THE POTENTIAL OF CATIA IN STRUCTURAL DESIGN

ESCUELA TÉCNICA SUPERIOR DE INGENIERÍA DEL DISEÑO

DEGREE IN MECHANICAL ENGINEERING

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Valencia, September of 2018

Abstract

The purpose of this thesis was to explore the possibility of using CATIA with an end goal of producing simulations for structural elements. The program requires technical input from the user with the addition of some geometry provided in the operational catalog of CATIA. In addition to the mentioned end goal, this thesis is pretended to be a useful resource for the future of the structural analysis world, not just being a one-use project that loses its utility once the project is done.

The information of this thesis was collected from literature, internet sources, practical experience, and by conversations with people knowledgeable in either the field of machining or programming. The system works by logically deciding which machining operations are necessary, which processes should come first and last, the desired insert shape and size, and the machining parameters. This information is all uploaded to CATIA which does the actual simulation through one if its modules, allowing the user to focus on the most stressed parts.

The results of this study demonstrate that this is a very viable option for structural industry production. It is recommended that more work be carried out to improve the capabilities of the program, exposing CATIA to different geometries, loads, and stress situations.

KEYWORDS: CATIA, structural elements, loads, stress.

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Nomenclature

CATIA	Computer-aided three-dimensional interactive application.
SAP2000	Finite element program for modeling, analysis and sizing of any structure.
CYPECAD	Software created to design, calculate and dimension reinforced concrete and metal structures for buildings and civil works.
CAD[1]	Computer-aided design.
CAM[2]	Computer-aided manufacturing.
CAE[3]	Computer-aided engineering.
PLM[4]	Product Lifecycle Management.
DS[5]	*Dassault Systèmes.*
CATIA V5 R21	CATIA Version n°5 Release n°21.
FEM[6]	Finite Element Module.
Part Design[7]	Part Design workbench.
Assembly Design[8]	Assembly Design workbench.
ArcelorMittal[9]	Luxembourgish multinational steel manufacturing corporation.
Axial effort	Effort that is perpendicular to the plane on which the tensile or compression force is applied, which is evenly distributed throughout its surface. Also called normal effort.
Shear effort	A shear effort is known as an internal force developed by a body in response to a cutting force that is tangential to the surface on which it acts. Also called shear force.
Bending Moment	Moment that occurs when flexing the section of a beam or other structural element; equivalent to the sum of the moments with respect to the center of gravity of that section.
"PAD" Command	CATIA command used to extrude sketches and create solid bodies.

"Hole" Command	CATIA command used to create holes in body surfaces and solid bodies.
"Rectangular Pattern" Command	CATIA command used to replicate operations implemented on a CATIA solid/product.
"Pocket" Command	CATIA command used to create holes based on profiles. This command requires pre-created profiles before executing the operation.
"Axis System Definition" Command	CATIA command used to insert customized axis systems on a Part Design or Assembly Design file.
"Manipulation" Command	CATIA command used to test the mobility of a product inside of the Assembly Design workbench.
ISO[10]	International Organization for Standardization .
US	Regulations used in the United States of America.
EN[11]	Regulations used in Europe. Known as "European Standards".
JIS[12]	Japanese Industrial Standards.
"MultiPAD" Command	CATIA command used to create different extrusion from a single sketch.
"Groove" Command	CATIA command used to remove material around a defined axis.
"Thread/Tap" Command	CATIA command used to create threaded surfaces on solids for analytical purposes.
"Clash" Command	CATIA command used to check surface contacts, overlaps or interferences within a product.
"Apply Material" Command	CATIA command used to add material properties to a body for analysis purposes.
GSA	Generative Structural Analysis workbench.

"Clamp" Command	CATIA command used to define different body parts as clamped for analysis purposes.
"Distributed Force" Command	CATIA command used to apply distributed loads on different faces of a body.
"Compute" Command	CATIA command used to start the analysis process of the GSA workbench.
Von Mises stress	Used to determine whether an isotropic and ductile metal will yield when subjected to a complex loading condition.
"CCP" Command	CATIA command used to administrate contact properties to different constraints in the GSA Workbench.
"*SCP" Command*	CATIA command used to administrate smooth adjustments to different constraints in the GSA Workbench.
"RCP" Command	CATIA command used to declare different constraint as rigid in terms of analysis calculations.

Extended definitions of the most specific normative [] can be found at the end of the thesis, in the Annex: Extended Definitions (Nomenclature).

Introduction

This thesis investigates the possibility of running simulations of structural elements within CATIA V5 using its different modules to design and create said elements. The purpose of using CATIA V5 instead of other programs such as SAP2000 or CYPECAD is because the wide possibilities the program offers in terms of designing. CATIA possesses more than 70 workbenches distributed in 9 independent modules. In this section of the thesis I'll talk about the program itself, objectives of the thesis, project scopes and delimitations and thesis structure.

Background: CATIA

CATIA is a multi-platform software suited for CAD, CAM, CAE and PLM. Developed by the French company *Dassault Systèmes (DS)*, CATIA was born in 1977 as an in-house development.

Figure 1. CATIA logotype (extracted publically from DS' webpage).

With more than forty years of experience, CATIA is the World's Leading Solution for Product Design and Experience. It is used by leading organizations in multiple industries to develop the products we see and use in our everyday lives (Such us BMW, Volkswagen Group, Tesla Motors or Airbus).
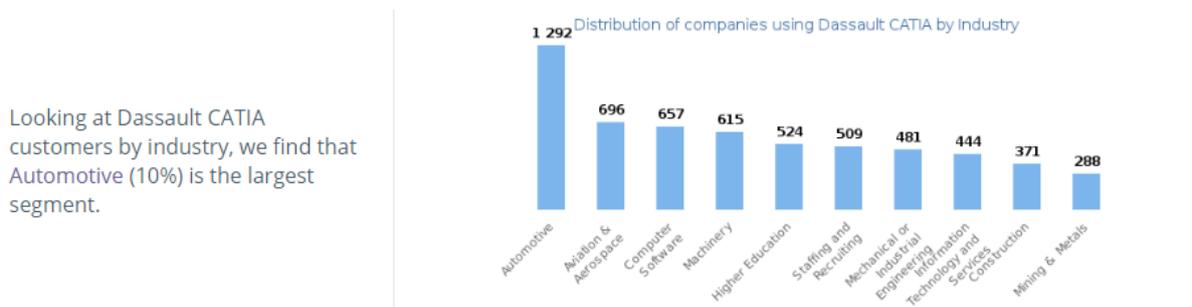
Figure 2. Distribution of companies using CATIA by industry (Powered by iDatalabs).

CATIA delivers the unique ability not only to model any product, but to do so in the context of its real-life behavior (complex simulations). Systems architects, engineers, designers and all contributors can define, imagine and shape the connected world.

The reason why CATIA has been successful and desired for the users it's because all what the program itself delivers. This can be summarized in 3 points:

- A Social design environment built on a single source of truth and accessed through powerful 3D dashboards that drive business intelligence, real-time concurrent design and collaboration across all stakeholders including mobile workers.

- An Instinctive 3D experience, for both experienced and occasional users with world-class 3D modeling and simulation capabilities that optimize the effectiveness of every user.

- An Inclusive product development platform that is easily integrated with existing processes & tools. This enables multiple disciplines to leverage powerful and integrated specialist applications across all phases of the product development process.

CATIA's Design, Engineering and Systems Engineering applications are the heart of Industry Solution Experiences from Dassault Systèmes to address specific industry needs. This revolutionizes the way organizations conceive, develop and realize new products, delivering competitive edge through innovative customer experiences.
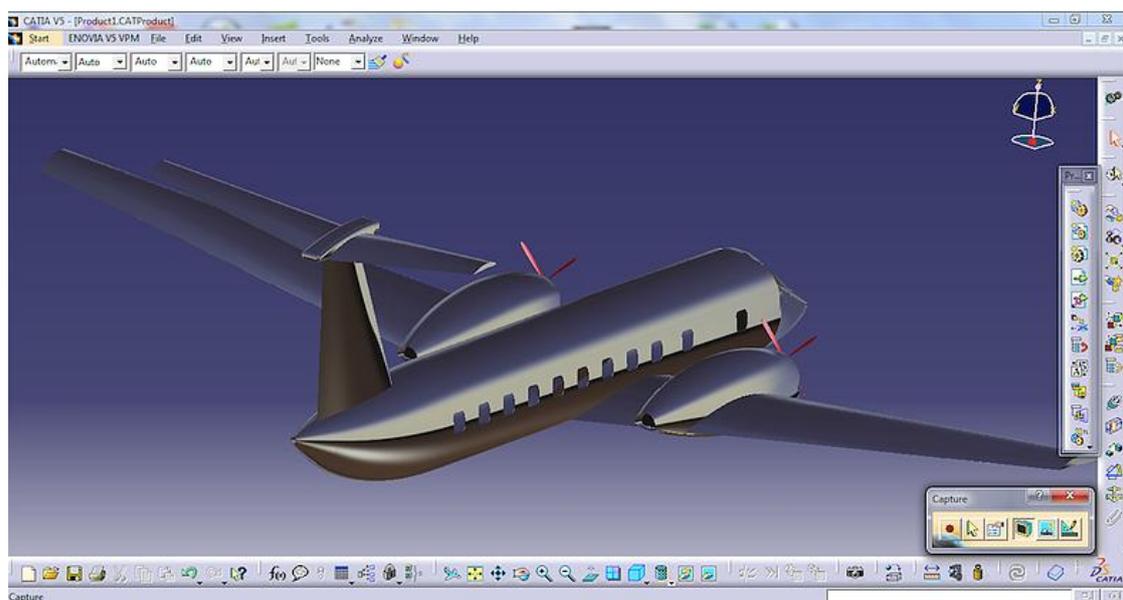
Figure 3. Example of a plane designed using CATIA.

CATIA V5

Since CATIA was created more than fifty years ago, DS has been releasing different versions through the years. CATIA V5 was released in 2001 and had different version updates since then until in 2011 CATIA V6 was released.

Due to the complexity of this program some companies are still working with CATIA V5 although the V6 version came out seven years ago. One of the additional reasons is that CATIA V6 doesn't allow you to store files in a hardware device such as a hard disk or a USB drive, you can only store the files in the cloud. I will use CATIA V5 R21 for the development of the thesis.

Table 1. CATIA release history.

Name/Version	Version History Value	Release Date
Catia v1		1981
Catia v2		1984
Catia v3		1988
Catia v4		1993
Catia v5		1998
Catia v5	R7	2001
Catia v5	R17	2006
Catia v5	R18	2007
Catia v5	R19	2008
Catia v6	R2010	2009
Catia v5	R20	2010
Catia v5	R21	7/2011
Catia v6		2011
Catia v6	R2012	2012
Catia v6	R20	2013

CATIA V5

Objectives and aims

The objective of this thesis is to demonstrate the viability of using DS' CATIA for simulating structural elements such as pillars or beams. One thing that CATIA offers that others programs don't (like SAP2000) is that we will be able to simulate unions between said elements, testing all the union elements at the same time (screws, nuts or washers). For doing this I have separated the course of the project in three different stages.

- First, I'll design and create all the elements (solids) I'll be using in the demonstration.
- Second, I'll assembly all the elements, applying the pertinent restrictions to the elements. This way we will close the breach between our simulation and a real-life simulation.
- Third, after the example model is ready, I'll start with the simulation. Once the model offers coherent results I will take the simulation as a success and I will write my conclusions about it.

Project Scope

As mentioned before, this thesis aims to demonstrate the viability of CATIA for doing simulations of structural elements. However, as this thesis it's only the beginning, we have to talk about the thesis' scope. Since the idea of using CATIA for structures is pretty new this thesis will serve as an introduction for the upcoming years. I've summarized the main delimitations of the project in the following list:

- CATIA is an expensive program. We have mentioned all the capacities that CATIA offers, however everything has its price and CATIA is not a cheap one. The lowest priced license starts around 35.000 € (tells source WorldCAD.com).
- Each version of CATIA (V3, V4, etc.) offers different software layouts, which makes extremely complicated to jump from one version to another. Of course, you cannot open files with previous versions if the file was created with a newer version.
- As CATIA is composed of more than 70 workbenches it's nearly impossible to get to know all of them, which means that you will have to invest a lot of resources in learning before it starts paying off.

- ▪ This thesis offers an example of CATIA's said capabilities applied to a union between structural elements. For more complex simulations you will need a potent hardware/software in order to obtain successful results.

Thesis Outline

This section gives a brief introduction to the chapters in the thesis.

Chapter 1, Introduction

This chapter describes the background to the program, objectives and aims and the scope of the project.

Chapter 2, Theoretical framework

This chapter contains the technical information needed to understand the examples chosen for this thesis. It presents information on the theoretical areas regarding the example (unions, pillars, beams, etc.).

Chapter 3, Designing and Modeling

The third chapter consists on describing the process followed to create and design the testing examples. It will include step-by-step screenshots and extended explanations of the used commands. Also, all the command will be explained on the Nomenclature annex located at the beginning of the thesis.

Chapter 4, Analysis (FEM)

In this chapter we will enter the already mentioned analysis module (FEM), concretely one of the workbenches inside of the module, due to CATIA modules are quiet extend. As done in the previous chapter, we will include explanations and screenshots of the process. At the end, we will see the results of the analysis in 3D graphic.

Chapter 5, Conclusion

Chapter six will consist on an overall review of what the thesis has been. Talking first about what we wanted to demonstrated and ending with the debate of the conclusion *"Is CATIA useful for Structural Analysis?"*.

Theoretical Framework

We have already presented and explained all the facilities and features that CATIA has to offer. In this chapter I will explain the theory of structural design needed to understand all the examples that I'm going to present in chapter number three.

We will begin with the explanation of the structural typology of our example. Once that is completely explained we will get into which materials we are using and different technical details we will have to consider in order to evaluate correctly the results obtained in the FEM module.

Structural Typology

The main example that we will be designing will consist in a union between a pillar and two beams, where apart from the pillar and beams themselves we will have small elements such as screws, nuts and washers. The example will consist will be based on the following example:
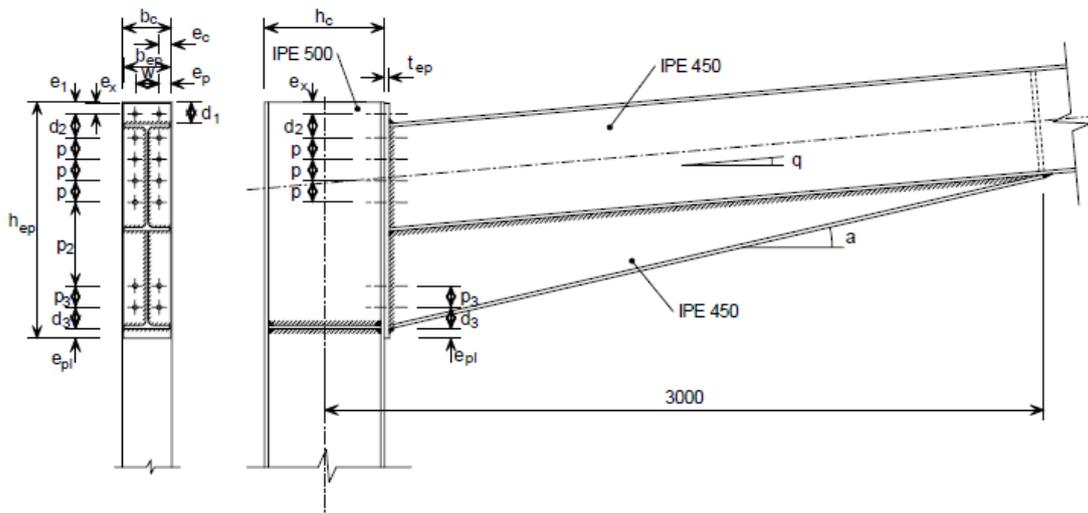
Figure 4. "Lintel union" sketch.

List of parameters (reference to Figure 4. "Lintel union" sketch):

a   Effective "throat thickness" of the weld.

b_p   Width of the edge plate.

d_1 Distance from the upper part of the wing drawn from the lintel to the end of the edge plate.

d_2 Separation between the row of screws in the extended area of the edge plate and the first row of screws below the trailing wing of the lintel.

d_3 Distance from the last row of screws "to shear" to the bottom of the compressed wing of the reinforcement.

e_1 Vertical distance from the edge of the pillar wing to the first row of screws.

e_c Horizontal distance from the edge of the wing of the pillar to the line of screws.

e_c Horizontal distance from the edge of the "edge plate" to the line of screws.

e_{pl} Distance between the lower part of the compressed wing of the lintel and the end of the edge plate.

e_1 Vertical distance from the edge of the edge plate to the first row of screws.

h_c Pillar height.

h_p Height of the edge plate.

n_s Number of screws "to shear".

n_t Number of rows of horizontal screws to tension.

p Separation (distance) between rows of screws in the traction zone.

p Separation between rows of screws in the shear area.

p_2 Separation between the last stressed screw and the first sheared screw.

q Inclination angle of the Lintel union.

r Number of rows of screws. The rows of screws are numbered from the row farthest from the compression center.

t_{fb} Thickness of the lintel wing.

t_{fc} Thickness of the pillar wing.

t_p Thickness of the edge plate.

t_{wb} Thickness of the lintel soul.

t_{wc} Thickness of the pillar body.

w Distance (horizontal distance between strew columns).

All the parameters and distances will be delimited inside CATIA, in the designing and modeling chapter (chapter number three).

Following the theoretical model (Figure 4. "Lintel union" example) we can clearly see that the lintel union will be treated as **rigid**, taking into consideration all the screws and the multiple welds to help the union stay stable. Also, we can notice that said union is made of two main elements, pillars and beams, and secondary elements such as screws, washers and nuts. All the elements will be chosen from catalogs provided by *ArcelorMittal*.

Main element 1: Pillar.

For the pillar we have chosen an IPE 500 profile. We will display said measures in the next chapter, while we explain the design of the profile.

Main element 2: Beams.

For the beams we will use IPEs 450 profiles. However, one of them will be trimmed, forming a "triangle" profile like-showed in the image below:
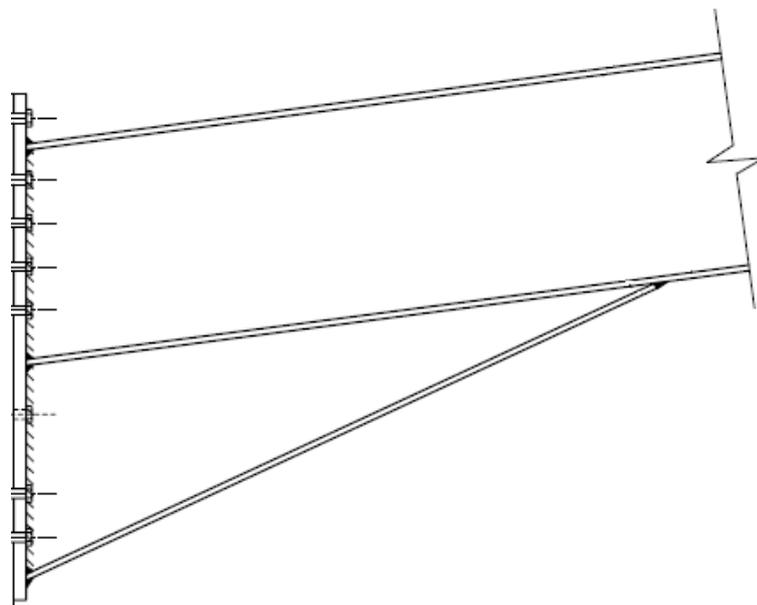
Figure 5. Trimmed beams sketch.

This type of assembly between beams is frequently used to reinforce the union and to help avoid having an unnecessary over-stressed pillar. This reinforcement will help will Axial and Shear efforts, and mainly with the bending moment.

Secondary elements: Screws, nuts and washers.

All this union elements will be chosen based on the rules of these specific profiles. We will have to choose the most recommended ones for our union.

Now that we know how the examples are going to look like we can talk about theoretical parameters we will have to consider in order to understand the results taken from the FEM module.

Materials. Properties and Structural Utility

As we are working with structural elements, the material we will be using for our demonstration will be **Steel**. Luckily CATIA offers the possibility of using this material in its simulations. However we won't be able to decide which kind of steel we will be using in terms of deformation stress, but this will not be a problem because we will be getting the maximum stress out of the whole body thanks to the results taken from the analysis module. Once we have that we only have to choose the stress-level steel that matches our necessities. In the figure below, we can see the main and common properties for all kinds of steel.

Properties	Symbol	Value	Units
Thermal conductivity	k	25	W/m^{0}C
Density	ρ	8000	Kg/m^3
Specific heat	c_p	400	J/kg^{0}C
Solidus temperature	T_s	1500	^{0}C
Liquidus temperature	T_l	1525	^{0}C
Latent heat of fusion	H_m	1.93×10^9	J/m^3
Absorption coefficient	α	1×10^5	m^{-1}
Staring temperature of Martensite	M_s	350	^{0}C
Austenization temperature	T_A	750	^{0}C

Figure 6. Steel physical properties (provided by ResearchGate).

The metal steel serves to denominate a mixture of iron with a quantity of variable carbon between 0.03% and 2.14% in mass of its composition, depending on the degree. If the alloy has a carbon concentration greater than 2.14%, the resultant steel cannot be forged. Depending on the percentage of carbon, the properties of the resultant steel will change. In the figure below we can appreciate how steel changes based on the carbon content and the temperature
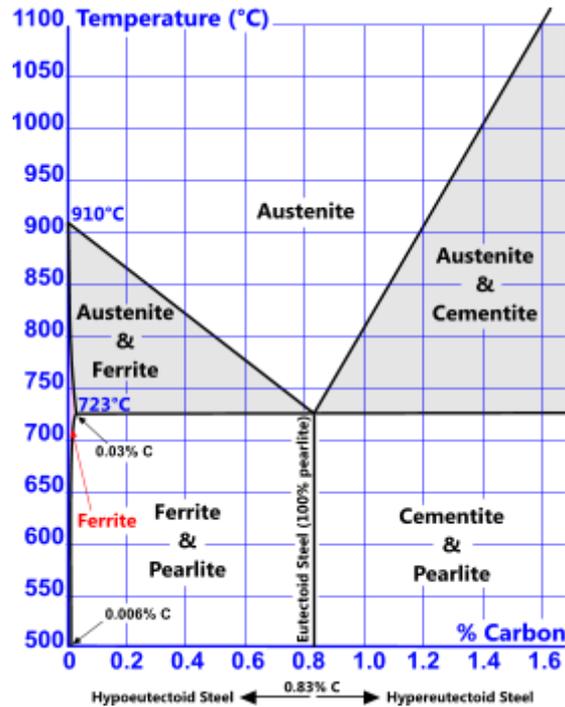

Figure 7. Steel composition depending on the Carbon percentage.

Depending on the field of work, we will use different combination of iron and carbon to create our most desired steel, being the one with a 0.83 % of Carbon the one called Eutectoid Steel (composed only of pearlite).

Designing and Modeling

I'll be presenting three different examples. The first two will be simple examples such as pillars and beams and the third one will be the lintel union we have already explained. For creating these examples we will be using two of the main CATIA modules: *Part Design* and *Assembly Design*. The simple examples will be useful to show how the FEM module works and how to interpret the results (we will go deeper in the FEM module in the fourth chapter).

Example 1: Pillar IPE 500

In this first example we will be designing and testing a pillar following the standard measures established by *ArcelorMittal* for an IPE 500 (Steel):

Perfiles I de alas paralelas (Perfil europeo) (continúa)
Dimensiones: IPE 80 - 600 según la Euronorma 19-57; IPE A 80 - 600; IPE O 180 - 600; IPE 750
Tolerancias: EN 10034: 1993
Estado de la superficie según norma EN 10163-3: 2004, clase C, subclase 1

European I beams (continued)
Dimensions: IPE 80 - 600 in accordance with Euronorm 19-57; IPE A 80 - 600; IPE O 180 - 600; IPE 750
Tolerances: EN 10034: 1993
Surface condition according to EN 10163-3: 2004, class C, subclass 1

Travi I ad ali parallele (continua)
Dimensioni: IPE 80 - 600 in conformità con le Euronorm 19-57; IPE A 80 - 600; IPE O 180 - 600; IPE 750
Tolleranze: EN 10034: 1993
Condizioni di superficie in conformità con EN 10163-3: 2004, classe C, sottoclasse 1

| Denominación Designation Designazione	Dimensiones Dimensions Dimensioni						Dimensiones de construcción Dimensions for detailing Dimensioni di costruzione					Superficie Surface Superficie	
	G	h	b	t_w	t_f	r	A	h_i	d	$\emptyset$	P_{min}	P_{max}	A_L	A_G
	kg/m	mm	mm	mm	mm	mm	mm²	mm	mm		mm	mm	m²/m	m²/t
							x 10³							
IPE A 500*	79,4	497	200	8,4	14,5	21	101	468	426	M 24	100	112	1,741	21,94
IPE 500	90,7	500	200	10,2	16	21	116	468	426	M 24	102	112	1,744	19,23
IPE O 500+	107	506	202	12	19	21	137	468	426	M 24	104	114	1,760	16,40

Figure 8. IPE 500 Measures (Provided by ArcelorMittal's catalog).

Taking on the measures displayed by the catalog, we will initiate and start with the designing the profile. For that, we will start "sketching" the profile in our program. We will sketch the face of the profile as a start. However, CATIA allows us to create simple sketches and form that, we can adjust the profile with different operations. We'll do it this way because this will allow us to quickly re-define the sketch if we decide to change the profile from an IPE 500 to another.

As described before, we will start with the sketch of the profile's face ending with the following result:
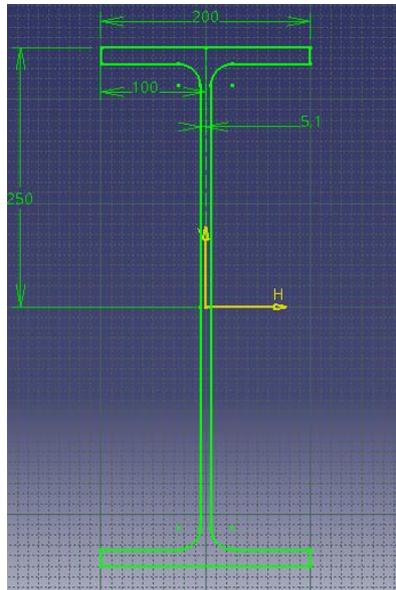
Figure 9. IPE 500 face sketch.

Once we have the sketch completed, we will use the *"PAD" command* to create the profile in three dimensions. We'll be using the PAD command frequently in structural design to extrude all kind of solids such us pillars, beams, screws, etc.

In the following figure we will see all the options offered by the PAD command. In our case we have decided to go with a 3 meters IPE 500 pillar as displayed in the command menu:
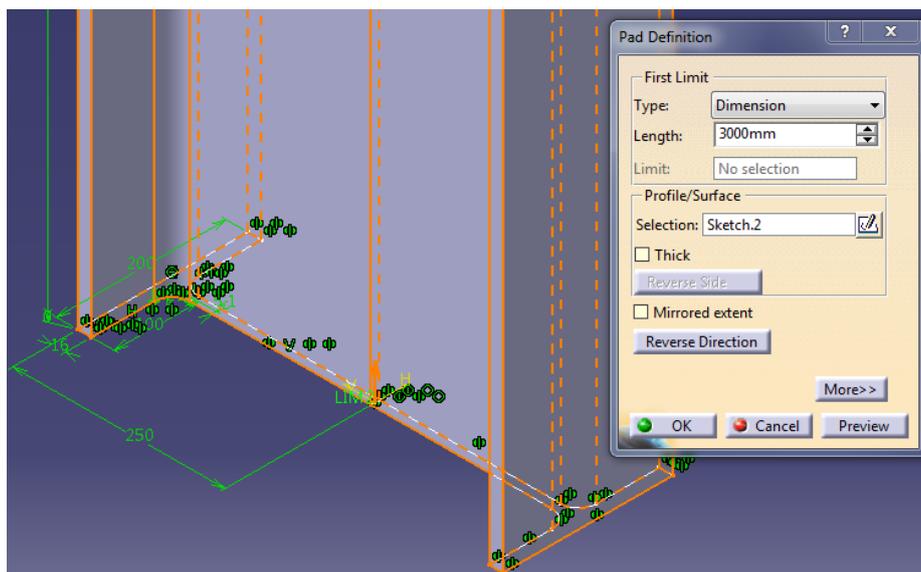

Figure 10. IPE 500 PAD operation.

If we look at the PAD command menu, we can see that the command offers different options:

- **Type:** Inside of the "type" menu, The PAD command offers us 5 different options regarding how we want to create our solid body:
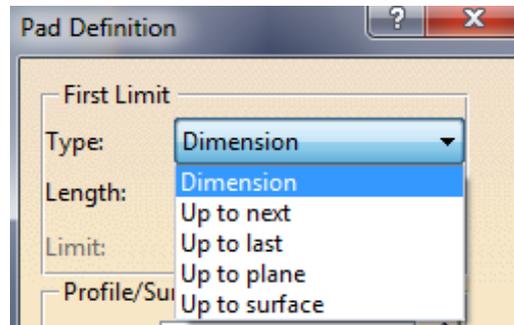

Figure 11. Pad command menu options.

Dimension: This option allows us to decide the length of the operation. This is the option we have used.

Up to next: With "Up to next" the PAD operation will continue until it faces the next body in the same workspace. This option would not work for us due to we only have the current body in our workspace, so theoretically the command will continue to the infinite (of course CATIA does not allow you to use this option if there is not another body to work with).

Up to last: "Up to last" works in a similar way as "Up to next". The PAD operation will continue until it faces the last surface of the last body in the same workspace. As it would happen with "up to next", this option would not work for us due to we only have the current body in our workspace, so theoretically the command will continue to the infinite (of course CATIA does not allow you to use this option if there is not another body to work with).

Up to plane: Like "Up to next" and "Up to last", The PAD operation will continue until it faces a plane. This plane will have to be designated previously by us. This is a really common option because it allows you to use auxiliary planes for this operation if, for example, you don't know the exact length of the path, or you don't have a body to refer to.

