliças planas na disciplina de mecânica computacional. In: XXXV Congresso Brasileiro de Educação em Engenharia (COBENGE), Curitiba, PR, 2007.

SILVA, E. C. Ensino do desenho nos cursos de engenharia e design. In: XXXIX Congresso Brasileiro de Educação em Engenharia (COBENGE), Blumenau, SC, 2011.

SILVA, J. R. S., MÉNDEZ, C. A. C., JUNIOR, H. M. T., SANTOS, M. D., RODRÍGUEZ, A. S. M. Projeto e análise dos resultados em elementos finitos de equipamentos didáticos para disciplina mecânica dos sólidos. In: Congresso Técnico Científico da Engenharia e da Agronomia (CONTECC), Fortaleza, CE, 2015.

WAGUESPACK, C. Mastering Autodesk Inventor 2014 and Autodesk Inventor LT 2014. Sybex, 2014.

ZEID, I. Mastering CAD/CAM. New York: McGraw-Hill, 2005.