



A comparison between the analytical solution of a single cantilever beam fixed at one end and the use of the finite elements method (FEM) with 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 aims to show a comparison between the **analytical solution** of a single cantilever beam by using **conventional formulae** and the solution to the same problem by using **Finite Elements Analysis (FEA)** in the **SolidWorks** platform. The main aim of this article is to show the **power of the finite elements method (FEM)** to solve not only simple engineering problems but also, complex engineering situations.

2 Introduction

In the last decade, engineering tools have invaded industry. These tools include one, two of almost all the stages related to the design and development of new engineering parts and/or assemblies. The use of these tools is growing continuously as they can help engineers to reduce the **“time-to-market”** [1] as conventional analytical approaches are substituted by computers. It is worthy to note the increasing use of CAD (Computer Aided Design), CAM (Computer Aided Manufacturing) and CAE (Computer Aided Engineering) tools.

Computer Aided Design and Engineering tools (CAD-CAE) are involved in the training process of current engineers. **Autocad®**, **CATIA®**, **Solid Edge®**, **ANSYS®**, **Abaqus®**, **Matlab®**, **Comsol Multiphysics®**, **SolidWorks®**, among others, are widely used by engineers [2-4].

Today, some software packages **contain several tools related to design, manufacturing and engineering** tools. SolidWorks, created by Dassault Systèmes is an interesting tool for engineers as it is highly versatile and provides a fast learning curve.

This article focuses on solving a simple engineering problem related to a single cantilever beam by two different approaches: the classical approach by conventional formulae and the approach that uses Finite Element Analysis (FEA) provided by the “Simulation” Add-In of SolidWorks.

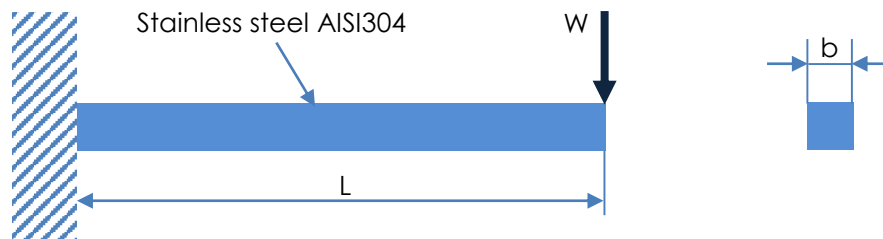
3 Objectives

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

- **Solve** a simple engineering problem by using **conventional formulae** and **materials selection**.
- **Draw/design** the problem into the **SolidWorks** platform.
- **Define** the **INPUTS** for a finite element analysis (FEA) using SolidWorks.
- **Analyse** the **OUTPUTS** that the FEA tool in SolidWorks provides.
- **Compare** the effectiveness of both analytical and FEA solutions.

4 Development

First let's define the problem to be solved. This consists on a single cantilever beam 100 mm length (L) with a squared section ($b = 4$ mm). This beam will support a load of 10 N in the cantilever edge and it is manufactured of stainless steel AISI304. In these conditions, the maximum deflection (δ) must be calculated. A scheme of the problem is as follows:



4.1 Analytical solution

Traditional approach ^[5, 6] to this problem gives the following expression to solve it:

$$\delta = \frac{W \cdot L^3}{C \cdot E \cdot I}$$

Where

- δ stands for the vertical deflection in the cantilever edge (in mm)
- W** represents the applied load (in N)
- L** is the beam length (in mm)
- C** stands for a constant that depends on the beam fixtures. For a single cantilever, this constant is 3 (non dimensional).
- E** is the Elastic modulus of the material (MPa or N/mm²)
- I** is the Moment of inertia of the cross-section (in mm⁴)

For this particular problem, $W = 10$ N, $L = 100$ mm, $C = 3$. With regard to E , this value can be obtained by searching the technical datasheet (TDS) of a stainless steel ($E = 197.000$ MPa or N/mm²). **Try to find** a technical data sheet for this particular stainless steel grade and you will see that you can get the elastic modulus from it. Maybe you will find slightly different values. You can try to solve the problem with your particular values.