Up to surface: As we can deduce, this command works the exact same way as "Up to plane" but with surfaces. It is important to mention that we can use both

another body's surface and a plane. So this command can be used in substitution of "Up to plane" in some occasions.

- **Length:** This feature allows us to introduce the desired length of our operation. This is because we have previously chosen the "Dimension" option. This feature will vary of we choose for example "Up to next".

- **Limit:** Limit works the same way "length" does, depending on the Type selection, it will be deactivated or not. In our case it is deactivated because there is no limit needed to be defined.

- **Selection:** In selection we will have to select the sketch we have created previously. Notice that CATIA will op an error up if the sketch is not adequate, for example in PAD operations, we cannot have open profiles.

- **Thick:** We will activate thick if we want to add some extra material in the corners of our body. In our case we will deactivate this option

- **Mirrored Extend:** This option is really useful in PAD operation because it allows us to extrude the body in both directions, not only one like we are doing. In case we activate this option, a secondary menu will display and we will be able to adjust it exactly like we have done it with this one.

Once we have finished adjusting the menu we will execute the command obtaining the following result:
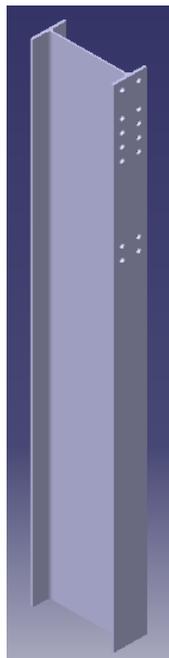
Figure 12. IPE 500 PAD operation result.

We can notice that there are holes in one of the face of the profile. This is due to that we will use the solid later on our third example, where we will assemble different solids and we will need the holes for the screws. For creating these holes, we will start with creating a single hole. Then, based on the first one, we will create the rest.

First of all we will create the sketch for the "Hole" command (remember that in CATIA you always need a sketch in order to use commands for solid bodies):
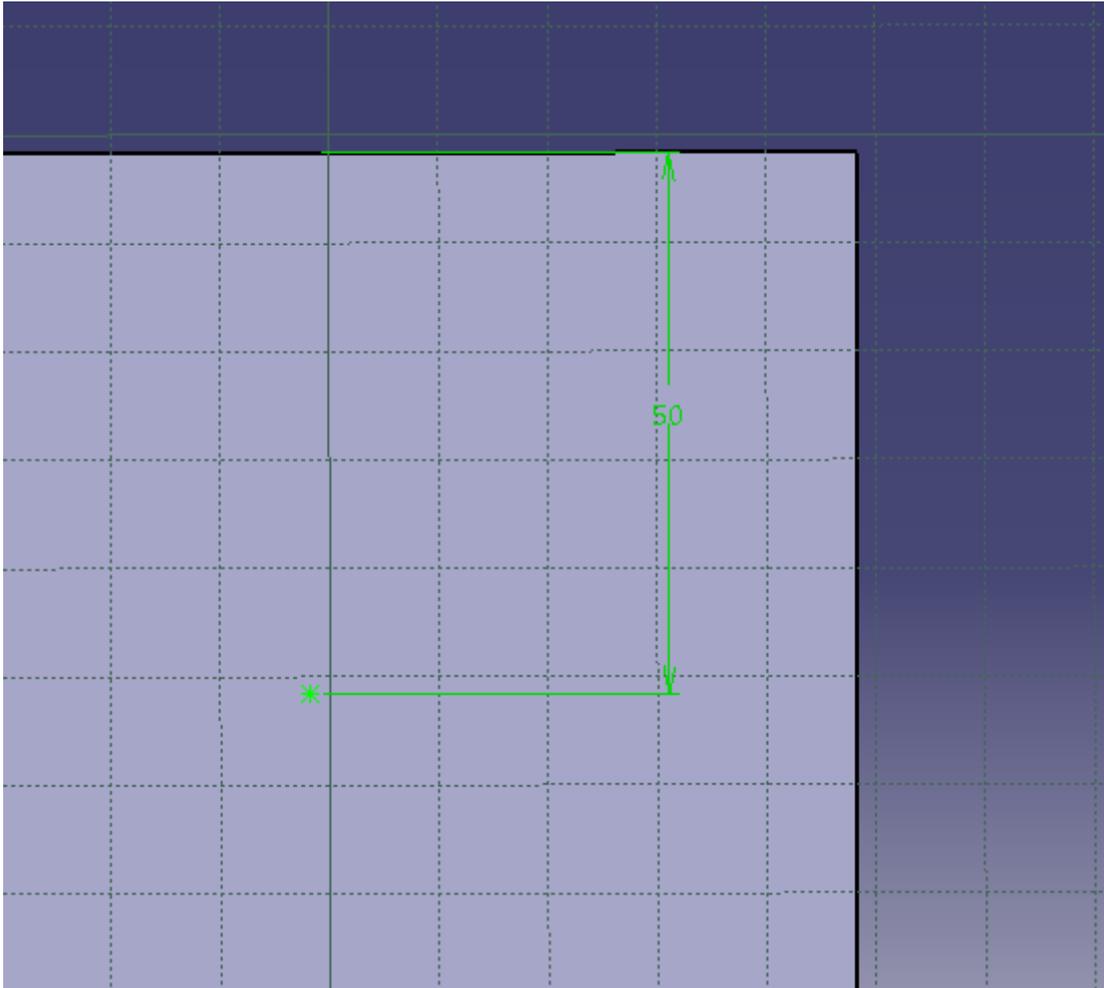

Figure 13. Hole command sketch.

The Figure 11 may be a bit confusing because the only 2 elements we are able to appreciate are a 50 mm dimension restriction and a point with an asterisk shape. This sketch is this simple because for the "Hole" command we only need the location where we want to make the hole.

Said that, we will open the Hole command menu and, as we have done with the PAD command we will explain all the option and features included. Once we have selected the sketch we have seen before this is what we get:
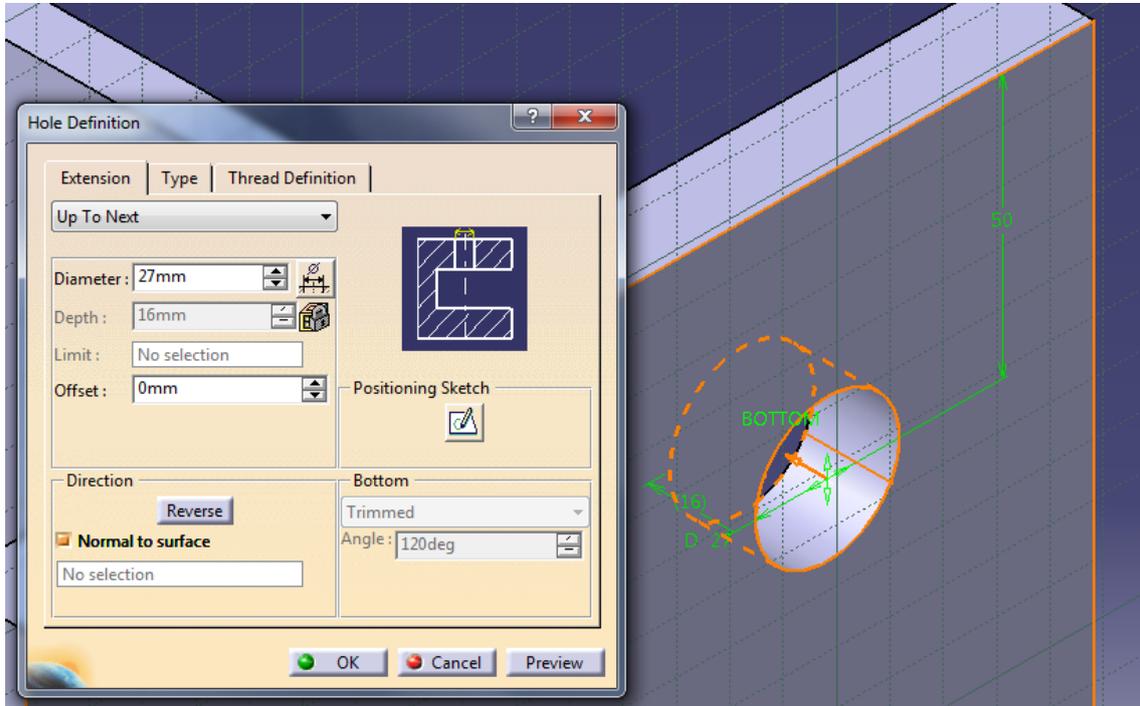

Figure 14. IPE 500 Hole operation.

As commented before, we are going to proceed with the explanation of the Hole command menu:

Extension

- First of all we bump into several options that will decide mainly how we are going to make the Hole. As you can see in the Figure 15 below, these options correspond with the PAD operation "Type" options.
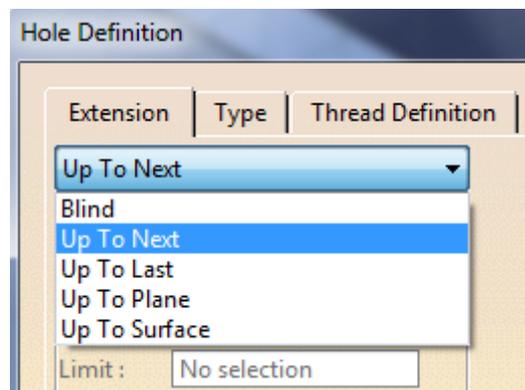

Figure 15. Hole command "Extension" menu options.

The "Blind" option corresponds with the "Distance" option we saw when explaining the PAD command menu.

- **Diameter:** This option will determine, as its name says, the diameter of the hole. This option has to be mandatorily fulfilled.

- **Depth:** Depth will be activated or deactivated depending on the selected option in the window we have just explained. In our case, as we have selected "Up to next", the depth option will get filled automatically (16).

- **Limit:** The Limit option works in a similar way as Dimension does. It will be deactivated unless we select commands such as "Up to plane" or "Up to surface".

- **Offset:** This feature is really useful when operation with commands such as PAD or Hole. It allows us to, despite the selection we have previously made, increase or decrease the depth of the hole or the extrusion. For example, you can select "Up to last" and then apply a dimension in the offset option to modify the selection.

- **Direction:** As the name says, we can indicate the command the direction of the hole. This can be done with an auxiliary profile (line) or like we have done it in this example, selecting "normal to surface" and creating a hole normal to the profile we want to make the hole on.

- **Bottom:** This feature will only activate when we select the "Blind" option in the first window. It will allow us to change the ending of the hole (in case we are doing holes for nails and they need a "V" ending.

- **Positioning sketch:** If we have not done it before pressing the Hole command we will have to select the sketch with the location to indicate where we want to make the hole.

Type

- As we can see in the figure below (Figure 16) inside of the "Type" window we have another bunch of options. This window basically allows us to define which kind of hole we want to make (simple, tapered, etc.). As we will be going for simple wholes, we will not touch anything.

- The anchor point is used to determined where to apply the angle deviation in case we are working with one (top or bottom).
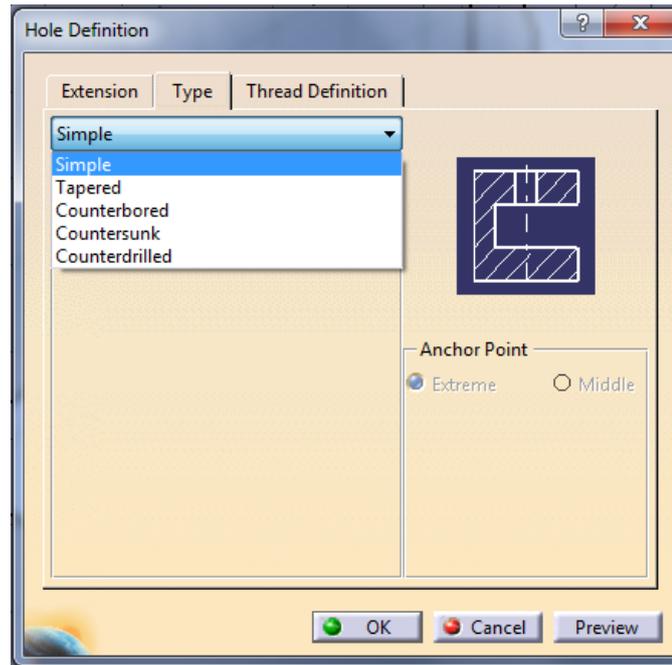
Figure 16. Hole command 'Type' menu options.

Thread Definition

▪ The thread definition window allows us to create a threaded hole. It's important to know that the threaded effect will only by for analysis purposes, nothing will change visually talking. Following the figure below we can see multiple features:
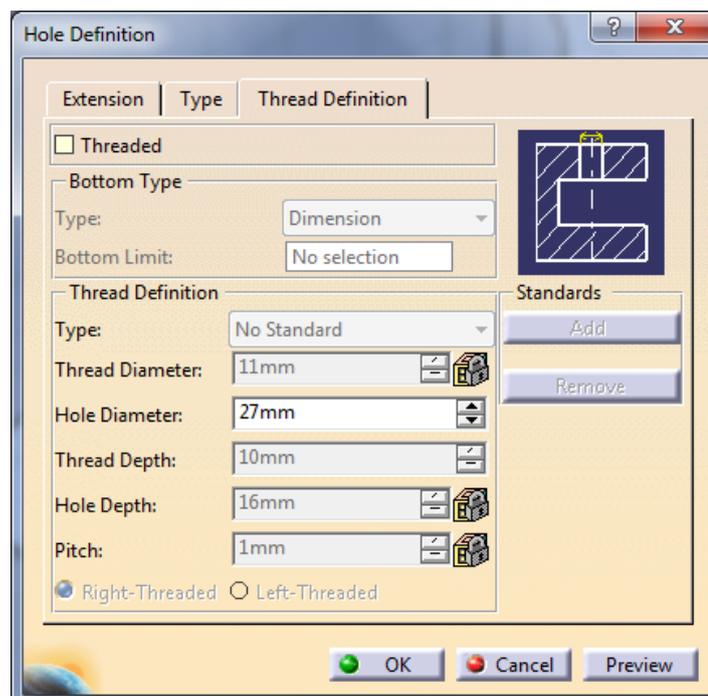

Figure 17. Hole command 'Thread Definition' menu options.

As we can already see, we have to activate the Threaded option if we want to edit this window. I will not go deeper in the explanation due to we are not working with threaded holes.

I will just mention that in the **'Type' option (inside Thread Definition)** we have three options:
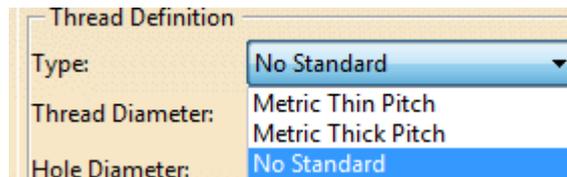
Figure 18. Thread Definition 'Type' menu options.

The first two options refer to standardized ways of threading the hole (thin or thick pitch); we will go deeper in here when we explain the screws in the third example. The "No standard" option is useful in case we want to create our own custom thread.

Now that we have adjusted all the settings we can just complete the operation obtaining the following result:
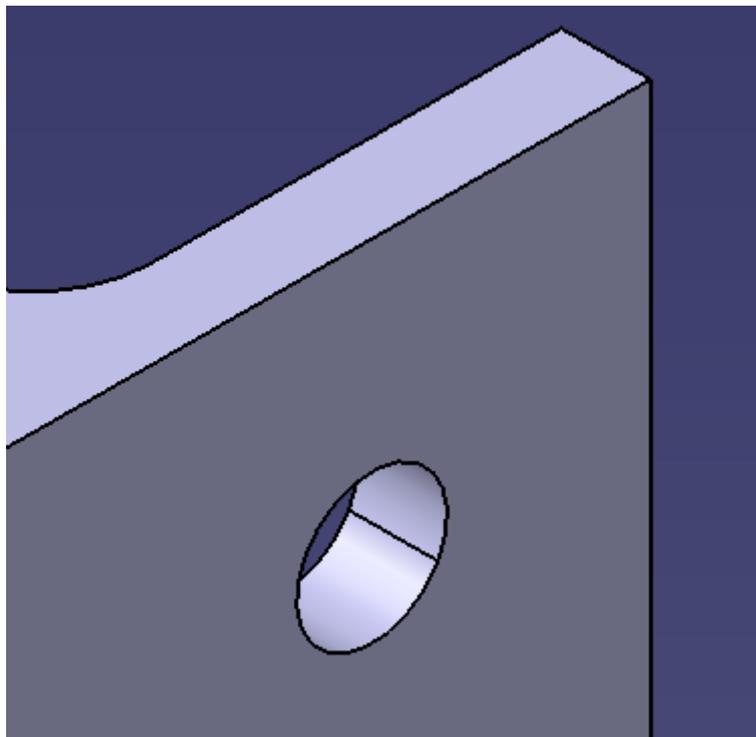

Figure 19. IPE 500 Hole operation result.

We have finished creating the hole we will use in further examples to introduce the screws. However we have only created one of them and we will need fourteen for the example 3. Luckily this is CATIA and we have a command useful to replicate the same operation all over the solid body.

This command is *Rectangular Pattern*. Now, we will go through the command and its multiple menu options:
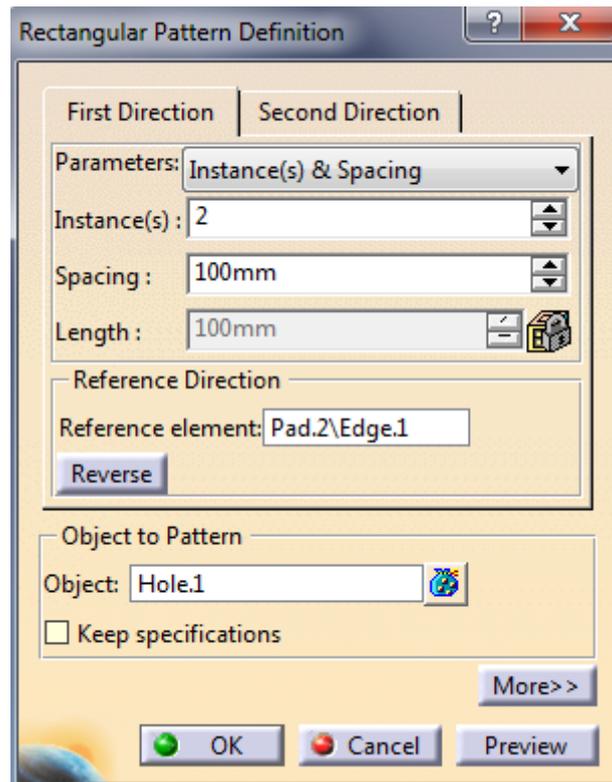

Figure 20. Rectangular pattern command menu 'First Direction'

As we can see in the menu, we have two main windows named **First and Second Direction**. We will start explaining the first window and all the features that come with it:

First Direction

- **Parameters**: As we have seen in previous commands such as PAD, the first thing we have in the menu area is the window to determine how we want to specify the command the way of working. This command is not different:
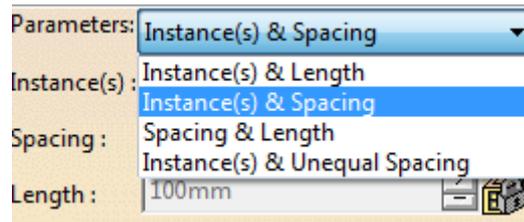
Figure 21. Rectangular pattern command menu 'Parameters' options.

Instance(s) & Length: The first option will ask us to determine the number of instances we want to create and the total length of the command. For example, if we want to have 15 instances in 150 cm of length, CATIA will distribute an instance every 10 cm.

Instance(s) & Spacing: This option works in a similar way as the *Instance(s) & Length* option does. First we will have to decide the number of instances that we want to create and then, the distance between instances.

Spacing & Length: In this option we will be able to work with distances instead of instances. We will have to tell the program the distance between instances and the total length of the command. Obviously, if the first distance cannot be placed exactly in the second distance, the last instance won't be at the same distance from the end as it is from the previous instance.

Instance(s) & Unequal Spacing: The last option is only used in very specific cases. It allows us to create a defined number of instances with different spacing distance between them. This option is very useful when we need a lot of holes but they do not follow a pattern itself.

Now if go back to the Figure 20, we will be able to see that we selected Instance(s) & Spacing for our pattern. We will go for two instances separated 100 mm.

▪ **Reference Direction:** In the following option we will have to tell CATIA the surface or object where we want to apply the pattern. For our case we will select one of the external surfaces in the profile:
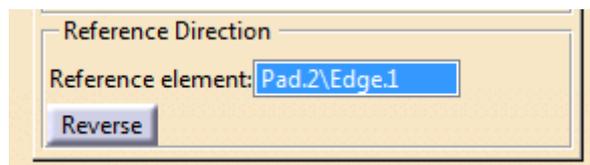

Figure 22. Rectangular pattern command menu 'Reference Direction' options.

- **Object to pattern:** In this section we will have to determine what shape or profile we want to create a pattern with. In our case we will select the hole we have previously created:
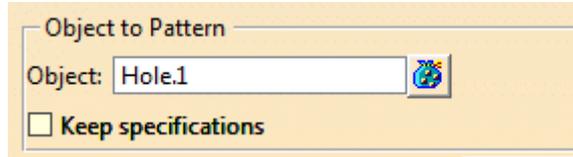

Figure 23. Rectangular pattern command menu 'Object to Pattern' options.

The *keep specifications* button allows us to create the pattern keeping all the specifications defined in the object command. If we select this option all the holes will have the same specifications as the original one.

Second Direction

- As we can see in the figure below (Figure 24) we can see that both menus look exactly the same. Because of this, we will explain the date we have introduced in the *second direction* menu:
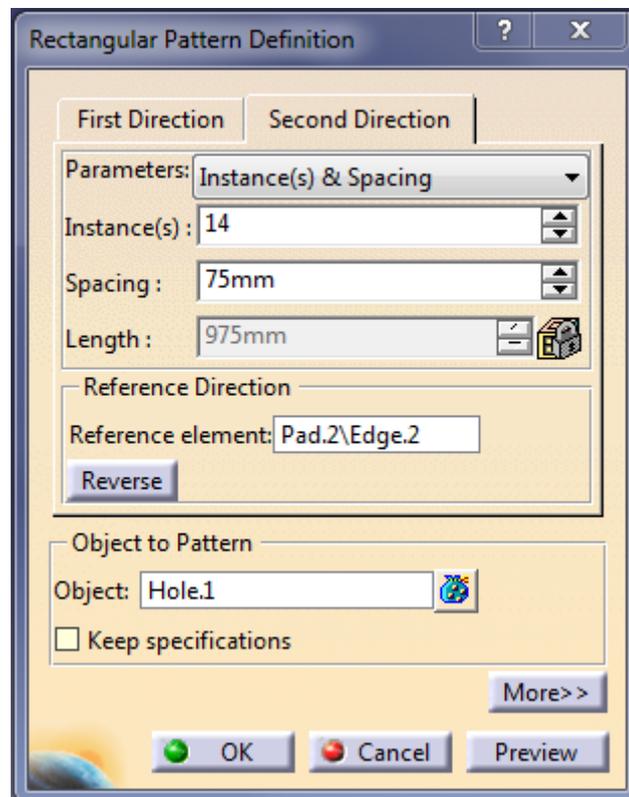

Figure 24. Rectangular pattern command menu 'Second Direction'.

As we can see in the figure we have chosen the same reference element and object to pattern. However we will have fourteen instances separated 75 mm each. This will lead us to the following preview:
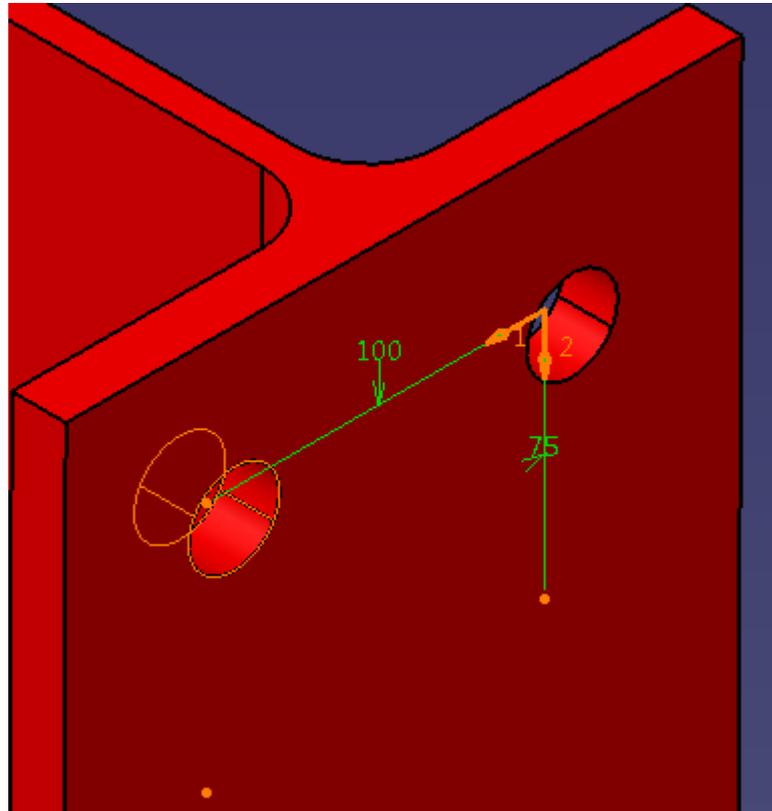
Figure 25. Rectangular pattern command final preview.

As we can appreciate, the command places small orange dots where we have defined that we want the instances to be. If we want the command to execute the object to pattern in one of the instances we just have to click on the dots. Once we have finished selecting, we will click ok and the command will be executed, obtaining, in our case, the following result:
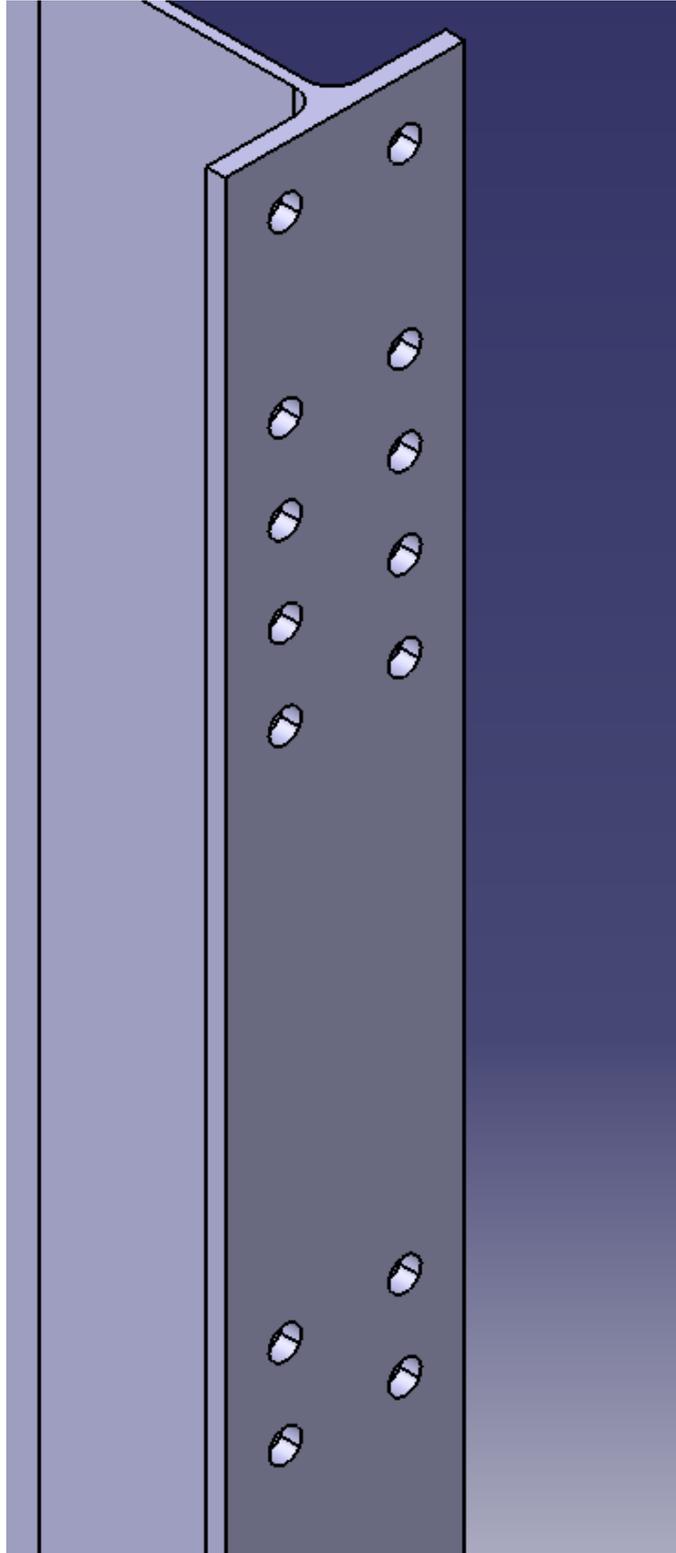
Figure 26. Rectangular pattern command final result.

In terms of designing and modeling, we have finished the first example. Now we will review the second and the third one respectively, taking into consideration that a lot of commands will be repeated.

Example 2: Beams IPE 450

The Example 2 will consist of a union between 2 beams, one of them trimmed (just as presented in the Figure 5). For explaining the process of creation we will explain each beam separately and then, we will assemble them, completing our example.

