UNIVERSITAT POLITÈCNICA DE VALÈNCIA

Interpretation of the results obtained by Finite Element Analysis (FEA) in SolidWorks

Surname, name	Balart Gimeno, Rafael Antonio (rbalart@mcm.upv.es) Quiles Carrillo, Luís Jesús (luiquic1@epsa.upv.es) Néstor Montañés Muñoz (nesmonmu@upvnet.upv.es)
Department	Department of Mechanical and Materials Engineering (DIMM)
Center	Escuela Politécnica Superior de Alcoy (EPSA) Universitat Politècnica de València (UPV)

1 Summary of the key issues

This article focuses on the potential of **Finite Elements Analysis** (FEA) as **complementary tool** in the design and development of an engineering part with SolidWorks. This article aims to give a **critical interpretation** of the results a Finite Element Analysis tool can give.

2 Introduction

Design, calculation and development of engineering parts and assemblies is not easy. With regard to engineering, **Computer Aided Design** (**CAD**) tools and **Computer Aided Engineering** (**CAE**) tools have become important as they can positively contribute to **reduce** the "**time-to-market**" thus, leading the company to a privileged position. With regard to CAE tools, it is worthy to note the increasing use of "**Finite Elements Analysis**" (**FEA**) tools as they allow the simulation of a designed part or assembly in similar conditions to service (geometry, supports and fixtures, connections, external loads, and so on). The correct use of these tools avoid continuous calculus iterations, thus leading to a remarkable decrease in the "time-to-market"[1].

Nevertheless, these tools are just so, "**tools**" that can help the engineer but they never can substitute the own engineering process. These tools require some **INPUTS** to make calculations and simulations and after these, a series of **OUTPUTS** are provided to the engineer. The **quality** of the obtained results or **OUTPUTS** is directly related to the quality of the **INPUTS** (geometry, manufacturing tolerances, fixtures, external loads, materials, and so on). For this reason, these **CAE tools** will never substitute engineers but they are very powerful to assist engineering during the whole engineering process. It is important to remark that many industrial **software packages** combine both **CAD** and **CAE tools** thus making easy design and finite elements analysis. Today it is possible to find a wide range of software packages such as **ANSYS, SolidEdge, SolidWorks, MatLab, Abaqus, Comsol**, etc. which include powerful FEA tools to assist engineers in the design and development process [2-4].

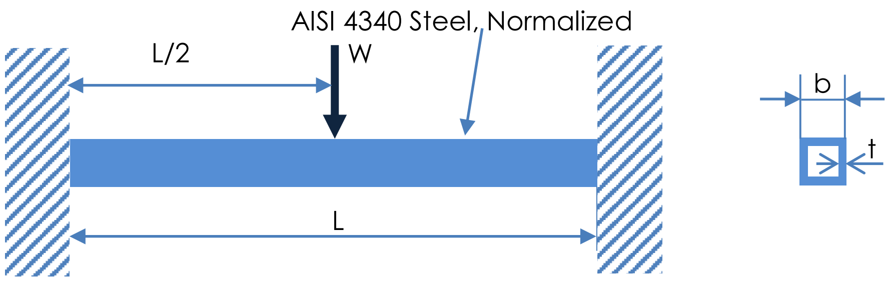
3 Objectives

Once you have read this article, you will be able to:

- **Define** the **INPUTS** for a Finite Element Analysis (FEA) using the Simulation Add-In of SolidWorks.
- **Analyse** the **OUTPUTS** that the FEA tool in SolidWorks provides from a critical point of view.

4 Development

First let's define the problem. This consists on a single beam fixed at both ends with a total length of 1 m (L) with a squared section (b = 5 cm) and a wall thickness of 2 mm. This beam is intended to support a load of 20000 N concentrated at the centre edge. The selected material for this beam is an AISI4340 steel, normalized. Design restrictions stablish that only elastic deformations can occur and a Factor of Safety (FoS) of 1,4 must be considered.



First you must try to construct this beam in SolidWorks for further analysis. Once you have done it, you will be able to solve it by Finite Element Analysis (FEA).

