

PMI/MBD DATA IN ROBUST DESIGN

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ABSTRACT

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PMI/MBD data in robust design

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The main purpose of this thesis was to check the robustness of a model, in other words, to check the model behaviour depending on the PMI data that the model had. However, to achieve this objective it was necessary to investigate and explain how to include “Product and Manufacturing Information” data in three-dimensional models as well as how to consider these data in “Finite Elements Analysis” in order to get the model results.

The definition of the PMI in three-dimensional models were made by using two tutorials, one for each CAD software used. In this case, Solidworks 2018 and Siemens NX 11.0. On the other hand, ANSYS software was used to simulate the model and configured to take in account PMI data that was included before.

As a result, the software gave the contact pressure behaviour of the model using a response surfaces in which some information stored in the model was considered. Therefore, the model robustness could be check and analysed.

In conclusion, the study shows the importance of considering PMI in any simulation of a model because otherwise the user could miss much information. In the same way, it would be interesting to make a real world study to check the differences between the simulation and the manufactured part.

PMI: Product and Manufacturing Information.

Robustness of a model: A model that can perform in a wide range of conditions.

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GLOSSARY or ABBREVIATIONS AND TERMS (choose one or other)

CAD	Computer-Aided Design.
CMM	Capability Maturity Model
DOE	Design of Experiments
FEA	Finite Element Analysis
GD&T	Geometric Dimension and Tolerance
IGES	Initial Graphics Exchange Specification
ISO	International Organization for Standardization
MBD	Model Based Definition
PMI	Product and Manufacturing Information
STEP	Standard for the Exchange of Product Data
STL	Stereo Lithography
VRML	Virtual Reality Modelling Language
2D	Two dimensions
3D	Three dimensions

1 INTRODUCTION

It is common to see different companies working in their 2D drawings and sharing the files so that the provider can manufacture the pieces drawn in them. However, sometimes the 2D drawings can generate some ambiguousness due to that they work with projections of the 3D model. This fact produces a lot of wasting time trying to discover their real meaning. In order to avoid any confusions in the interpretation and solve this problem, some CAD tools have developed an option in their software in which the user can include all the Product and Manufacturing Information (PMI) annotations in the 3D models (Model Base Definition), without the need to convey them into 2D drawing. This procedure give to the user a more understandable representation of the model as well as no ambiguousness when interpreting its information. However, it is not that common to use this tool. So, to normalize this way to work, this project introduce the user step by step on how the Model Base Definition work in two common 3D-CAD programs such as Solidworks and Siemens NX 11.0. Providing tutorial material showing how to include PMI data in 3D models in each program.

Another real world problem is how to validate robust models. The simulations of Finite Elements programs are for the nominal values of the dimensions of the model, and they do not consider the tolerances set for the manufacturing process, but the fact is that the models manufactured never have all the dimensions in their nominal value. So, the next step in the project is how to include the Product and Manufacturing Information in FEM models so that the program, in this case we have worked with ANSYS, can consider the range of variation of the dimensions to check how the model behaves.

However, simulating the variation of the dimensions in FEA programs is not enough to full check the robustness of a model. There are some other factors in real world that affects in the behaviour of the model such as manufacturing variation, component deterioration or environmental variation during product usage. Finally, the last step in this project was to use a statistic method called Six Sigma that analyses the results and provides the noise parameter need so that the robustness of the model can be checked.

2 THE 3D MODEL

First, before focusing in the main tasks of the thesis it was thought that creating a 3D CAD model was a good idea so that following the whole process with the same model would be easier to understand. It was decided that creating an O-ring would be a good example for thesis purposes because it would provide the chance to go through all the process checking every single issue that it would present.

The reason why the O-ring was chosen is because its geometry is simple and its material is an elastomer, this means that is an elastic material quite sensitive to pressure. So, checking how variations in the O-ring dimensions affect the contact pressure with other parts would be easier than doing it with other harder materials. Moreover, to study the pressure that a model makes means that some other parts must suffer that same pressure in the opposite direction, that's why it was decided to create an assembly in which the O-ring was included.

First of all, the idea of the general O-ring dimensions was that they should be around the following values.

$$\varnothing_{d1} = 45 \text{ mm}$$

$$\varnothing_{d2} = 3 \text{ mm}$$

In accordance with ISO 3601-1 and AS 568 the dimensions and tolerances that fit better with our initial idea were defined. Therefore, the dimensions used are the ones in the *224 Size Code* of the *Trelleborg* catalogue (T.Trelleborg Sealing Solutions...2016):

$$\varnothing_{d1} = 44.04 \pm 0.44 \text{ mm}$$

$$\varnothing_{d2} = 3.53 \pm 0.10 \text{ mm}$$

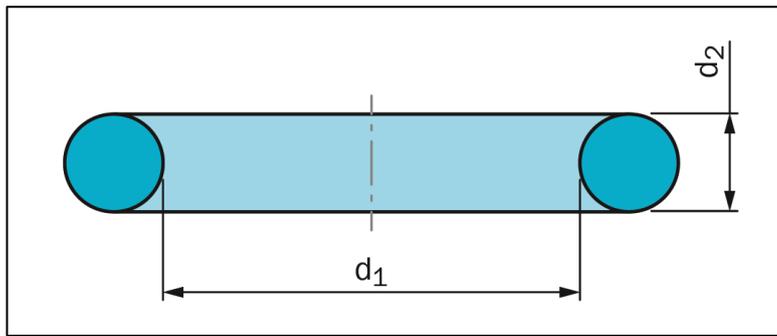


Figure 1. O-ring dimensions – (T.Trelleborg Sealing Solutions...2016, 52)

An important point to create the O-ring was what material use. According to the objective of the project, the material chosen was not so relevant because the goal was to be able to simulate any kind of material. However, some GD&T can change depending on the material. So, it was decided to use the standard one, Acrylonitrile-Butadiene Rubber (Nitrile Rubber), NBR in 70 Shore A.

A standard quality O-Ring has dimensional tolerances to Trelleborg Sealing Solutions standard TBS-00024 in accordance with ISO 3601-1, class B and standard surface quality according to Trelleborg Sealing Solutions standard TBS-00005. Based on ISO 3601-3, grade N. (T.Trelleborg Sealing Solutions. 2016, 52).

One of those geometric tolerances was the flatness. To define the value of the tolerance the rules established by *Trelleborg* catalogue were followed (T.Trelleborg Sealing Solutions...2016).

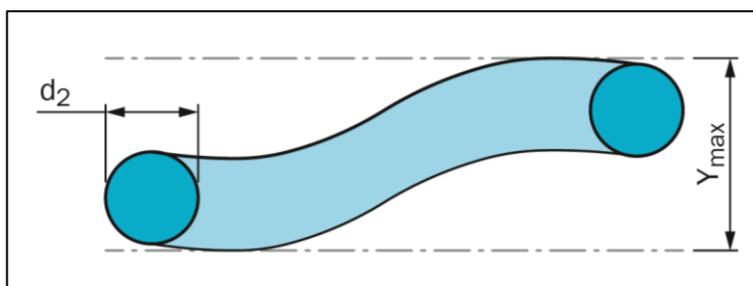


Figure 2. Flatness tolerance Y_{max} - (T.Trelleborg Sealing Solutions...2016, 52)

Taking in consideration the Inner diameter d_1 and the Cross-section d_2 , depending on the ratio of these two dimensions provided, the value of the flatness could vary. The formula used was (T.Trelleborg Sealing Solutions...2016):

$$Y_{max} = 1.5 * d_2 \quad \text{for } 0.11 \leq \frac{d_2}{d_1} \leq 0.21$$

$$Y_{max} = 1.3 * d_2 \quad \text{for } \frac{d_2}{d_1} > 0.21$$

The problem was that the value of the ratio given from the dimensions is not inside the values given in the O-ring catalogue (T.Trelleborg Sealing Solutions...2016).

$$\frac{d_2}{d_1} = \frac{3.53}{44.04} = \mathbf{0.08} < \mathbf{0.11}$$

Although it was not possible to calculate the flatness properly, contacting with *Trelleborg* company, the problem was exposed to try to find a solution for this issue. After some discussions, considering that the cross-section was so small compared to the inside diameter of the O-ring, the O-ring was going to bend and the flatness tolerance was possible for to be quite high. So, it was decided that was safer to go with the flatness tolerance equation according to the ratio of 0.11-0.21 (T.Trelleborg Sealing Solutions...2016).