IPE 450 (Normal)

As we have done with the first example, we will start taking a look into ArcelorMittal's catalog for an IPE 450 steel profile:

● **Perfiles I de alas paralelas (Perfil europeo)** (continúa)
   Dimensiones: IPE 80 - 600 según la Euronorma 19-57; IPE A 80 - 600; IPE O 180 - 600; IPE 750
   Tolerancias: EN 10034: 1993
   Estado de la superficie según norma EN 10163-3: 2004, clase C, subclase 1

● **European I beams** (continued)
   Dimensions: IPE 80 - 600 in accordance with Euronorm 19-57; IPE A 80 - 600; IPE O 180 - 600; IPE 750
   Tolerances: EN 10034: 1993
   Surface condition according to EN 10163-3: 2004, class C, subclass 1

● **Travi I ad ali parallele** (continua)
   Dimensioni: IPE 80 - 600 in conformità con le Euronorm 19-57; IPE A 80 - 600; IPE O 180 - 600; IPE 750
   Tolleranze: EN 10034: 1993
   Condizioni di superficie in conformità con EN 10163-3: 2004, classe C, sottoclasse 1

| Denominación Designation Designazione		Dimensiones Dimensions Dimensioni						Dimensiones de construcción Dimensions for detailing Dimensioni di costruzione					Superficie Surface Superficie	
	G kg/m	h mm	b mm	t_w mm	t_f mm	r mm	A mm²	h_i mm	d mm	Ø	Pmin mm	Pmax mm	A_L m²/m	A_G m²/t
							x 10³							
IPE A 450*	67,2	447	190	7,6	13,1	21	85,6	420,8	378,8	M 24	100	102	1,603	23,87
IPE 450	77,6	450	190	9,4	14,6	21	98,8	420,8	378,8	M 24	100	102	1,605	20,69
IPE O 450+	92,4	456	192	11	17,6	21	118	420,8	378,8	M 24	102	104	1,622	17,56

Figure 27. IPE 450 Measures (Provided by ArcelorMittal's catalog).

Taking on the measures extracted from the catalog, we will initiate and start with the designing the profile. We will start with the sketching just like we did in the Example 1.

Remember that we have decided to sketch one of the faces of the profile but CATIA allows us to create simple sketches and form that, we can adjust the profile with different operations.

As described before, we will start with the sketch of the profile's face ending with the following result:
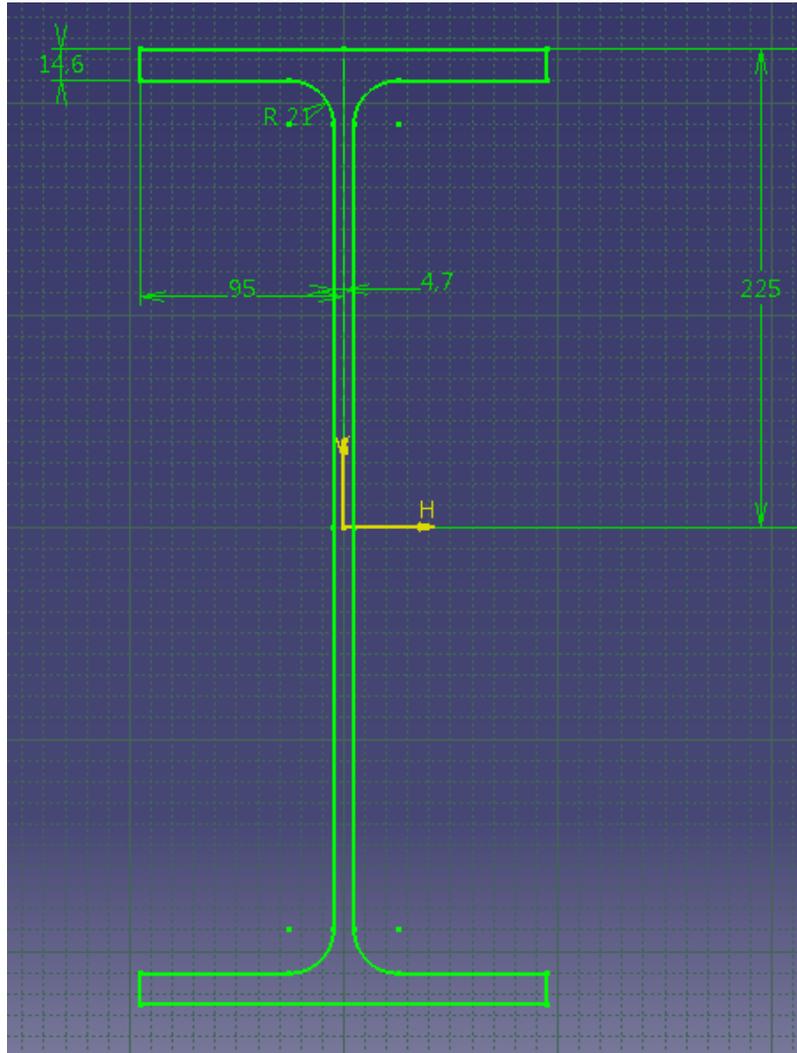
Figure 28. IPE 450 face sketch.

Once we have the sketch completed, we will use again the PAD command. As well as we did in the first example, we will introduce the data in the command's menu:
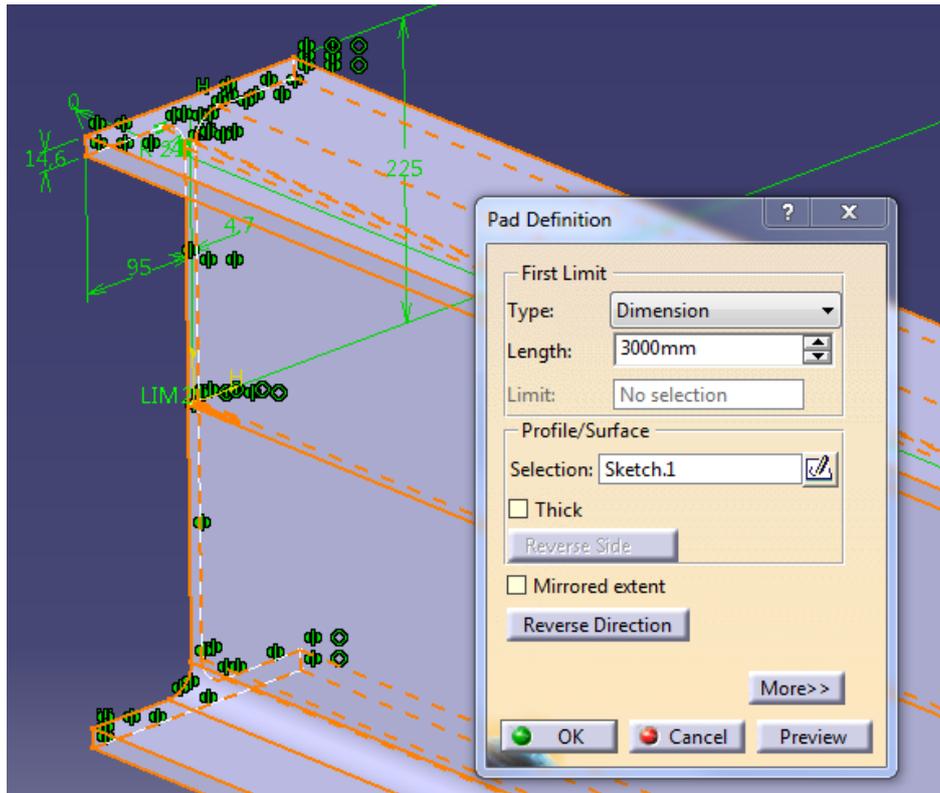
Figure 29. IPE 450 PAD operation.

We have selected a 3 meters beam for the example, just like we did with the pillar. Because we have already seen all the different options we have in the PAD operation menu, I will summarize the options we have selected for the operation:

Type: We will go with dimension, putting a length of three thousand millimeters.

Selection: Of course we will select the sketch we have previously created.

After this we will execute the command obtaining the following result:
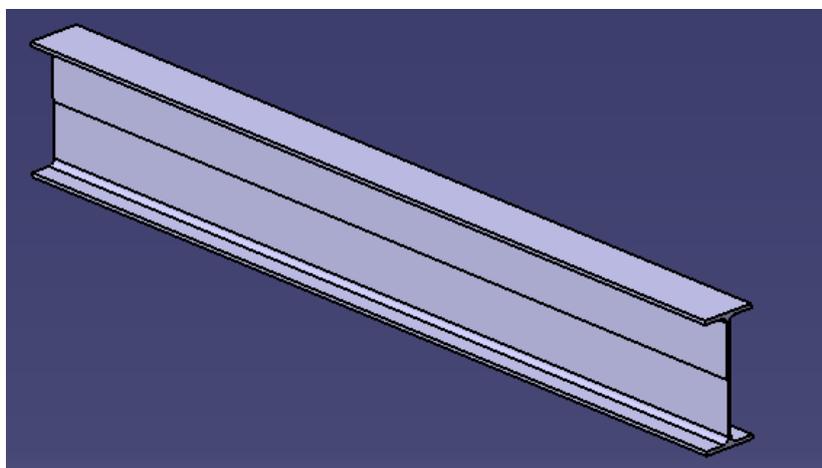
Figure 30. IPE 450 PAD operation result.

As this example is about beams, our work here is done. Again we can see how quickly we can create elements with CATIA, in about 5 minutes you can have your 3D profile ready to jump into the FEM module.

IPE 450 (Trimmed)

For creating the trimmed profile we will follow the same procedure as we did with the normal profile. If we check again the Figure 27 we can jump directly into the sketch phase. Of course, the resultant sketch will have the same design as the one we used for the normal one:
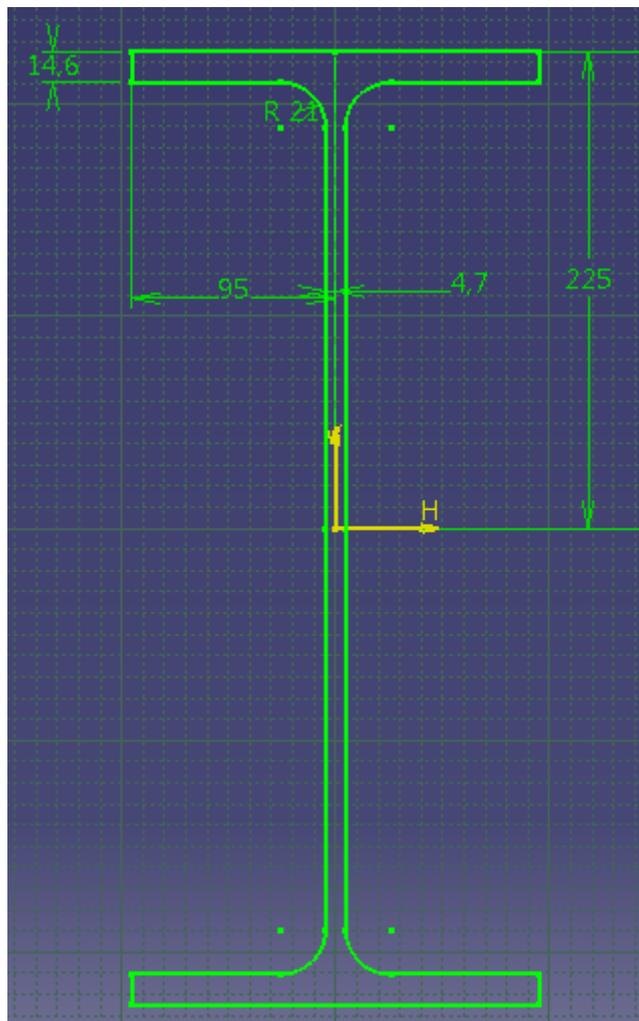

Figure 31. IPE 450 Trimmed face sketch.

Once we have the sketch, we will do as we did previously. We will execute the PAD command with the same data we chose for the normal profile:
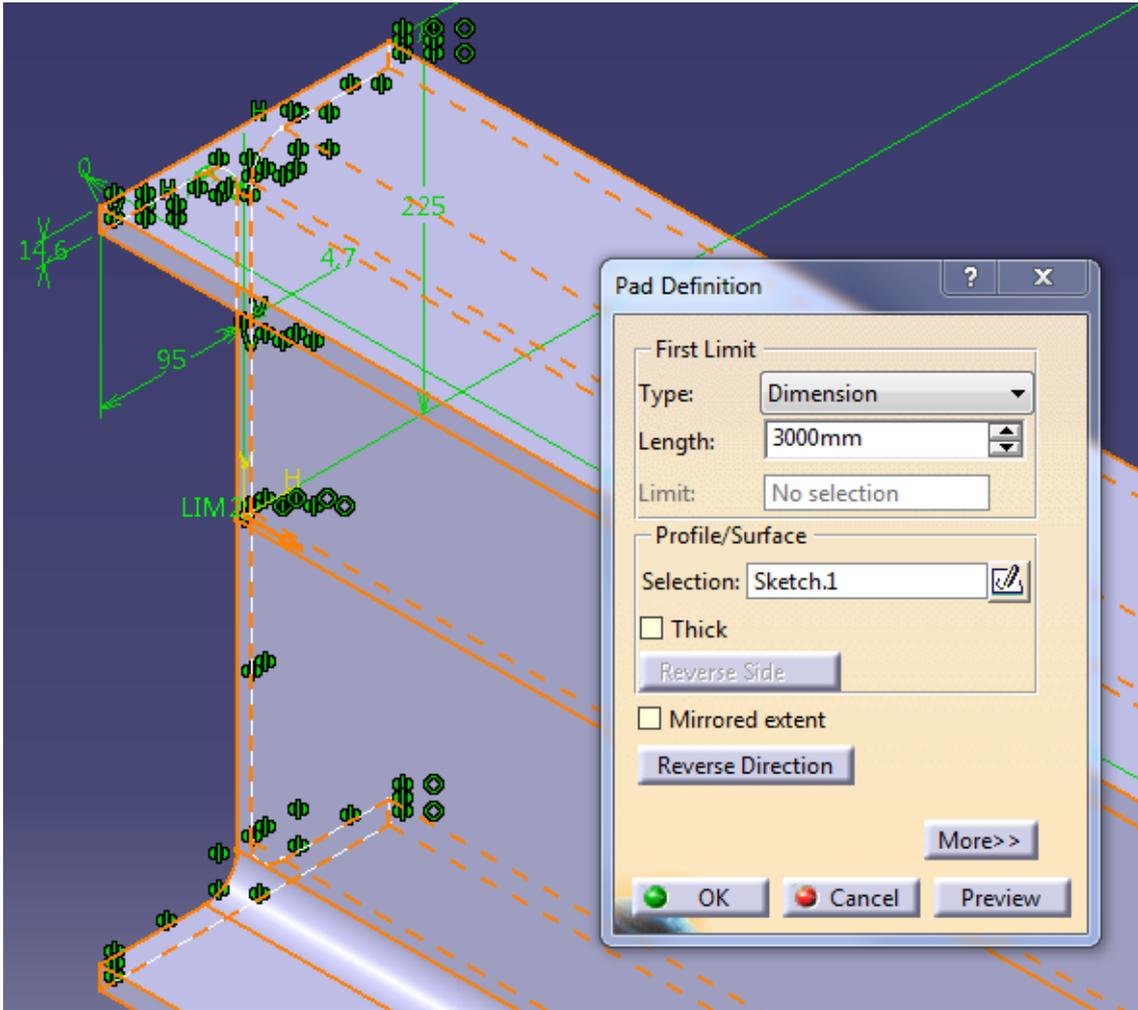
Figure 32. IPE 450 Trimmed PAD operation.

Like we said, we introduce the exact same data and then we execute the command obtaining the same result we obtained previously. After this I will explained how I decided to trim the profile, introducing a new command.
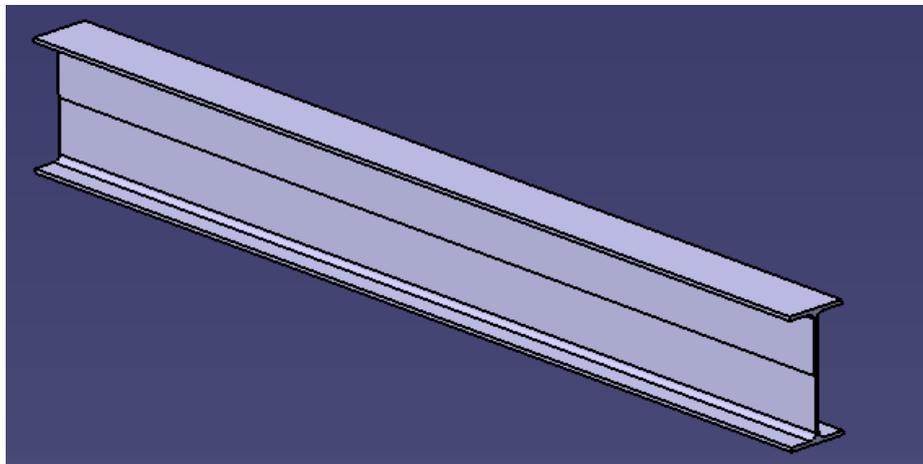

Figure 33. IPE 450 Trimmed PAD operation result.

For the trimming process we will use a new command called *Pocket*. The Pocket command works in a very similar way as The Hole command does. The difference is that Hole only works with circles and with Pocket you can select pretty much any profile.

Like we just said we will need to create the sketch for the command. We have to keep in mind the kind of pocket we want to make. In our case we want to trim half of the profile:
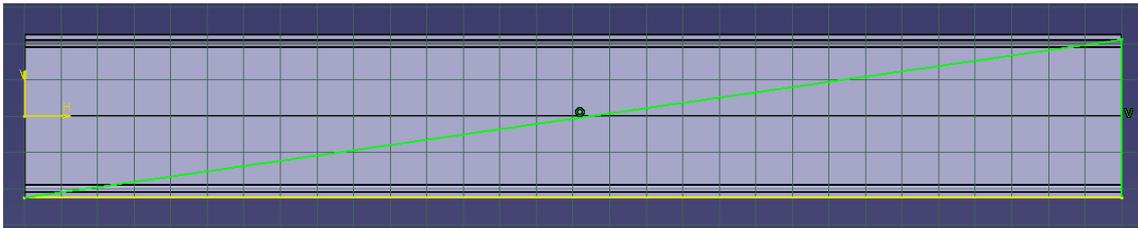

Figure 34. IPE 450 Trimmed Pocket sketch.

Based just on the Figure 34, the sketch may be unclear. What we have created is a triangle based on the geometry of the profile so when we execute the command, the trimmed will happen perfectly.

Now that the profile is ready we execute the command:
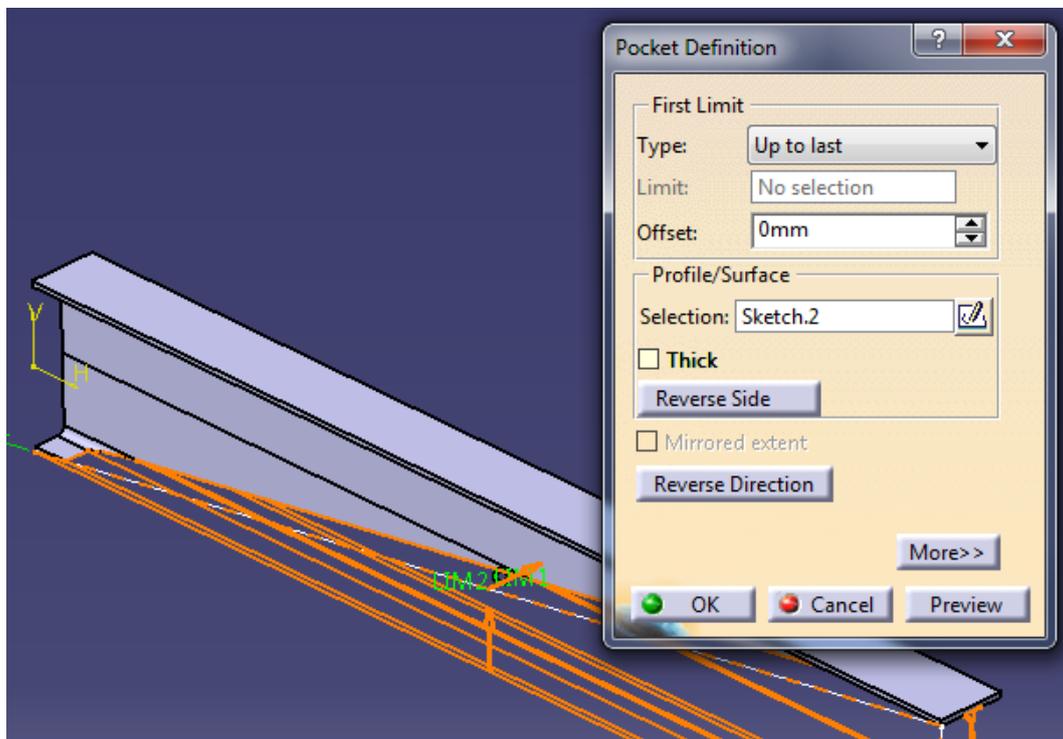
Figure 35. IPE 450 Trimmed Pocket operation.

As we can see, the command's menu is really similar to the PAD command's menu. In CATIA the majority of the commands inside of a module follow the same structure in their menus.

Just like we did with the PAD command, we will explain the menu step by step. However as the menus are almost the same we will not go deeper in the options (in case you need to refresh the definition or the function of one of the options, please refer to the page 20, where we explained the PAD command menu options).
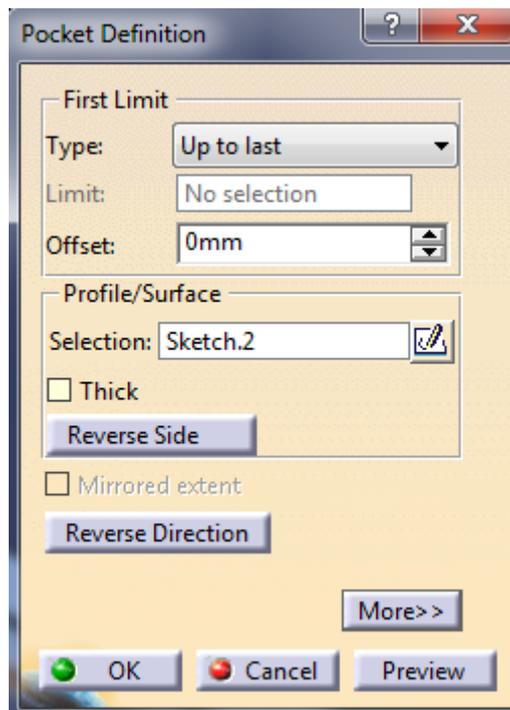

Figure 36. IPE 450 Trimmed Pocket menu options.

Inside of the First Limit section we have the following options inside of **Type**: Dimension, Up to next, Up to last, Up to plane and Up to surface.
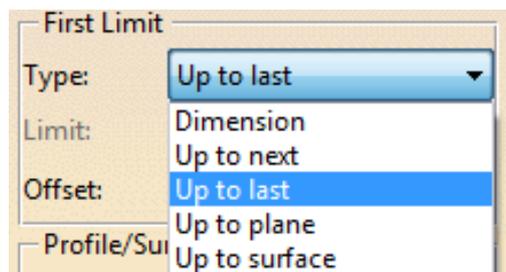

Figure 37. IPE 450 Trimmed Pocket 'Type' menu options.

As I said before, we have already gone through all the options. In this case we will choose the option "Up to last" as we want to trim the whole profile. This option is really useful because even if we change the dimensions of the profile in the future, the command will execute the operation exactly the same way.

If we take a look into the Figure 35, we will see that when we select "Up to last" the offset option appears. Remember that we already went through this feature in Page 24. In our case, we will select zero millimeters of offset because we are not interested in prolonging the operation.

Once we have selected the type, selecting the profile is the last thing to do (Because we are not selecting the Thick option here). We will select our sketch we commented before (Figure 34) and execute the command obtaining the following result:
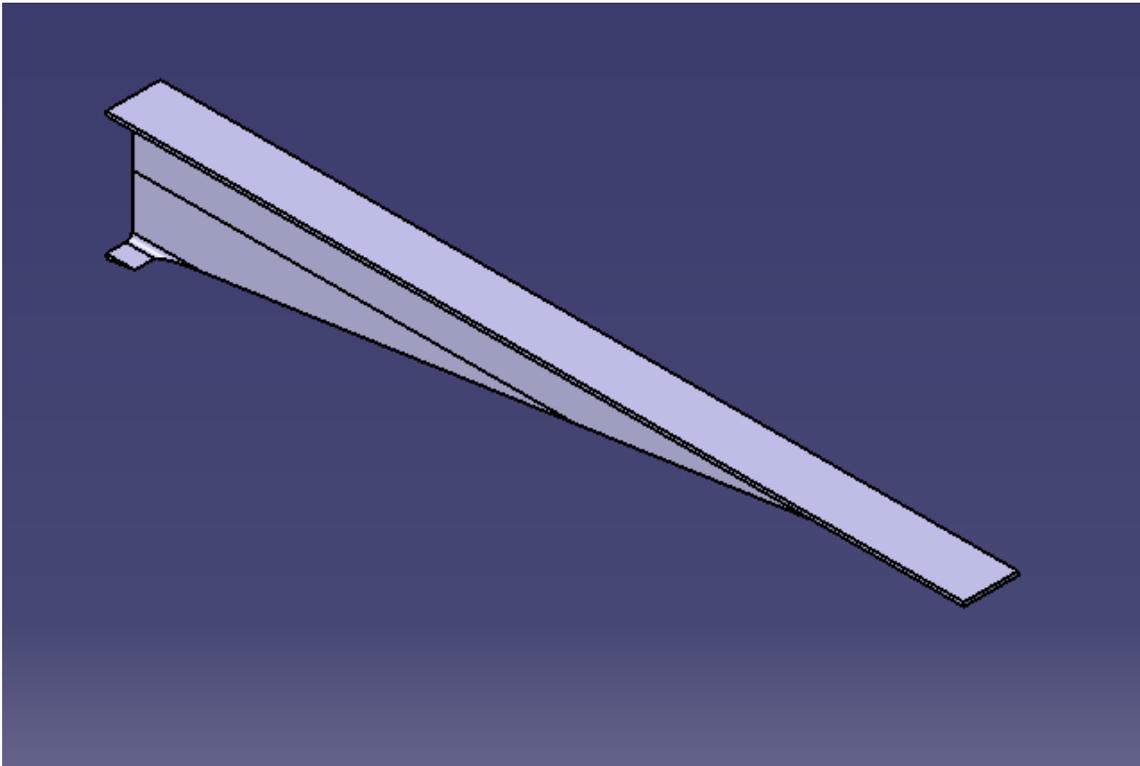
Figure 38. IPE 450 Trimmed Pocket result.

Now we have finished both IPE 450 profiles. What is next is to change into the Assembly Design workbench and assembly both profiles. Once we finish with that, the Example 2 will be done.

Assembly Design (Union)

CATIA offers a pretty simple way of changing between modules. If we click on the left corner of the interface, we will see all the modules, including the workbenches:
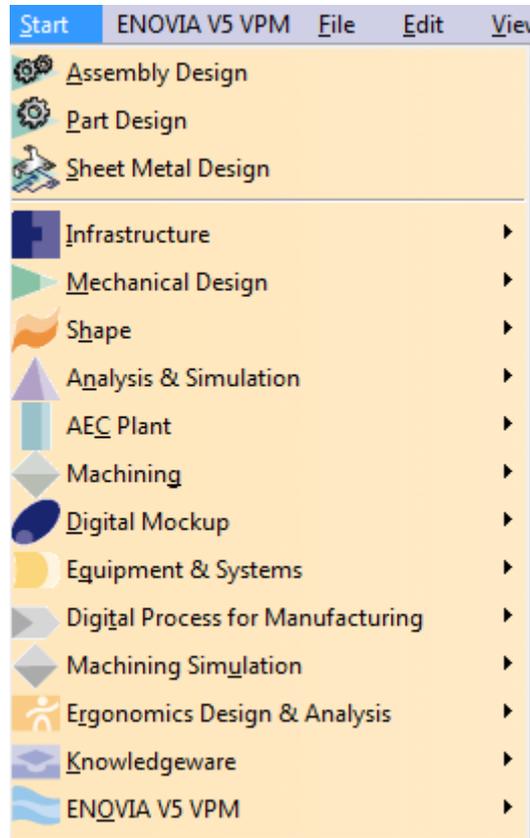

Figure 39. CATIA Interface facilities.

For structural design the main workbenches normally used are located in the modules **Mechanical Design, Shape and Analysis & Simulation**. For example, if we are in the Part Design workbench and we want to change to the Assembly Design module:
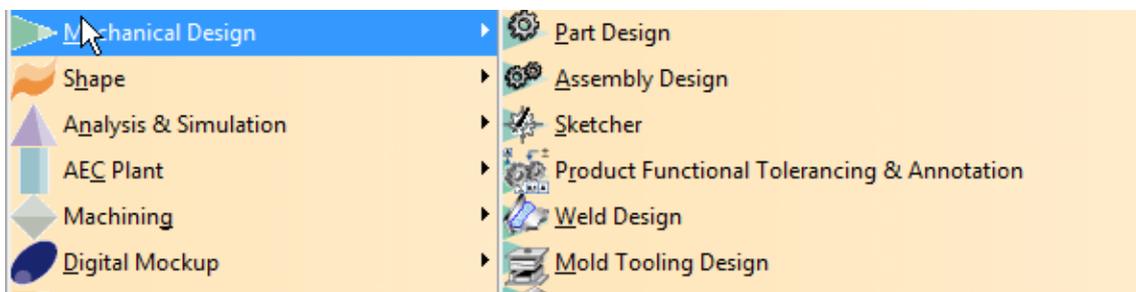

Figure 40. CATIA Interface 'Mechanical Design' module.

We can just click on the desired workbench and CATIA will change its interface. In our case we will click on Assembly Design.

Once we are inside of the Assembly Design interface, the first thing we notice is that in our work tree, we have what we call a **Product**:

Figure 41. Assembly Design Interface.

A Product in Assembly Design is how CATIA names the final result of combining all the different Part Design solids with all the different constraints. "Product 1" is the default name CATIA gives to the Product.