4.1. Finite Element Analysis with SolidWorks.

You need the “**Simulation**” Add Inn available in SolidWorks. If you click on the “**Simulation**” label, then you will be able to choose the “**Study Advisor**” which will guide you in all the stages or just select the “**New Study**” to select the appropriate study without any guidance.

We are going to analyse this problem in static conditions, so that, you can click on “**New Study**” and select “**Static**” from the list and then, write the desired name for this study (for example **Initial Study**) and, finally, click on the green check arrow as indicated in Figure 1.

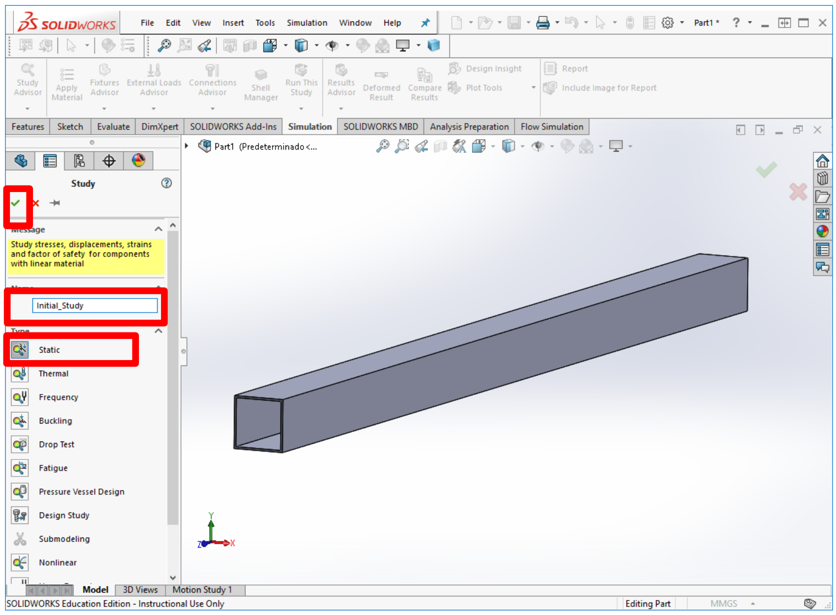
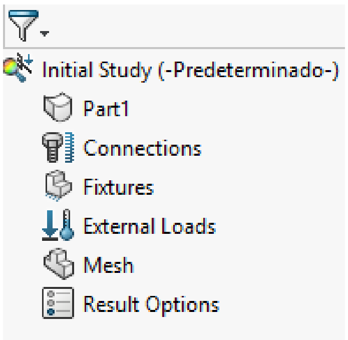


Figure 1. Selecting a “Static” study for FEA analysis.

A series of elements appear in the left side of the window. These elements are essential to define the problem in terms of fixtures, materials, loads, etc.



Part1 corresponds to the beam you have previously sketched

Connexions are used for assemblies

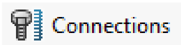
Fixtures are used to define how the beam (or engineering part) is supported

External Loads will be used to define external loads (forces, stresses, gravity, etc.)

Mesh to define the dimensions of the mesh which will divide a complex design in a finite number of small problems

Results Options to evaluate the OUTPUTS of the simulation

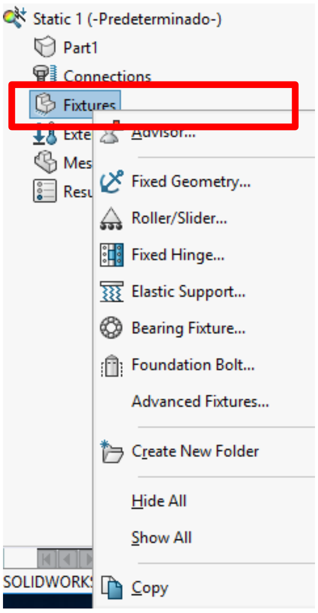
The first step for a FEA simulation is to **define the material**. To this, right-click on "**Part1**" and a menu will be displayed. To assign the material, click on "**Apply/Edit Material**" and you will have access to the **Materials Database** of SolidWorks. Then you have to search the database and when you find the corresponding material, click on "**Apply**" and "**Close**". Before closing this database, you will see all mechanical parameters needed for calculation. In this particular case, an **AISI 4330 Steel, normalized**, it is worthy to highlight its "**Yield strength**" with a value of **710 N/mm²** (MPa). This is an important value for the optimization process we will discuss later. Now try selecting this steel in SolidWorks and check the yield strength value.