$$Y_{max} = 1.5 * d_2 = 1.5 * 3,53 = \mathbf{5.295 \text{ mm}}$$

Another important geometric tolerance to consider was the roundness, to calculate its value the following formula provided in the O-ring catalogue was used (T.Trelleborg Sealing Solutions...2016):

$$X_{max} = (1.1 * d_1) + 2 * d_2$$

$$X_{max} = (1.1 * 44.04) + 2 * 3.53$$

$$\mathbf{X_{max} = 55.5 \text{ mm}}$$

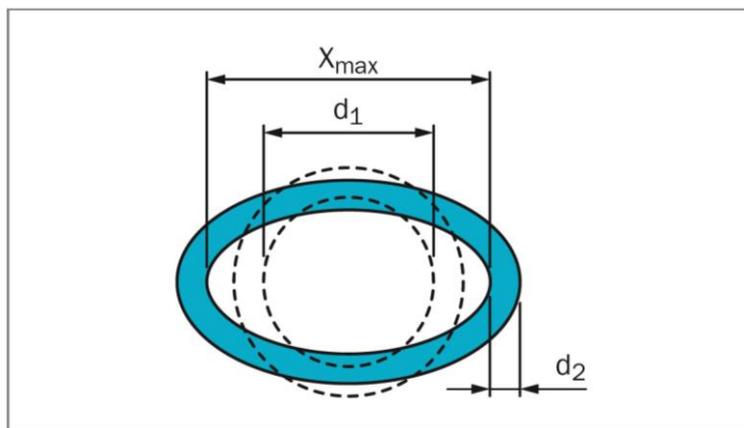


Figure 3. Roundness tolerance X_{max} - (T.Trelleborg Sealing Solutions...2016, 52)

At this point, the O-ring dimensions and tolerances were defined. The next step, as it has said before, was to create an assembly with other parts in order to check how the variation on dimensions of the O-ring effects the contact pressure.

Taking in consideration the O-ring dimensions and tolerances specified before, and using the O-Rings + Housing calculation of the *Trelleborg Sealing Solution*, a lay-out of the assembly was generated automatically as it can be seen in Figure 4. The results given showed all the important housing dimensions with their respective tolerances, Figure 5.

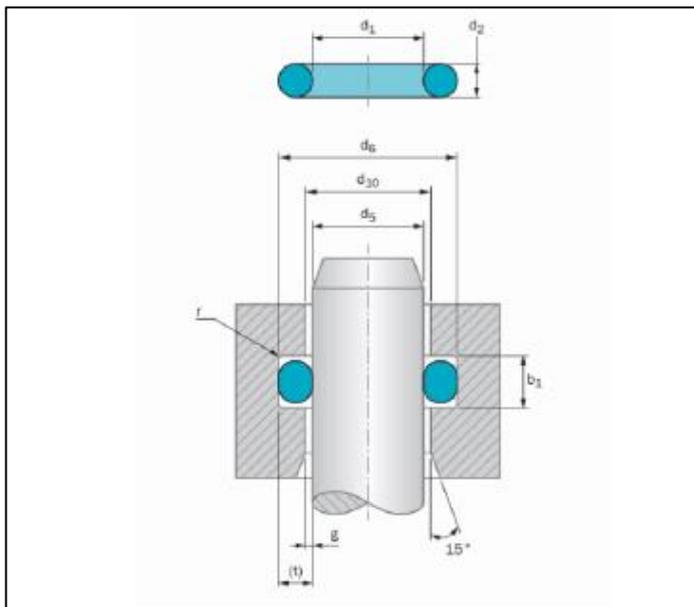


Figure 4. O-ring assembly layout – (Calculation Results Concentric Position...)

Input (mm)	Nominal Size	Standard/Fit	Lower Tolerance Limit (23°C)	Upper Tolerance Limit (23°C)	Min. Dimension (23°C)	Max. Dimension (23°C)
Bore-Ø	d_{10}	H8*	0.000	0.039	44.500	44.539
Rod-Ø	d_5	f7*	-0.050	-0.025	44.450	44.475
Groove-Ø	d_6	H8*	0.000	0.062	49.900	49.962
Groove Width	b_1	recom.	0.000	0.250	5.000	5.250
Groove Radius	r	recom.	-0.100	0.100	0.200	0.400
O-Ring Inside-Ø	d_1	ISO	-0.44	0.44	43.60	44.48
OR Cross-Section-Ø	d_2	ISO	-0.10	0.10	3.43	3.63

Figure 5. O-ring calculation results - (Calculation Results Concentric Position...)

The next step was to create the O-ring housings following the values that the calculation provides. The housings consisted on a Radial Inner Sealing, a group of cylinder-piston in which the cylinder had a groove where the O-ring was placed getting pressure from both bodies, the piston and the cylinder.

3 EXAMPLE CASE

In order to be introduced in one of the main parts of the thesis it is good to define what CAD programs consist on.

“CAD is the use of computers to create 2D and 3D designs. Common types of CAD include two-dimensional layout design and three-dimensional modeling.” (CAD Computer-Aided Design...2014).

One of the main applications of 3D modelling is the product designs, in other words, the 3D model is created using the software so that check how the theoretical final model will be is easy.

At this point, the dimensions defined before were used to create the different parts of the O-ring assembly. In the following pictures, Figure 6 and Figure 7, it can be seen a section of the assembly created using the two CAD programs that were in the scope of the thesis, Solidworks 2018 and Siemens NX 11.0

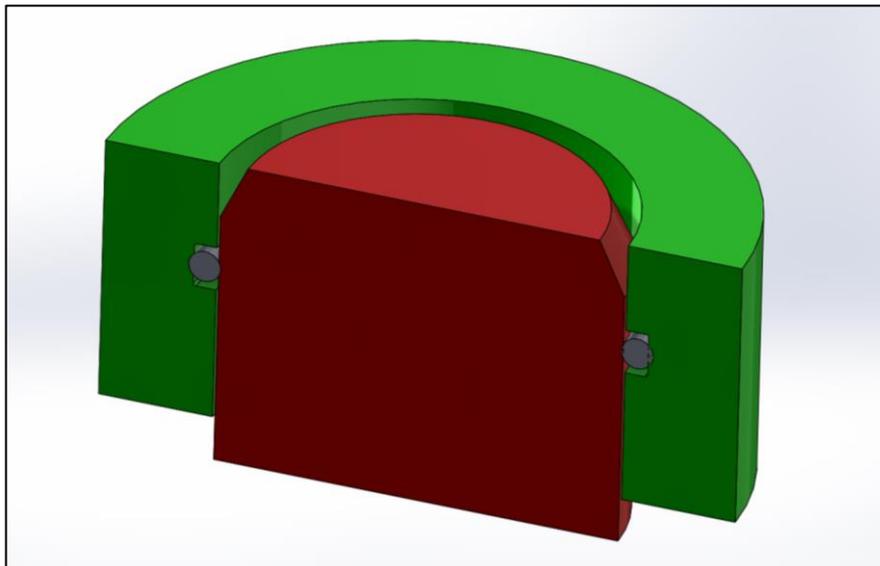


Figure 6. Section of the O-ring assembly – Solidworks 2018

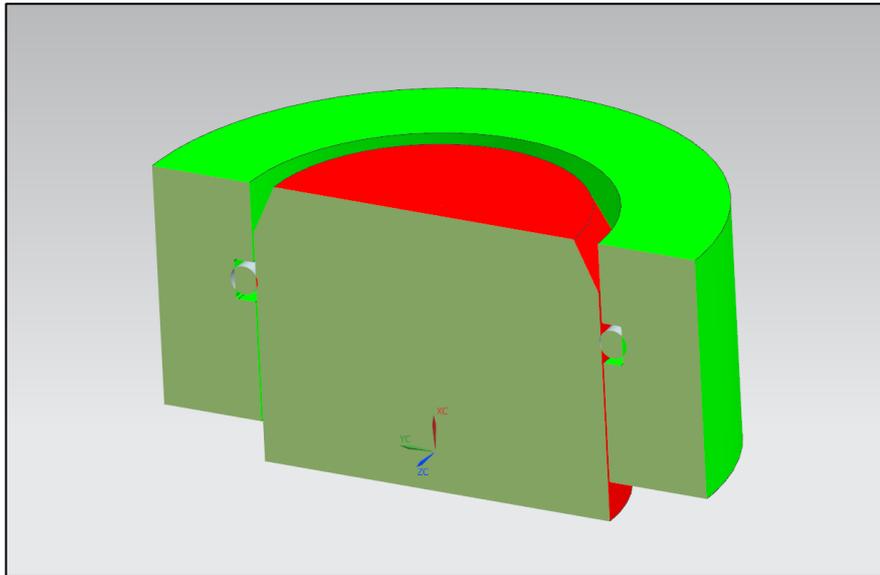


Figure 7. Section of the O-ring assembly – Siemens NX 11.0

Regarding the pictures, it is easy to see that there is an interference between the O-ring and the cylinder. This is because CAD programs cannot simulate the real situation that would happen when the O-ring contacts with other parts. This issue is solved in the Finite Elements analyses step, where checking what changes the O-ring suffered is easier.