Finally, you can calculate the **Momentum of inertia (I)** of the cross-section, by the corresponding equation for a squared area:

$$I = \frac{b^4}{12}$$

As indicated in the problem, the squared section has a b side of 4 mm; therefore, the Momentum of inertia is **$I = 21,3333$ mm⁴**.

The following step is to substitute all the above-mentioned numerical values into the general expression to calculate the maximum vertical deflection.

IMPORTANT NOTE. Check if you are working with the appropriate units to avoid mistakes.

$$\delta = \frac{10 \text{ N} \cdot 100^3 \text{ mm}^3}{3 \cdot 197.000 \frac{\text{N}}{\text{mm}^2} \cdot 21,3333 \text{ mm}^4} = 0,793 \text{ mm}$$

4.2 Solution by Finite Element Analysis (FEA) using SolidWorks Simulation Add-In.

First, you will have to **sketch** the problem into the **SolidWorks** platform. To this, open SolidWorks [7] program and click on start a new "**Part**". After this, you will have to select the plane in which you want to sketch the beam, for example, the "**FRONT**" view (click on front view). Then, the "**FRONT**" view will be ready to sketch.

First, you have to **sketch a square** with a side of 4 mm. To this, you must click on the label "Sketch" (blue rectangle) and select the "square/rectangle" (red rectangle) tool as indicated in Figure 1. You can select the option "Corner Rectangle" or the option "Center Rectangle". By choosing whatever option, sketch a rectangle (*do not mind about the dimensions*) as shown in Figure 1. When finished, click on the green check arrow located at the upper-left side.

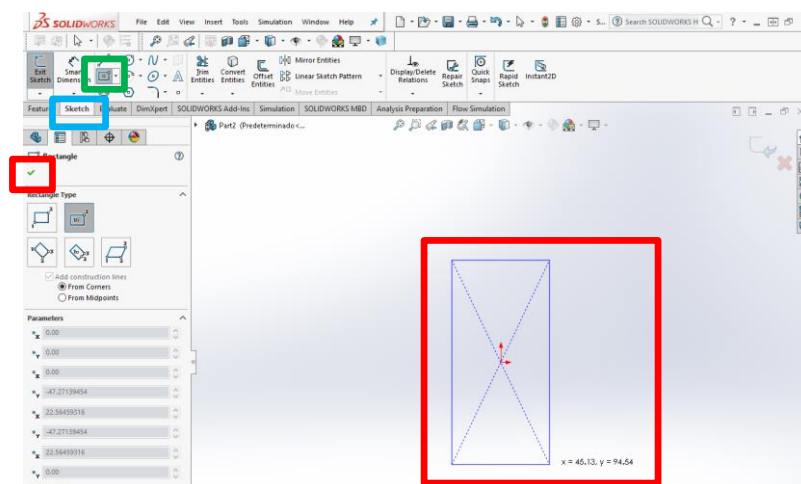


Figure 1. Initial sketch of a rectangle that will be the base of the beam cross-section.

Then, you have to adjust the current dimensions (rectangle) to a square with a side of 4 mm. To this, you must click on the corresponding side (for example, the up side) and you can write the real side length (4 mm) as shown in Figure 2. A new window appears at the left side with several parameters you can change. You have to change the current length of this side (50 mm) to the real length (4 mm).

Then you have to do the same with the right or left side (it doesn't matter). After doing this, you will have a square with a size of 4 mm.

