



Creating a CAD model of a single beam for engineering analysis with SolidWorks

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1 Summary of the key issues

This article aims to show the potential of **Computer Aided Design (CAD)** as **complementary tool** in the engineering process. This article focuses on converting a simple mechanical engineering problem (a beam fixed at both ends) into its corresponding CAD file for further analysis.

2 Introduction

In the last years, the **consumer society** we live has increased its demand on new and high technology content products. In this **new paradigm**, engineering developments need to be **faster** than time ago. "**Time**" has always been considered as gold but in the last years this concept has invaded design and development of engineering parts and assemblies. The "**Time-to-market**" has become one of the most relevant issues in the strategy of a company to reach a particular market. Every day, the market demands new, original, high technological and "**faster**" solutions ^[1].

It requires high levels of technical formation. Nevertheless, in the last years a wide variety of **tools** have been developed **to assist engineers** during this process. These tools include **all the stages** in a typical engineering development: *market analysis, concept, proofs of concept, initial designs, validation, calculation, dimensions, manufacturing process and tolerances, cost analysis, release of the product*, among others.

With regard to engineering, a new concept has arisen, *i.e.* **CAD-CAM-CAE** which means **Computer Aided Design (CAD)**- **Computer Aided Manufacturing (CAM)** and **Computer Aided Engineering (CAE)**. These tools have become really important for engineers as they allow reducing development times by using **computers**. It is worthy to note that some tools include all three approaches: CAD, CAM and CAE, thus giving powerful tools to engineers. One of these tools is **SolidWorks** which stands out for a quick learning curve, compared to other software packages.

3 Objectives

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

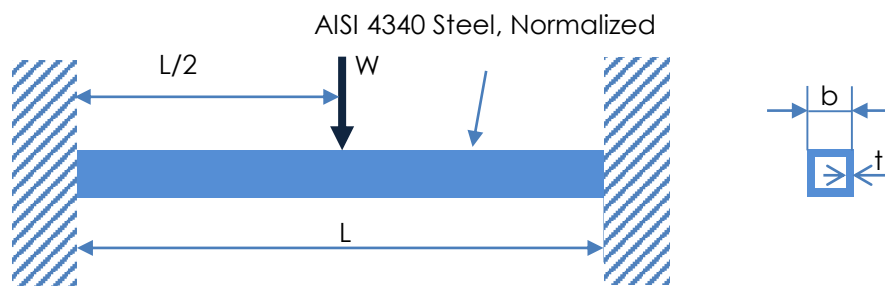
- **Identify** the **main parameters** to be translated into a CAD design.
- **Draw/design** a problem into the **SolidWorks** platform considering shape parameters.

4 Development

First let's define the problem, a typical problem related to beam analysis.

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 establish that only elastic deformations can occur and a Factor of Safety (FoS) of 1,4 must be considered.

As you will agree this is a typical beam analysis problem and it can be solved by using several analytical expressions [2-4]. Maybe you can think that the fastest way to solve this problem is by using the corresponding expressions and probably you are right. Then, **why** is it interesting to build a **CAD model** related to an engineering problem? In this particular case, it will not give you important advantages as the problem can be easily solved by using a formula. Nevertheless, **real engineering problems** are remarkably **more complex** than a simple beam. So that, it is convenient to use **computers** to solve these more complex problems which will allow time to develop new parts and assemblies.



4.1 Initial design with SolidWorks

To obtain a CAD model of the above-mentioned problem, firstly, you have to **sketch** the design in SolidWorks. To start with the sketch, open SolidWorks and click on start a new "Part" as indicated in the following figure. First you will have to design the beam with its particular shape and geometry and after that, you will be able to define all other information as applied forces, fixtures, etc.