Now that this is clear, we will proceed with inserting the beams we have created. For doing that, we will go to the "Insert menu" (located in the upper toolbar of CATIA's interface) and select "Existing component":
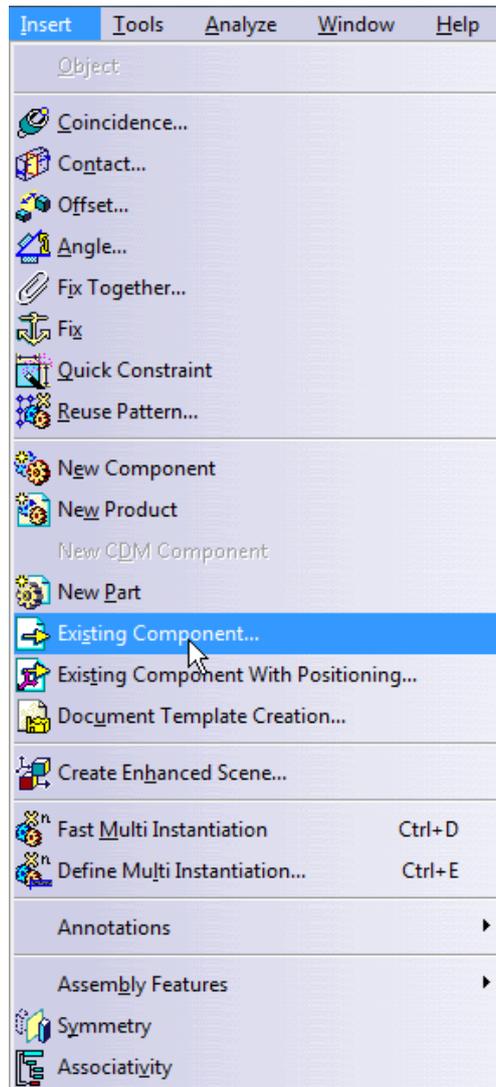

Figure 42. Assembly Design 'Insert' menu options.

When we click on the "Existing Component" option, we will have to click on which Product inside of Assembly Design. Remember that we can have more than one product inside of the same Assembly Design file. Said this, we will insert three components in our file. The first two will be the IPE 450 beams we have created:
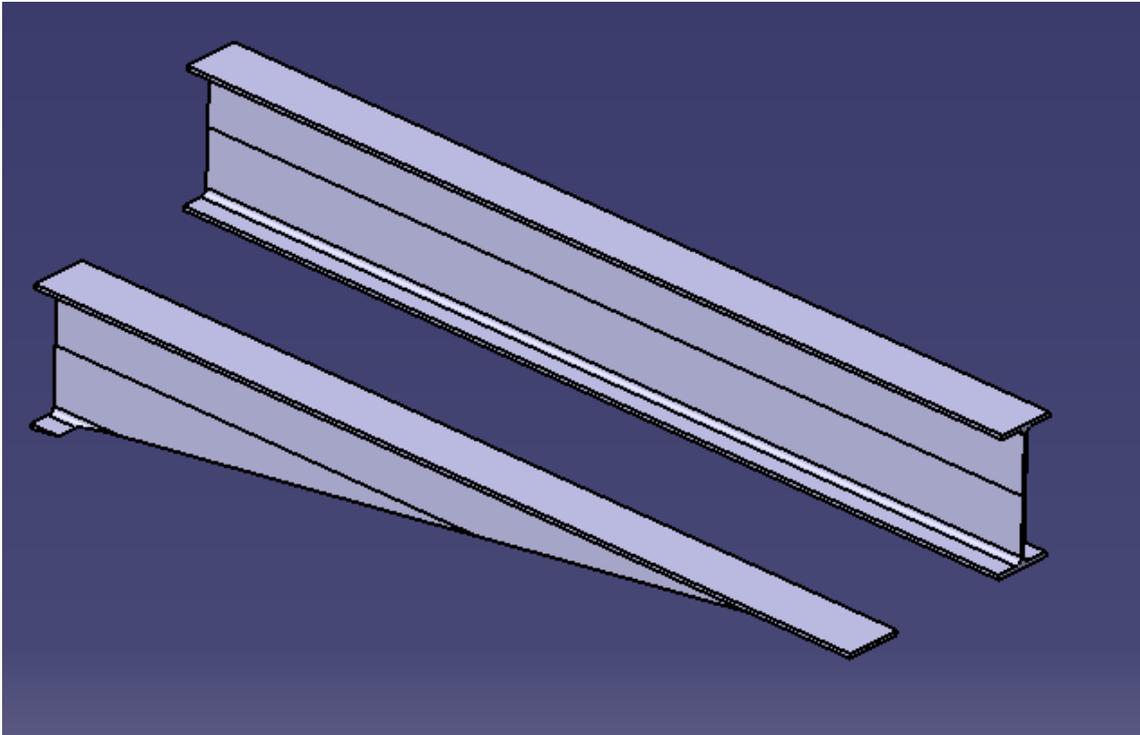

Figure 43. Assembly Design IPE 450 profiles inserted.

As we can see the beams are freely positioned in the space. We will set them constrained according to our analysis later. For doing that we will need the third component, a reference axis.

This requires pretty much no effort to create; we just have to insert an axis system with the same directions that the default axis. In order to do that we will go back to Part Design in all of our files and click on the command called *Axis System Definition.*

First of all, we will click on the command to review all the options possible inside of the command.

Note: We could have done this at the beginning of the designing process to avoid complications. However is not always necessary because CATIA already offers a default axis system. We are doing this because we will be able to make the simulation even more real in terms of results.
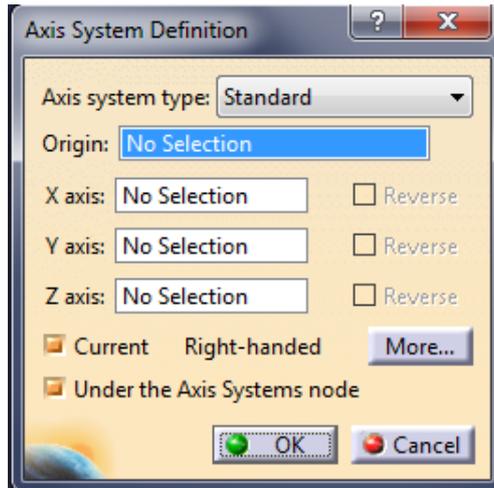
Figure 44. Axis System Definition menu options.

The only thing we are going to change here is the "origin". We will use the origin of the default Axis System to place the new one. We are not changing anything else because the only thing we want is to have a system exactly like the default one but the difference is that we will be able to constrain this axis. This is the obtained result:
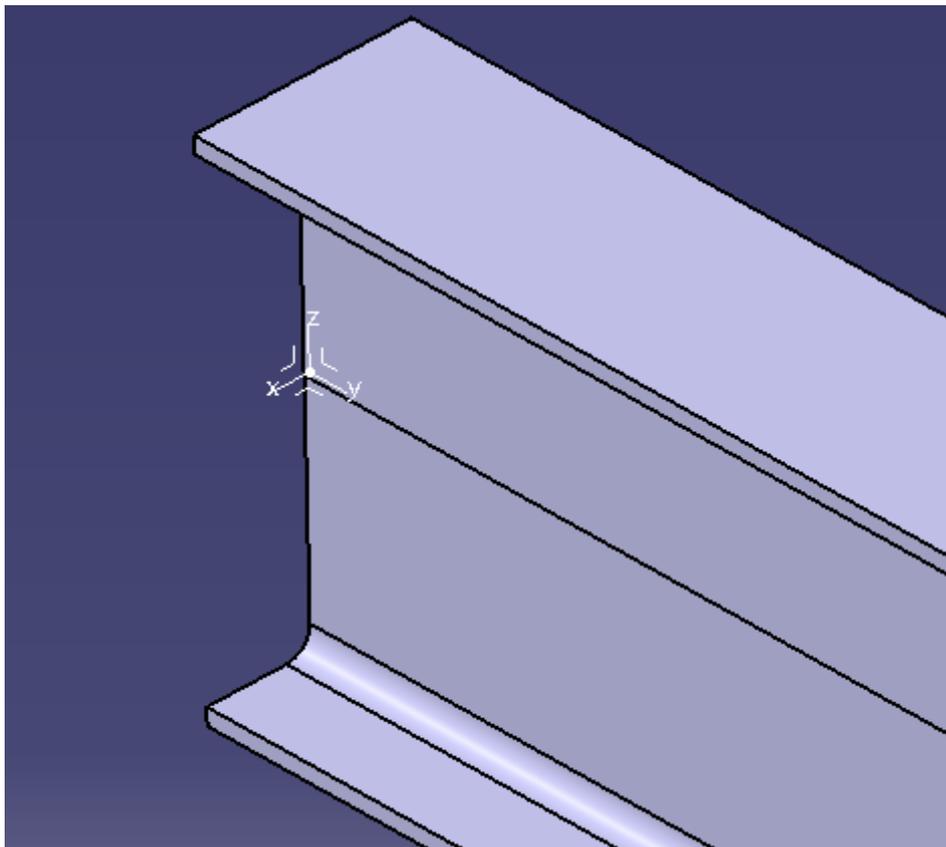

Figure 45. Axis System Definition result.

Like we said before, we will do the exact same thing with the Example one and pretty much all the profiles we have defined. However is necessary to say that what we just did is not strictly mandatory for all profiles (we are not going to do it for the screws and nuts).

In addition to both beams we already said that we were going to insert another component to the Product. This component will be just an Axis, like we can see in the picture:
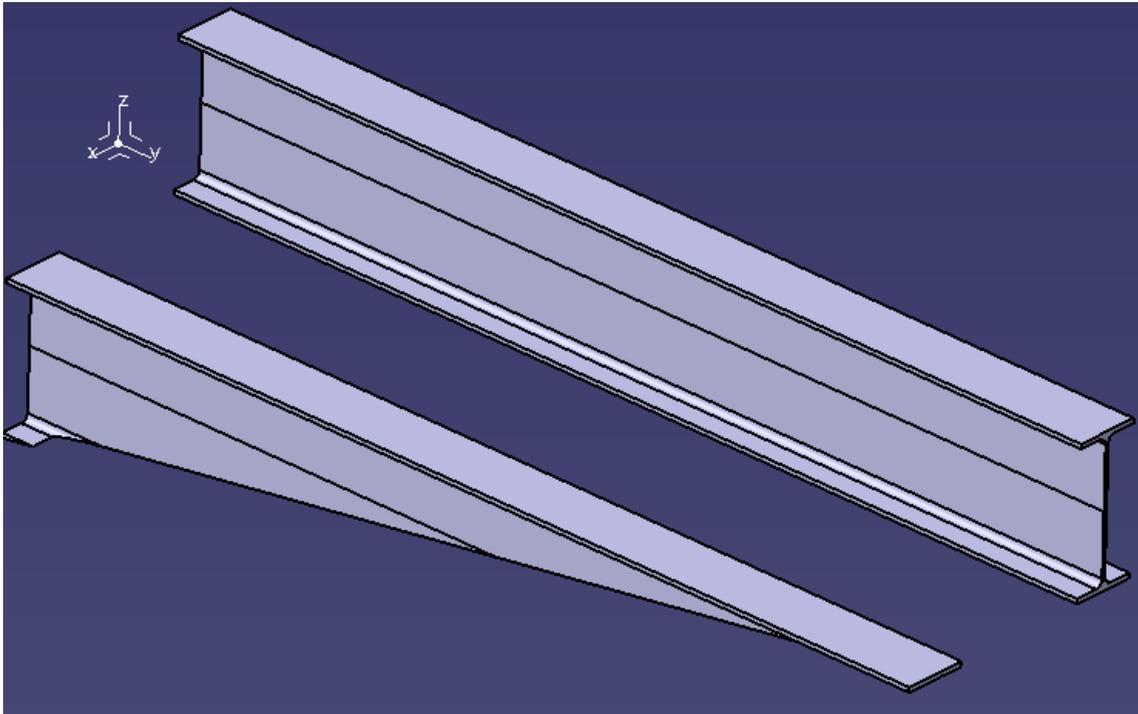

Figure 46. Components inserted for Example 2.

Now if we take a look into our work tree we will have our 3 components there:
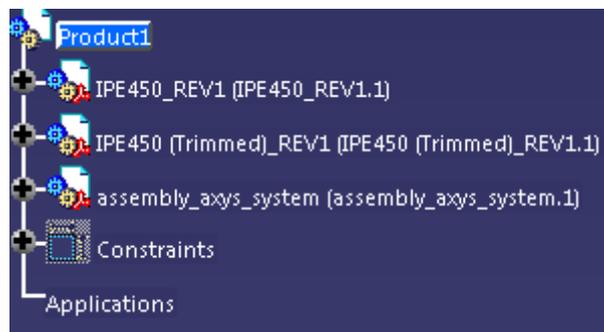

Figure 47. Product 'Work Tree'.

In addition to the work tree we see a fourth "component". However this is not a component. Constraints refers to all the relations we are going to create in between the components.

Our goal is to keep the number of constraints to the minimum (Later, in the FEM module, we will have to tell CATIA how to interpret this constraints in terms of analysis). For example two we will not have a lot of constraints. The main constraints we will be normally using in CATIA are:
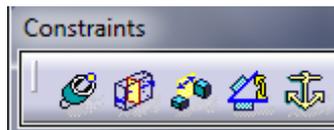

Figure 48. Constraint toolbar.

- **Coincidence Constraint:** This type of constraint is used to align elements. Depending on the selected elements, you may obtain concentricity, co-axiality or co-planarity. When using Coincidence you are telling CATIA that the selected elements have to remained connected

- **Contact Constraint:** The Contact constraint can be created between two directed surfaces. Directed means that an internal side and an external side can be defined from a geometrical element, a face of a pad for example. This definition excludes surface element and wireframe surface because they are not directed. This constraint is really useful to simulate welds between solids.

- **Offset Constraint:** This constraint allows us to set a determined distance between two elements, or even faces. For example, if you need a washer to be 10 mm away from the head of the screw, you can use offset for that.

- **Angle Constraint:** As its noun says, this constraint sets a specific angle between two elements or faces. This is also a very popular constraint in all kind of work spaces.

- **Fix Constraint:** This is a very dangerous constraint. When using Fix, you have to take into consideration that whatever you use this command into, it will get completely constrained (of all six freedom degrees). Because of this constraint is why we inserted the third component, the Axis.

Now that we know all the constraints that we will be using. We will start constraining the elements.

We will start using the Fix constraint on the Axis. This way we will have the axis completely constrained and it would be useful to place the rest of the elements without affecting the analysis:
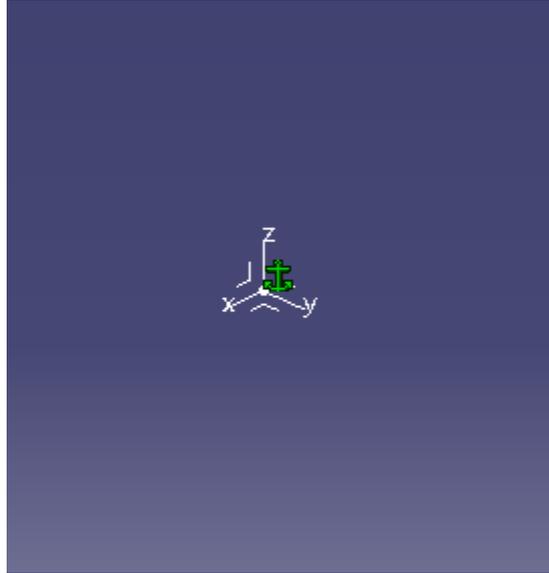
Figure 49. Axis Fixed.

Just to remember, we will recognize a full-operative constraint when is completely green. Next we will make a coincidence between the Normal IPE 450 and the fixed axis; using of course, the Coincidence constraint:
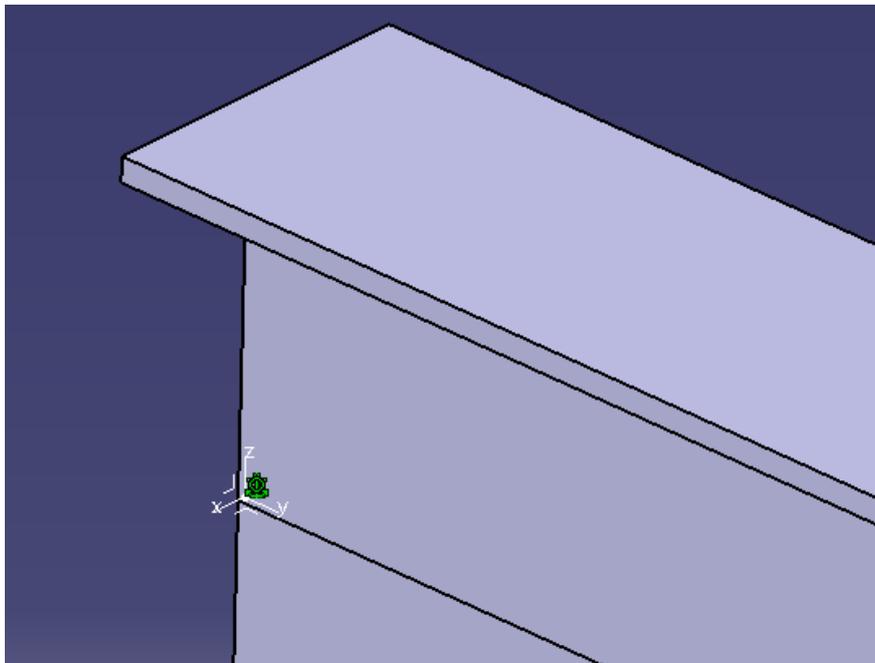

Figure 50. Coincidence between Axes.

Now that we have our main element attached to the axis we can start constraining the rest of the elements to our main one. As both beams are one on top of the other we will apply a Contact constraint between the bottom face of the normal IPE 450 and the top face of the trimmed IPE 450, obtaining the following result:
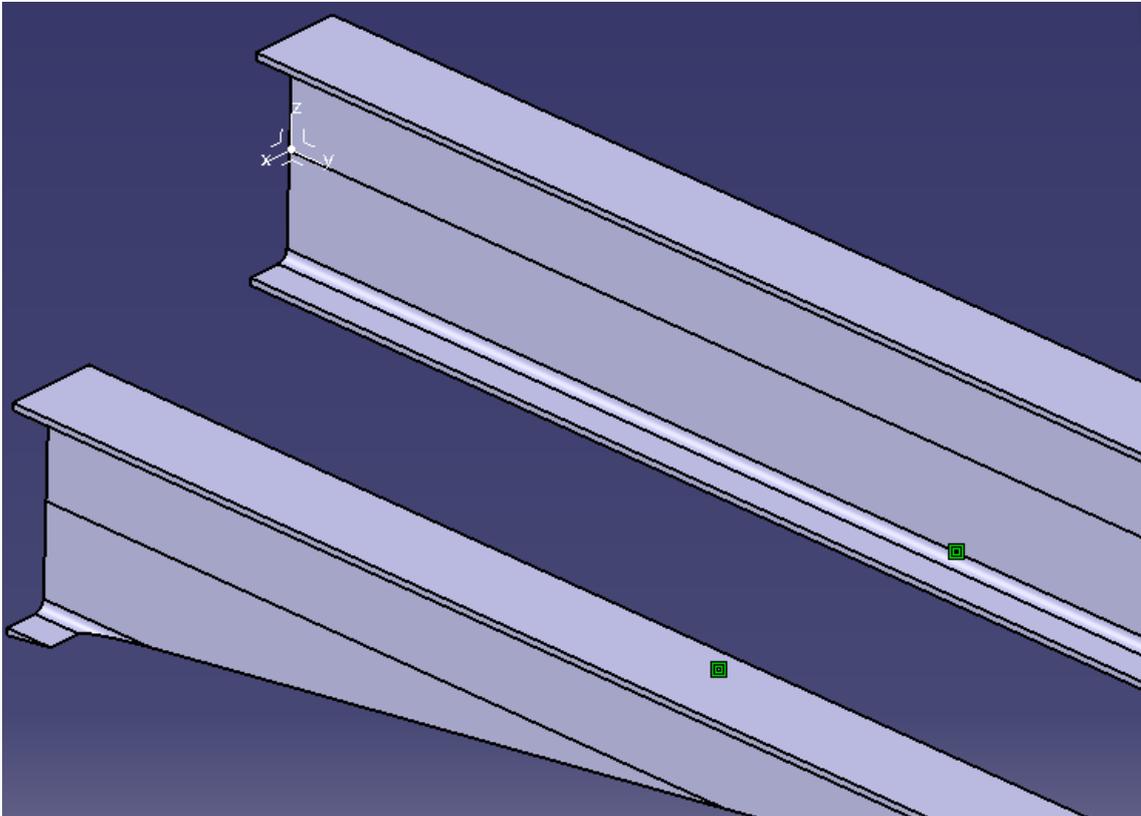

Figure 51. Coincidence between IPEs 450.

As we can see, the Contact constraint does not work as we pretended (It does not work like glue). Both faces are "in contact" or better said, at the same level but they are not touching each other. The same thing happens in the Y axis direction. Right now the beams can move freely in the X and Y axis direction. However, this is easily solvable by applying a coincidence constraint between two faces that have to be at the same level (parallel).

We will apply two Coincidence constraints to said faces to solve the problem, obtaining the desired result:
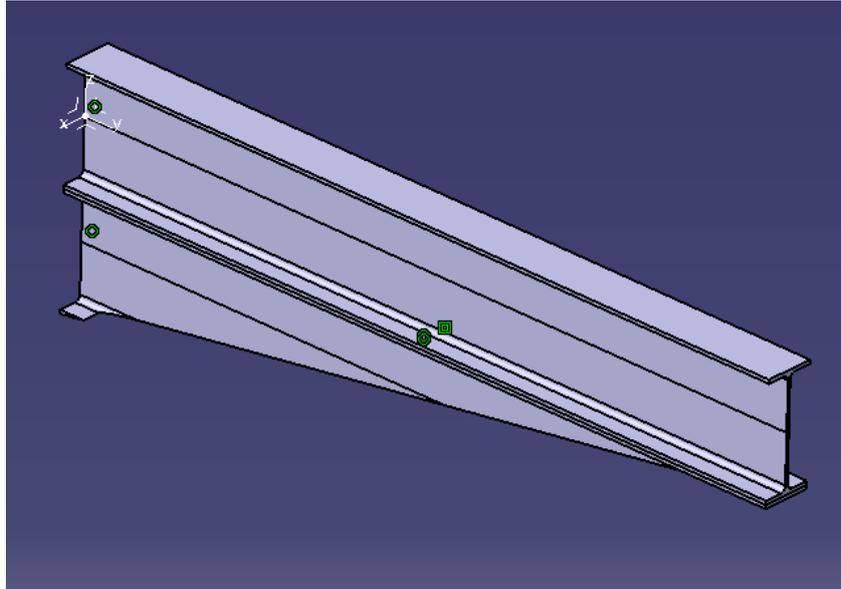
Figure 52. Example 2 Assembly Design completed.

Visually we can tell that the example is complete. However we are going to introduce a new command that will allow us to manipulate the assembly taking into consideration the constraints.

This command is *Manipulation.* Manipulation will allow us to check if the constraints we have placed are good enough (when using the command, the option "with respect to constraints have to be activated):
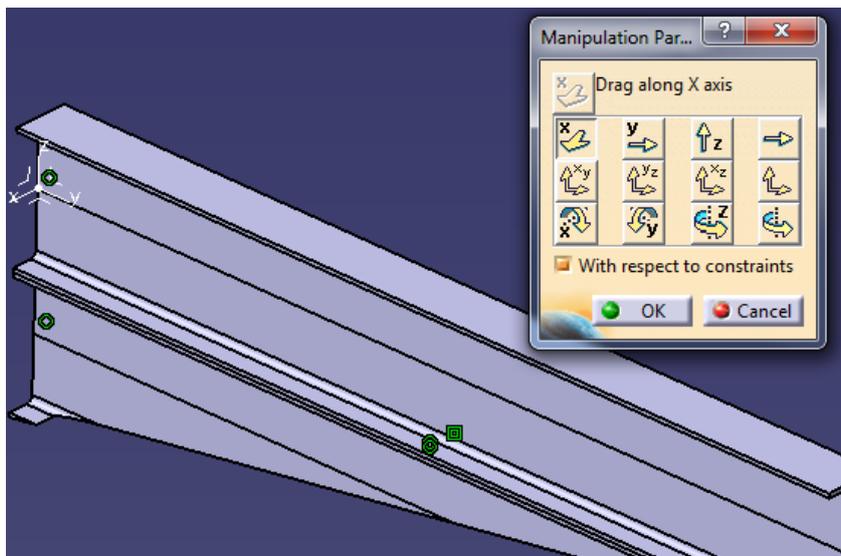

Figure 53. Example 2 Assembly Design Manipulation test.

In our case, the simulation has been successful which mean our product is ready to go into the Analysis module.

Example 3: Lintel Union IPE 500 + IPE 450

This example will be the most difficult one we will be presenting in this thesis. As we have already explained we will be modeling a lintel union between an IPE 500 and two IPEs 450, one of them trimmed. Taking into consideration the regulations in terms of construction we will design auxiliary elements that are necessary in these types of unions. These elements would be really simple, geometrically talking.

First of all we will have a twenty mm. sheet that will go between the pillar and the beams. We will go briefly through the process because we will use the same command we have already seen.

20 millimeter Sheet

As usual, we will start with a sketch (In these types of elements it's not really mandatory to introduce an axis but is always a good idea just in case). The sketch has no mystery as it is a rectangular shape:
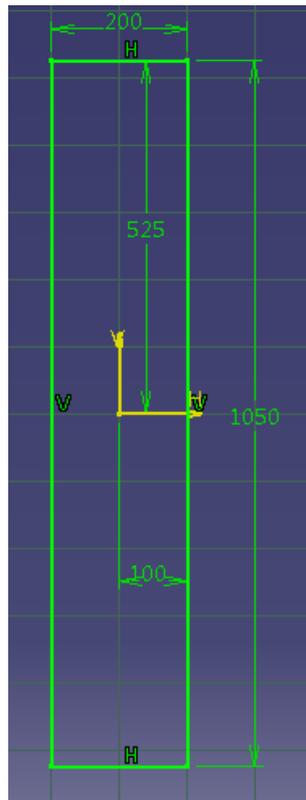

Figure 54. Example 3 Sheet sketch.

Once the sketch is finished we will proceed with the PAD operation obtaining the already mentioned 20 mm gross Sheet.
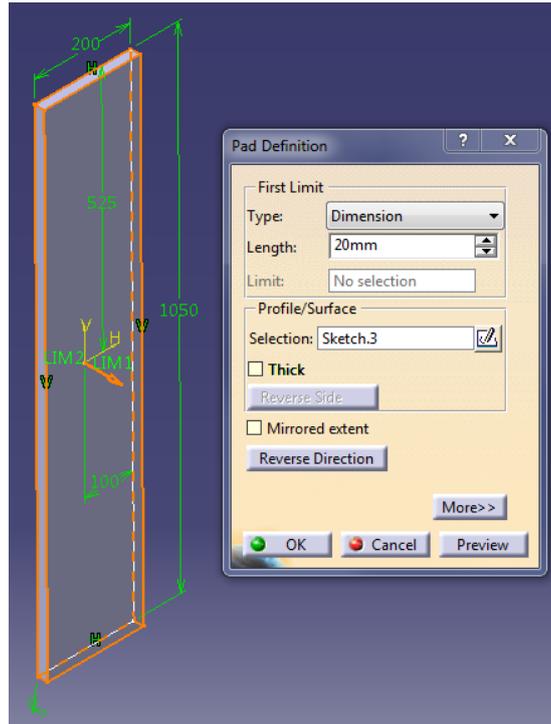
Figure 55. Example 3 Sheet PAD operation result.

The last thing left to is to apply the same pattern of holes that our pillar has. It has to be exactly the same because otherwise the screws will not go through both solids. Just as a reminder, we have already explained the whole process in the Example one (Page 23). Here is the obtained result:
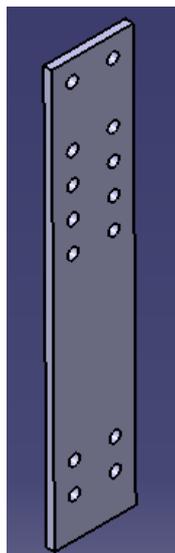
Figure 56. Example 3 Sheet Rectangular pattern result.

In terms of designing and modeling the sheet is ready to go into the analysis module. The following elements would be the screws, washers and nuts.

M24 Screws

Before getting into how I designed the screws, it is important to know that CATIA has its own catalog following four different regulations (*ISO, US, EN* and *JIN*).
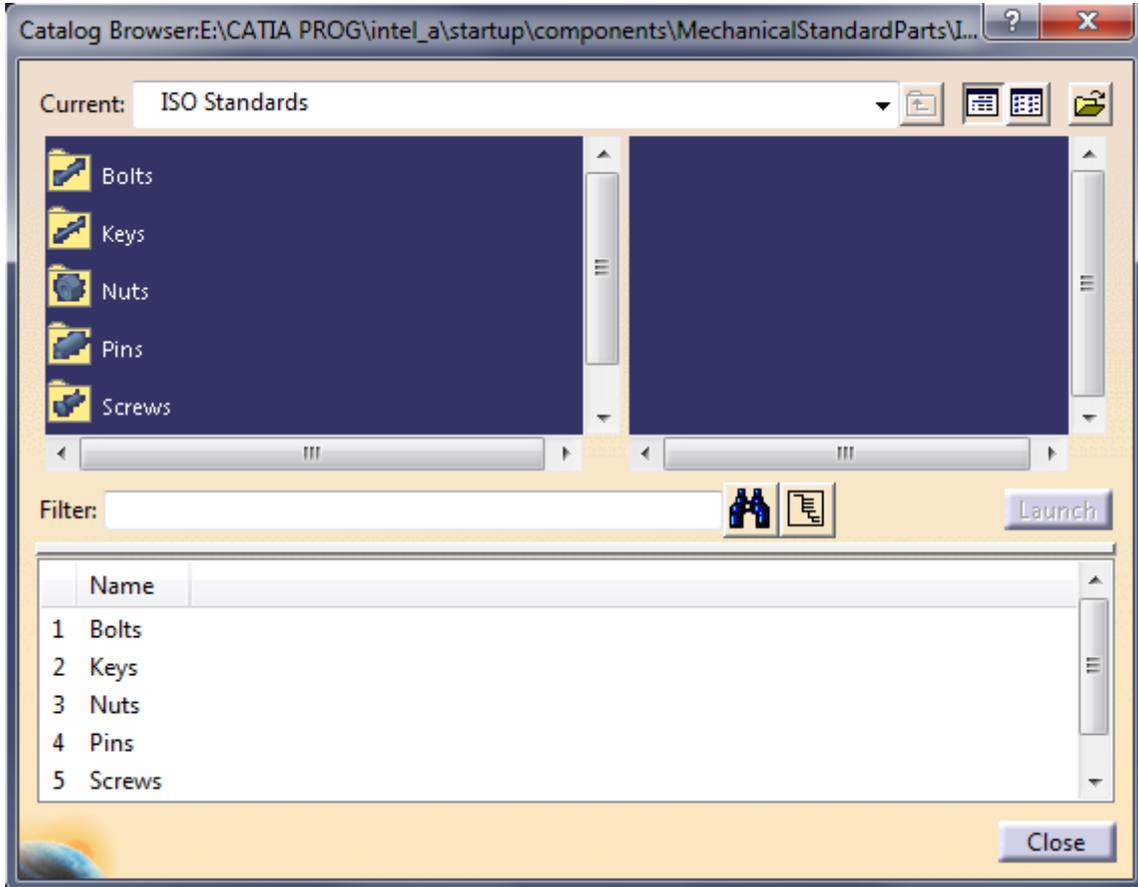
Figure 57. CATIA Catalog preview.