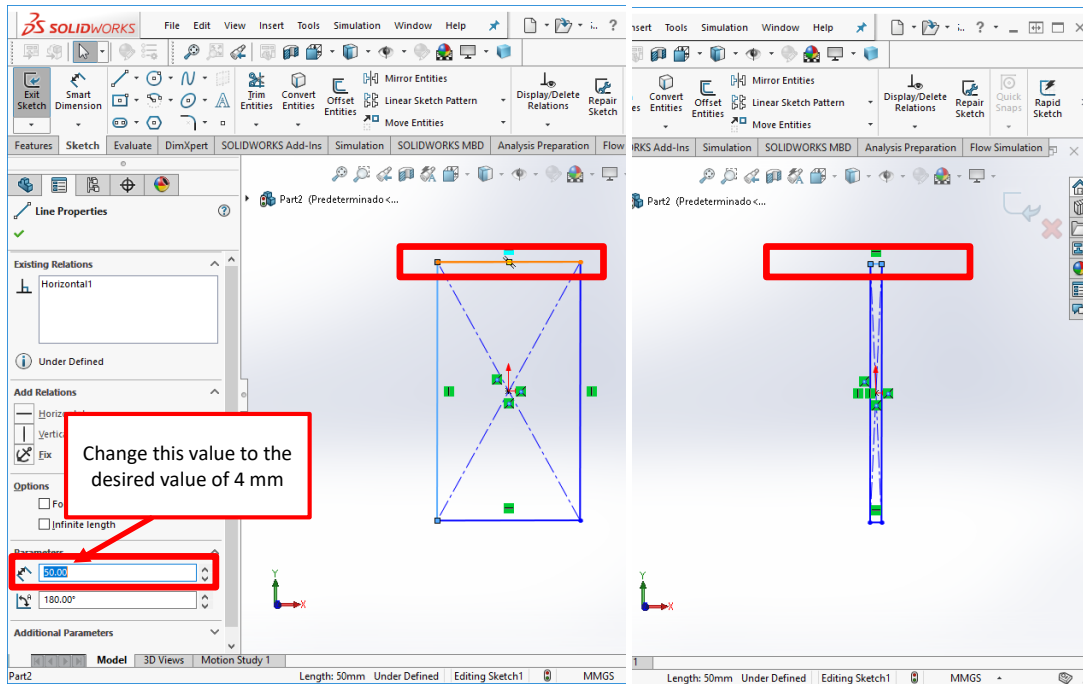
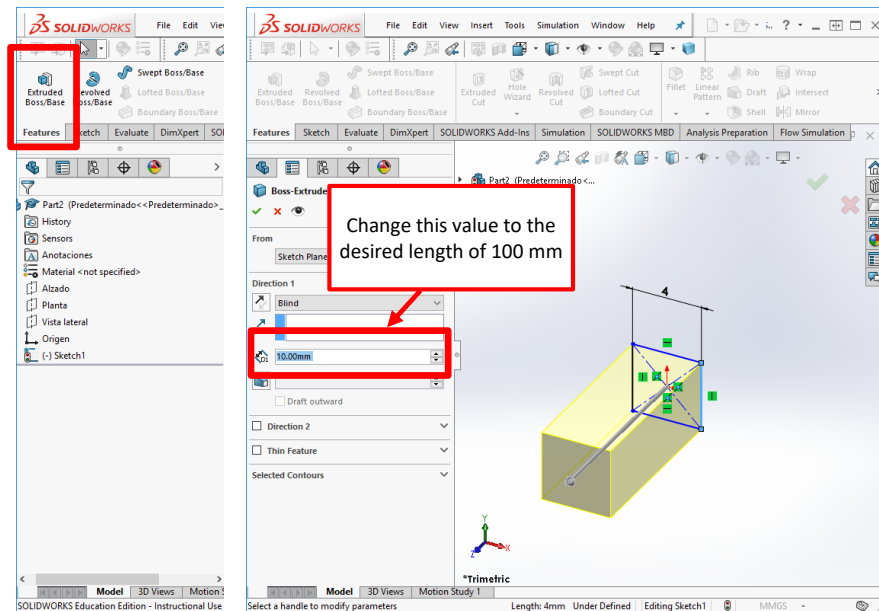


Figure 2. Procedure to change the upside dimension of a rectangle.

Once you have well defined the square cross-section, you have to give a specific "Feature" to this cross-section. In particular, an extrusion (beam length of 100 mm) in the normal direction (See Figure 3a). To this, you click on the "Features" label, located in the upper-left side and select the "**Extruded Boss/Base**" icon. This feature will extrude the base perpendicularly up to a certain value defined by **D1** as observed in Figure 3b.



a)

b)

Figure 3. Using the "Extruded Boss/Base" feature to convert a squared cross-section into a solid beam.