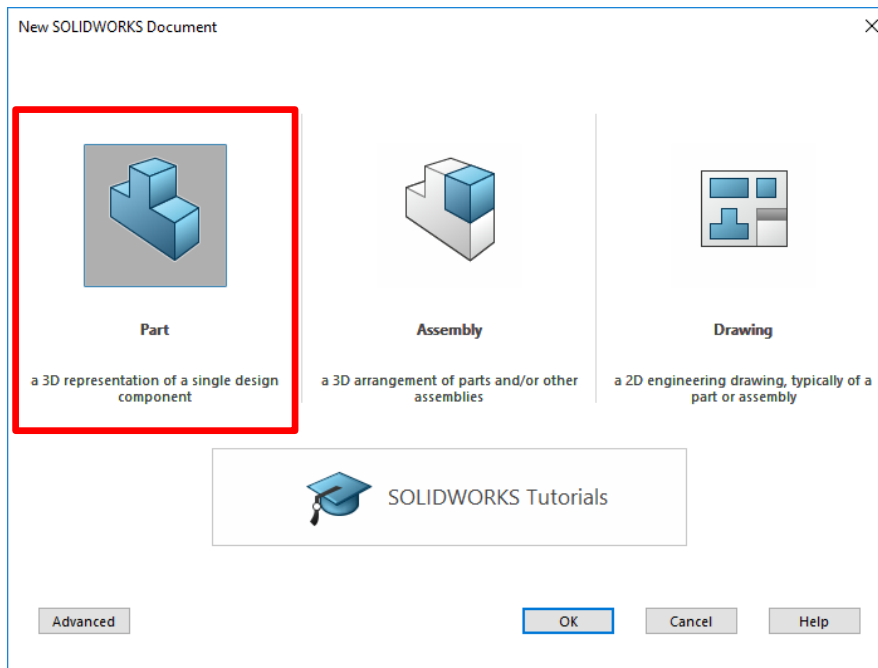


Figure 1. Selecting an engineering "Part" to start working with SolidWorks.

Then you can start with the sketch. To sketch a **squared hollow geometry**, you must select a **view** in the left panel. For example, you can select the "**FRONT**" view to start sketching as shown in Figure 2.

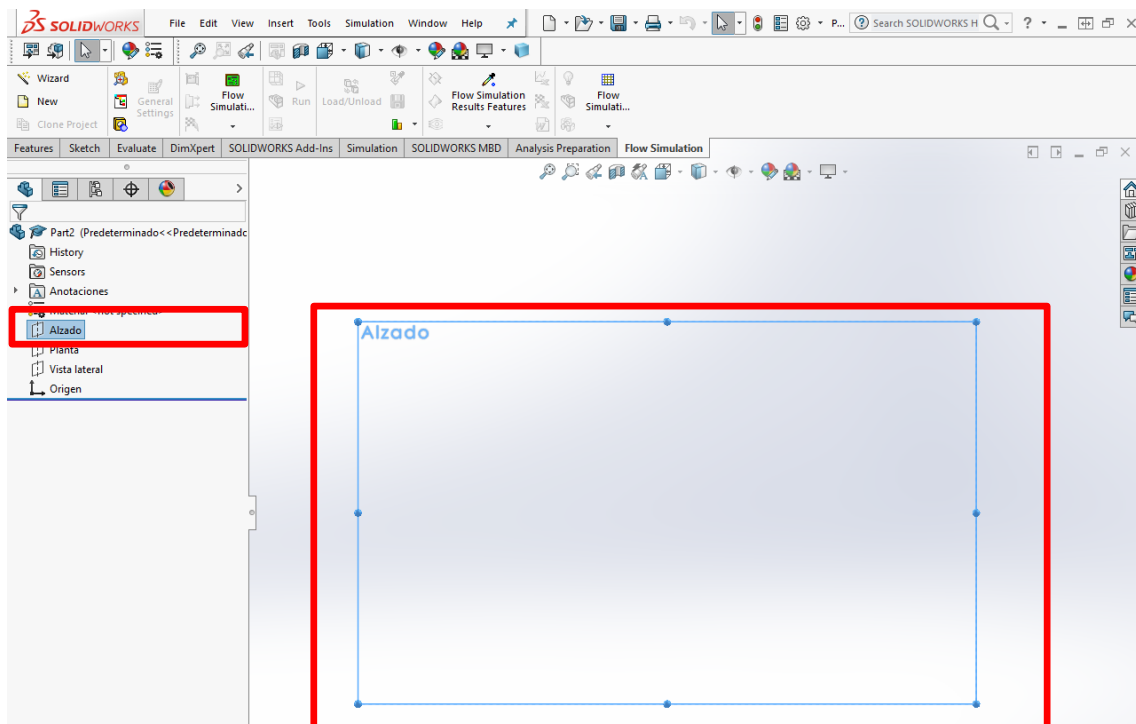


Figure 2. Selecting the appropriate view to work with.