On the other hand, it was also important to consider that the manufacturing process is not 100% precise. That is why most of the CAD programs have a tool to include that extra information to let the provider know the range of actuation he has when the model has to be manufactured, in other words, Product and Manufacturing Information (PMI).

Product Manufacturing Information (PMI) contains dimensional, tolerance, datum, as well as GD&T information. This information has been in the 2D drawings from long time ago. However, it is not that common to include PMI in 3D model. So, the purpose of the next step was to get introduced at PMI data and to define some guidelines that explain how to define and consider this information in 3D models.

3.1 PMI/MBD data in 3D-CAD

As it is said before, PMI data is very important in every manufacturing process, to include the PMI data in the same 3D model give the possibility to full understand the part, with no ambiguousness and no confusions. This way to work is so helpful while setting up the manufacturing process, the MBD helps to speed up this process, avoiding the need of 2D drawings.

On the other hand, MBD is also used to assess the quality of the manufactured models due to it is much more effective getting CMM results than the CAD models, what means reducing the manufacturing costs and more productivity.

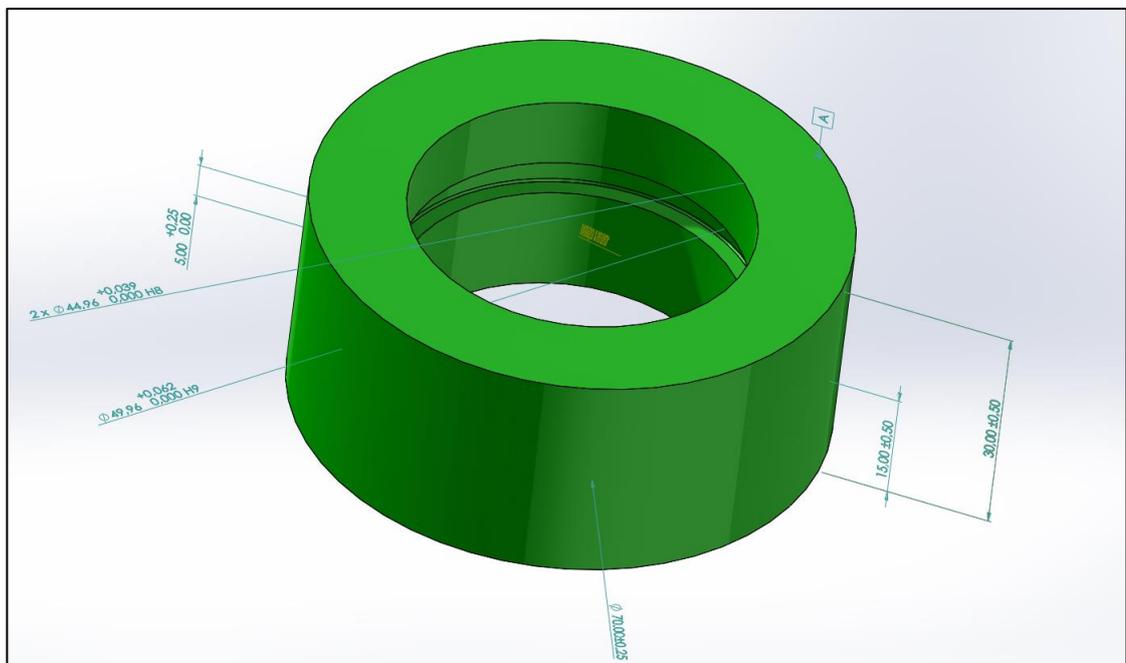


Figure 8. PMI data included in a 3D model – Solidworks

In Figure 8, there is a 3D model of the cylinder in which the PMI data was included. In order to explain the whole procedures for setting up all the different annotations that PMI has, two tutorials were created, one for each program used.

In Solidworks tutorial, it can be found how to configure the software so that default GD&T annotations will be placed automatically in the model interface without the need to modify them. However, the rest of annotations that PMI include must be set up separately, each one with its settings and procedure. Although this is the fastest way to include

PMI data, there is another possibility to include them in the 3D model. This second procedure avoids to previously configure the software with default GD&T, but the user has to define all the annotations separately, which means more time. So, to know the full procedure step by step, it can be found in Appendix 1.

On the other hand, the Siemens NX 11.0 tutorial has a different interface with some other processes but in a general idea, it is very similar to the second option of the Solidworks tutorial, the user has to define one by one all the dimensions, tolerances, and other annotations need to include them in the 3D model. So, to know more about how to include PMI data in Siemens NX 11.0 3D model it can be check in Appendix 2.

3.2 PMI/MBD data in CAD-neutral files

What makes the neutral files useful is the possibility to share these models between customers and suppliers, even though they do not use the same CAD tool. Actually, this is the main purpose of neutral files, import CAD data from any CAD software, and store it under a standard so that it could be exported to any other CAD tool. Figure 9 shows a representative scheme of neutral file utility (A closer look at CAD... 2014).

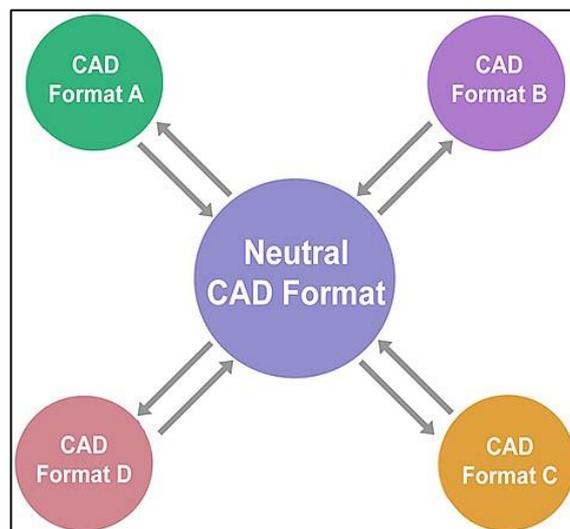


Figure 9. Scheme of how neutral files work - (A closer look at CAD... 2014)

Nowadays, there are many neutral files such as Parasolid, STL, and VRML but the two most common neutral files used are IGES and STEP. The fact is that IGES is good for representing 2D CAD drawings and 3D surface models, but not for 3D solid models.

STEP does a much better job with representing 3D solid models (A closer look at CAD... 2014). Considering the objectives of the Thesis, it was decided to work with STEP format to export the O-ring assembly model. In the next point, it is explained more concretely what is STEP about.

3.2.1 STEP

In the eighties, the International Standards Organization decided to create ISO 10303 also known as STEP in order to improve what IGES standard was. The purpose was to create a global standard to relate most of all the different CAD data. Due to its complexity, it took some years to develop the standard. Actually, this standard is constantly upgrading. That is why it has some different versions with different functionalities.

Some STEP standards such as AP203 and AP214 are very common in 3D mechanical design, AP214 is the improved version of AP203 that store the geometry, topology and configuration management data of solid models for mechanical parts and assemblies as well as colours, layers, geometric dimensioning and tolerance, and design intent. However, in December 2014 a new standard was developed including some new functionalities, STEP AP242 (Why AP242? – ePLM Interoperability). The major technical impact of the new standard covers the following areas:

- Model Based Development (MBD),
- PDM integration and PDM services,
- Long Term Archiving (LTA),
- supply chain integration,
- engineering design data exchange including composites,
- advanced Product Manufacturing Information (PMI),
- Mechatronics and requirement management...