Now you are **ready to solve the problem** by using the “**Finite Element Analysis- FEA**” Add-In included in SolidWorks. This can be loaded into the working tools by clicking on “**Options icon**” and then, selecting “**Add-Ins...**”. A new window will open and you will be able to select several Add-Ins. In particular, for Finite Element Analysis (FEA), the “**SOLIDWORKS Simulation**” Add-In has to be selected (Figure 4).

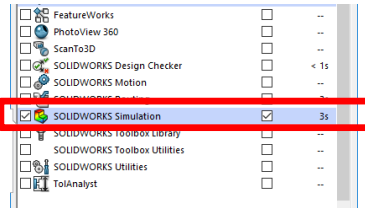


Figure 4. Selecting the “Simulation” Add-In for FEA in SolidWorks.

Once this Add-In has been installed, a new label appears (Figure 5). If you click on the “**Simulation**” label, then you will be able to choose the “**Study Advisor**” which will guide you or just select the “**New Study**” to select the study without any guidance.

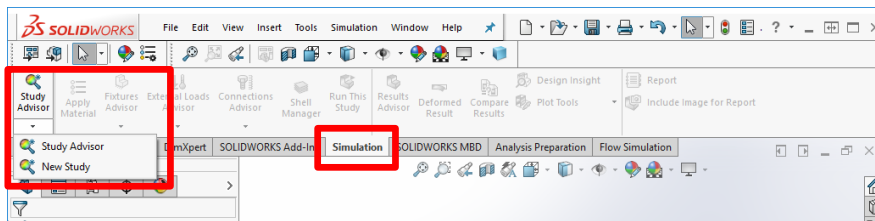


Figure 5. “Simulation” Add-In and options to carry out a simulation study.

As we have to solve a static problem, you can click on “**New Study**” and select “**Static**” and then, click on the green check arrow as indicated in figure 6a.

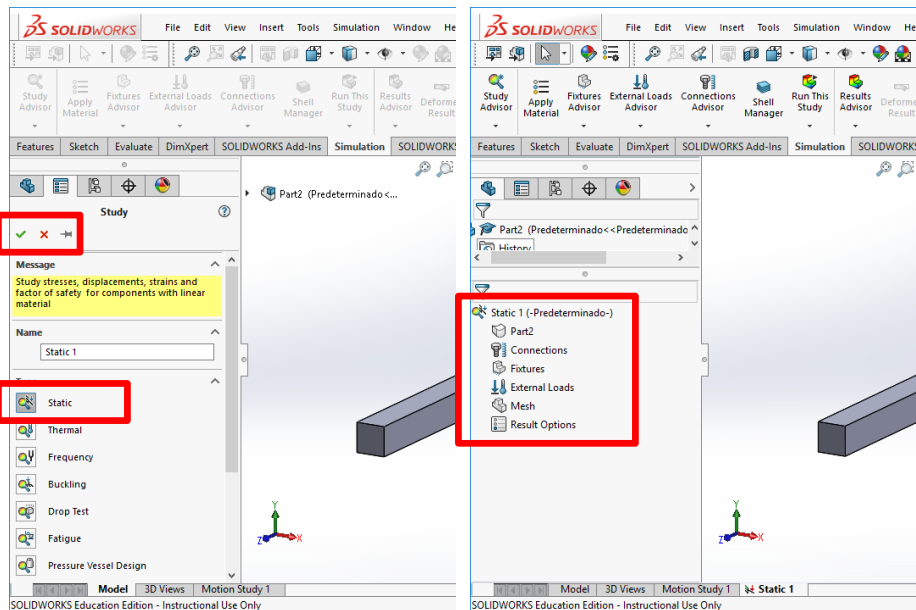


Figure 6. a) “Static” study for FEA analysis and b) Elements of a “Static” study.

The first step for a FEA simulation is to **define the material**. To this, just right-click on “Part2” and a menu will be displayed (Figure 7a). To assign the material, just click on “**Apply/Edit Material**” and you will have access to the Materials Database of SolidWorks. You can **search** for a specific material as the database contains metals and alloys, plastics, ceramics and composites. If you click on a specific material, its corresponding properties are provided as shown in Figure 7b. As you can see it is possible to find the AISI304 stainless steel in this database but its Elastic modulus is slightly lower (190.000 N/mm²) than that obtained by a Technical Data Sheet (TDS) (197.000 N/mm²).