The following step is to sketch a **square** with a side of 5 cm. To construct this square, you just click on the “**sketch**” label and then, select the square tool as indicated in Figure 3. This tool has a drop-down list with several options. To sketch a square, for example, select the “**Center Rectangle**” options. With this option, firstly you define the center and then, you move the mouse and a rectangle is constructed. Don't worry about the dimensions that you can obtain by using the mouse.

NOTE. Check the default units to avoid wrong definitions and calculations.

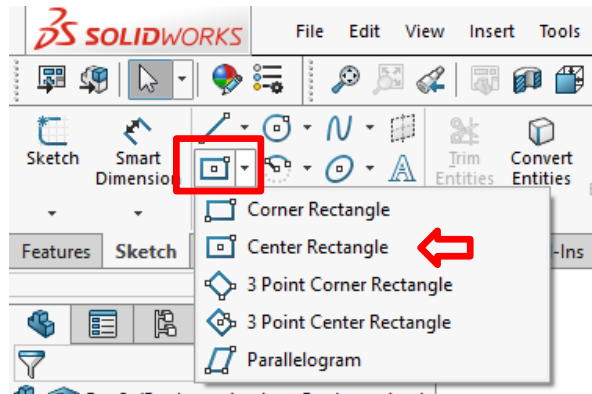


Figure 3. Selecting a sketch tool and its corresponding drop-down list.

As it can be seen in Figure 4, the sketched square is not really a square; it is a rectangle with dimensions 139,62 mm x 113,68 mm. It is not a problem. Sketch your rectangle and click on the green check arrow to complete this operation.

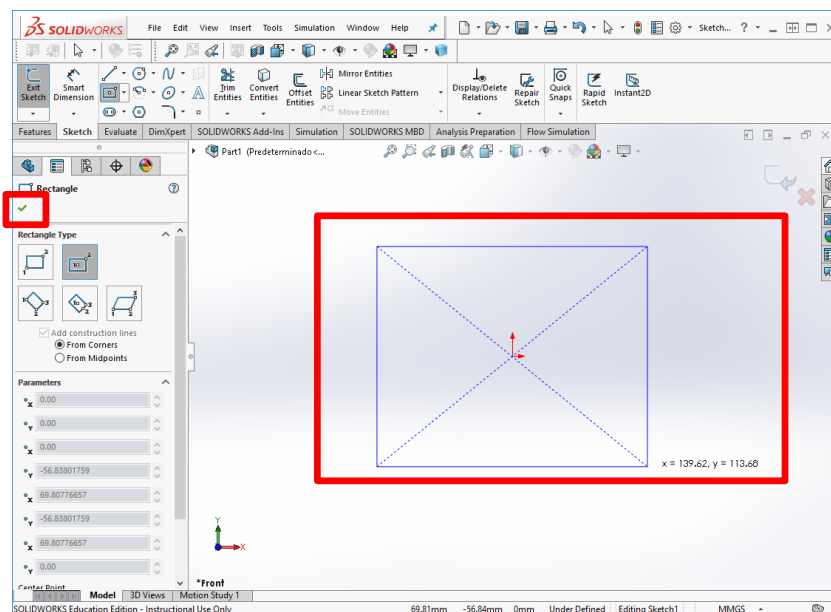


Figure 4. Initial sketch of a rectangle that will be the base of the squared hollow beam.

To provide the appropriate dimensions it is highly recommended to use the “**Smart Dimension**” tool in the “**Sketch**” icons, as shown in Figure 5. Click on “**Smart Dimension**” and then click and drag on a side. A “**Smart Dimension**” will be created when you finally click on the mouse again and then, you will be **prompted to provide** the appropriate

value (in this case, 50 mm) and then click on the green check arrow. Then, you will repeat the same process for the other side to obtain a square with a side dimension of 50 mm.