As we can already see in the figure above, we have 5 main elements we can just take from CATIA's catalog. This tool is very useful as you can just take the elements according to your needs and those elements would be completely according to the regulations selected. This is another reason why CATIA is very suitable for structural design.

However, I have decided to create my own screws, washers and nuts, following the ISO regulation just to show that CATIA also offers this possibility in terms of designing and modeling.

Starting with the screws, we will design the screws juts like we can find them in CATIA's catalog (following the ISO regulation). As usual, we will start with a simple

sketch but this time we will not use the PAD command for turning the sketch into a solid.

Because screws have 2 very different parts (the head with a hexagon shape and the body with a round shape) we will draw both shapes directly into our sketch:
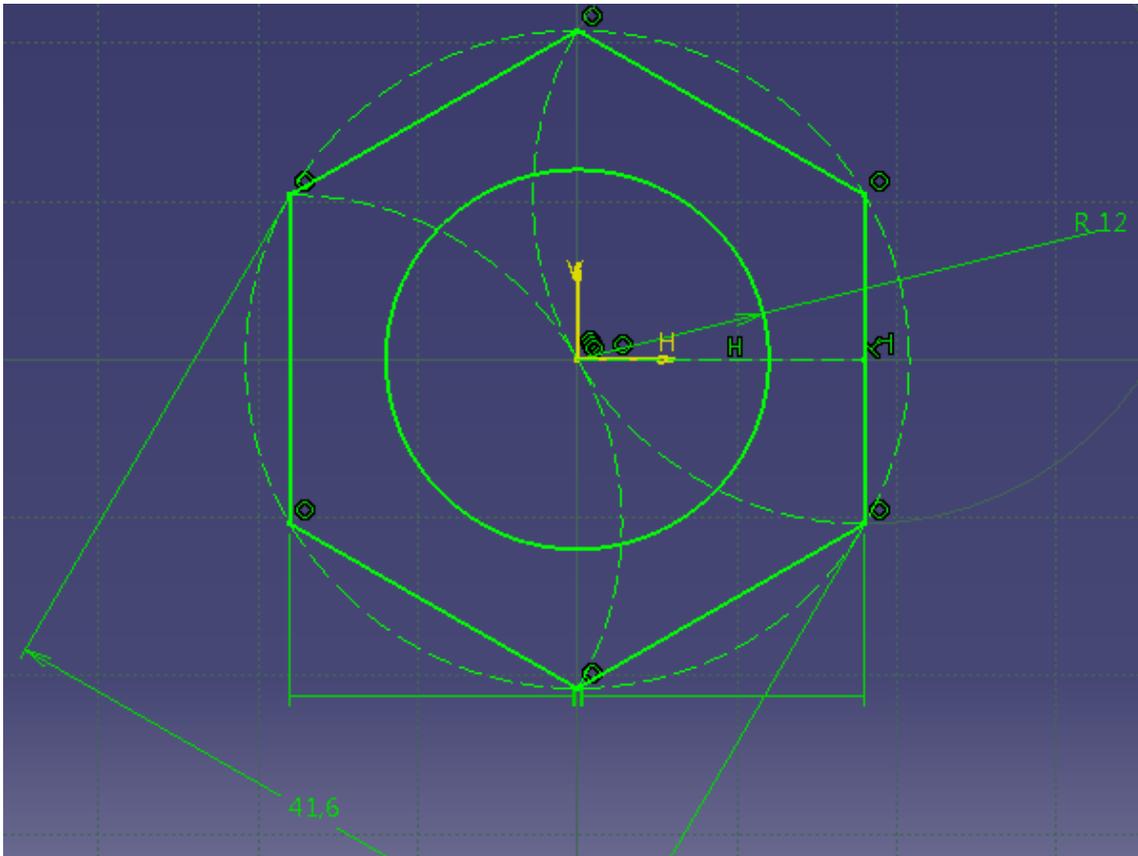

Figure 58. Example 3 Screw sketch.

Following with the sketch we will introduce a new command called *MultiPAD*. We will use said command to create the basic shape of the screw out of one operation. Once you enter the command you have to select shapes not sketches.

In our case we have the shape of a hexagon for the head and the shape of a circle for the body.
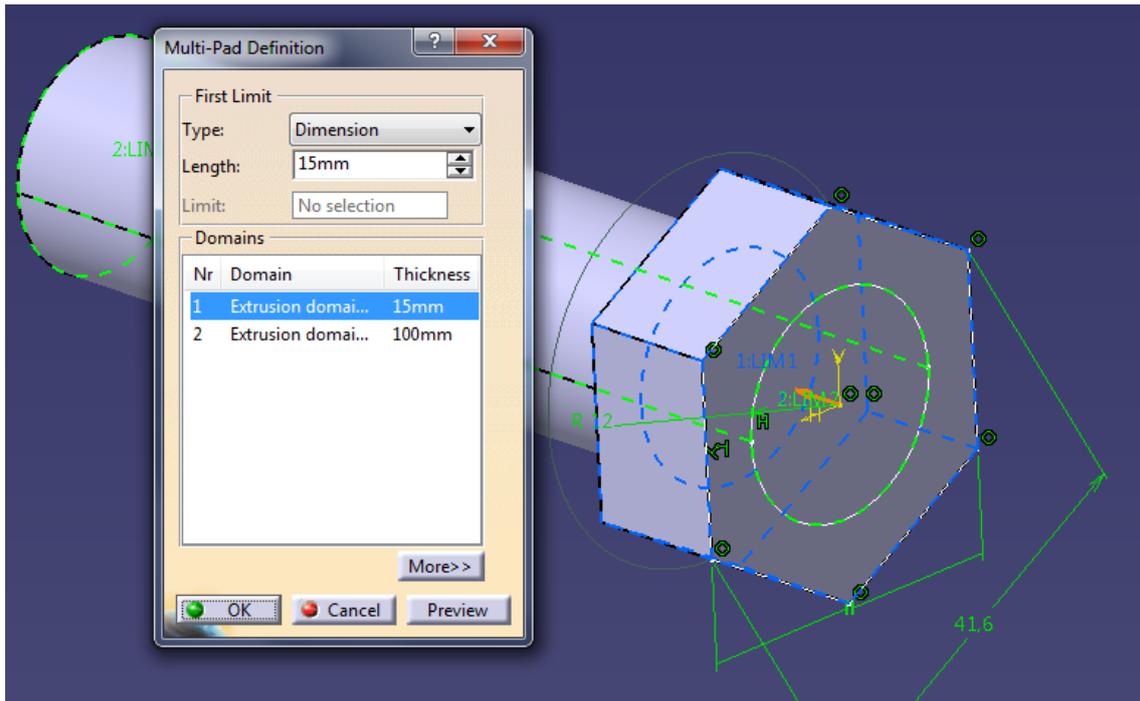
Figure 59. Example 3 Screw MultiPAD operation result.

AS we can see in the Figure 59 we have two independent extrusions, one with a dimension of 15 mm. (head) and the second one of 100 mm. (body). It is important to mention that the second extrusion starts where the shape taken as a model is drawn, which means that the total length of the screw is 100 mm. Also we have to mention that the only possible option when using this command is the Dimension option, there is no Up to next, Up to last, etc.

What follows next is a little visual detail pretty much irrelevant in terms of analysis. We are going to explain how to make the little round shapes on the corners located on top of the screw's head. We will make these little notches with a new command called *Groove.* As usual, before using the command we will create our little sketch
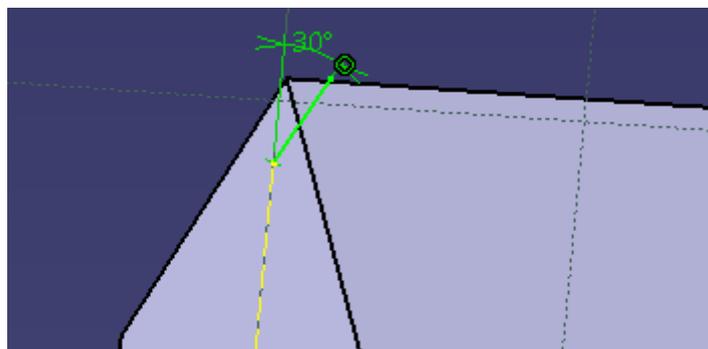
Figure 60. Example 3 Screw Groove sketch.

We don't need to create a complex sketch because groove allows us to work with just a line. In our case we want to remove material in one direction so our sketch is more than enough.

Once we are done with the sketch we will open the Groove command and we will select our sketch. The most important part comes when we have to select the direction of the removal.
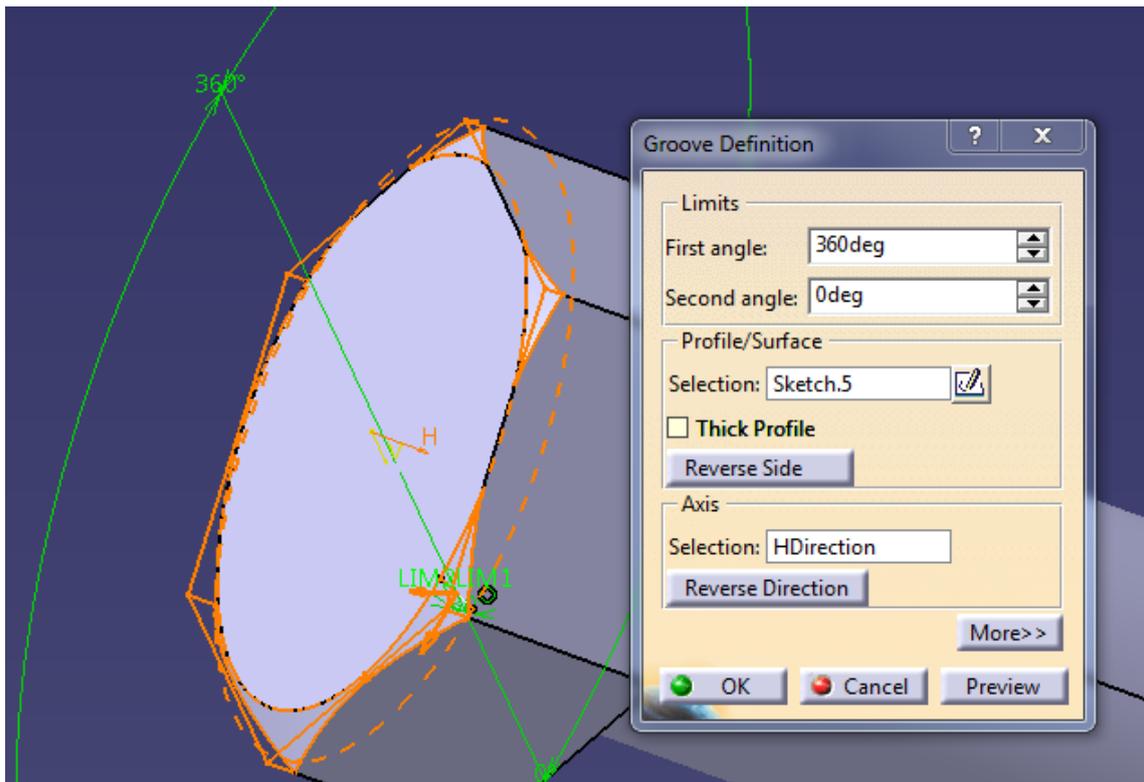
Figure 61. Example 3 Screw Groove operation.

Inside of the command we can choose if we want a full groove (360 degrees), or the angle we want, the sketch we are using for the groove, the axis we are using a reference (If we have drawn using the correct axes, we can use these axes to make our grooves) and last but not least, we have the reverse direction button. Before pressing anything I would recommend clicking on preview and if the result showed it is not what we wanted we can just click on reverse direction and problem solved.

Once we have adjusted everything as we wish we can execute the command by clicking OK and we will have our screw visually perfectly defined:
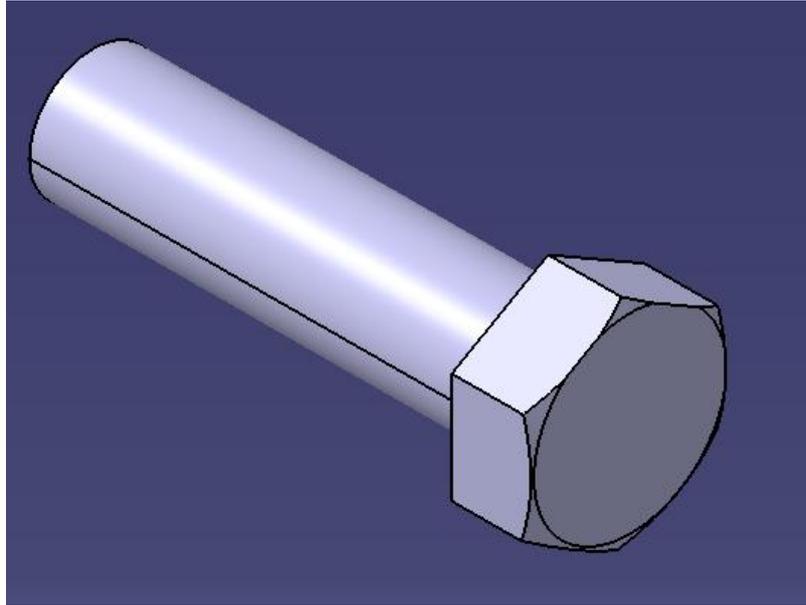
Figure 62. Example 3 Screw visually completed.

Now our screw is complete in terms of bodies and solids. However if we want to tell CATIA that our screw has a thread on its body. Visually talking nothing is going to change but when we go to the analysis module, CATIA will take into consideration the threaded body.

For doing that, we will need a very useful command called *Thread/Tap*. It is called this way because with this command you can both Thread (screws) and Tap (nuts) bodies (we will go deeper with the Tap option in the Nut section).
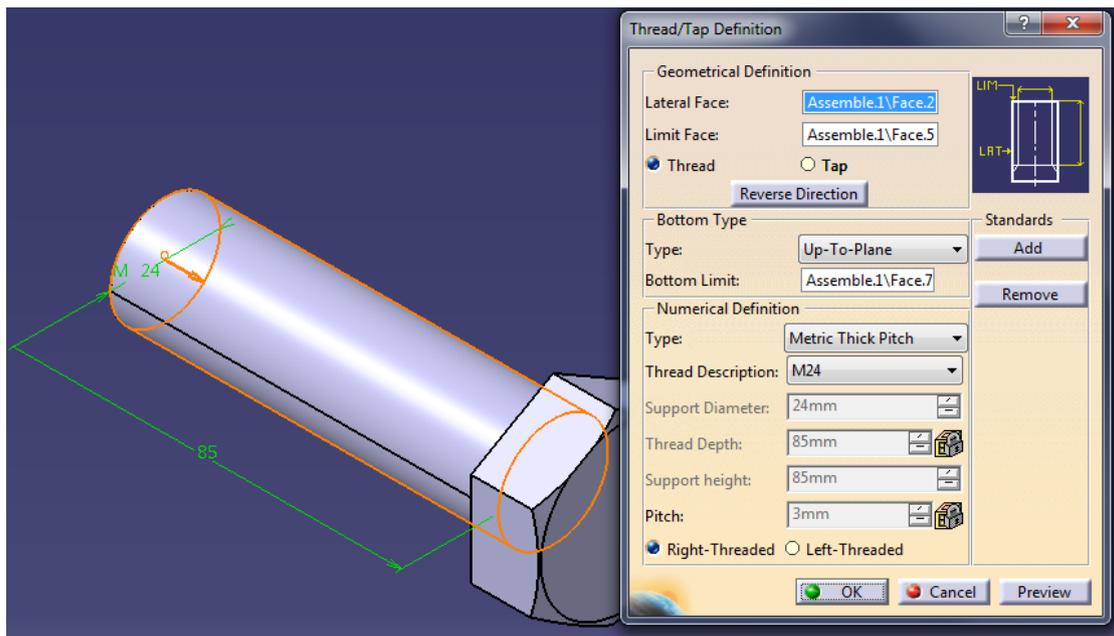
Figure 63. Example 3 Screw Thread command.

As we can already see, there are a lot of options inside this command. First of all we have to set the limits for the thread. In our case those limits will be the faces that delimit the body of the screw. However, the main part comes when we have to choose between Thread and Tap. When choosing Thread, the first thing to do as usual is to set the kind of delimitation we are using (The limits we have previously defined can be automatically defined by CATIA).

Second, we have to choose between Metric Thin Pitch, Metric Thick pitch and No standard. In our case we will go with Metric Thick pitch which means our screw will have a 3 mm pitch (certified by the ISO regulation), we can see in the last option that it says "Pitch: 3 mm", this is completely automatic defined by CATIA.

Third, we have to define the metric. Like we have said before we will go with screws M24 ISO. It is important to decide this before you start modeling in CATIA because if for example now that we have designed everything based on a M24 screw, we choose M26 the command will give us an error saying that numbers do not add up.

Fourth and last, we will have to choose between right-threaded and left-threaded. This feature is really cool because you can change between these two options really quick just in case originally the direction of the threaded was one and it changed into the other one.

M24 Nuts

Following the same path that we have been explaining during the designing process of the screws, we will proceed with the nuts.

We will start with the sketch. As this a really simple solid the sketch has no mystery, it will have the shape of a hexagon. However, in order to avoid using the PAD command and the having to make a pocket using the Hole or Pocket command, what we have done is creating already the shape of the circle and then we will use the MultiPAD command directly:
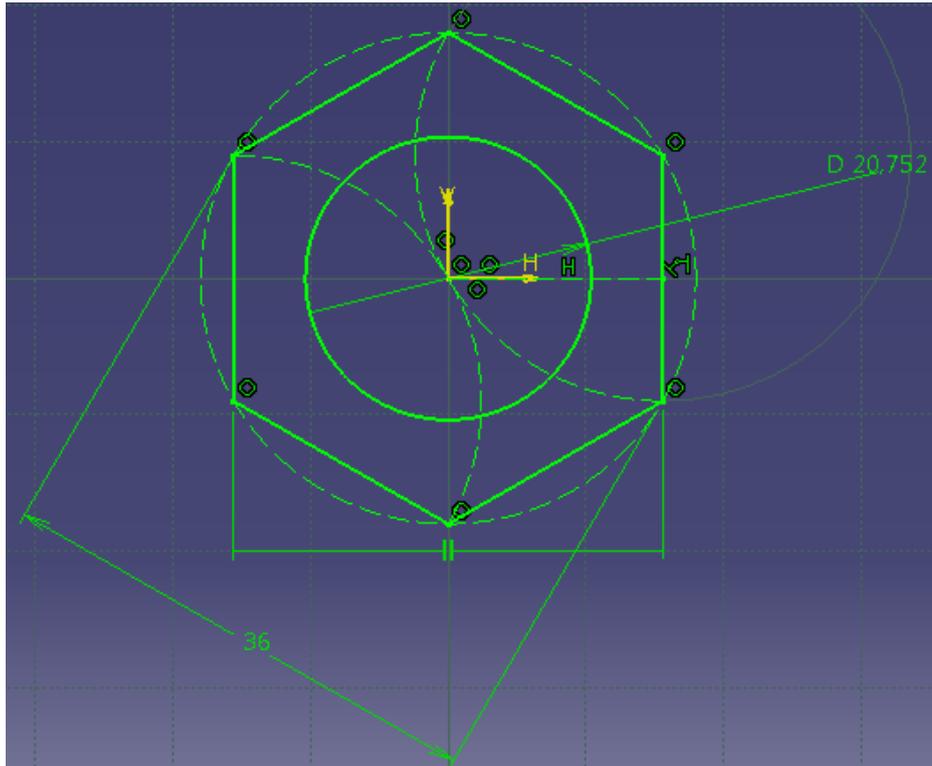
Figure 64. Example 3 Nut sketch.

Once we have our sketch we will execute the MultiPAD command as we have said before and create our solid:
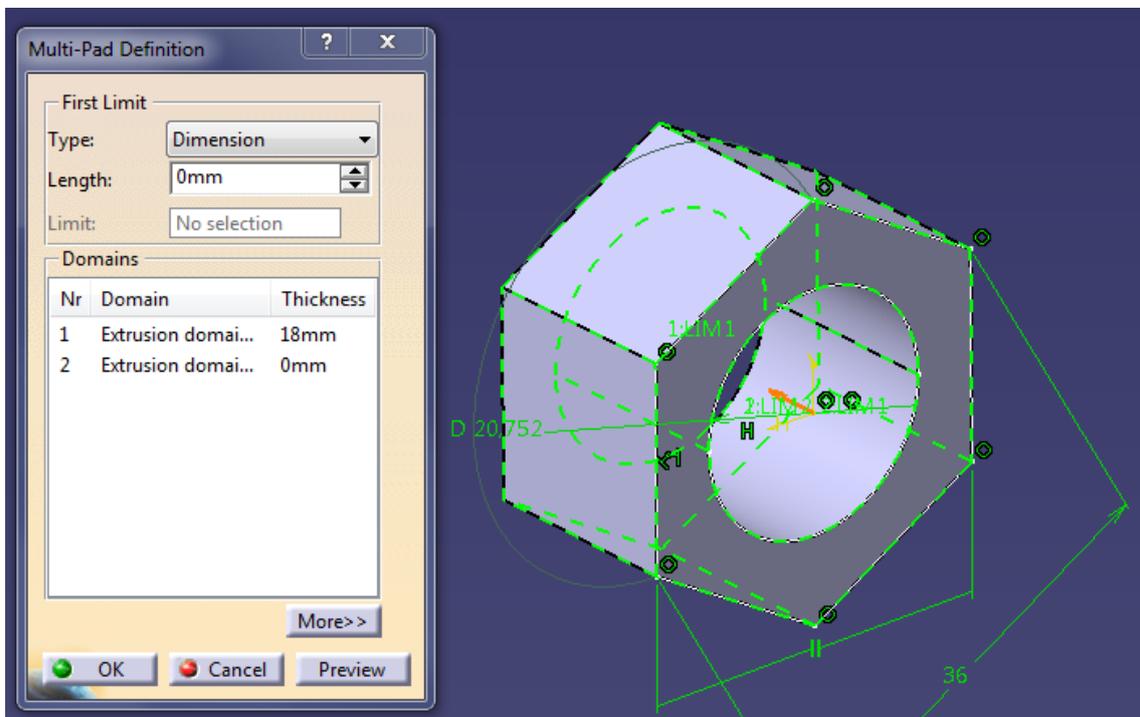
Figure 65. Example 3 Nut MultiPAD operation result.

As we can see in the figure above, we have two extrusions in the MultiPAD command. However the second one has the value of 0 mm. because that's how we made the hole, indication that the selected shape (circle) has to have an extrusion of 0.

Next, we will go back to the Groove command and create the exact same notches we created for the nuts. As we have already seen the necessary sketch (Figure 60), we will jump directly into the command:
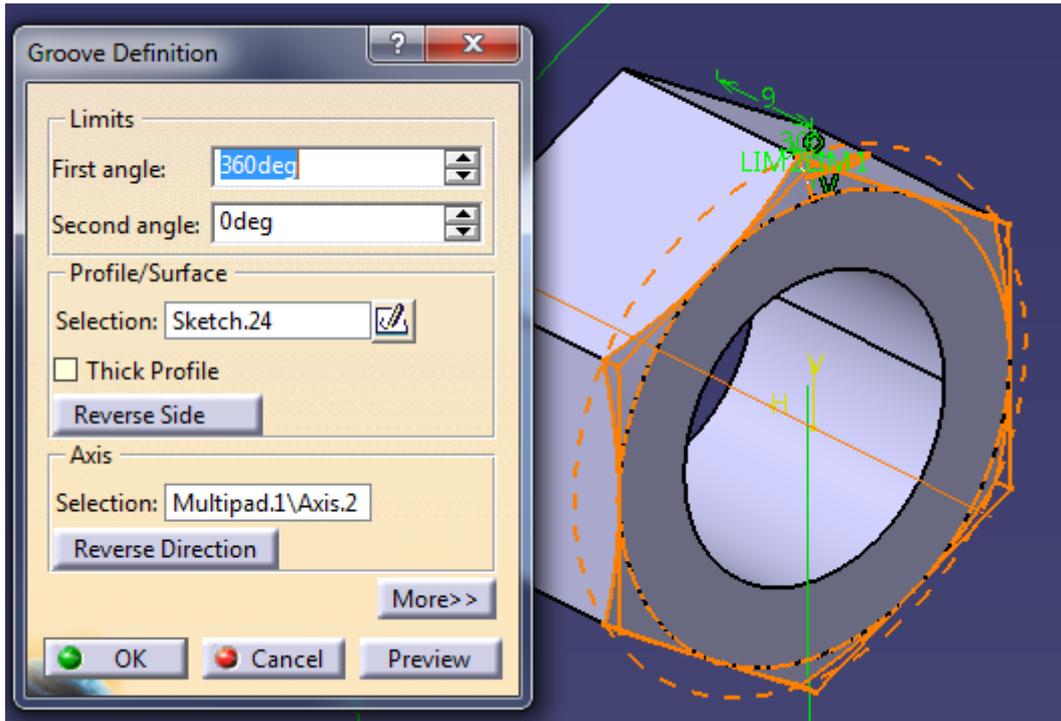
Figure 66. Example 3 Nut Groove operation result.

The same configuration we wanted for the Screw. Only one angle (360 degrees), selecting the sketch and, like before, we want to groove using the horizontal axis. Notice that we have two sides here as this is a nut, not a screw. This means that we will have to execute the groove command twice, one per each side.

Now, visually speaking the nut is completed. The only thing left to do is to use the Thread/Tap command again and apply the same configuration that we did with the screw.
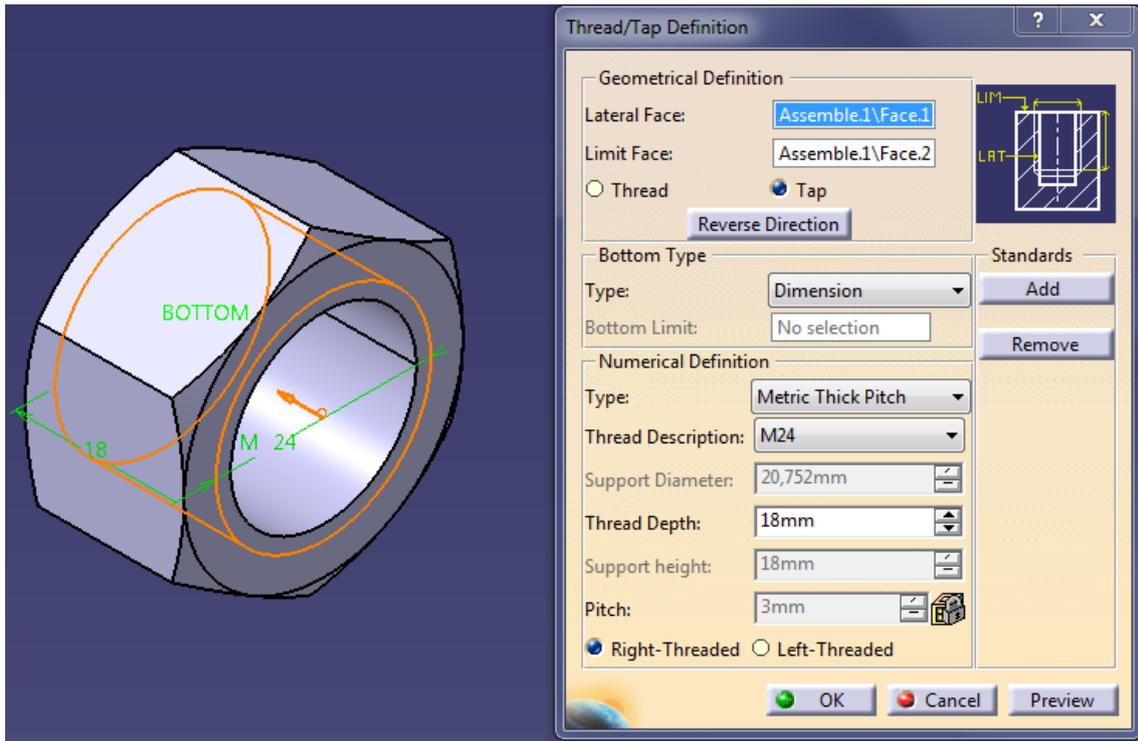
Figure 67. Example 3 Nut Tap operation result.

Of course, this time we will select the Tap option. Even if we select this option we can check with the Figure and realize that the rest of options are exactly the same. We will select the most convenient ones and execute the command. It is important to realize (looking at the Figure 64) that we had to create a diameter of 20,752 mm; this is because when you are using the Tap option, you are telling CATIA that the threaded would be from-inside-to-outside, completely the opposite of the Thread option.