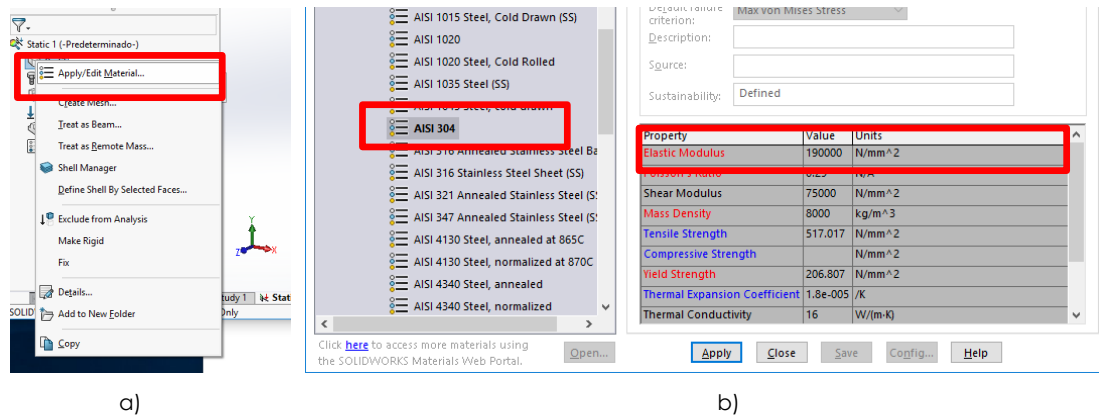


Figure 7. a) Selecting “Apply material” and b) Materials Database in SolidWorks.

Do not worry about that! This is not a problem. Probably this AISI304 is giving an average value of some commercial grades, so that its Elastic modulus is not the same as the previous commercial grade. Nevertheless, you need to use an Elastic modulus of 197.000 N/mm² to compare results. Therefore, you can **CUSTOMIZE** your materials. To customize a material, you right-click on the material you want to modify and click on “**Copy**” (Figure 8a). Then, you move to the bottom of the database and you will see an option corresponding to “**Customized Materials**”. Then you create a “**New Category**” (Figure 8b) and, finally, you can “**Paste**” the selected material (Figure 8c).

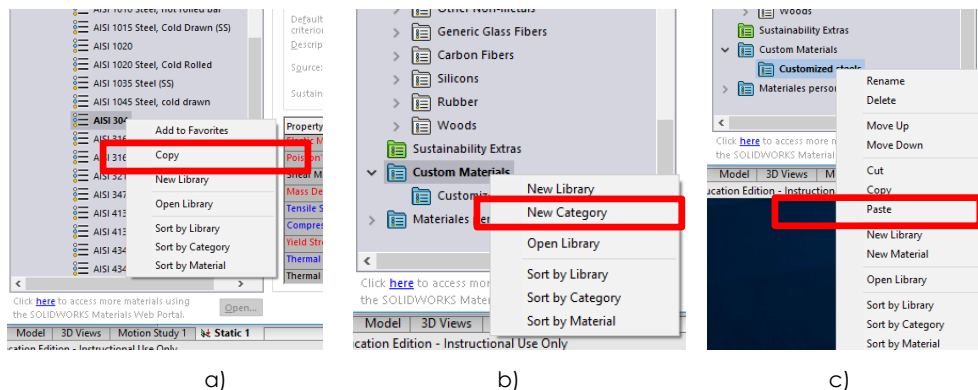


Figure 8. Defining a customized material in SolidWorks.

This customized material can be modified by changing its current Elastic modulus to the desired value (197.000 N/mm²). Once this has been done, you click on the “**Apply**” button and this material will be applied to the designed part. Then, you click on “**Close**” button (Figure 9).