Considering the scope of the thesis, it is important to explain how to export the 3D-CAD model from each CAD tool to a STEP format. The procedure can be checked in (Appendix 3).

3.3 FEM models

Finite Elements Method is a numerical method to solve engineering and physics problems. Mostly used by the mechanical, aerospace, civil, and automotive industry, the Finite Elements Analysis programs provide the structural and stress analysis of the geometry studied. It is very useful for complicated geometries, loadings, and material properties where analytical solution is impossible to obtain. FEA programs work by discretization, it divides the domain into small and simple parts called finite elements which everyone has their own simple algebraic equation, and then the program groups all the finite elements equations into a bigger system that solves the problem.

One of the thesis objectives was to check the robustness of a model, what FEA programs provide is the possibility to analyse the response of the model depending on the variation of its dimensions. The program used for this purpose was ANSYS 19.1, in which following some steps it comes up with interesting results.

3.3.1 PMI/MBD data in ANSYS

In order to consider PMI data in the FEM model that ANSYS generates, the user has to define the important dimensions as parameters itself because the software cannot include the PMI data from the model imported automatically. These parameters have their own tolerances, which were considered later in the DOE study for robust design purposes. So, after simulating different values inside the tolerances range of the parameters, the software came up with a response surface that showed the behaviour of the model and gave the first information about its robustness.

3.3.2 ANSYS procedure

To get the results that ANSYS provides and check the behaviour of the model, it was important to first set up the model with some different conditions and constraints such as the materials, the contacts between parts, meshing, and some other points that are explained in the following stages.

First of all, the ANSYS Workbench 19.1 was opened and the common workbench interface appeared. **Statistic Structural** inside the **Analysis Systems** was chosen because it is the usual procedure when working with solid parts. Immediately, the different steps to follow appeared as you can see in Figure 10. In the following pages, it is explained what decisions were taken in every step of the process and their reasons.

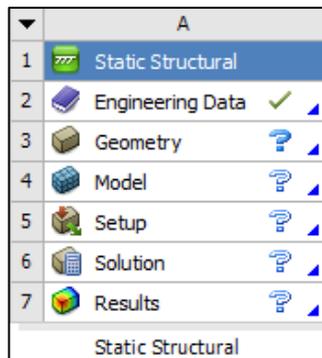


Figure 10. Set up steps - Ansys Workbench 19.1

- **Engineering data**

This point consisted in defining what materials were going to be used so that they could be assigned later to each part of the assembly. In this case, the materials chosen were **Rubber 1** and **Structural Steel**.

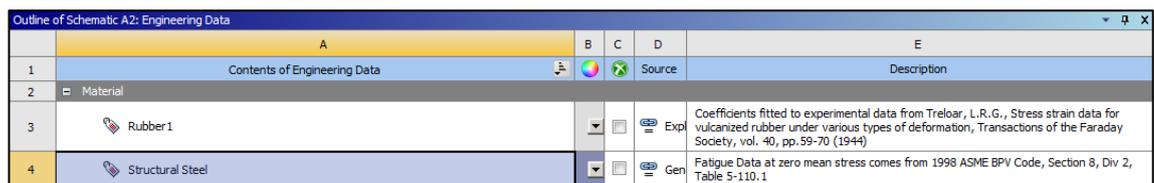


Figure 11. Engineering Data – Workbench 19.1

These materials were chosen because they are two of some default materials that were stored in the software, so it was not need to modify any property. Although the results of the simulation depend on the material properties, the real goal of this study was to check that is possible to get information about the robust design independently of the material used. That is why the defaults ones were chosen.

- **Geometry**

Considering that the geometry was axisymmetric it was decided to just use a 2D plane surface that represented the behaviour that every point of the model would have. At this point, before open the **SpaceClaim Geometry** interface it was need to switch the geometry properties from 3D analysis type to 2D one.

Afterwards, to import the geometry to the software, the geometry interface had to be opened; this new interface is called SpaceClaim. When the user is inside it, the STEP file is imported and the 2D design is shown in the screen like in Figure 12. When the model is imported, the software saves automatically the design geometry so that the user can go for the next steps.

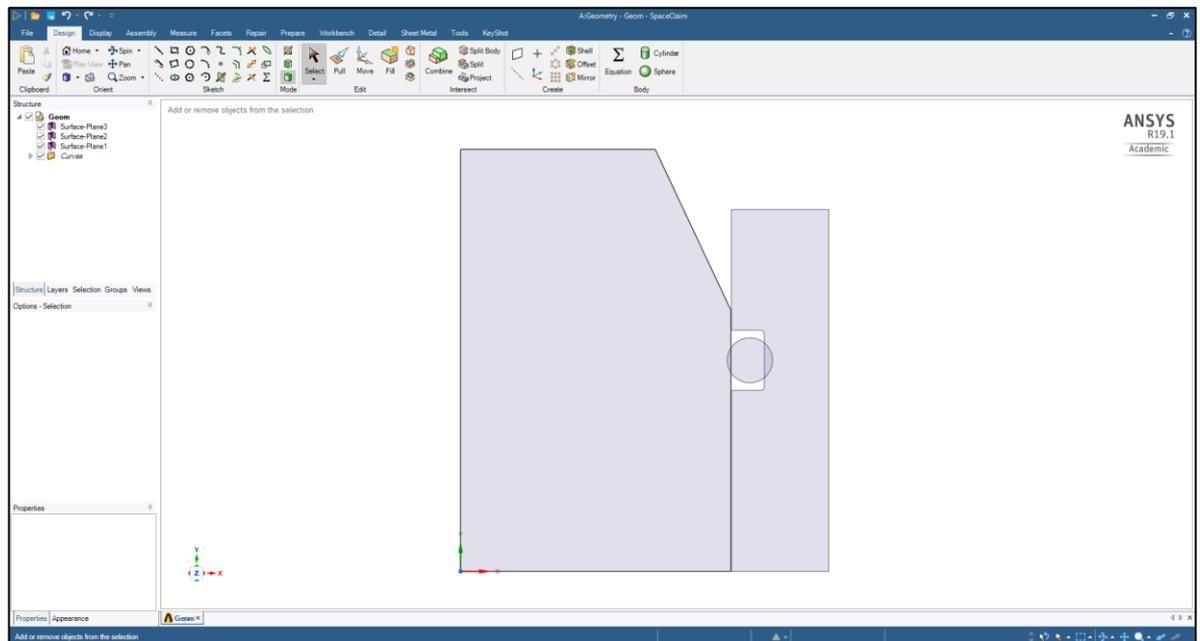


Figure 12. 2D O-ring assembly design – SpaceClaim

- **Model**

With the geometry imported the model can begin to be configured, defining some conditions and constraints to make the simulation as much real as it can be. In order to define the O-ring model the **Mechanical** interface was opened. The first thing was to select **Geometry** in the left dropdown and go to Details of “Geometry” to switch the 2D behaviour to **Axisymmetric** to be coherent with the geometry uploaded.