As previously indicated, **connections**, 

are used for engineering part assemblies and are not applicable to this beam. Then you will have to define the Fixtures to indicate how this model (beam) is supported. In this case, the beam is fixed at both ends. To indicate this situation in SolidWorks, right-click on the Fixture icon 

, and a pop-up menu will be displayed. This menu shows all types of "**Fixtures**" you can use in SolidWorks: Fixed Geometry, Roller/Slider, Fixed Hinge, and others. For this particular beam, the problem says it is fixed at both ends. For this reason, you will have to select the "**Fixed Geometry**" icon 

.



Then a menu with several options will appear at the left side. Now you just have to click on the surfaces you want to fix and they will be displayed in the blue rectangle. Maybe you will need to make some zoom to appropriately select the face you want to fix. On the other hand, probably you will need to rotate the model to select the opposite face as it is indicated in Figure 2. After this, the beam will behave as a beam fixed at both ends. Once you have finished, you can click on the green check arrow and the Fixtures will be displayed as GREEN ARROWS.

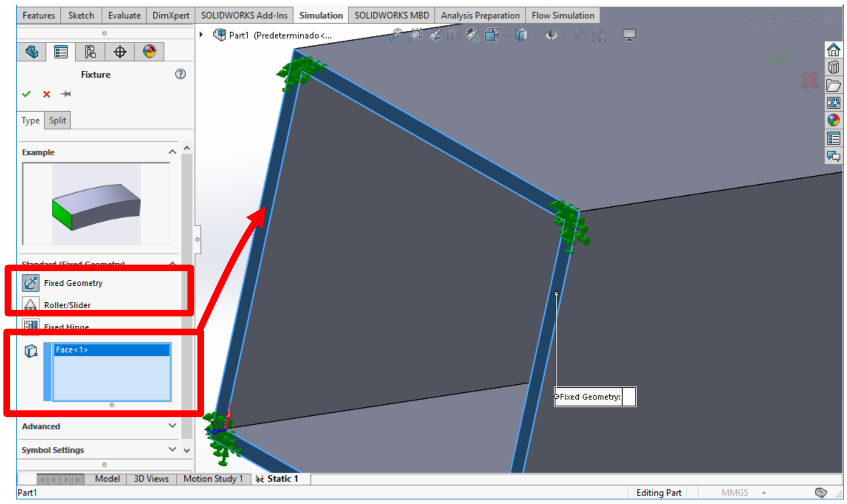
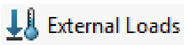
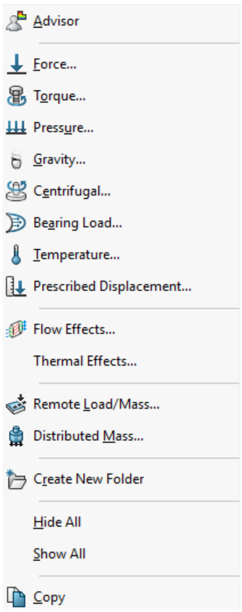


Figure 2. Applying fixtures to an engineering part. In this example, a "Fixed Geometry" is defined in both ends of a beam.

The following step is to define de externals loads. As it has been indicated, the beam is going to support a force of 20000 N in the center. To represent this with SolidWorks, right-click on the "**External Loads**" icon  and a pop-up menu with different possibilities will be displayed.



You can use the "**Advisor**" option if you are not sure about defining External Loads. In addition, you will see a wide range of external loads such as forces, torques, pressure, gravity, and so on. In this case, you are going to **select a force** to be applied. To this, you click on the **Force... icon** and its corresponding menu will be displayed at the left side. Then you check on the "**Force**" icon.