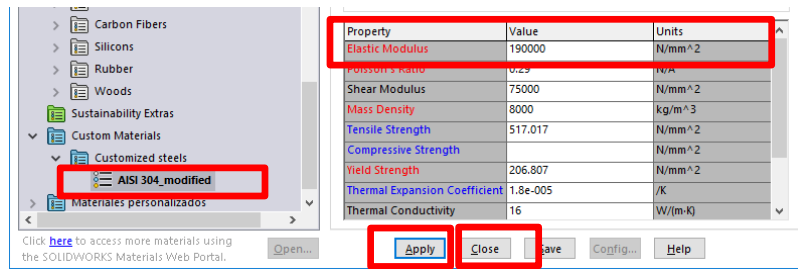


Figure 9. Defining and applying new/different properties in customized materials.

As this single cantilever is not a part of an assembly, then the "Connections" item is not necessary. Now let us define the **Fixtures** that support the beam. As we have said, the beam consists on a single cantilever, which is fixed, in one end. To define the fixture, you right-click on "**Fixtures**" and select "**Fixed Geometry...**". Then, you will have to select a vertex, and edge or a face to fix. In this particular case, you fix one of the two end surfaces as shown in Figure 10, and click on the green check arrow.

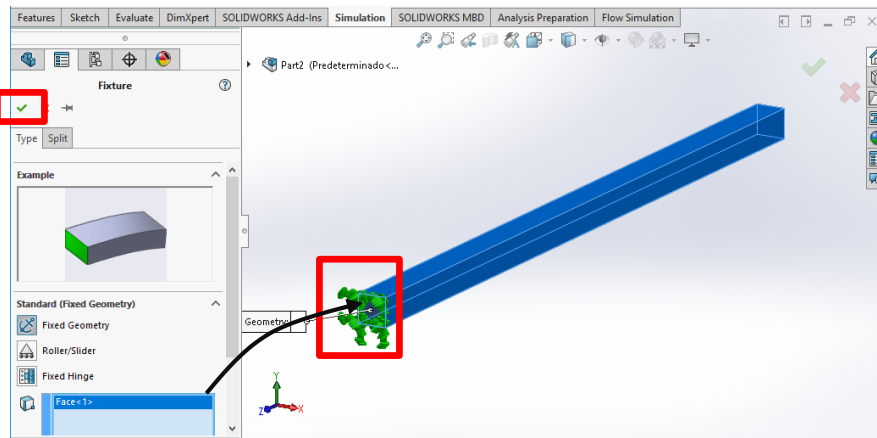


Figure 10. Fixtures in a single cantilever beam.

The following step you must follow is to define the external loads. As indicated in the problem, a 10 N force is applied in the cantilever edge. To represent this, you can right-click on "**External Loads**" and a pop-up menu will appear, showing the different loads you can use in SolidWorks. Then, select "**Force...**" (Figure 11a) and a sub-menu will appear at the left side (Figure 11b). First, you select the edge on which, the force will be applied (red arrow). Then, click on the "**Selected direction**" to appropriately define the direction and select an edge with the desired direction (green arrow). Then you write the value of the force (10 N) and click on "**Reverse direction**" to show the force in an appropriate direction.