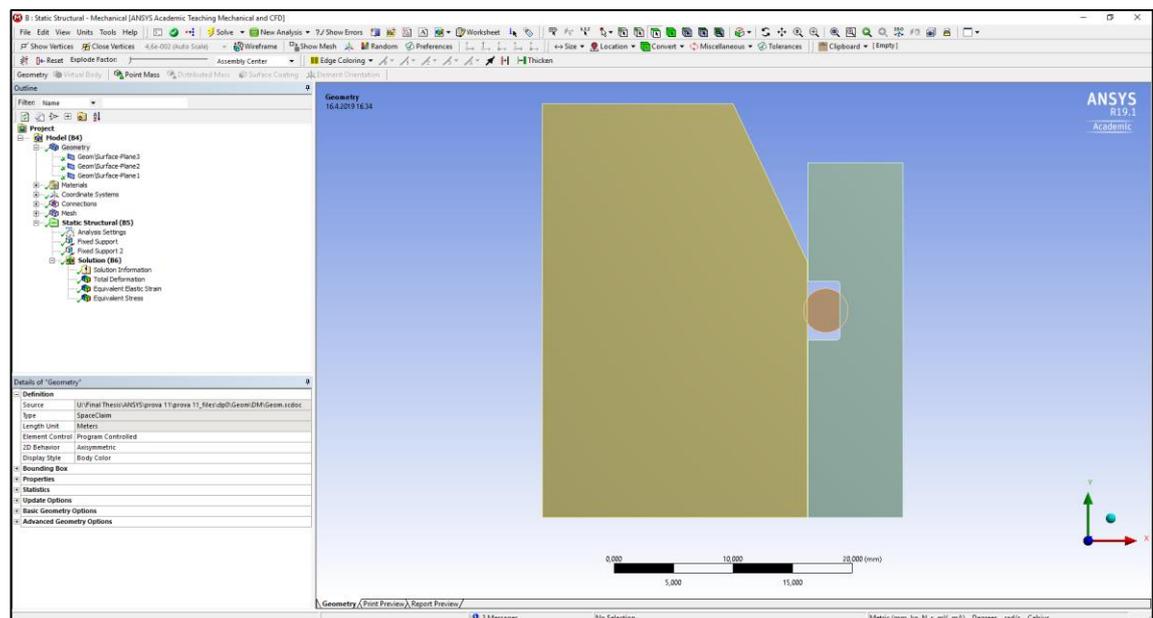


Figure 13. Mechanical interface – Workbench 19.1

Afterwards, in the geometry dropdown, it was defined what material belong to each part, and this was the assignation:

- O-ring: Rubber 1
- Cylinder: Structural Steel
- Piston: Structural Steel

In the left dropdown, the **Materials** icon was selected to edit some properties and modify the material characteristics, but as it was said before the default material were chosen so that they would not have to be modified. On the other side, the **Coordinate Systems** is to check where the origin of reference was. In this case, it was important to check that all

the geometries were on the right side of the Y-axis because otherwise the software would not simulate the results properly.

- **Connections**

This step is one of the most important ones, all the contacts between the different geometries were set up at this point and a big percentage of the final result depends on how these contacts were. So, the **Connections** dropdown was opened, the same with the **Contacts** one and **Contact Region** was selected. Then, in the bottom left side of the screen, the **Details of “Contact Region”** were defined.

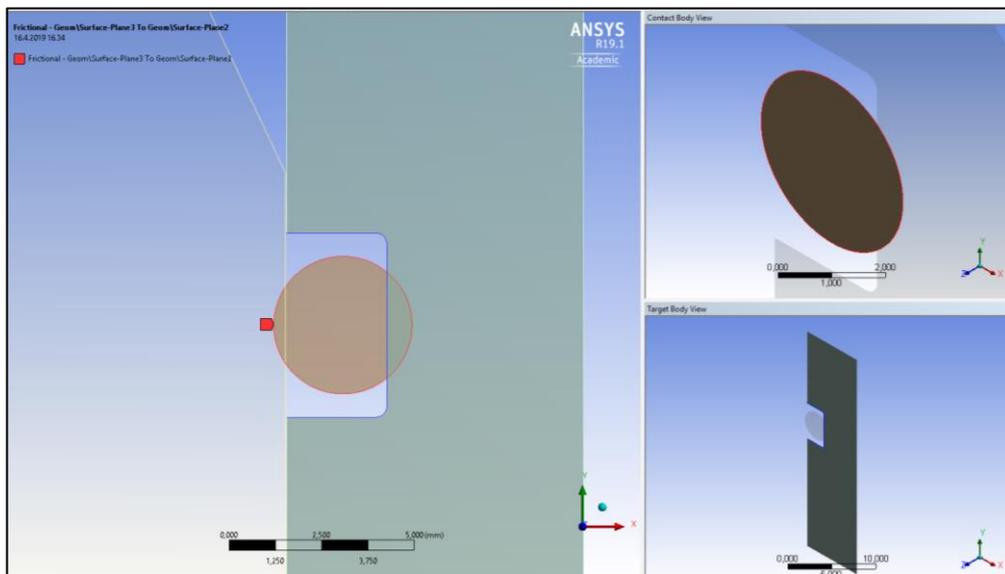


Figure 14. Contact Region 1 set up – Mechanical interface

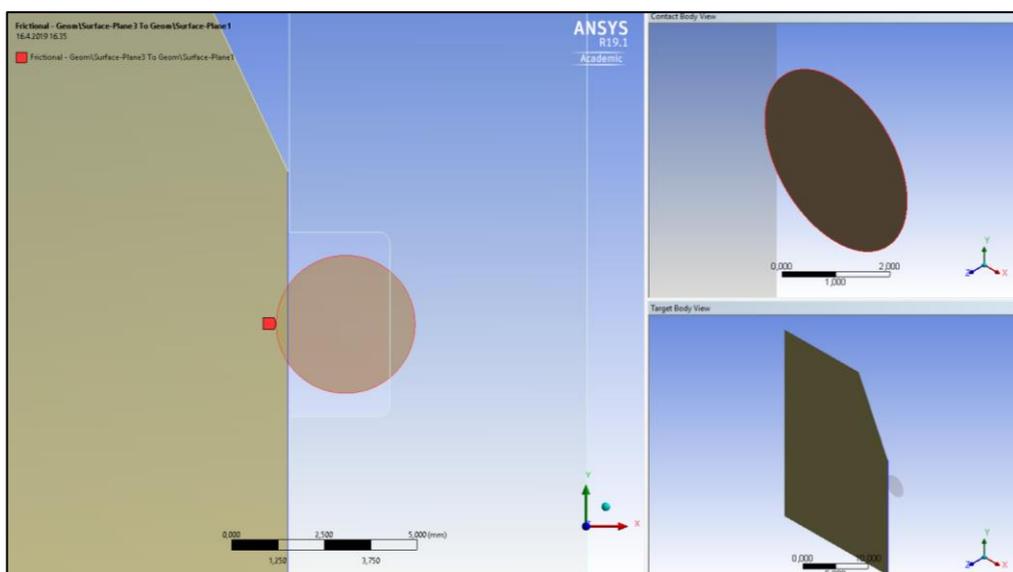


Figure 15. Contact Region 2 set up – Mechanical interface

The contact in Figure 14 was a contact between the O-ring and the cylinder that the same software detected, so it was need to modify some points in the **Details of “Contact Region”**. On the other hand, the contact in Figure 15 is the O-ring contacting the piston and it was need to modify some properties of the contact as well. The topics modified can be checked in Appendix 4.

- **Mesh**

In order to optimize the software running time and considering the quality of the results, some different meshes were created for each geometry depending on what information was of our interest. The goal of this simulation was to study the O-ring behaviour. That is why the Element Size of the O-ring was smaller than the piston and cylinder ones. The element size of the different geometries can be checked in Appendix 4.

To customize the O-ring mesh it was need to right click the **Mesh** icon in the left side screen dropdown and select **Insert** and **Sizing**, then one of the O-ring surfaces is selected and its element size defined. In Figure 16 you can see the different sizes of the meshes.