Then you will have to indicate where the force will be applied, an edge, a face, a point, uniformly distributed, etc. In this particular case, you select the edge in the center of the beam as shown in Figure 3. If the default direction is not the appropriate, then you will have to "**select**" the adequate direction. To achieve this, you click on "**Selected direction**" and after this you click on an edge with the desired direction. Finally, if the orientation is not appropriate, you will have to click on "**Reverse direction**" and indicate the value of the applied force. In this case, 20000 N. All this procedure is summarized in Figure 3 with several boxes.

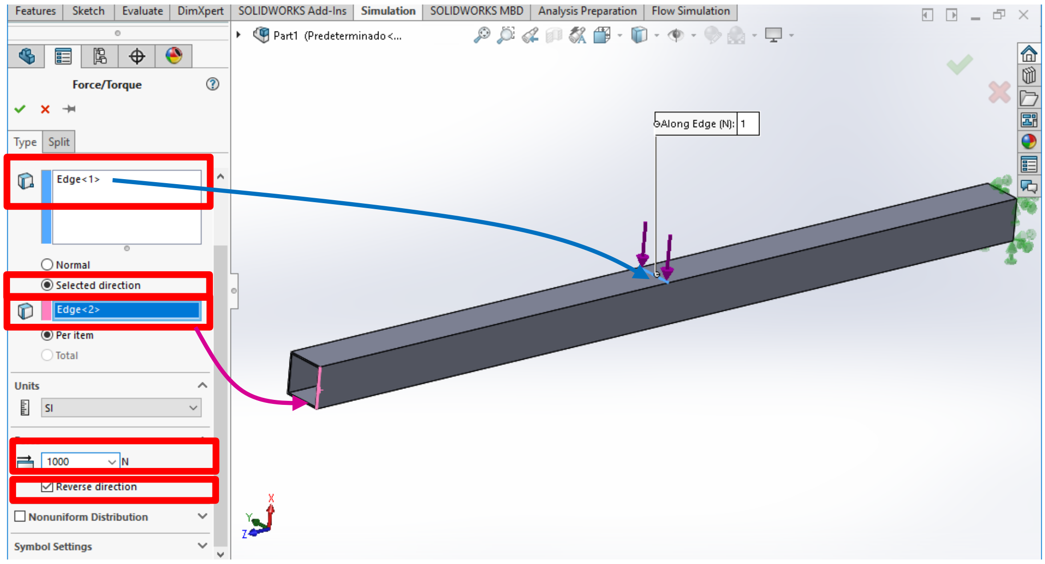


Figure 3. Applying External Loads to a beam fixed at both ends. The force is applied along the center edge.

The model is almost ready for the simulation. Now you have to mesh it. To this, you can right-click on the **Mesh icon**, 

and can explore different possibilities. The simplest procedure if you are not an expert in meshing is to select the **"Create Mesh …"** option. A new menu will be displayed at the left side and you can select the mesh density by moving the blue arrow towards to "**Coarse**" or "**Fine**". The finer mesh density involves higher computational times. As this is a relatively simple model, you can select the "**Finest**" mesh density and click on the green check arrow. Then the meshed model will be displayed and will look like Figure 4.

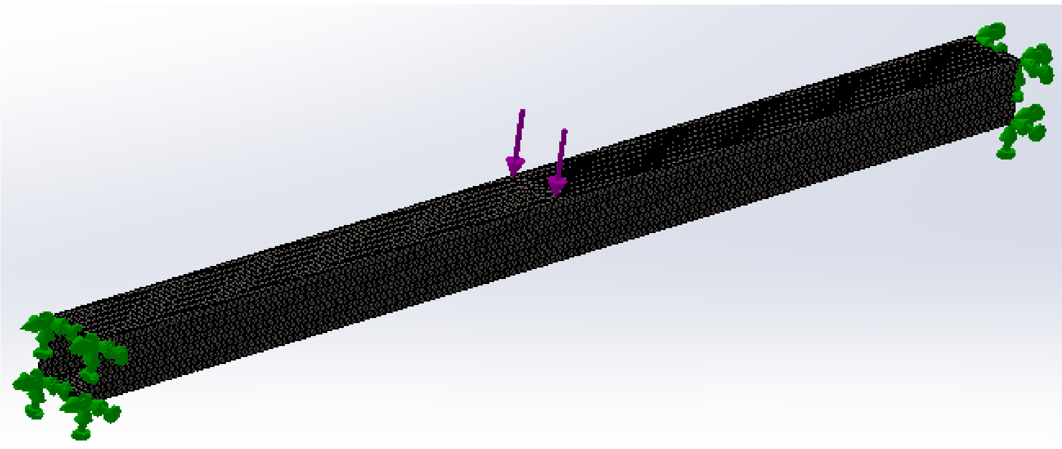
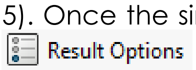