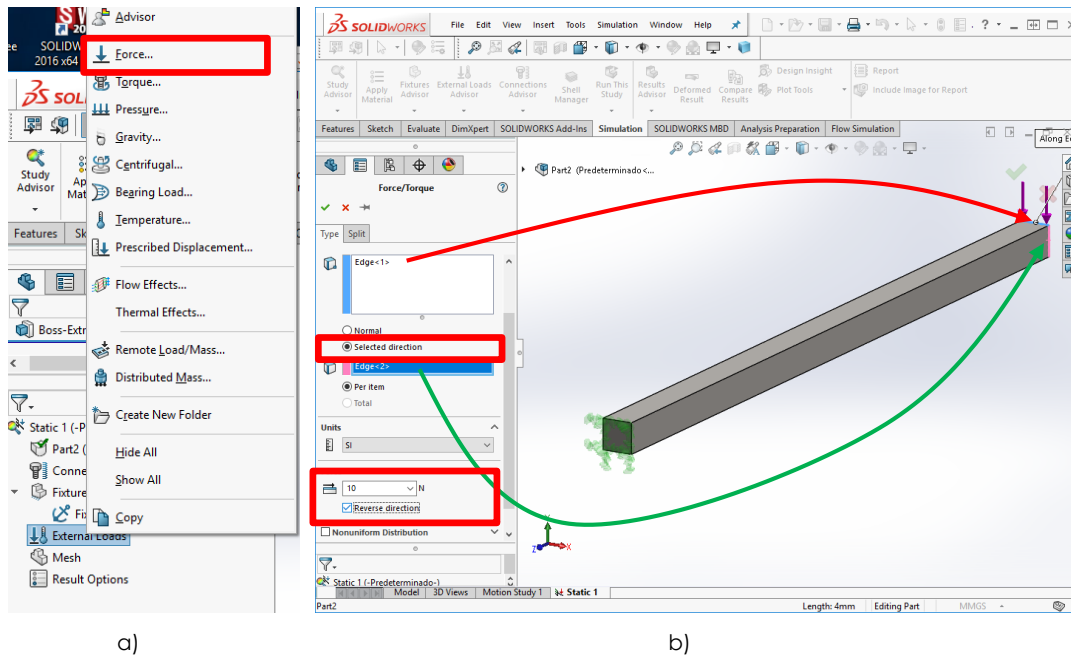


Figure 11. a) Selecting the “External Loads” menu and b) Selecting the applied force, value, direction.

The problem is completely defined. The Finite Elements Method is based on decomposing a complex problem into a Finite number of smaller and easier problems, which are easy to solve. To obtain a finite number of elements, the geometry must be **meshed**. To mesh your problem, right-click on “**Mesh**” and select “**Create Mesh...**”. Then a new window with information and mesh parameters appears. You can decide the element size (the finer elements, the longer computing time). After doing this, the solid beam has been divided into a finite number of nodes and elements as you can see in Figure 12b.

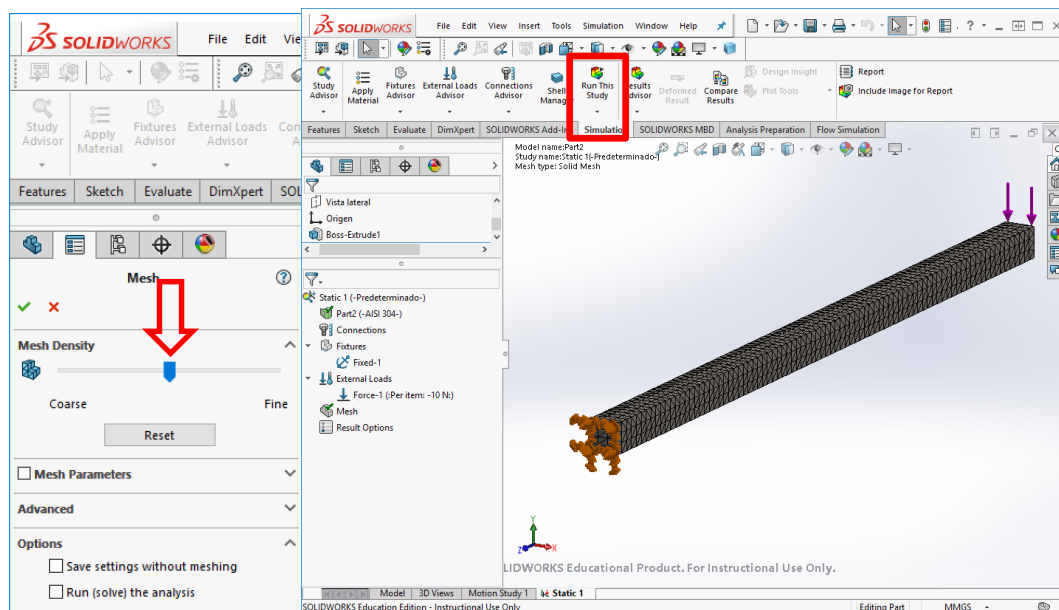


Figure 12. a) Selecting the size of the mesh element and b) Meshed solid.