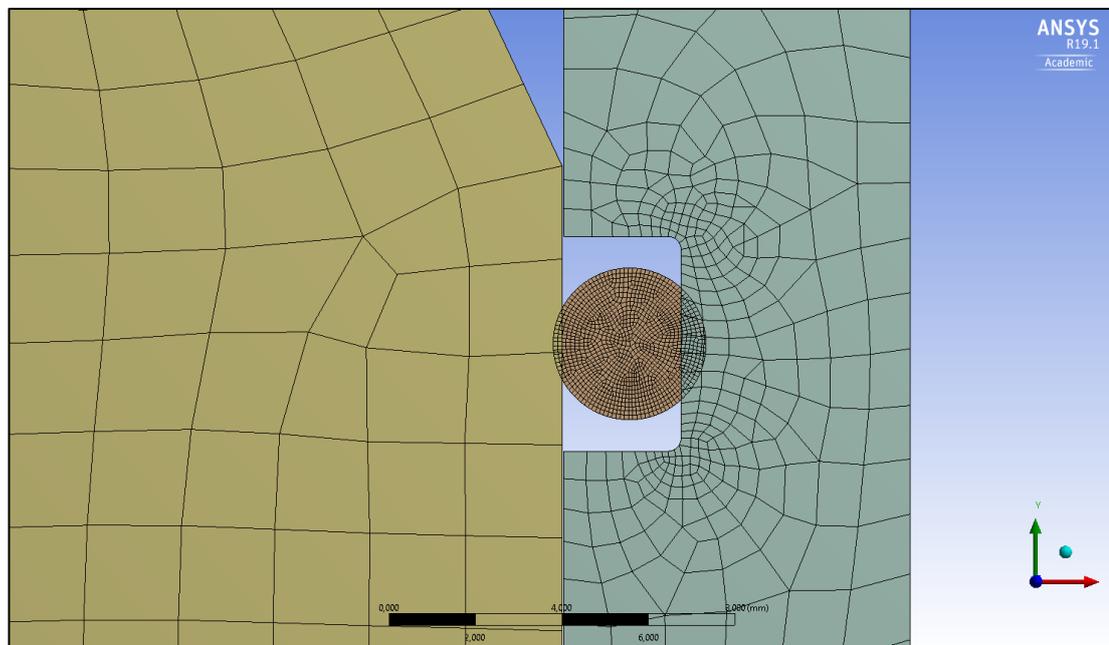


Figure 16. O-ring geometry meshed – Mechanical interface

- **Static Structural**

The next step was to set up some conditions such as fixing the different parts. So, a **Fix Supports** had to be included, one of the cylinder edges was selected and the constrain was applied, then the same had to be done with the piston geometry. Afterwards, it was need to modify some points of the **Analysis Settings**, those modifications can be checked in Appendix 4.

- **Solution**

The final step before get with the first results was to decide what kind of results the software had to calculate. So, considering that the goal was to get the pressure values, two contact tools were inserted in the Solution simulation in which the contact pressure of each contact existent could be checked. In Figure 17 and Figure 18 the simulation results are shown.

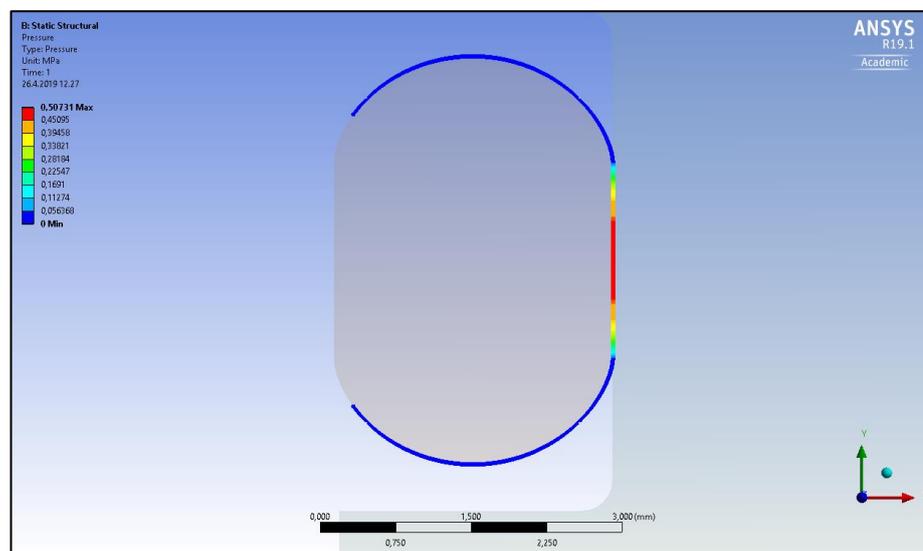


Figure 17. Contact pressure of the O-ring and the cylinder – Mechanical interface

The results said that the maximum pressure that the O-ring suffered was **0.50731 MPa**. Regarding the colour scale in the left side of the screen, it was easy to see that the zone of the O-ring that suffered more pressure was the red one.

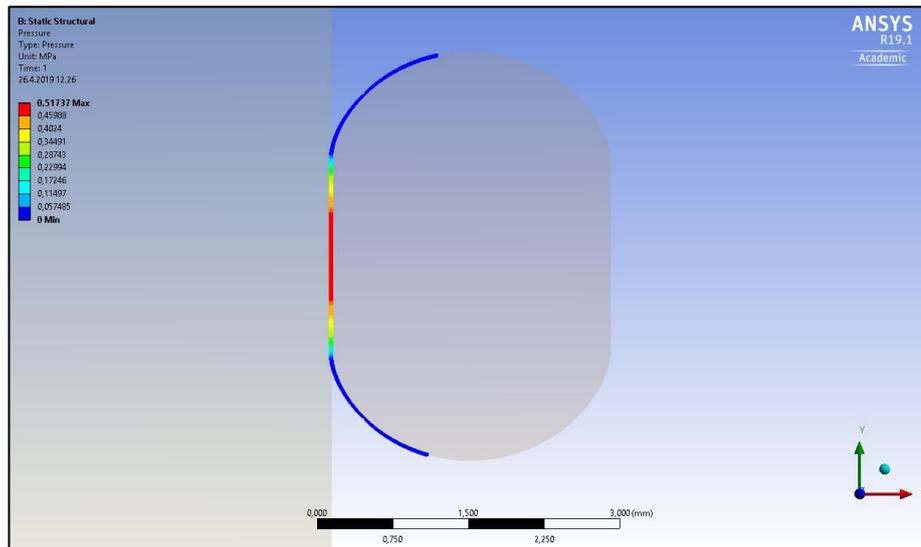


Figure 18. Contact pressure of the O-ring and the piston – Mechanical interface

In this case, the maximum pressure calculated that the O-ring suffered is **0.51737 MPa**, following the same general checking procedure as before it was checked that the O-ring zone where the O-ring suffers the biggest pressure was in the middle of the contact.

This simulation was made with the nominal values of the O-ring model. So, these values are for one particular situation that most of the times does not happen. In order to consider the whole range of tolerances that the O-ring parameters had and check how it performs in any case inside that range, it was need to set up a Design of Experiments study (DOE).

3.4 DOE study

Before getting into this stage of the thesis, it is need to clarify what is a robust design. A robust design is a model that can perform in a wide range of conditions, it has longer life and it keeps consistent in front of external conditions change such as the temperature and some others conditions. The purpose of this study was to provide the first information about the robustness of the O-ring model. This study consist on define the parameters that the solution depends on, and the software generates the theoretical values of the response. To make this study properly it should take many different values of the parameters. However, in this case only three values were taken, the maximum, nominal and minimum tolerances of each parameter because the study was going to be made by the same ANSYS software, so it was important to optimize the simulation running time, without forgetting the quality of the simulation.

Therefore, the next stage explains how to set up the Design of Experiments tool in order to get the final response surface.

3.4.1 Design parameters

The design parameters of this model are called input parameters in ANSYS software. Normally, these parameters represent the important dimensions and tolerances that the behaviour of the model depended on. In the O-ring case the parameters were:

- P1: O-ring cross-section

$$3.53 \pm 0.10 \text{ mm}$$

- P2: O-ring inner diameter

$$44.04 \pm 0.44 \text{ mm}$$

Firstly, to define the values of the parameters in ANSYS the user has to go back to the geometry interface, the SpaceClaim. At this point is important to define the parameters with the **Ruler** tool in the left side of the screen, otherwise the program could take wrong references to define the parameters and it could generate errors in the simulation. As you can see in Figure 19 and 20 the nominal values of the parameters defined were the radius instead of the diameter. Nevertheless, this was not a problem because the only thing that it had to be made was to take care with the dimensions and tolerances values defined.