Figure 4. General appearance of the meshed beam ready for simulation.

Finally, you can run the simulation by clicking on the "**Run This Study**" icon (Figure 5). Once the simulation has finished and all calculations are done, the **Results Options**, 

will include several plot results: stresses, displacements and strains as it can be seen in Figure 6a (stress Von Misses) and Figure 6b for displacements.

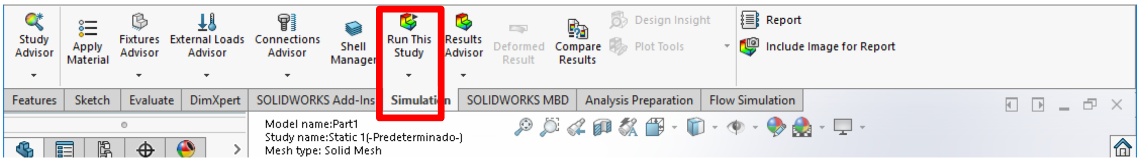


Figure 5. Running the simulation by Finite Element Analysis-FEA in SolidWorks.

a)

b)

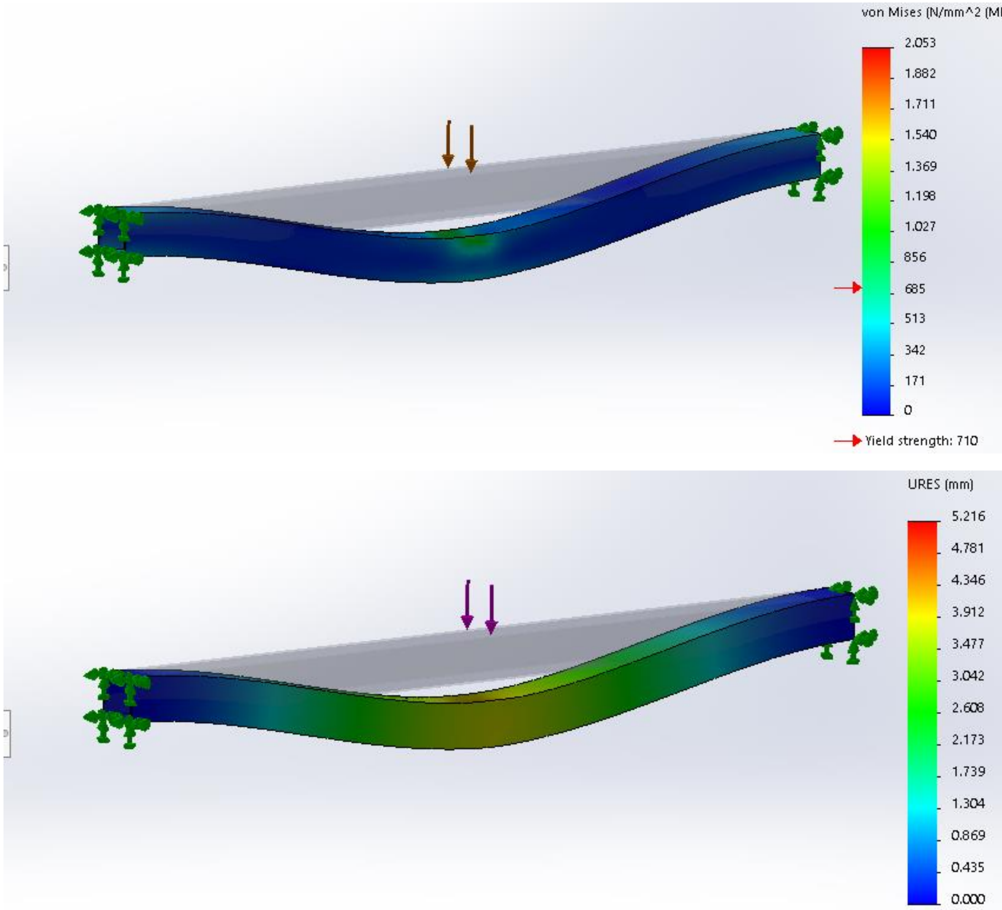


Figure 6. Results for Finite Element Analysis (FEA) in Solid Works. a) Stress plot and b) displacements plot.

As we can see, the FEA tool in SolidWorks give colourful graphics and 3D representations. Usually, we could think these results are correct as they have been calculated by a computer. But as an engineer **you should be critical** with the obtained results.

As you can see, the maximum deflection is **5.216 mm** as observed in Figure 6b. With regard to the **stress plot**, it is worthy to note, that some areas are subjected to higher stresses than the yield strength. This indicates plastic deformation occurs and, potentially, failure can occur.

If we take into account the real stress values, some areas are subjected to more than 1500 MPa (green colour) and, even more, some areas are subjected to almost 2000 MPa. If we go back to the properties of an AISI 4340 steel, its tensile strength is 1110 MPa and as we can see in Figure 6a, not only the yield strength is reached but also extremely high stresses of 2000 MPa are obtained, thus indicating **the beam will break**.