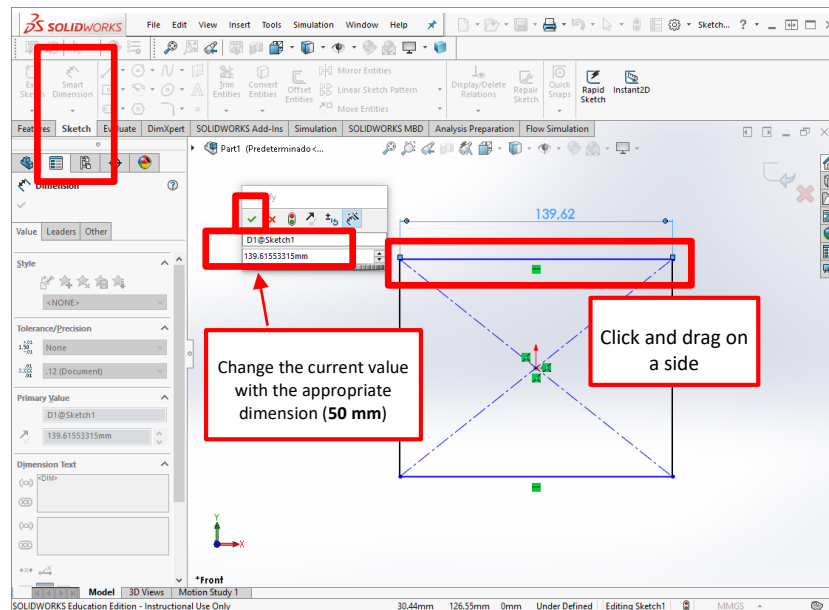


Figure 5. Using “Smart Dimensions” in a sketch to define the correct dimensions.

After this, you are going to learn how to create the hollow base. To this, we are going to use the “**Offset Entities**” that sketches a parallel shape. To use this tool, first you have to select all the entities you want to offset. To select the four sides, you must click on the “**SHIFT**” key in the keyboard and then click on each side. All the selected lines will change their colour to **blue**. Once all four sides have been selected, you can click on the “**Offset Entities**” icon as shown in Figure 6. As you will see, the **default offset is 10 mm external** to the sketched square. As the wall thickness is 2 mm, you just have to click on “**Reverse**” and change the default offset to 2 mm. Finally, you must click on the green check arrow to complete this operation.

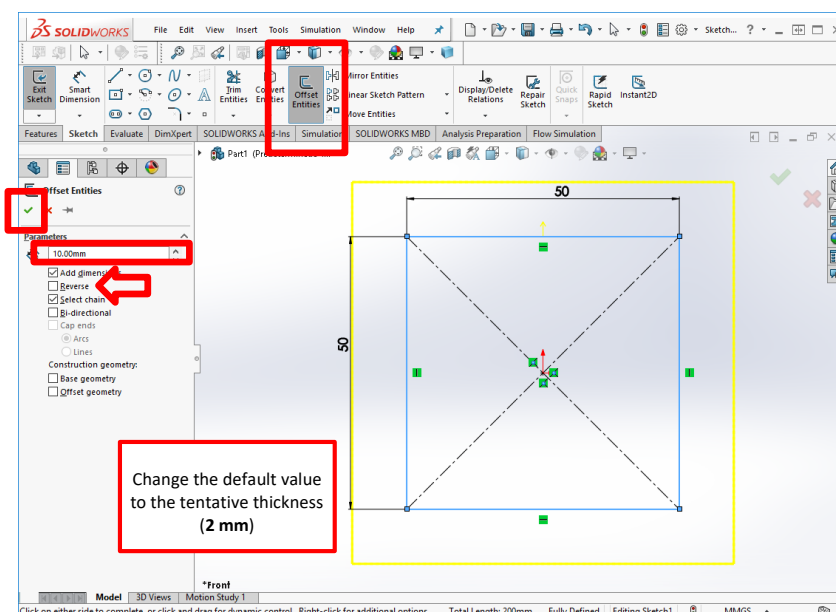


Figure 6. Using the “Offset Entities” tool to construct the hollow squared base.

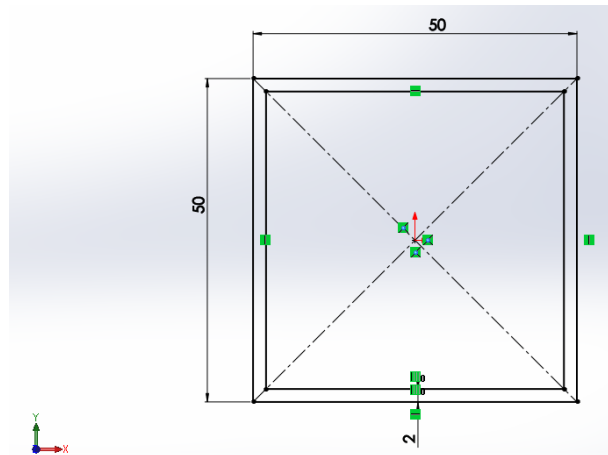


Figure 7. Squared hollow base with a thickness of 2 mm, obtained using the "Offset Entities" tool.