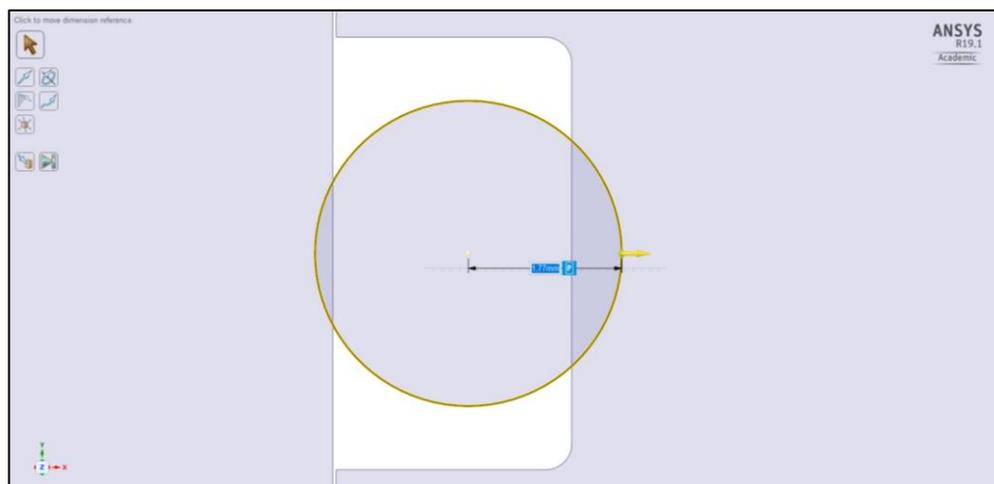


Figure 19. Creating cross-section parameters – SpaceClaim interface

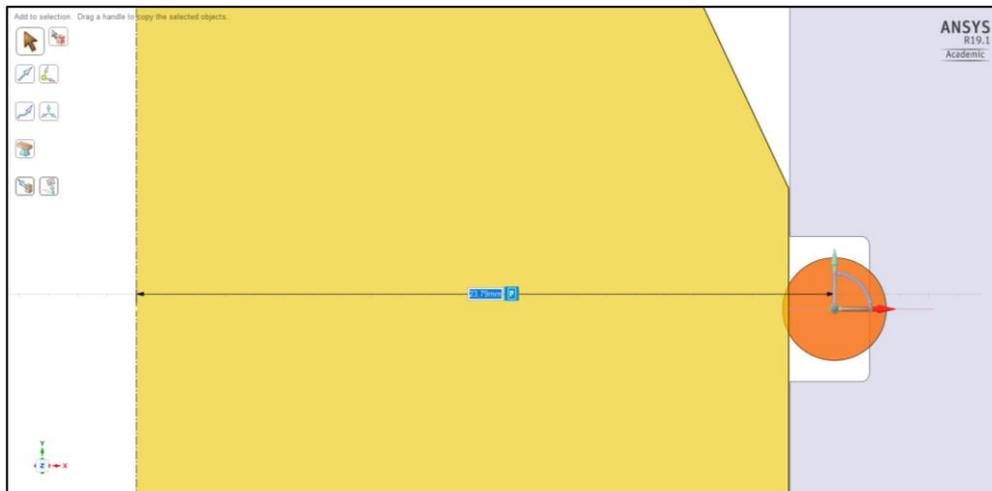


Figure 20. Creating inner diameter parameters - SpaiiceClaim interface

In conclusion, the parameters were set up with the following values:

- P1: O-ring cross-section, with the ruler the reference in the middle of the O-ring was taken so the values to set up the simulation were:

$$1.765 \pm 0.05 \text{ mm}$$

- P2: O-ring inner diameter, the software got the reference on the middle axis of the whole assembly, or what is the same, the radius:

$$23.785 \pm 0.22 \text{ mm}$$

Secondly, it was necessary to decide which output parameters were needed from the simulation. Taking into account the results wanted, the most coherent parameter to define as output was the Pressure Maximum of both sides of the O-ring model.

On the other side, before updating the Design of Experiments it was important to configure the properties like are shown in Figure 21. It was an important step because it is the way to order the program to calculate all the random points in between the upper and lower values of the different parameters. Otherwise, the simulation could give some wrong information.

Design of Experiments	
Design of Experiments Type	Central Composite Design
Design Type	Face-Centered
Template Type	Standard

Figure 21. Properties – Design of Experiments tool

When all the properties and settings were defined, the DOE tool generated a combination of design points (maximum tolerance, nominal, minimum tolerance) of both parameters between each other. It generated 9 different combinations (3^2), 2 parameters with 3 values of each one. Then, the file needed to be updated and the software started running the simulation. In Figure 22 the design points with the **Pressure Maximum** values calculated can be checked.

Table of Outline A2: Design Points of Design of Experiments					
	A	B	C	D	E
1	Name	P7 - cross-section (mm)	P9 - iner diameter (mm)	P10 - Pressure Maximum (MPa)	P11 - Pressure Maximum (MPa)
2	1	1,765	23,785	0,51696	0,50709
3	2	1,715	23,785	0,46415	0,45612
4	3	1,815	23,785	0,56838	0,55834
5	4	1,765	23,565	0,5075	0,50598
6	5	1,765	24,005	0,52367	0,52794
7	6	1,715	23,565	0,4558	0,45059
8	7	1,815	23,565	0,55589	0,55716
9	8	1,715	24,005	0,47124	0,46725
10	9	1,815	24,005	0,58171	0,58242

Figure 22. Design Points – Design of Experiments tool

3.4.2 Surface response

At this point, when the response values were calculated, the **Response Surface** tool was opened in order to graph them. Previously, before getting the surface, some settings needed to be modified such as defining what type of response surface is required, in this case the type chosen was a “Full second order polynomials”. Moreover, activating the verification with three different points was needed as well; these points are randomly defined by the software to check if the response surface shape fit well with all the points. Afterwards, when all the setting were applied the simulations starts to run again in order to get the verification points results, and the graphs were automatically generated. Finally, the software shows the results, which can be studied using the different tools that ANSYS

has. In the next stage is explained what tools the software has in order to analyse the results and get the firsts conclusions.

The first type of result is the “Response”, there are different output graphs such as 2D, 3D and 2D Slices, depending on what are you thinking to study some of them will be more useful than the others. In the O-ring case, two of them were used to understand the behaviour, the 3D and the 2D graphs.

In order to have a general idea of how the contact pressure behave, the most useful representation is the 3D graph. These graphs in Figure 23 and Figure 24 represent the contact pressure between the O-ring and the piston, in the first case, and the O-ring contacting the cylinder in the second case. They show how the maximum pressure (Z-axis) depends on the both design parameters, the inner diameter (Y-axis) and the cross-section (X-axis). In conclusion, regarding the graphs it is easy to see that the maximum pressure that the O-ring suffers has variations inside the parameters tolerances, in addition it is also seen that the parameters do not have the same effect to the final response, the cross-section has a bigger weight to the contact pressure that the inner diameter.