So that, the **deflections are wrong** since the beam will break working in the defined conditions. So, despite the graphical results are very colourful and interesting, you must give **coherence** to these results, and conclude they are wrong. In some cases, they could be quite accurate and coincident with the real situations but in other cases, as this, the obtained results must be processed by an engineer.

You can check the effect of several changes on the final results and be critical with the new results. For example, try to solve the same problem with a wall thickness of 4 mm. Another possibility is to reduce the applied load to 5000 N. Please, take your time to do these alternative problems and analyse the obtained results from a critical point of view.

Finally, we can check the **Factor of Safety** of this design but it is expectable a low value (lower than 1 as the yield strength is reached). To check the **Factor of Safety**, you can click on the **Results icon** and a pop-up menu will be displayed with several plot options. Then you select "**Define Factor of Safety Plot …**" (Figure 7).

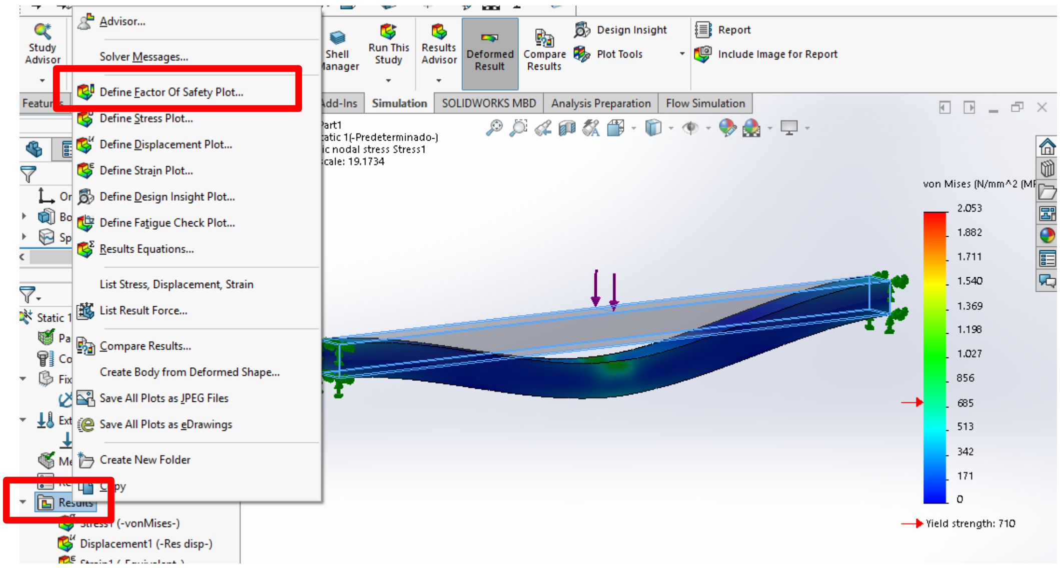


Figure 7. Selecting a plot of the Factor of Safety in SolidWorks after FEA simulation.