Once this base geometry has been defined, we are going to use the "**Extruded Boss/Base**" Feature (see Figure 8) to extrude the base to the desired length (1000 mm). You have to change the default extrusion length (D1) up to **1000 mm** as indicated in the problem, and then, click on the green check arrow to complete the operation.

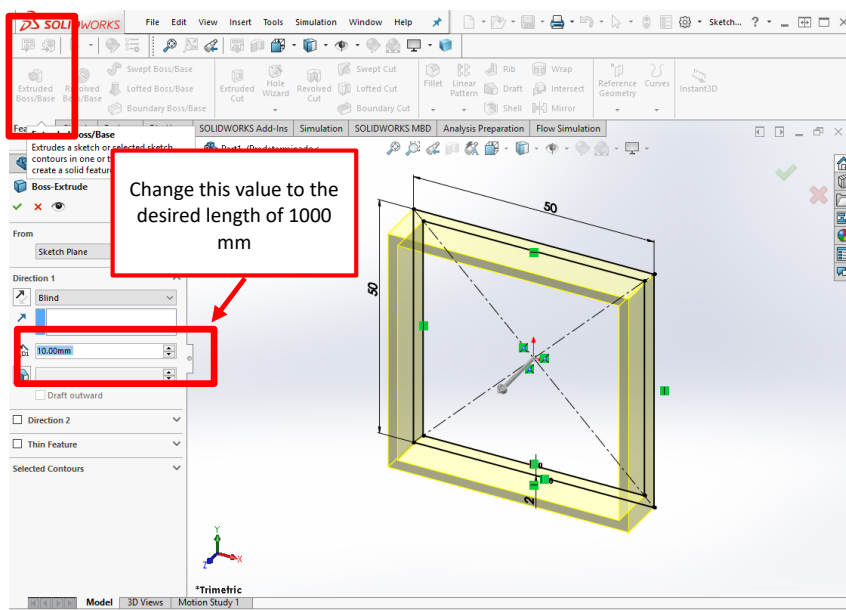


Figure 8. Using the "Extruded Boss/Base" feature to convert a hollow square cross-section into a solid beam.

Once you have completed all the previous steps, you will have a hollow squared beam as shown in Figure 9.

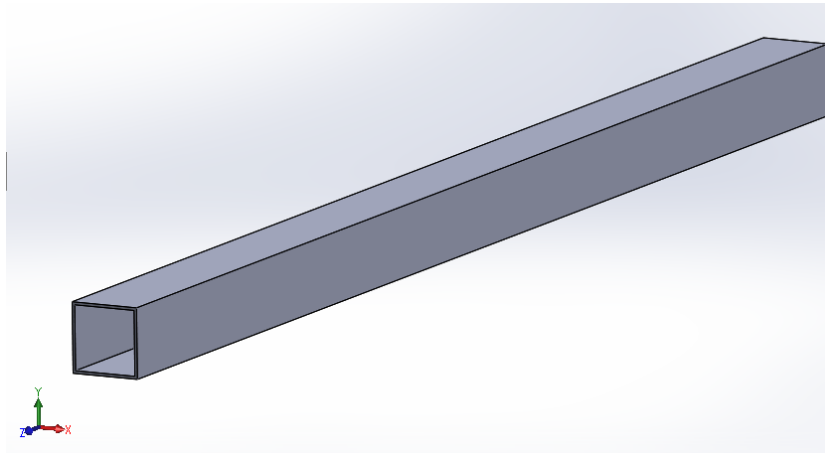


Figure 9. 3D plot of a beam with a length of 1000 mm and a hollow squared cross-section with an external size of 50 mm and an internal thickness of 2 mm.

In addition, you will have **to define the edge** on which, the **load will be applied**. To do this, you just have to create other **sketch** with a line and then, you will project this line over the solid to give an additional edge. The line in the new sketch must be located in the center of the beam as shown in Figure 10.