This problem will cause an error that we will have to ignore in the analysis part. We will come back to this at the end of the explanation of the Example 3, when we finish assembling all the pieces together. With that been said, the nut is finished
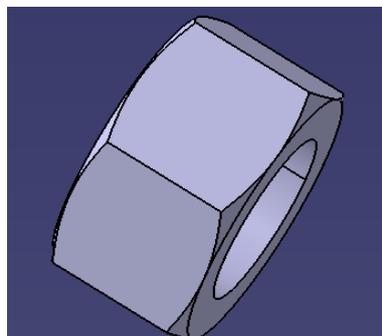
Figure 68. Example 3 Nut final result.

Washers

The washer is just a supportive element that helps screws and nuts gaining grip with different surfaces. In our case, the washer has no mystery; it will be four millimeters thick.

We will start with the sketch as usual:
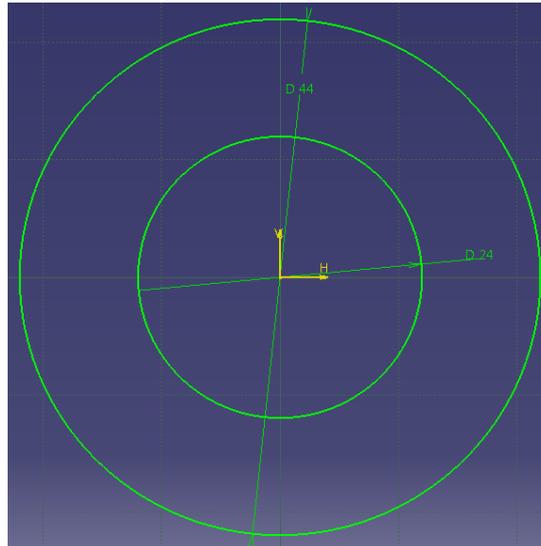
Figure 69. Example 3 Washer sketch.

Just like we did with the nut, we will sketch the hole too and use the MultiPAD command to extrude everything directly:
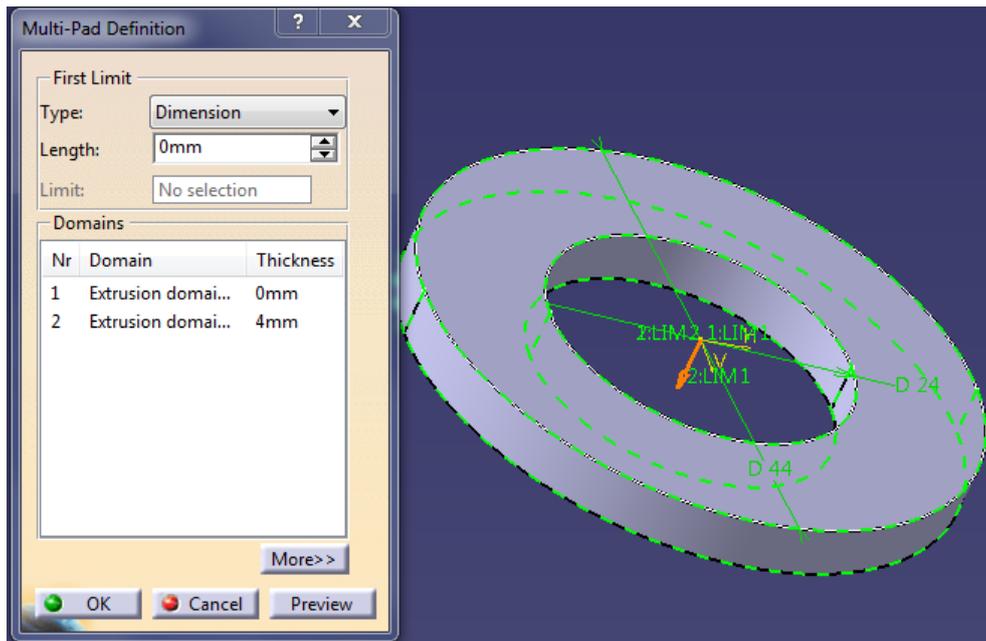
Figure 70. Example 3 Washer MultiPAD operation result.

Like we said before, it will be a four millimeter thick washer. Following the MultiPAD command we will add a little groove on one of the sides of the washer, using the same sketch we used with the screw and nut (Figure 60):
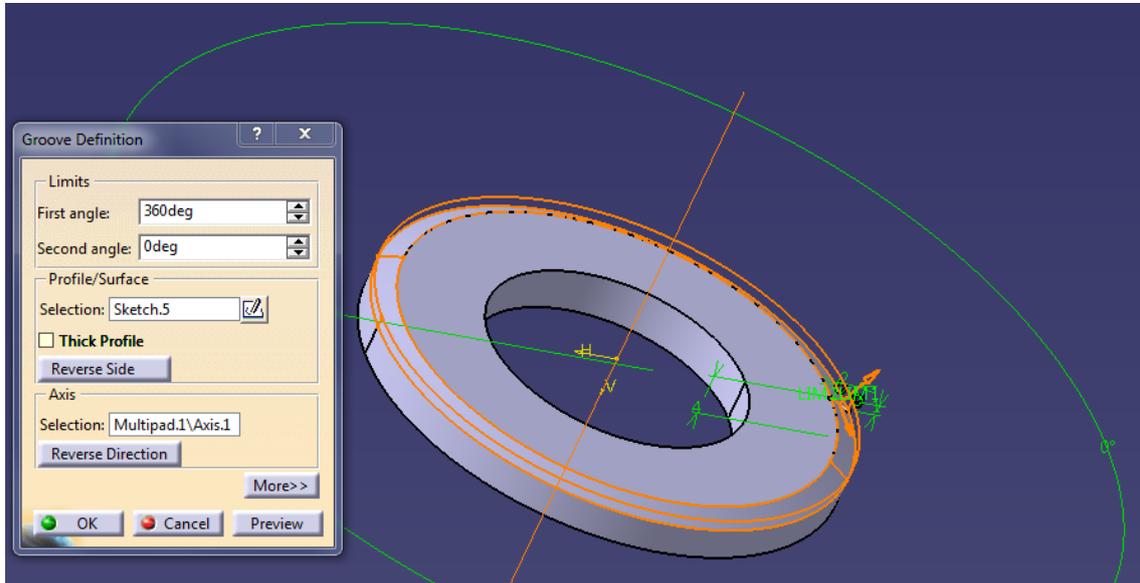
Figure 71. Example 3 Washer Groove operation result.

Same exact configuration than the nut except from the axis selected. This time we will go with the vertical axis.

This concludes everything related to the designing and modeling of all the solids that we will be using in all three examples. The only thing left regarding to the example number three is to assemble everything before jumping into the next chapter (analysis). However as we will be using more than ten screws, nuts and washers, we will assemble the nuts and the washers just to work with them as one solid instead of two.

Assembly: Nuts and Washers

The first thing we have to do is to insert both bodies in one product inside of the Assembly Design workbench. This time we will not need a custom Axis (like we did in Example 2) because these solids have a very simple geometry.

As we have already gone through the basis of Assembly Design we will not explain everything again (Please refer to Example 2, page 41), I will just mention all the command and steps we are going to take with their configuration.
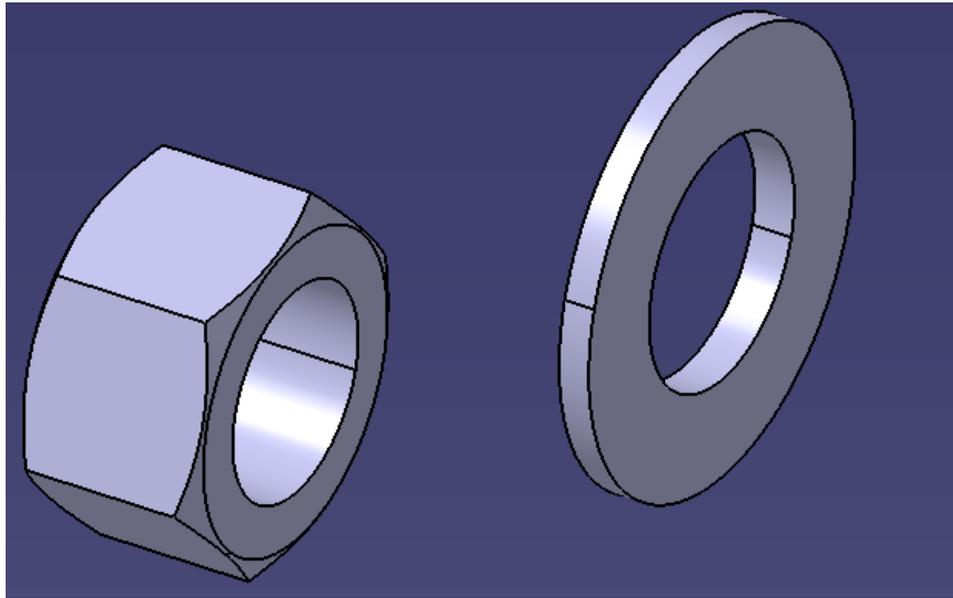
Figure 72. Example 3 Assembly Nut + Washer inserting components.

If we pay attention at the geometry of the bodies and our necessities in terms of constraints, we only need two constraints for restraining all the freedom degrees we need.

The first one will be coincidence of axes between the nut and the washers because they need to be aligned in order to fit the screw. The second will be a contact coincidence between one of the faces of the nut (as its symmetric both faces work) and the face of the washer where we applied the groove. Once we do this we obtained our finished assembly:
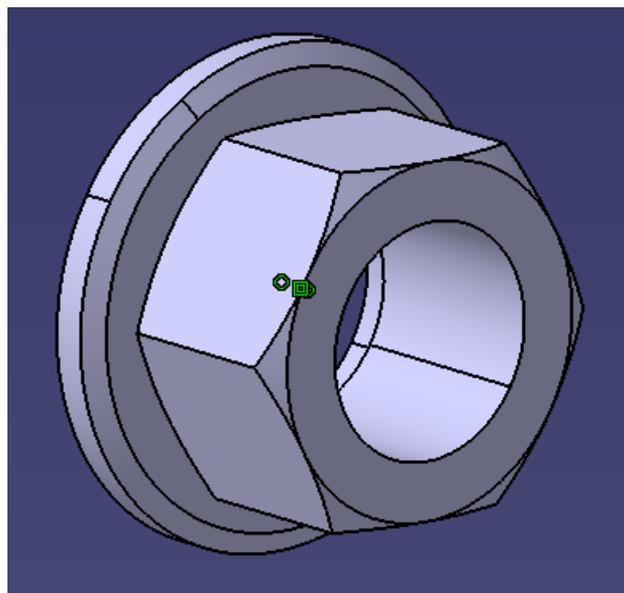
Figure 73. Example 3 Assembly Nut + Washer assembled.

Assembly: Example 3

This is the last semi-chapter inside of Example number three. Now that we have all the elements we need we will assemble and constrain everything perfectly so we will not have to face any problems in the analysis module.

We will start inserting the main and the biggest elements: The pillar, the assembly of the beams, the sheet and the axis. We will do it this way because we don't need all the elements to start with the assembly process, because we can assemble some bodies and once we finish we can insert the rest and assemble them more easily.
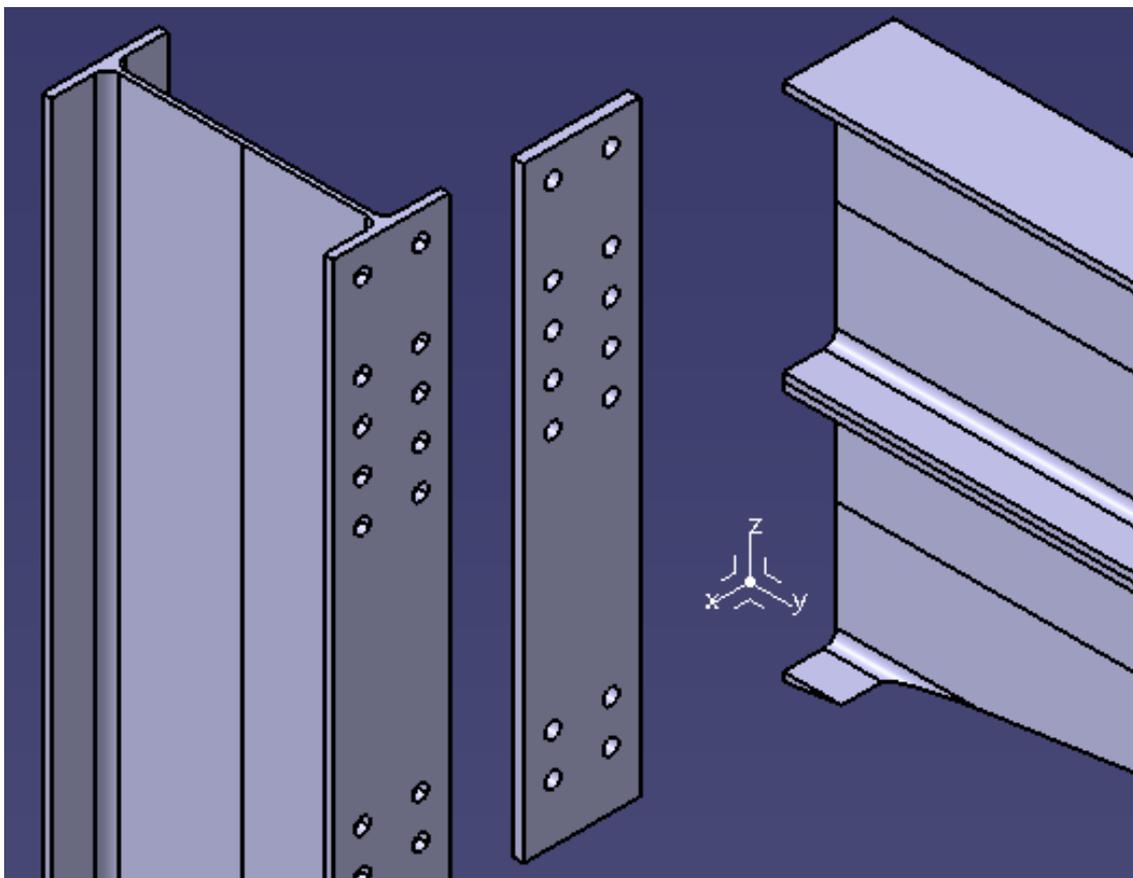
Figure 74. Example 3 Final Assembly main elements inserted.

Now we will start with constraints of fix, coincidence, contact and offset. It would be important to check that the holes in both bodies (IPE 500 pillar and 20 mm. sheet) are completely aligned.

Fix constraint

1. - We will only fix one element in the whole process. This will be the external Axis and we will use it as a reference to assemble the rest of the main elements.

Coincidence Constraint:

1. - Now that we have our Axis fixed in space, we will use it to make a coincidence between the external axis and the axis of the pillar. This way we will have our pillar restricted in space without affecting the analysis results

2. - We will apply a coincidence constraint to the upper face of both the pillar and the sheet, indicating that those faces have to be on the same level.

3. - Also with the pillar and the sheet, we will use a coincidence constrain for the left face of both bodies, locating perfectly the sheet in CATIA's space.

4. – The last coincidence constraint would be between the pillar and the beams. Like we did with the sheet, we will constrain the left face of the pillar with its respective in the beams assembly.

Contact Constraint:

1. - Once the sheet is located in the space, we will restrict its movement by applying a contact constrain between the front face of the pillar and the back face of the sheet. This way, the sheet gets completely constrained.

2. – We will make a contact restriction between the front face of the pillar and the front face of the beams (notice that both faces have not the same geometry). However, the beams will not be completely constrained with these to constraints

Offset Constraint:

1. - Only one offset constraint will be needed in this product. We will apply an offset constraint between the upper face of both the pillars and the beams assembly and we will set the value of the offset in 100 mm.
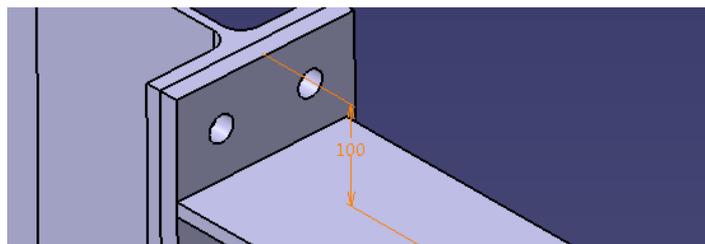
Figure 75. Example 3 Final Assembly Offset constraint value.

With all the constraints applied this is the result we obtain (it is important to use the Manipulation command to check if our assembly is restricted as we please):
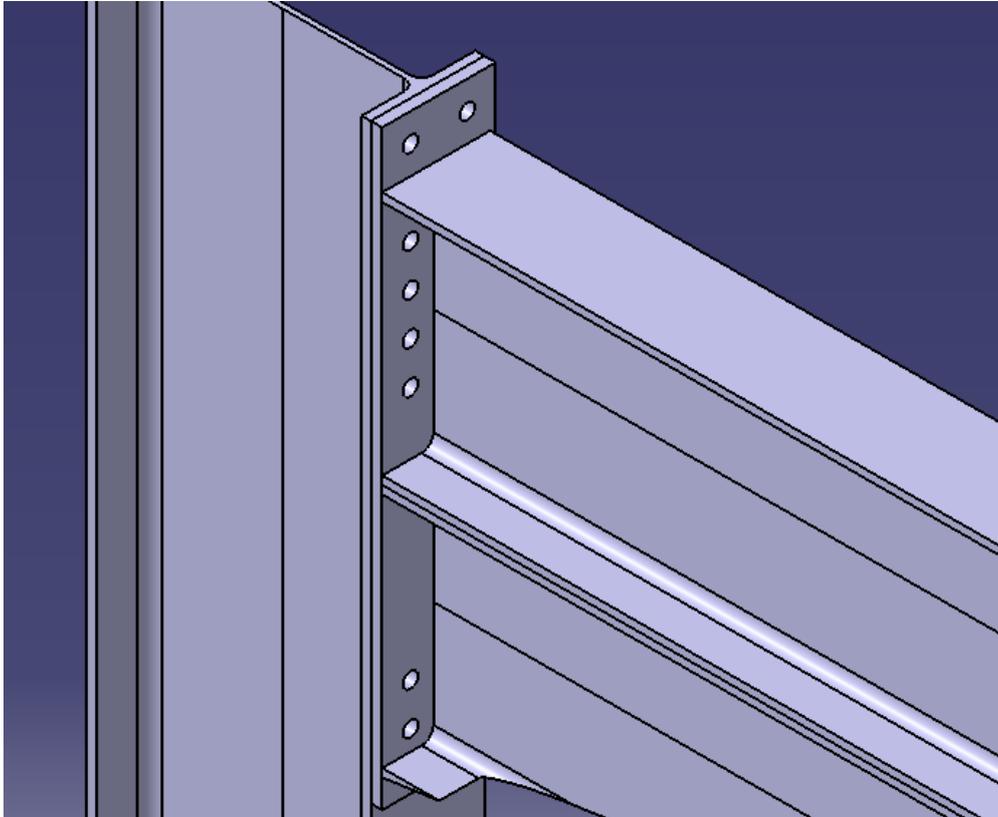
Figure 76. Example 3 Final Assembly Main elements assembled.

Now, if everything went correctly with the Manipulation command and all the measures fit as expected we can insert the rest of the elements and assemble them. Please notice that we will insert the screw and the assembly of the nut and the washer to avoid extra-constraints:
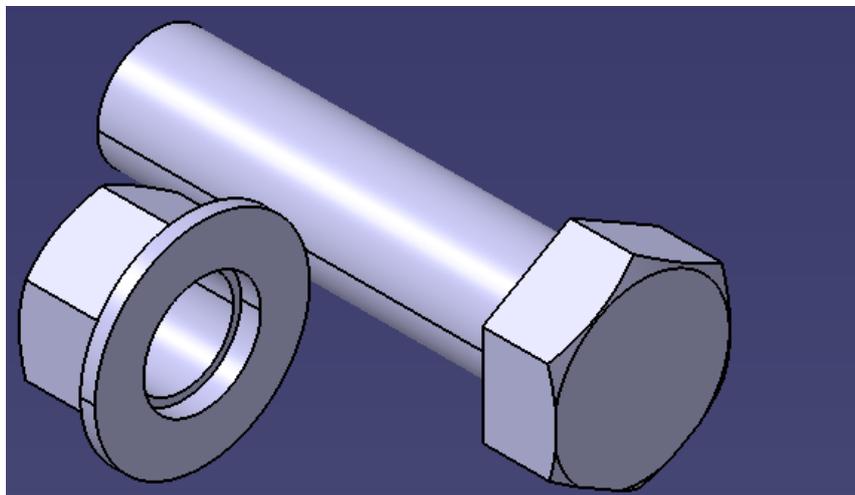
Figure 77. Example 3 Final Assembly Secondary elements assembled.

With everything inserted we will start with the constraints needed for the secondary elements. We will only need coincidence and contact constraints for said elements:

Coincidence Constraint:

1. - The first coincidence constraint will be between the edge of the screw and the edge of one of the holes. This way the screw will be almost constrained, only able to move in said edge.

2. - The second one will be exactly like the first one but between the Washer + Nut assembly and one of the holes (The same hole as the screw).

Contact Constraint:

1. - To completely constrain the screw we will have to use a contact constraint between the inner face of the head and the front face of the sheet. The screw is completely constrained now.

2. - Following the same style of contact constrain we applied to the screw we will set between the free face of the washer and the back of the front face of the pillar. This way the washer and the nut get completely constrained.

The result obtained is the following:
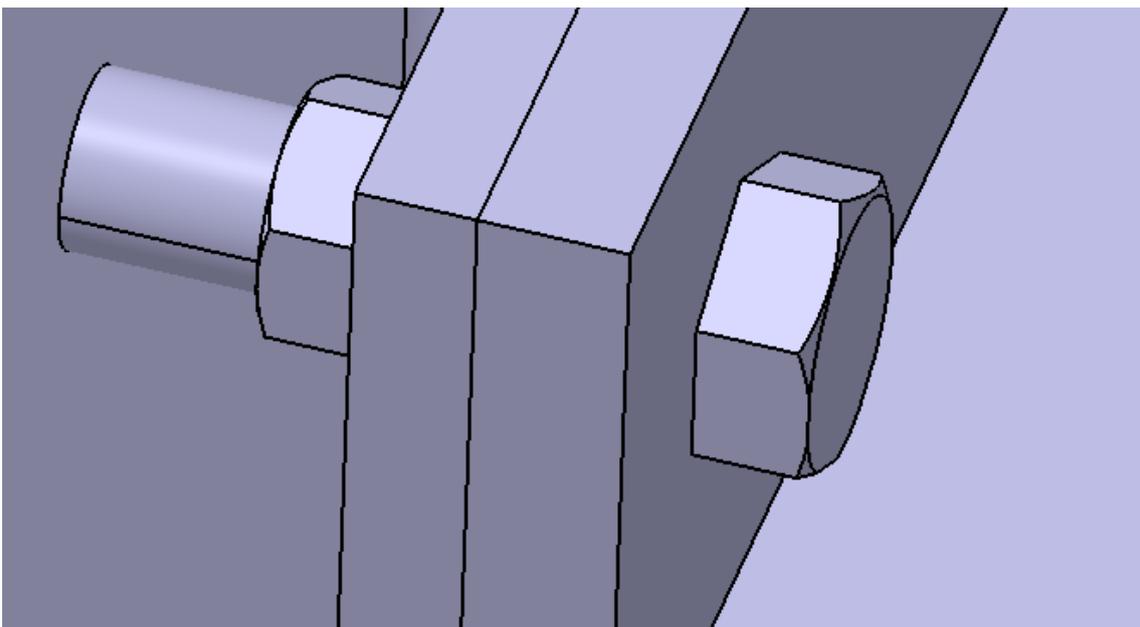
Figure 78. Example 3 Final Assembly Secondary elements assembled.

It is important to mention that <u>we haven't constrained all the freedom degrees of the secondary elements</u>. This is because all the elements are allowed to spin taking the edge of the hole as a reference (we do this to simulate how a real screw works).

The last thing to do is to insert as many screws, washers and nuts as needed and constrain them the same way we did with the first pack. Once we finish with all of them this is the result we obtain:
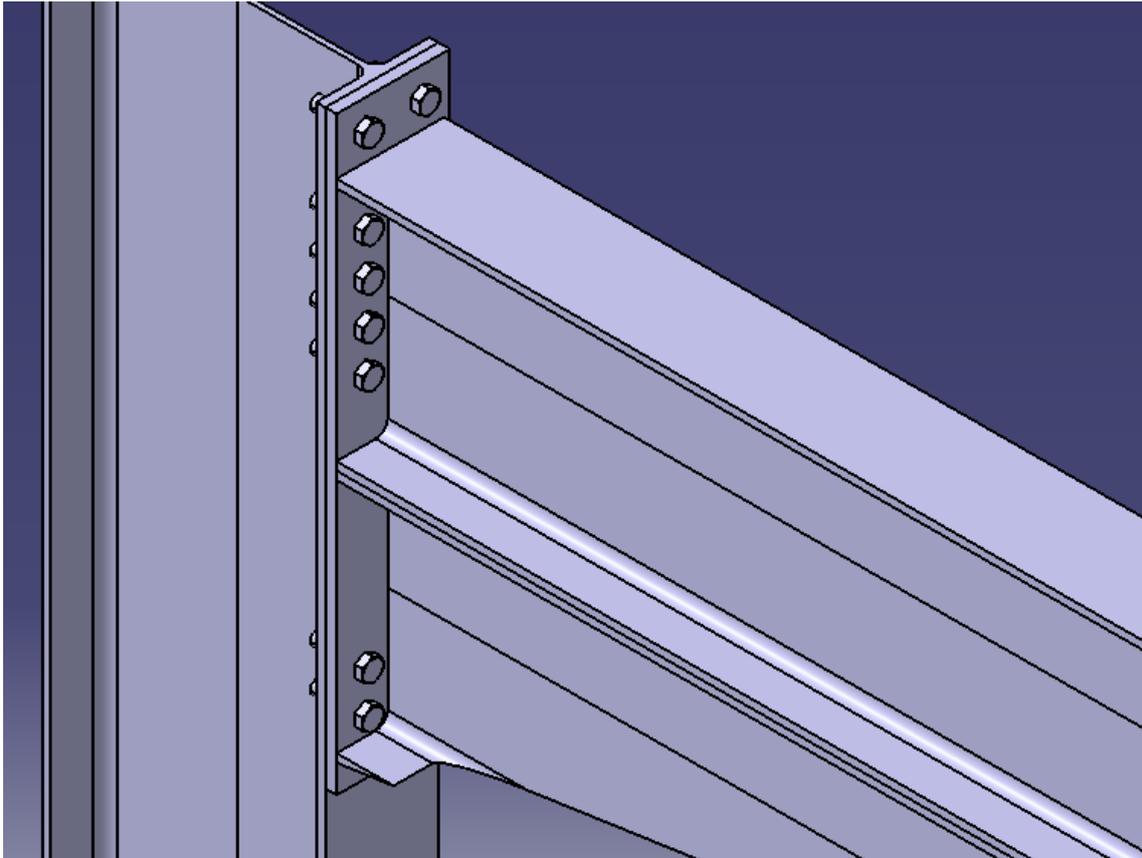
Figure 79. Example 3 Final Assembly completed.

With that, the example three is ready to go into the analysis module. However if we are doing an assembly with multiple elements or a bit complex we can use a command to check if there is any kind of overlap between elements. This command is called *Clash*.

If we click on our product and then we execute on the command we will get an analysis of all the contacts and overlaps ("clashes") in the product. Of course we already know that there are going to be a lot of contacts so we will not take them into consideration. We will only check the overlaps.

When executing the command, we will select only the clash option. This is the obtained result:
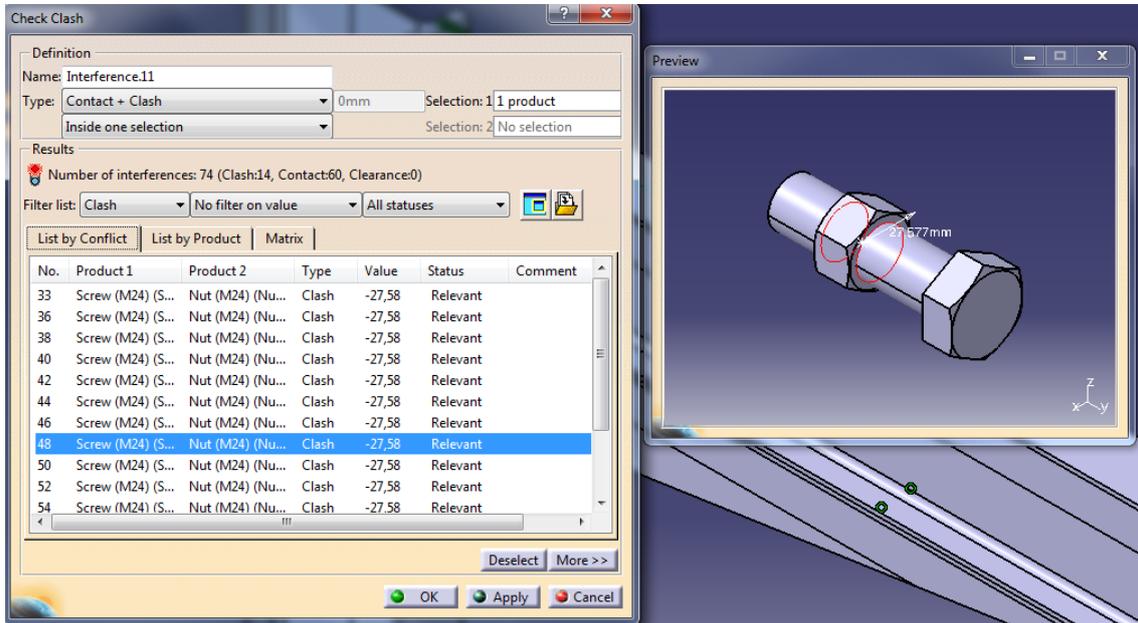
Figure 80. Example 3 Final Assembly Secondary elements overlap.