A new menu will be displayed at the left side. You can explore the different options this plot can provide but if you just click on the green check arrow, a plot of the **Factor of Safety** will be displayed. As you can see (Figure 8) the minimum Factor of Safety is **0,35** which is a poor value (and quite far from that indicated in the problem).

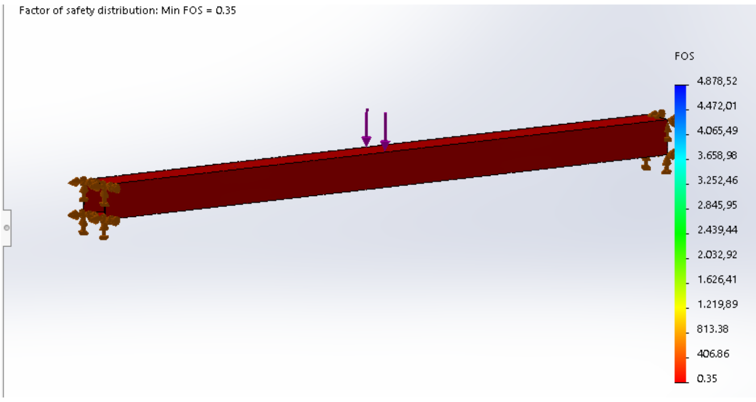


Figure 8. Plot of the Factor of Safety obtained after FEA simulation of a single beam fixed at both ends that is supporting a load in the center edge of 20000 N. The beam is manufactured with an AISI 4340 steel, normalized.

For this reason, it is necessary to optimize the thickness of the beam to reach a minimum Factor of Safety of 1,4 that ensures good performance of the beam in the above-mentioned conditions. Check the Factor of Safety for the two alternatives above-mentioned.

5 Closure

Although these CAD/CAE tools provide fast calculations, it is important to remark that the **quality** of the results (**OUTPUTS**) are directly linked to the **quality** of the **INPUTS** (geometry, tolerances, materials, fixtures, external loads, and so on to simulate the real working conditions). Engineers are responsible for the INPUTS and good INPUTS require an intense previous formation, therefore, CAD/CAE tools will never substitute engineers but can help them in reducing the overall time for the development.

Engineers need **important training** to obtain **useful** interpretation from a Finite Element Analysis. It is important to take into account the FEA is based on a mathematical calculation procedure and it is the engineer who has to be **critical** with the output results. In some cases, these results will be accurate but in other situations, the results will not match the real behaviour and the engineer must detect this to avoid wrong designs and dimensions.

6 References

[1] Charmey, C. "Time to Market. Reducing Product Lead Time", Ed. SME, Society of Manufacturing Engineers (1991)

[2] Khennane, A. "Introduction to Finite Element Analysis Using MATLAB® and Abaqus", Ed. CRC Press (2013).

[3] Lee, H-H. "Finite Element Simulations with ANSYS Workbench 16", Ed. SDC Publications (2015).

[4] Adams, V; Askenazi, A. “Building better products with finite element analysis”, Ed. OnWord Press (1999).

[5] Krenk, S. “Mechanics and Analysis of Beams, Columns and Cables. A Modern Introduction to the Classic Theories”, Ed. Springer, (2001).

[6] Carrera, E; Giunta, G; Petrolo, M. “Beam Structures: Classical and Advanced Theories”, Ed. John Wiley & Sons (2011).

[7] Onwubolu, G. “Computer-Aided Engineering Design with SolidWorks”, Ed. Imperial College Press (2013).