NOTE. Be sure the line is correctly placed perpendicular to the beam and located in the center.

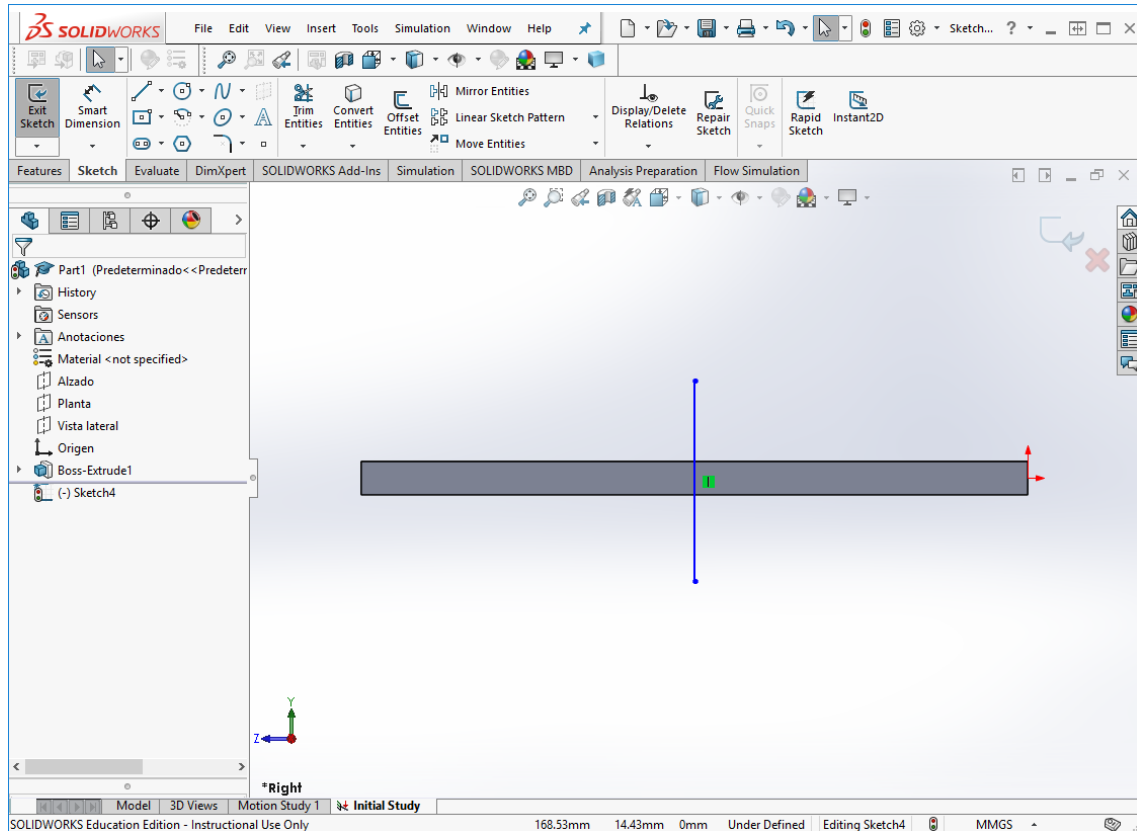


Figure 10. New sketch of a single line to create a partition line on the 3D solid that will represent the edge on which, external loads are going to be applied.

Once you have finished this step, you can go to the main menu and click on **“Insert”**. You will see a sub-menu with several options but to achieve the above-mentioned goal, you just have to click on **“Curve”** and subsequently, on **“Split Line ...”**.