As we can see we have plenty of clash interference in our project. This is caused by the screws and nuts; as we can see in the preview of the command the washer diameter is too thin for fitting the screw.

However we already knew this was going to happen, as we talked about it previously in this chapter (page 61). This error happens because if you want to do a Tap on a surface you to design the hole with the diameter needed for doing said tap, and meanwhile in the analysis module CATIA sees a threaded surface it doesn't work equally in the workbenches of Part Design and Assembly Design. The solution here is to change the status of this error to "irrelevant" inside of the Clash command and proceed like we were going to, ignoring this error.

It is important to mention that this error was reported to CATIA and they solved the issue in further releases but it is something we have to deal in CATIA V5 R21.

Analysis (FEM)

In this chapter we will review one of the workbenches inside of CATIA's analysis module. We will simulate all three examples displayed in the previous chapter with the objective of demonstrating different interpretations of the workbench.

However, before entering the analysis module we will introduce a new command from the Part Design workbench. We have not seen this command before because this command is not necessary unless you are going to test stress simulations, like we are going to do. This command is called *Apply Material*.

The command itself has no mystery. We will select our bodies (individually) and then we will select the desired material out of CATIA's catalog:
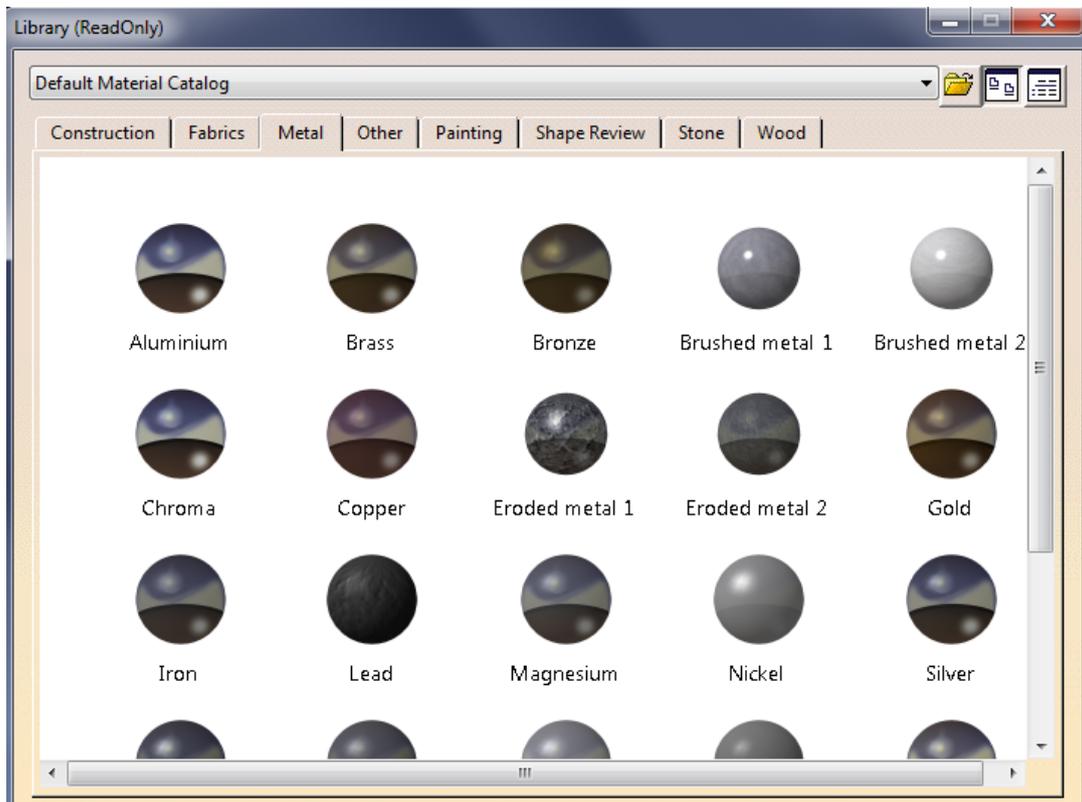
Figure 81. Analysis Applying materials.

As we can see we have a wide amount of materials distributed in chapters. In our case we will look for the Steel material inside of the Metal chapter (it is possible to check the properties of the material and change them if desired).

Like I said before, we will have to apply this command to all the bodies individually. Once we finish we will be ready to change modules and get into the analysis part.

Example 1: Pillar IPE 500

Starting with the Pillar, first of all we will open the file and quickly change from the Assembly Design workbench to *GSA workbench*. For doing that we will have to follow the same steps we did follow to change from Part Design to Assembly Design:
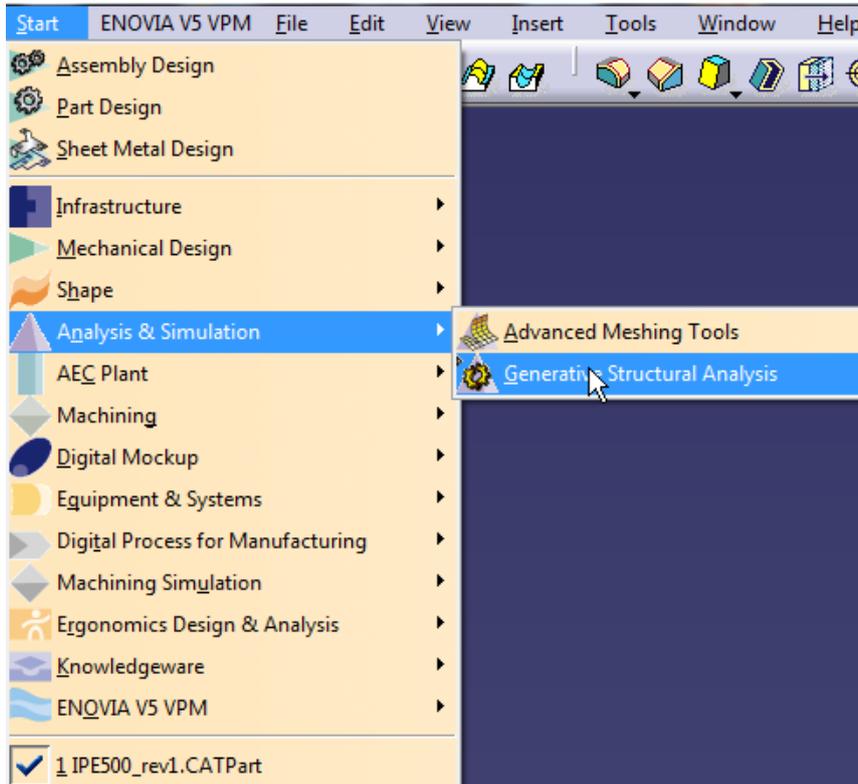
Figure 82. Analysis Example 1 changing to analysis module.

As we can see in the figure above we will have to go to the start menu on the toolbar located on the top side of CATIA's interface. Then, we will click on the Analysis & simulation module and then on the GSA workbench.

The first thing we have to do once we get in the module is to select the type of case we want to analyze:
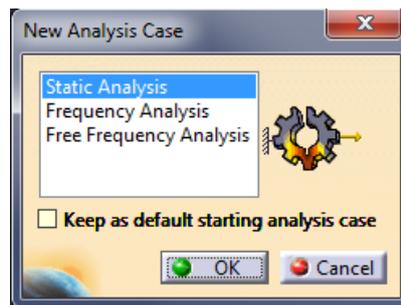
Figure 83. Analysis Example 1 Analysis case.

For our three examples we will select the Static case as we are analyzing structures.

However, as we already see in the Figure 83, CATIA offers two more cases inside of the same workbench (GSA) which allows us to extend our analysis if desired.

Next we will take a look at all the Meshes inside of the product. When we talk about meshes we refer to all the different analysis surfaces that the program is going to analyze. In our case we only have one mesh because the example one is just one solid body.

If we open the module we will be able to see the configuration we are able to customize inside the mesh:
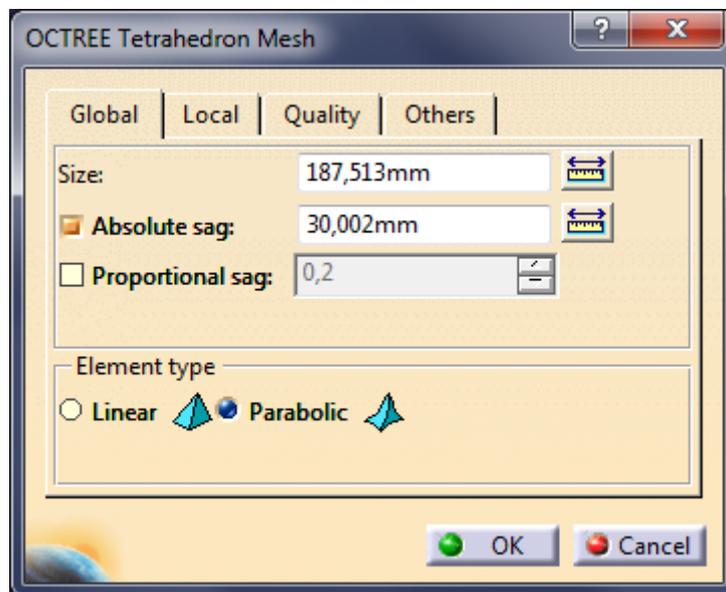

Figure 84. Analysis Example 1 Analysis case.

We can see in the figure above that we have four submenus where we can change the default configuration of the mesh. We will focus on the Global menu as it is the main one when it comes to settings.

Keeping the eyes on the Figure 84 we see two different measures. The first one refers to the size of the mesh, the bigger the mesh the most precise it would be. However there is a point where incrementing the mesh it will only delay the analysis. Normally the value that CATIA offers as default corresponds with that point. If we are struggling with the analysis we can reduce the mesh's size. The second numbers refers to the mesh sag (the deep it goes in terms on testing. It works the same way as the first number does, however we can change it to a proportional value depending on the initial mesh size.

Once we are done adapting the mesh to our requirements (we can always come back to the settings and change it as we please) what follows is to restrict the product, for example one of the faces has to be clamped in our case. For this we will use a new command called *Clamp*.

The clamp command is really simple as it only asks us to select a face to be clamped. In our case we will select the bottom face of the pillar obtaining the following result:
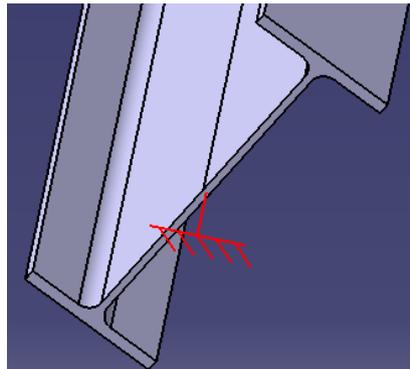
Figure 85. Analysis Example 1 pillar clamped.

Now that we have our product clamped we need to add at least one load to make the simulation visible. In our case we will apply two of them, the first one as an axial force distributed on the opposite side of the clamped face and the second one another distributed force located on the front face of the pillar.

For applying loads we will use a new command called *Distributed Force*. This command offers us different options inside its menu:
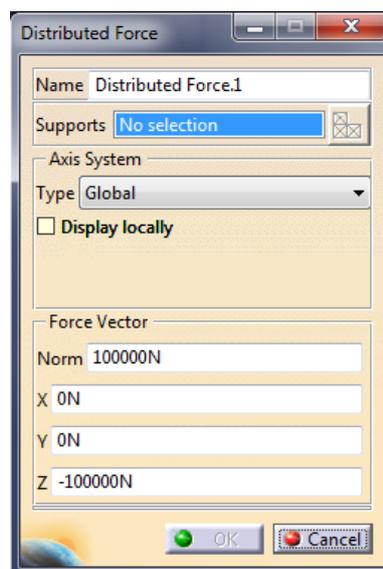
Figure 86. Analysis Example 1 Distributed Force menu options.

As we can see in the Figure 86 we can rename our loads in case we have more than one and we need to locate them. Second we have to select the face where the force is going to be applied. And last but most important, we can decide the axes system we want to use (local or global) and introduce the desired value. The command will give us a preview of the direction of the load. In our case we will apply 100 kN to the top face and 25 kN to the front face. Once we execute the command this the obtained result:
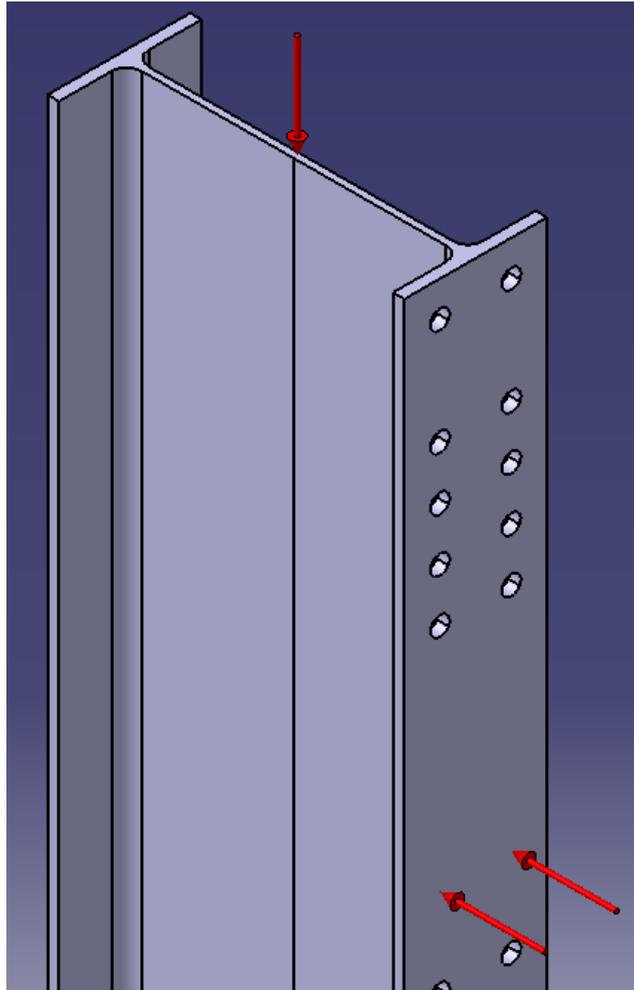
Figure 87. Analysis Example 1 Distributed Force implemented.

The red arrows show us that there are loads in those directions but they do not say anything about the value. In case we want to know it, we just have to double-click on them and the menu we just saw on the Figure 86 will pop up again showing us the value of the load. If we want to change it we just have to edit the value that appears once we click on the arrows.

Due to we do not have any constraints we don't have to do anything else before calculating. For the calculation we will use a new CATIA command called *Compute*. This command is not just a "run" bottom. Inside of its menu we have different analysis options:

Figure 88. Analysis Example 1 Compute menu options.

As we can see we can only select different part of the analysis in case we are only interested in one of the parts. In our case we will select all the parts and execute the command. As this is a very simple command the analysis will take seconds, obtaining the result we have been waiting:
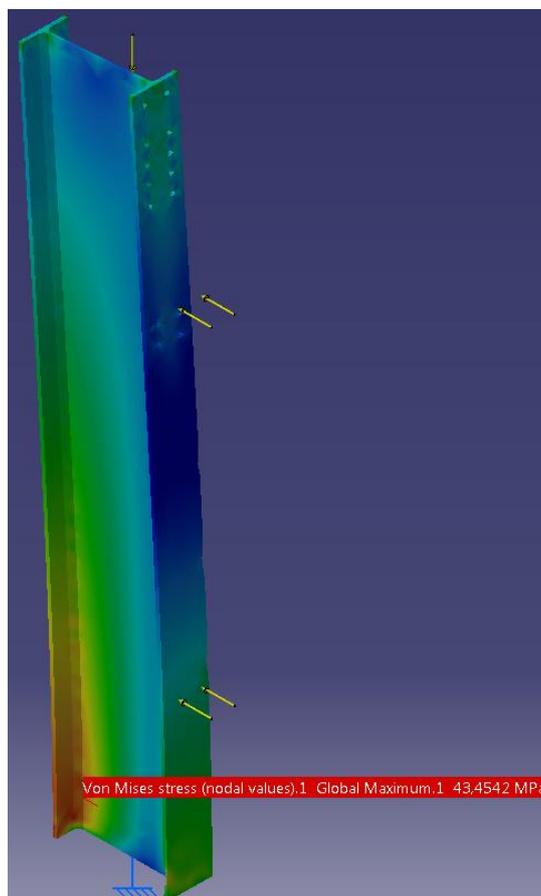
Figure 89. Analysis Example 1 Compute completed.

And there it is our 3D Stress (*Von Mises stress*) analysis. CATIA also offers us to see real deformations in the whole product as well as displacements. In this example we will be reviewing the Von Mises stress as it is what is going to indicate us if the product will resist the loads without major complications. As we can see we have a maximum of 43,4542 Mpa (N/mm^2) which is nothing for the steel.

However if we want to extract a report with a quick summary of the product and the analysis results is possible. If we go to the Tool sub-menu (located on the top toolbar) and click on information and then on the Von Mises display we will obtain the following summary:
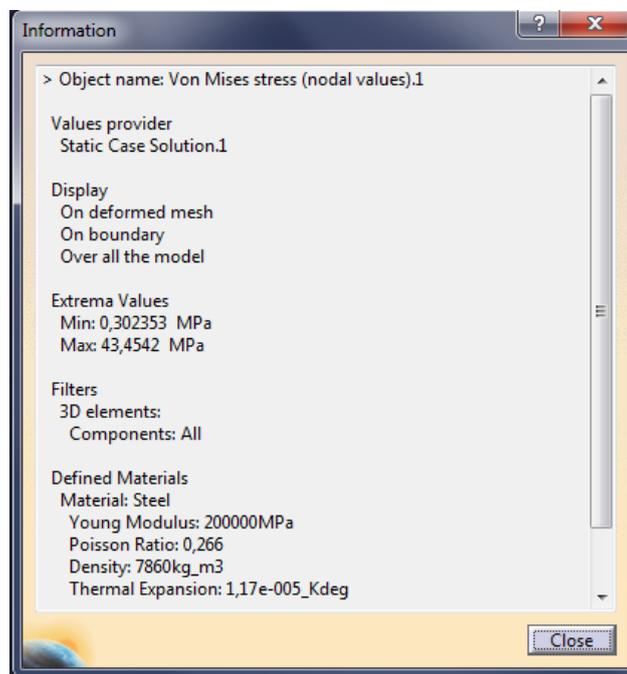
Figure 90. Analysis Example 1 Von Mises information summary.

Like we said, this summary offers a quick and simple view of the analysis. But, if we want a deeper and extended analysis CATIA has it also. For getting it we only have to save the file and we will get an extra file containing all the compilation and analysis results.

That concludes the Example 1. In the next example we will explain what we have to do when we have constraints in our product, basically we will tell CATIA how to interpret those constraints in terms of analysis.

Example 2: Beams IPE 450

Following with the analysis chapter we will get into the next example where we will be testing the assembly between two beams IPE 450, one of them trimmed. We will follow the same script as Example one, beginning with changing workbenches from Assembly Design to GSA. We will run the static case again.

If we do not want to change anything regarding the mesh we can jump right into the Clamp command (as we have two solids now, we will have 2 meshes). In this example we will clamp the biggest end of the profile, just to show the results CATIA offers regarding a clamped beam union:
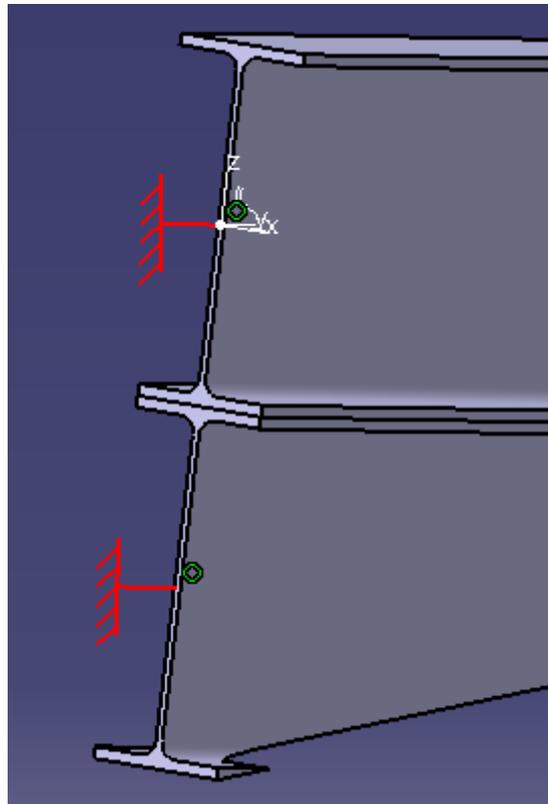
Figure 91. Analysis Example 2 Beams clamped.

Once we finish clamping the beams the only thing left to do is to assign a load to the product. As this is a beam union after all we will be only creating one load for this example. It will be a 100 kN load located on the top face of the union, obtaining this way a perfect simulation of a beam clamped.

When we finish introducing the loads into the program this is what we should obtain:
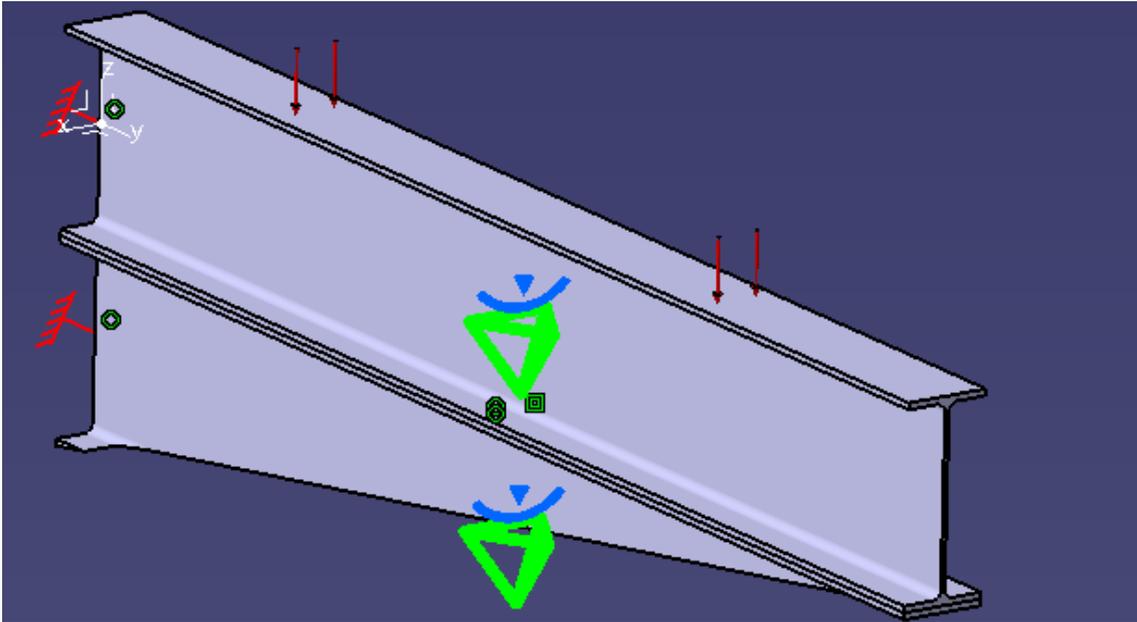
Figure 92. Analysis Example 2 Distributed Force implemented.

Now, we will talk about something new that we didn't see on the Example 1, how to deal with the constraints we implemented in the Assembly Design workbench. We will have to indicate CATIA how to interpret those constraints in terms of analysis. For doing this we will introduce the three commands we will be using: *Contact Connection Property (CCP), Smooth Connection Property (SCP)* and *Rigid Connection Property (RCP).*

All the contact restrictions we have placed will be assigned with the command CCP, to tell CATIA that the elements involved in the restriction have to remain in contact during the analysis. We will use the RCP command with those constraints that in real life are rigid, like a welding for example. And last but not least, we will use SCP for restraints that have geometrical purposes. What "geometrical purposes" means is that we have placed those elements that way but a force strong enough could move those elements and separate them.

With that being said we will apply all the command to their respective commands. Obtaining the following result:
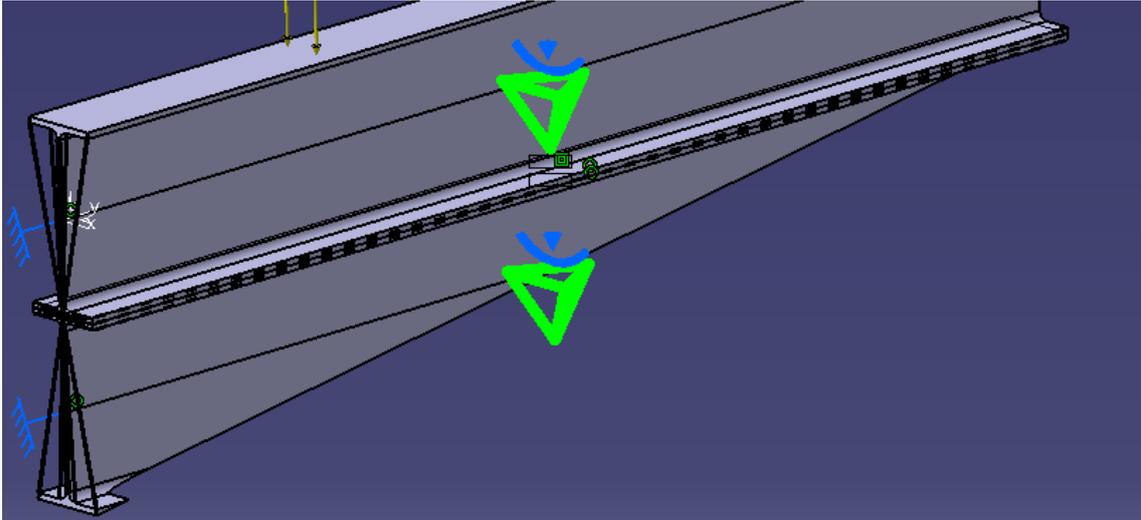
Figure 93. Analysis Example 2 Restraints assigned.

Regarding the preparation for the execution we are done. The last thing to do now is to execute the Compute command and obtain the analysis results. Like we did in the example one we will simulate all in a matter of seconds.

For a 100 kN load placed on the top face of the product this is the obtained result:

Von Mises stress (nodal values).1 Global Maximum.1 96,095 MPa
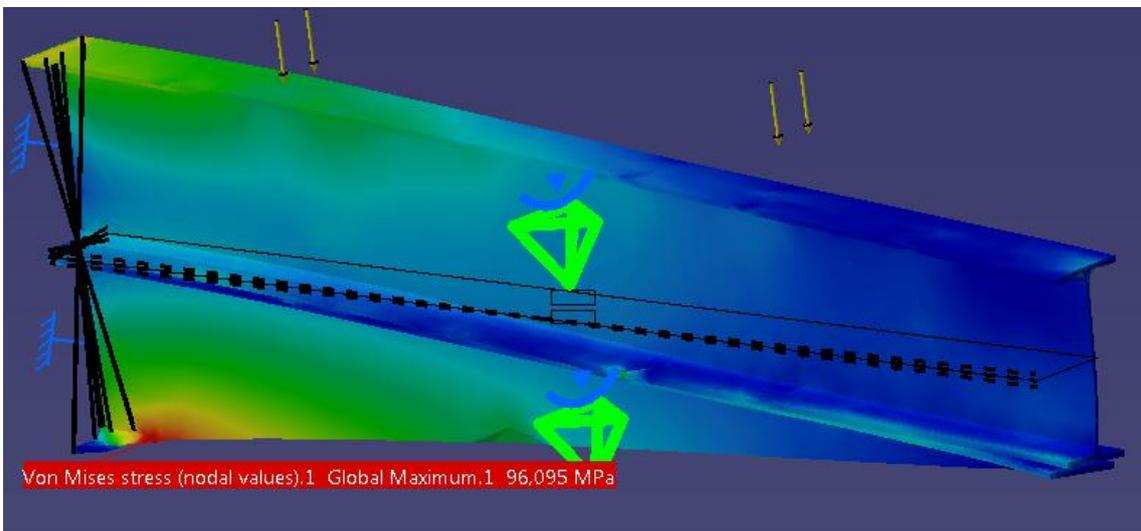
Figure 94. Analysis Example 2 Compute completed.

For said load we obtain a maximum of 96 MPa located on the bottom of the trimmed profile. We got a higher value than the one we got in the Example 1, mainly because the load was placed on a worse position in terms of enduring the load and the generation of stress. With that being said the Example 2 is completed so we will review the last one in this thesis, the Example 3.

Example 3: Lintel Union IPE 500 + IPE 450

Example three will be the last product we will be reviewing in this thesis. As we have already explained all the processes and commands we need in order to prepare the product for the analysis, we will just comment on the interesting facts we find during the process.

Starting like we did with the other examples, we will get right into the Clamp command. In this example we will clamp the base of the pillar, like it was completely restrained to the floor:
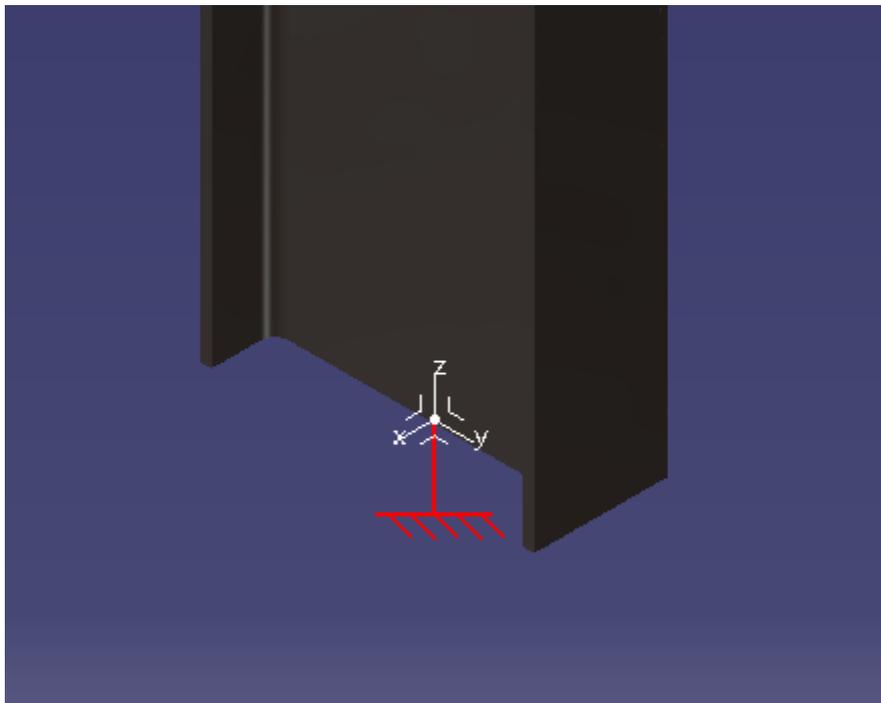
Figure 95. Analysis Example 3 Product clamped.