The simulation is ready; now you just have to click on “**Run This Study**” (Figure 12b) and the simulation will start. It takes some time to solve the system and, finally, once the solution has been found, some results appear in the “**Results Options**”: Stress, Displacement and Strain

If you “double-click” on “**Displacement**”, then you will see a colour plot of the beam with its corresponding deformation at each point (Figure 13). As you will see, the maximum displacement is **0,792 mm**, which is very close to the analytical solution (0,793). Obviously, there is an error related to FEA. In this case, this error (by taking the analytical solution as reference) is of about 0,12%.

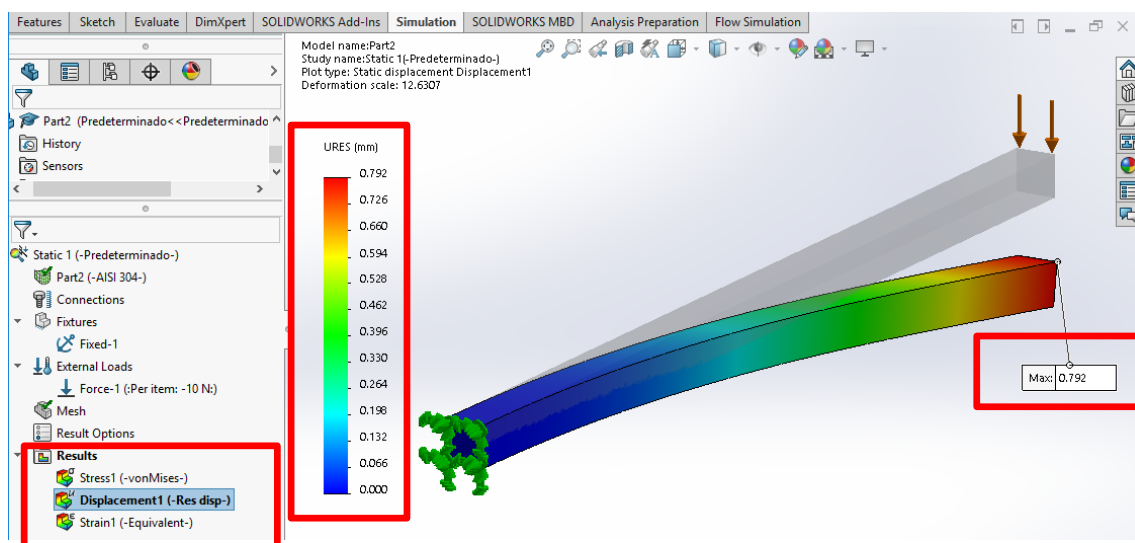


Figure 13. Graphical plot of the numerical simulation by FEA, corresponding to displacements on the beam.

You can continue working on “Results” to show stress, strain and select different visualization options.

Therefore, I hope you have understood all the steps. Now you can solve this problem from a conventional/analytical approach and by Finite Elements Analysis (FEA). I encourage you to change some parameters and evaluate the maximum deflection. For example, using a coarse mesh, using a fine mesh, etc.

5 Closure

As you have seen in this article, the solution of a simple engineering problem (single cantilever beam) can be solved by either analytical expressions and Finite Elements Analysis (FEA). The solution is almost the same but some deviation can occur by using numerical methods by FEA.

Perhaps you are now thinking that it is easier to solve this problem by using the analytical approach with its corresponding expression. That is true! but, why? It is just because the system we have described is very simple and the use of a simple formula is enough to solve it. Nevertheless, think about a complex engineering part or assembly. These complex engineering systems are not easy to be solved by analytical expressions and it is there where FEA takes some advantages.



FEA allows simulating complex parts in a reasonable time so that FEA represents an important tool for engineers. The use of FEA tools can positively contribute to reduce the design+development time of an engineering part or assembly.

However, you have always to take into account that the quality of the simulation results (OUTPUTS) will highly depend on your knowledge about this engineering problem, so that the INPUTS you define to the software, will define the quality of the OUTPUTS. So do not forget that CAD-CAE tools are just one of a broad range of tools than can assist engineers during the design+development process.

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).