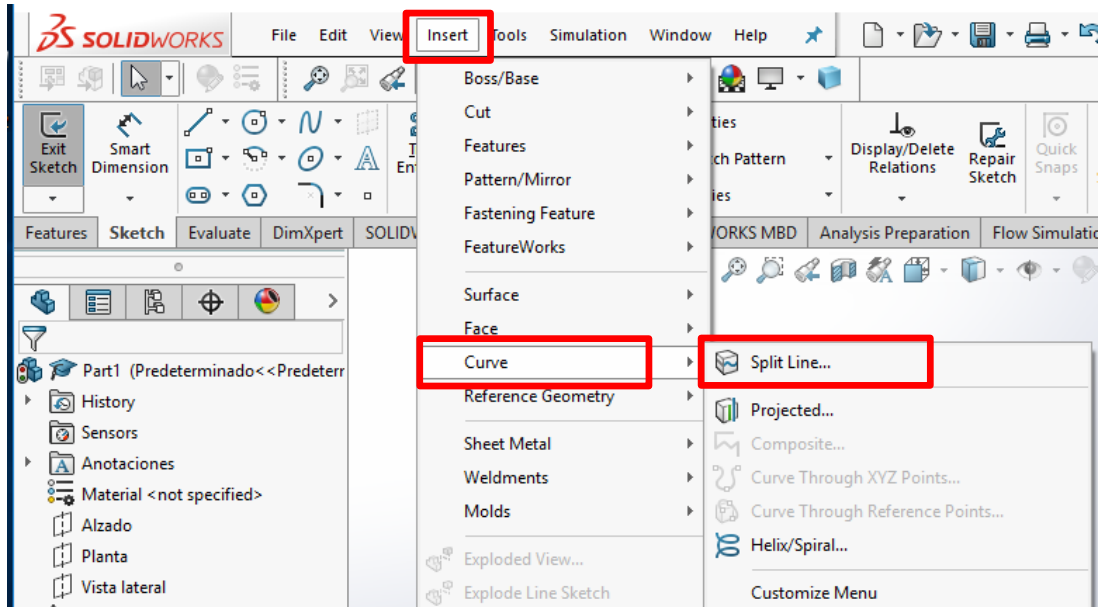


Figure 11. Access to the **“Split Line...”** feature in SolidWorks to make a projection of a sketch on a solid.

The **“Split Line”** menu will appear at the left side and you just have to select the sketch you want to project and the solid in which this sketch will be projected (Figure 12). You have several options you can try but in this case, you select the default option **“Projection”** and click on the sketch (pink line) and the solid part (blue line) and, finally, click on the green check arrow.

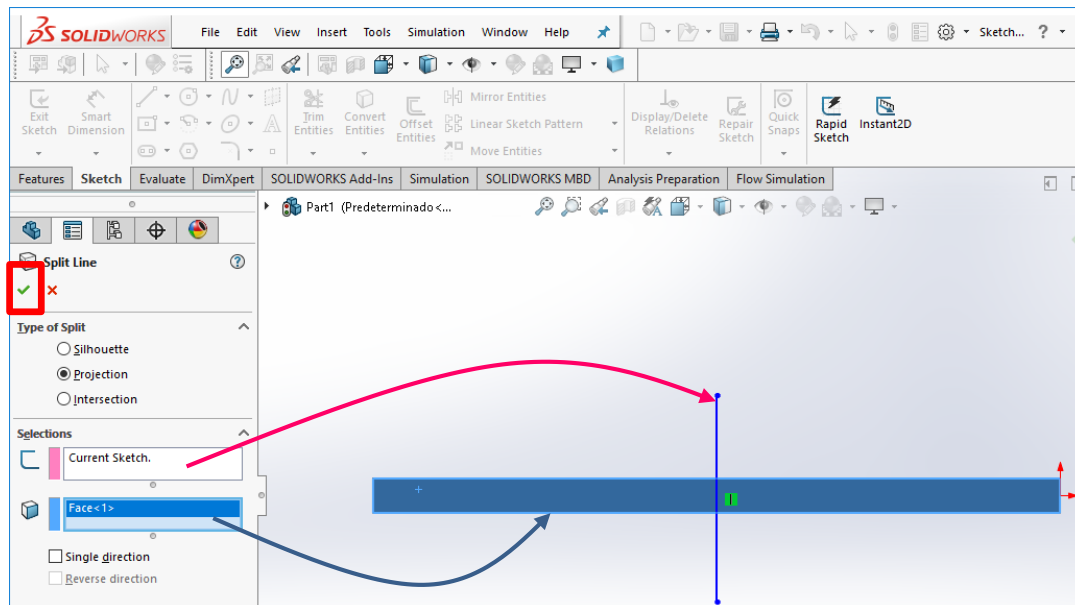


Figure 12. **“Split Line”** options to project a sketch over a solid geometry.

After this stage, you will have the same solid beam you have constructed but, additionally, an edge appears in the center (see Figure 13).