In terms of loads we will apply two different distributed loads on the product. This combination will be a result of both previous examples. We will apply 200 kN on the head of the pillar simulating an axial force; and we will add another one of 100 kN on the top face of the beams, simulating this way shear forces and momentum to be endure it by the product. We will not post a figure of the addition of the loads as we have already explained the look it has and how to work with the command once we have placed the loads in case we want to delete or change anything.

Example 3: Lintel Union IPE 500 + IPE 450

Instead, we will jump right into the interpretation of the constraints as in this example we have to be extremely careful with the different assemblies we have. On one hand we will have to deal with the beams the same way we did in the Example 2. On the other, we will have to take into consideration all the restraints we have in the screw+ nut + washer assembly, as we have fourteen of those assemblies each of them with their own restraints. Although it will take time, this is the preview we should obtain once we finish assigning all the constraints:
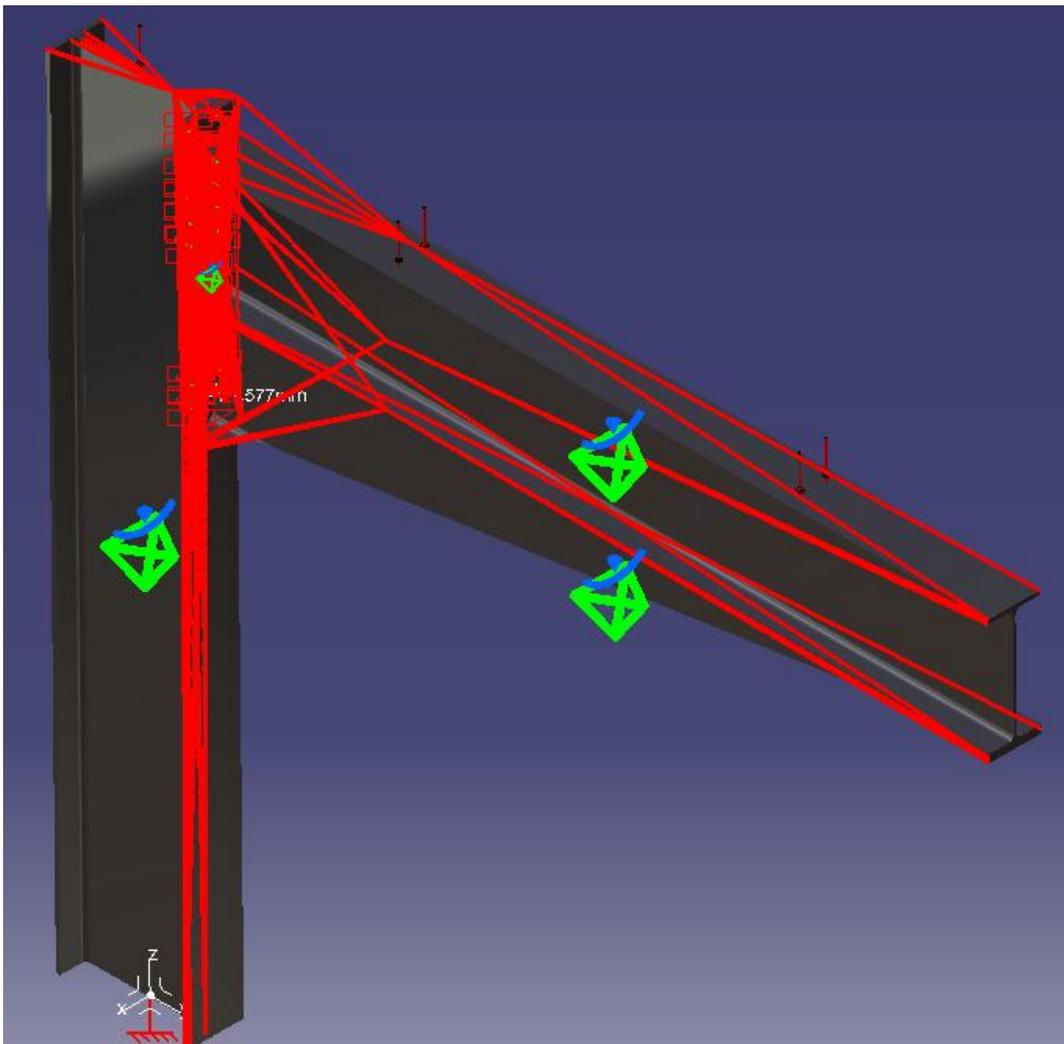
Figure 96. Analysis Example 3 Product restrained.

After setting all the properties to the restraints the only thing we have left to do is to execute the Compute command and obtain the Von Mises Stress diagram.

We have to understand that assigning all the constraints to the constraints is the most delicate task we have in CATIA, because depending on what we tell the program to interpret, the result will change drastically.

In our case we have to take into consideration that our model is <u>rigid</u>. What rigid means is that we will indicate CATIA that all the efforts and stress will be transmitted to the base of the pillar, where we located the clamped face. The figure below is the obtained result for our analysis:
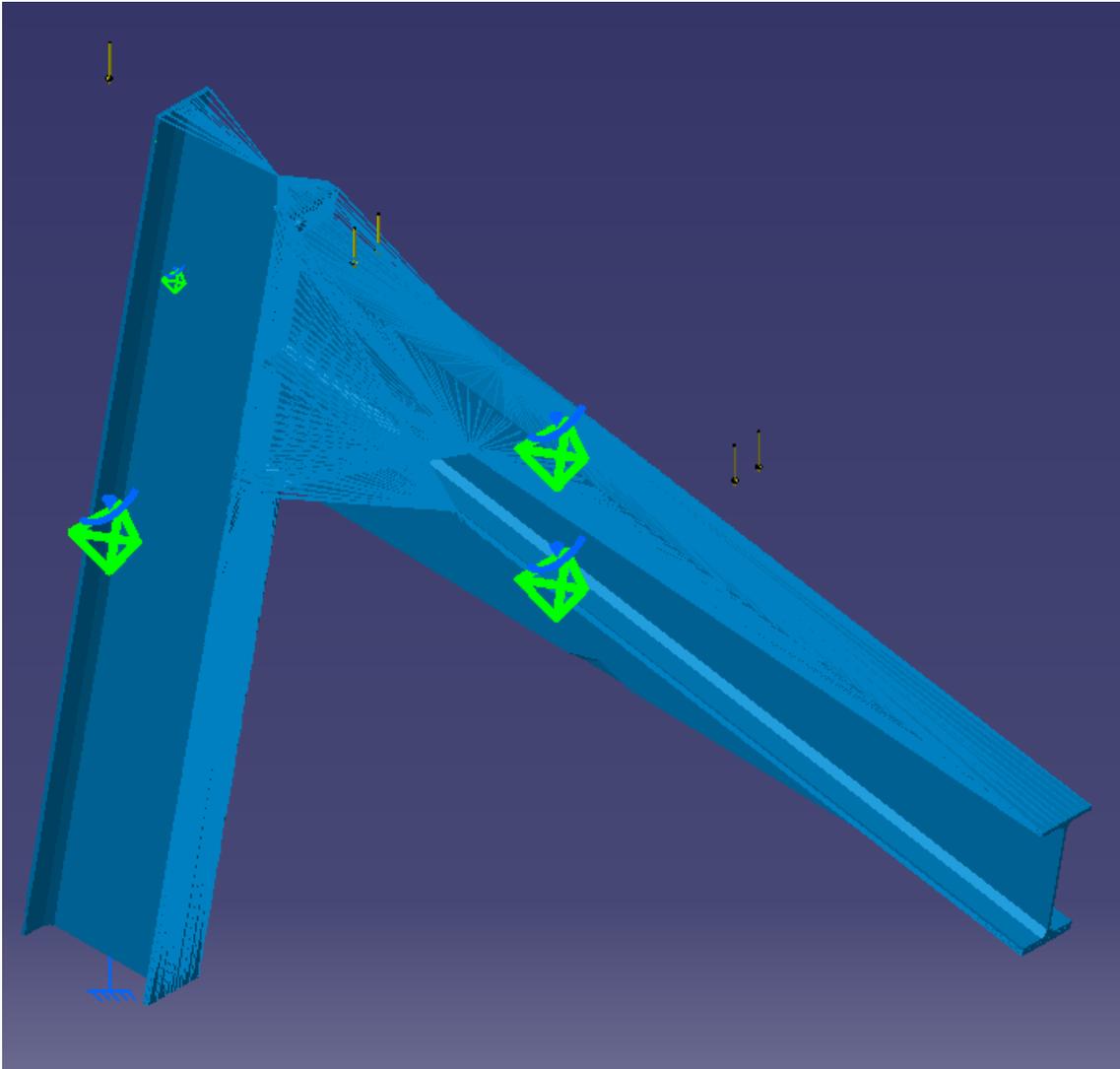
Figure 97. Analysis Example 3 Compute completed.

In this example we will be reviewing other option of visualization that CATIA offers. This is the Deformation map where we can see the direction and the values (if we hove the mouse on the product) in the designated unit of that deformation.

That concludes the analysis of the last example, and with that, the chapter number 4. We will jump into the last chapter of the thesis (Chapter 5: Conclusion), where we will debate if CATIA is truly an option to take into consideration in the Structural Design world.

Conclusion

In this chapter we will review the whole process we went through working with CATIA. We will start from the chapter number 3 (Designing and Modeling); talking about different facilities and struggles we faced in the process. Next, we will jump into chapter number 4 (Analysis FEM); where we will discuss if the results are worth of all the process. And last but not least, we will talk about the BIM regulation and how we can benefit from CATIA to satisfy its requirements.

CATIA: Is it worth?

We will start trying to answer the question above. The way we are going to face the question is making a comparison between pros and cons of CATIA. After that we will focus on Structural Design and we will value if it is really a worthy program:

Pros

1.- Multidisciplinary Approach.

CATIA provides a multidisciplinary approach when it comes to product design and development. It addresses the specific needs of various disciplines to improve how organizations and companies conceive, realize, and develop new products.

2.- Product Design (Styling).

As an example, the product design software offers advanced surface modeling capabilities to transportation and industrial designers. Because of this, they will be able to analyze and visualize aesthetic and ergonomic shapes to achieve the highest surface quality.

Creative designers can also access a set of tools that permits them to create and explore conceptual ideas. Using a unique 3D sketching technology, they can import 2D sketches and come up with design ideas. Likewise, virtual experience designers can produce high-quality renderings using global illumination, and reflections and shadows that are physically correct.

3.- Engineering.

CATIA allows mechanical engineers to gain insights into the factors that affect the performance and quality of mechanical products through the aid of its 3D modeling

tools. What's great about this is that they will be to obtain such insights early in the product development process. They can develop and analyze mechanical products in their operating environments through digital prototyping, analysis, and simulation.

To facilitate the creation and validation of complex systems and products, CATIA delivers a development platform to systems engineers and architects, and developers which combine cross-discipline modeling, simulation, verification, and business process support. CATIA provides them with the ability to integrate embedded systems and 3D design processes so they can better simulate complex systems and products.

In addition, through requirements engineering, they will be able to ensure compliance with market requirements and regulations, and at the same time reduce the overall cost of system and product development.

4.- Collaborative 3D Environment.

CATIA is built with a collaborative 3D environment which permits users and stakeholders to be actively involved in the design process anywhere and anytime. This design environment can be accessed through 3D dashboards, and it serves as a single source of truth.

As an example, designers can interact with other departments and collaborate with people from different locations around the world to boost design innovation. In the same manner, systems engineers and architects can collaborate across all disciplines as they design and develop the components and functions of complex systems.

Cons

1.- The main con of CATIA: Money.

CATIA is way too expensive for small and medium design enterprises to purchase the licenses. Even if you purchase a license or two, you are limited in functionality because of what 'configuration' you purchased. Many useful functions of CATIA might not be available just because an inferior configuration was selected at the time of purchase. Adding up the functionalities will require to purchase more licenses: more money.

Based on conversations with DS employees the price offer for CATIA V6 depend on how many modules you want to purchase, so the price will go from 9,000 to 65,000

thousand USD (Plus around 7,000 extra for maintenance every year) per license, which for a medium company makes it pretty much impossible.

However there are student offers and renting options depending on your requirements and needs. Of course, with a student license it is not possible to make profit out of designs and products made with this license.

2.- High level experience in 3D software

CATIA is not an easy software; you need to put a lot of work and hours on becoming a really expert with the program. Otherwise the work and performance become sloppy and slower and the program loses its worth.

Also, as we have already pointed out, CATIA has more than 60 workbenches distributed in 9 independent modules which makes almost impossible to master all of them. This is solvable by having a team with members specialized in different workbenches; but that will cause an increment on the price because we will need more than one license and as we have already seen, it is not cheap precisely.

3.- Top level processing software.

CATIA is a program that requires the 100% out of your computer. If your software is not updated to the last innovations you will struggle with CATIA, specially in the analysis module.

However, this is a con that all the 3D analysis programs have; the more powerful your software is the smoother everything will work. Specially on the analysis module, if your computer is not strong enough it can take hours analyzing because after all we are working with finite elements analysis.

CATIA in Structural Design

In the next chapter we will talk about our thoughts on CATIA based on the experience gained through the process of this thesis. We will start commenting on the designing chapter and finishing on the analysis part:

Designing with CATIA

From my point of view, CATIA is the most complete software for 3D designing on the versatility market. Maybe we could compare its designing skills with

Autodesk's Inventor, but when it comes to and the variety of options inside of the program, the competition is over.

As we have seen during the process, CATIA offers us countless of tools to make our designs come true. We have to keep in mind that we have only design using one workbench (Part Design), when the majority of industries using CATIA work with surfaces, one of the strongest points in the program's design skills. Also, in our examples we decided to make a complex sketch because were modeling IPE profiles and it would have been more complicated to change different command than just adjusting the sketch; but we have to remember that that was our choice, we could have gone other way and the result would have been perfect.

Analysis with CATIA

Now talking about analysis, we have seen in three different examples how we are able to visualize stress and deformation and a full 3D mesh diagram. Which allow us to identify exactly where to beef up the structure because we are able to identify the most stressed part of the product.

However, I have to admit that based on my experience I did struggle a lot more in the GSA workbench than I did in the Part Design or the Assembly Design workbenches. In my opinion the interface in the GSA workbench is a lot less intuitive than it is in other workbenches. For example, whenever you get an error in the GSA workbench it is way too difficult to find it and to fully understand what the problem is.

My conclusion for this thesis is that CATIA has shown that It is capable the structural design industry, but it is not a program that you can learn easily. It needs lots and lots of hours and like in my case professional formation to be able to start designing and creating content with the program. Even though the program has its pros and cons, my overall opinion is that this software is the most complete software right now. Of course, alongside quality we can find high prices (not always), but in this case it is more than worth.

BIM Regulation

In this section of the chapter we will talk about the BIM regulation as the final conclusion of the thesis. First, we will start introducing what the concept BIM stands for:

Building Information Modeling (BIM) is named after the process of generating and managing a building's data during its life cycle, using <u>dynamic building modeling software in three dimensions and in real time, to reduce the waste of time and resources in design and construction</u>. But what are the real benefits of the BIM regulation? I have pointed out the main advantages of this alternative:

Multidisciplinary work: BIM allows collaborative work through a single platform in which the different agents involved in the construction process (architects, surveyors, engineers, etc...) can work on the same tool integrating and sharing information in real time.

Real-time interference checking. Before starting to build, we have a complete model with all the information needed. This way, you can check if there is any integration problem between the different elements in a safe and fast way.

3D and 2D working along. This methodology integrates the 2 and 3 interconnected dimensions, improving the vision of the global project and avoiding possible errors.

Automatic document generation: budgets, planning structures, etc. achieving an increase in productivity and, therefore, saving time and costs.

Energy impact. Through BIM, we can know the energy impact of each of the materials that make up the work. In this way, smart buildings can be created that are increasingly integrated with the environment, sustainable and efficient.

What this brief definition means is that the BIM regulations demands us to add all the technical information regarding construction elements and materials of a project. What the BIM regulation aims is to help future workers in the task of reforming and rehabilitating structures. Nowadays is pretty common to start a reforming process of an old building and, if you are lucky enough to find the plans, there is no information about which profiles make the structure up.

CATIA allows us to do this. We can introduce all the information we want inside of our bodies in the properties section. We can change from the name of the profile to add technical information all inside of the body. CATIA accomplish pretty much all the requirements of the BIM regulations.

This is just another advantage of CATIA in structural design. As we already know, the world keeps on evolving and we have to adapt the best we can. From 2018, the BIM regulation will be implemented in Spain, that will join another European countries such as UK or Norway. This is the future changing the way we work in the structural field and we have to change with it.

Annex: Bibliography

In this annex I have compiled all the internet sites that I've consulted for information and opinions:

1.- Dassault Systemes (https://www.3ds.com)

2.- Quora forum (https://www.quora.com)

3.- WorldCAD Access forum (http://www.worldcadaccess.com)

4.- The CAD Forum (https://www.thecadforums.com)

5.- CATIA Community forum (http://catiadoc.free.fr)

6.- 3D Portal forum (www.3dcadportal.com)

7.- Wikipedia (https://en.wikipedia.org)

8.- CICE blog (https://www.cice.es/blog-landing/)

9.- CATIA tutorials (https://catiav5v6tutorials.com)

In addition to the internet sites mentioned above, I have also consulted documents provided by my tutor who kindly allowed me to use all the information in those documents for my thesis:

10.- Assembly Design guide (CATIA).

11.- Analysis guide for CATIA V.5.

12.- Dintel Union guide (ICNC).

Annex: Technical Specifications

As we have already commented on the outline of the conclusion, we need a powerful set-up in order to run CATIA comfortable. Preventing this problem, DS offers a free guide for hardware and software on their web page (https://www.3ds.com). If we go to the "Hardware and Software configuration" section we will get a preview like this:
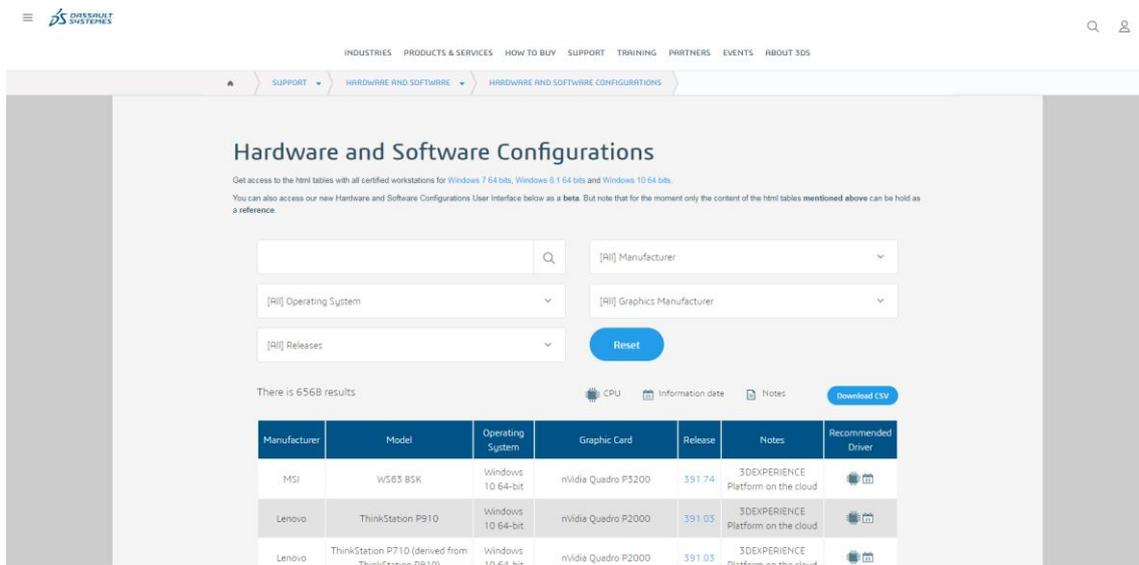

Figure 98. Dassault Systemes Software and Hardware search service.

With this tool you can look for the following requirements and the searcher will provide you with the most commercial brands and models that match your requirements:

Operating system: We will have to choose between Windows 7 64 bit, Windows 8.1 64 bit and Windows 10 64 bit. CATIA does not work with 32 bit software or other operating systems apart from windows.

Releases: The searcher allows us to discriminate between releases in case we are working with CATIA V5 (like our case) and you want to look for that specific release. These are all the releases we DS offers to look for:
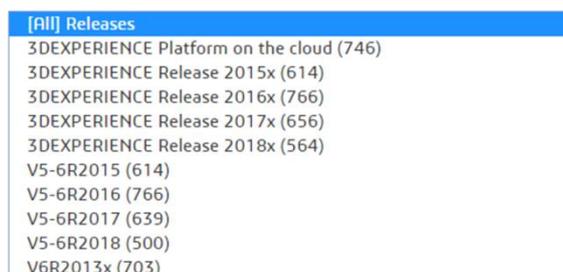

Figure 99. Software and Hardware search service Releases option.

Manufacturer: In this option we can choose between a bunch of manufacturers that usually work with CATIA, such as Dell, HP or MSI. Even though the options in terms of manufacturer are "limited" we can approach other brands and ask them for their technical configuration to see if they match our requirements.

Graphics Manufacturer: This section is limited to two options; AMD and nVidia. These are the largest producers of graphic software in the whole world. In my personal opinion, it is very unusual to find a decent computer with and nVidia graphic card inside, so I will go with nVidia 100%.

Once we have selected our requirements we can just look at the table generate with our requirements. However, CATIA also offers this table if we do not specify on some categories (for example we can have no preferences on the computer brand):

Manufacturer	Model	Operating System	Graphic Card	Release	Notes	Recommended Driver
MSI	WS63 8SK	Windows 10 64-bit	nVidia Quadro P3200	391.74	3DEXPERIENCE Platform on the cloud	
Lenovo	ThinkStation P910	Windows 10 64-bit	nVidia Quadro P	CPU Intel Core i7-8850H (2.60)		
Lenovo	ThinkStation P710 (derived from ThinkStation P910)	Windows 10 64-bit	nVidia Quadro P2000	391.03	3DEXPERIENCE Platform on the cloud	
Dell	Precision 3530	Windows 10 64-bit	nVidia Quadro P600	391.03	3DEXPERIENCE Platform on the cloud	
Dell	Precision 5530	Windows 10 64-bit	nVidia Quadro P2000	391.03	3DEXPERIENCE Platform on the cloud	
HP	Z440 (derived from Z840)	Windows 10 64-bit	nVidia Quadro K4200	391.03	3DEXPERIENCE Platform on the cloud	
Dell	Precision 5530	Windows 10 64-bit	nVidia Quadro P1000 (derived from Quadro P2000)	391.03	3DEXPERIENCE Platform on the cloud	
HP	Z640 (derived from Z840)	Windows 10 64-bit	nVidia Quadro K4200	391.03	3DEXPERIENCE Platform on the cloud	
HP	Z840	Windows 10 64-bit	nVidia Quadro K4200	391.03	3DEXPERIENCE Platform on the cloud	

Figure 100. Software and Hardware search service result.

Like we see on the Figure 100, on the last column called "Recommended Driver" we have a little icon with the shape of a Microchip that if we pass the mouse over it we will be able to see which Intel core is necessary (there no more options than Intel because CATIA only works with Intel processors).

I have referred to the search tool of DS because it is not possible to give only one valid option of configuration to run CATIA correctly. As we have seen in this chapter, we have many combination and possibilities.

Annex: Thesis Budget

In this annex we will get into the hypothetical price for this thesis as it was commissioned by a company as a research project. There are a few factors we have to take into consideration.

The first one would be price of the license of CATIA; because, even though this thesis is not based on designing and creating specific pieces for a selling purpose, we did use the program and of course we have to pay its price. CATIA has a student version for only 99 USD. However it is forbidden to sell or to make any profit out of CATIA using this license. For all the modules we used in this thesis, Mechanical Design and Analysis & Simulation (we can refer to Figure 39 to see all the modules) we would be talking about 30,000 Euros plus 2000 on maintenance per year (Renting) based on the feedback obtained from the DS employees via its web page.

The second factor will be the salary of the thesis' redactor as an Engineer. Based on my tutor's indications we would be talking about 20 Euros/hour for an Engineer salary. Taking into consideration that the working period will cover around 30 hours a week during 3 months. If we make the calculations we reach a result.

However, as renting CATIA's license covers the entire year, I have come up with 2 final budgets. One of them only counting the 3 month period for the license, and the second one counting the whole year:

Engineer Salary €/H	Hours per month	Months of work	Salary Gross	CATIA License 1 year price (€)	*Total Gross*
20,00 €	120	3	7.200,00 €	33.000,00 €	*40.2800,00 €*

Table 2. Budget Thesis alternative 1 (12 months CATIA).

Engineer Salary €/H	Hours per month	Months of work	Salary Gross	CATIA License 3 Months price (€)	*Total Gross*
20,00 €	120	3	7.200,00 €	8.250,00 €	*15.450,00 €*

Table 3. Budget Thesis alternative 2 (3 months CATIA).

Annex: Extended definitions (Nomenclature)

1- *Computer-aided design (CAD)*: CAD consists of using computer programs to create, modify, analyze and document two- or three-dimensional (2D or 3D) graphical representations of physical objects, offering an alternative to manual drafts and product prototypes. CAD is widely used in computer animation and media special effects as well as in product and industrial design.

2- *Computer-aided manufacturing (CAM)*: CAM commonly refers to the use of numerical control (NC) computer software applications to create detailed instructions that drive computer numerical control (CNC) machine tools for manufacturing parts.

3- *Computer-aided engineering (CAE)*: CAE refers to the use of computer software to simulate performance in order to improve product designs or assist in the resolution of engineering problems for a wide range of industries. This includes simulation, validation and optimization of products, processes, and manufacturing tools.

4- *Product Lifecycle Management (PLM)*: PLM is an information-management system that can integrate data, processes, business systems and, ultimately, people in an extended enterprise. PLM software also allows you to manage this information throughout the entire lifecycle of a product efficiently and cost-effectively from ideation, design and manufacture through service and disposal.

5- *Dassault Systèmes (DS):* Dassault Systèmes is a subsidiary of the Dassault Group created in 1981 by Avions Marcel Dassault. It is a European software company headquartered in Vélizy-Villacoublay, France. DS develops 3D design, 3D digital mock- up, and product lifecycle management (PLM) software.

6- *Finite Element Module (FEM)*: The Finite Element Module (FEM) is one of the options CATIA offers when it comes to Analysis. As its name already points, this module will run finite elements analysis over our design, showing in the end how the stress is distributed through the piece (3D of course).

7- **Part Design***:* The Part Design workbench makes it possible to design precise 3D mechanical parts with an intuitive and flexible user interface, from sketching in an assembly context to iterative detailed design. While using the Part Design workbench you will be able to accommodate your design requirements for parts of various complexities, from simple to advanced.

8- **Assembly Design**: Assembly Design allows the design of assemblies with an intuitive and flexible user interface. As a scalable workbench, Assembly Design can be cooperatively used with other current companion products such as Part Design.

9- **ArcelorMittal**: It was formed in 2006 from the takeover and merger of Arcelor by Indian owned Mittal Steel. ArcelorMittal is the world's largest steel producer, with an annual crude steel production of 98.1 million tons as of 2014. Lakshmi Mittal (owner of Mittal Steel) is the chairman and CEO. His family owns 40% of the shares and voting shares in the company. By the 21 April 2016 had a share value of 16,616 million Euros, distributed in 3,065,710,869 shares.

10- **International Organization for Standardization (ISO)**: ISO is an independent, non-governmental international organization with a membership of 162 national standards bodies. Through its members, it brings together experts to share knowledge and develop voluntary, consensus-based, market relevant International Standards that support innovation and provide solutions to global challenges.

11- **European Standards (EN)**: European Standards (ENs) are documents that have been ratified by one of the three European Standardization Organizations (ESOs), CEN, CENELEC or ETSI. An EN "carries with it the obligation to be implemented at national level by being given the status of a national standard and by withdrawal of any conflicting national standard".

12- **Japanese Industrial Standards (JIS)**: Japanese Industrial Standards specifies the standards used for industrial activities in Japan. The standardization process is

coordinated by the Japanese Industrial Standards Committee (JISC) and published through the Japanese Standards Association (JSA). Japanese Industrial Standards Committee is composed of many nationwide committees and plays vital role in standardizing activities in Japan.

13- *Generative Structural Analysis (GSA)*: Generative Structural Analysis allows you to rapidly perform first order mechanical analysis for 3D systems. Inside of the GSA workbench there are different possibilities in terms of structural analysis:

- The Generative Part Structural Analysis (GPS): offers the possibility to obtain mechanical behavior information with very few interactions.
- The ELFINI Structural Analysis (EST): Extension of the GPS product, fully based on the V5 architecture. It represents the basis of all future mechanical analysis developments.
- The Generative Assembly Structural Analysis (GAS): Extension of GPS. It enables the study of the mechanical behavior of a whole assembly.
- The Generative Dynamic Analysis (GDY): It allows us to work in a dynamic response context.