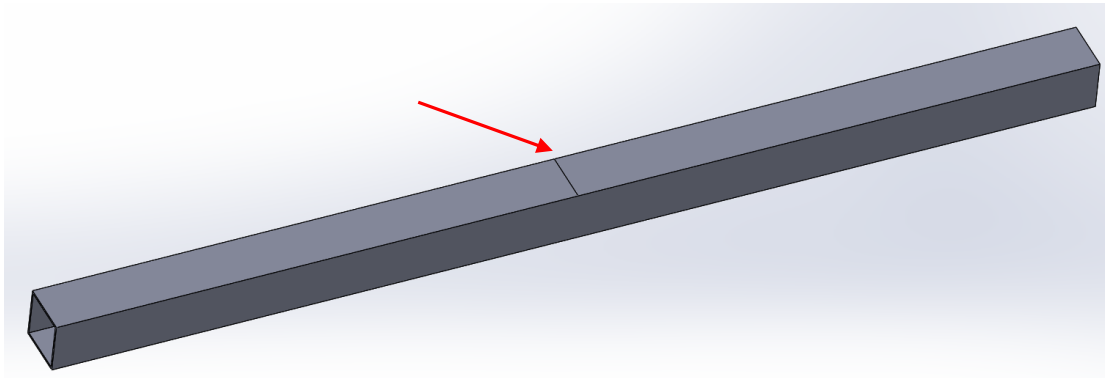



Figure 13. 3D plot of a beam with an additional edge in the center to apply external forces.

4.2. Defining other information to the CAD model.

In addition to the CAD model, it is necessary to define other useful information as materials, applied force, fixtures, etc. Nevertheless, all this additional information should be defined to perform a **Finite Element Analysis (FEA)** of the **CAD-model**. Finite Element Analysis is a CAE (Computer Aided Engineering) tool. Using FEA tools is out of this learning article.

To use **FEA feature in SolidWorks**, you need to install "**SOLIDWORKS Simulation**" **Add-Inn**. If you have already installed this Add-In, it is perfect. In case you have not previously installed this, you must click on the "**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 must be selected. Once this Add-In has been installed, a new label appears (Figure 14). 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 appropriate study without any guidance.

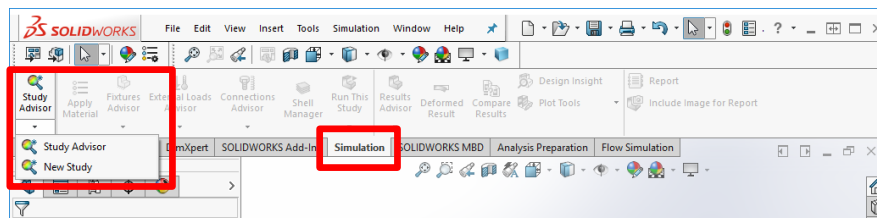
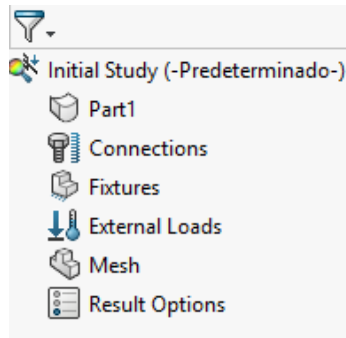


Figure 14. "Simulation" Add-In and options to carry out a simulation study.

A series of elements appear at the left side of the window. These elements are essential to define the problem in terms of fixtures, materials, loads, etc. (**CAE-model**).



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

5 Closure

Computer Aided Design (CAD) tools can **assist** engineers during designing new products, parts and/or assemblies or just to create a CAD-model of an engineering problem. Although the solution of simple problems (single beam analysis) can be carried out in a simple way by using several analytical expressions, construction of a parametric CAD-model is really useful as it will allow further analysis by using CAE (**Computer Aided Engineering**) tools or even CAM (**Computer Aided Manufacturing**) tools.

A good transcription of an engineering problem into a CAD-model is essential to obtain accurate solutions.

6 References

- [1] Charmey, C. "Time to Market. Reducing Product Lead Time", Ed. SME, Society of Manufacturing Engineers (1991)
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- [3] Carrera, E; Giunta, G; Petrolo, M. "Beam Structures: Classical and Advanced Theories", Ed. John Wiley & Sons (2011).
- [4] Onwubolu, G. "Computer-Aided Engineering Design with SolidWorks", Ed. Imperial College Press (2013).