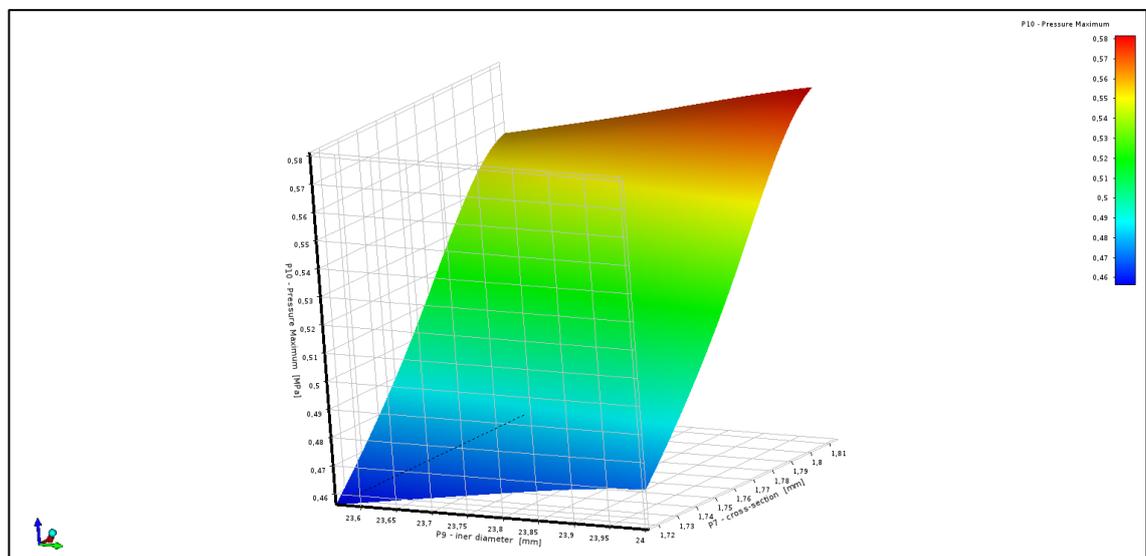


Figure 23. O-ring – Piston maximum pressure – 3D Response surface

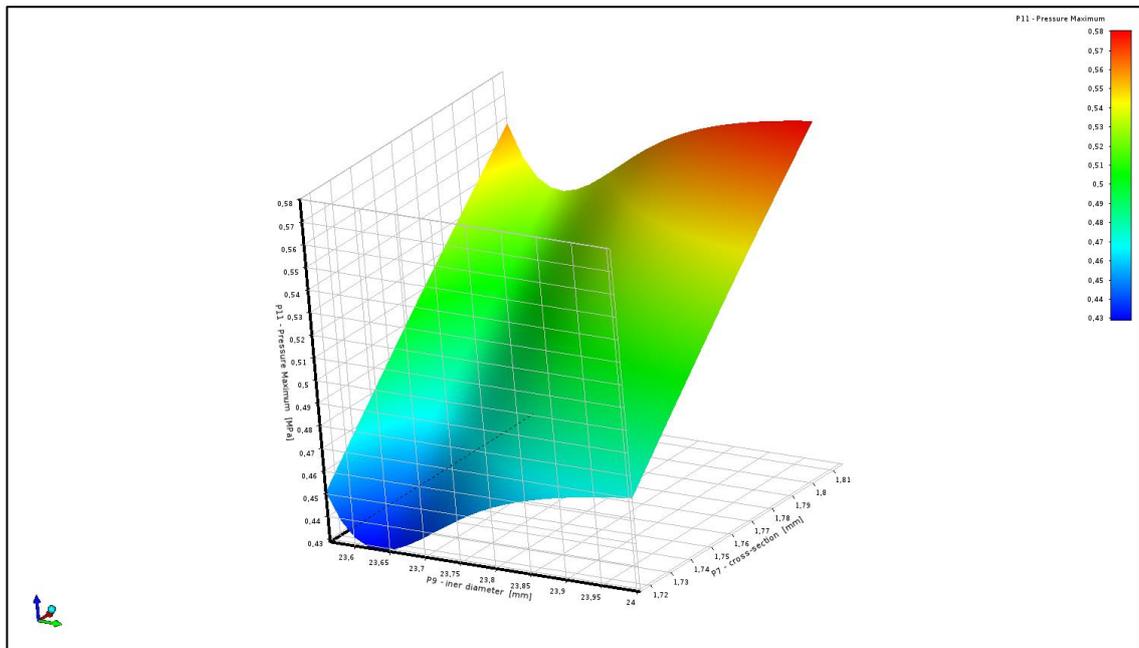


Figure 24. O-ring – Cylinder maximum pressure – 3D Response surface

Another important issue that the simulation gave is the chance to compare the two different contacts that the O-ring has in order to check their values and see the different effects that they have. These following values are the most significant ones of the contact pressure simulation.

- Contact 1: O-ring contacting the piston

$$P_{max} = 0.58171 \text{ MPa}$$

$$P_{min} = 0.4558 \text{ MPa}$$

- Contact 2: O-ring contacting the cylinder

$$P_{max} = 0.58242 \text{ MPa}$$

$$P_{min} = 0.45059 \text{ MPa}$$

As you can see, there is not a big difference between the contact pressure values of the two different contacts. In conclusion, considering how the contacts are and regarding this behaviour, it is confirmed that the symmetric style of the parameters tolerances make sense with the results.

Apart from these results, if a general idea is not enough for the purposes that the user can has, the option is the 2D graphs, where you can see the sensitivity of the response in front of the parameters separately.

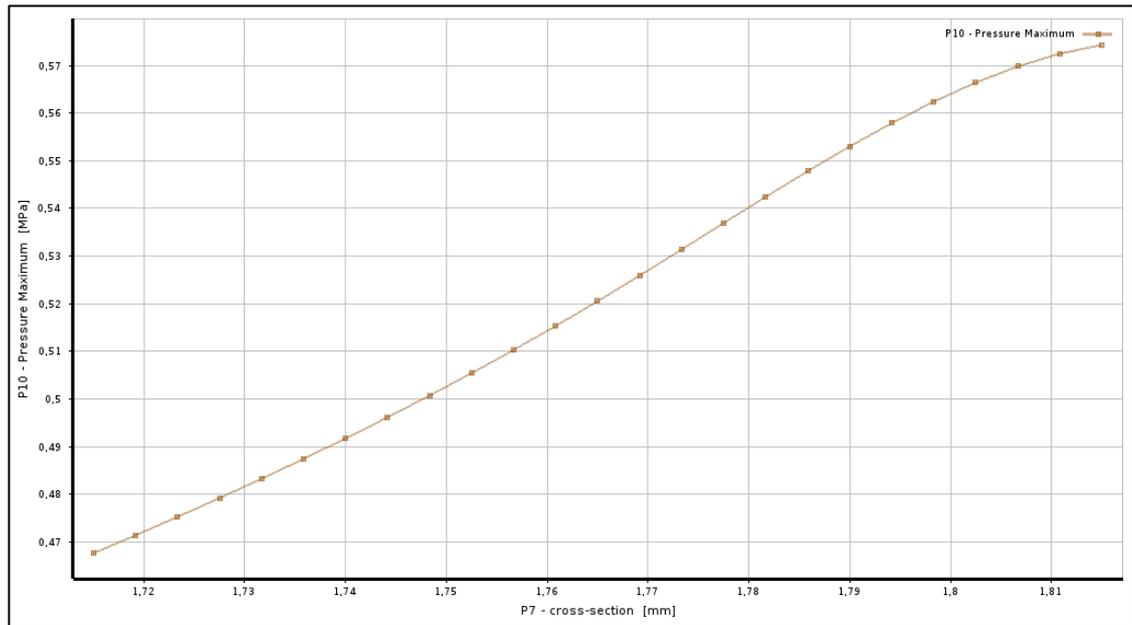


Figure 25. O-ring – Piston maximum pressure / cross-section – 2D response

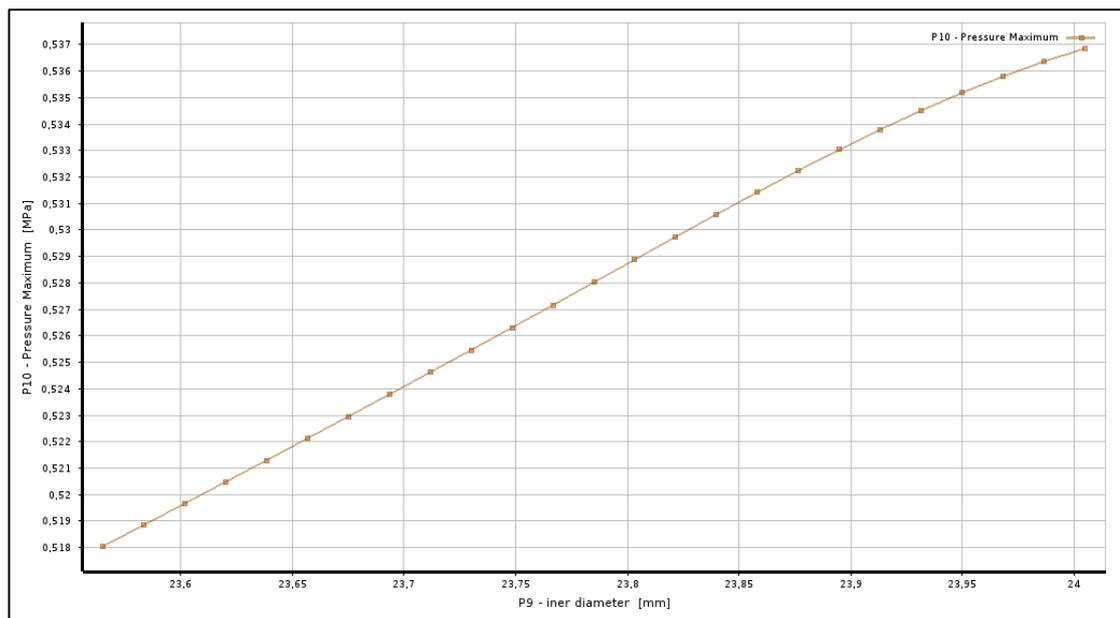


Figure 26. O-ring – Piston maximum pressure / inner diameter – 2D response

Using these graphs is easy to understand how the pressure vary depending on the parameters tolerances. As a result, these variations confirm that simulate only the nominal values has some risks because a lot of information is missing.

On the other hand, the user can also check the local sensitivity, which provides the percentage of weight that the parameters have depending on their exact values. In the bottom left side of the screen, there is a place in which the user can define the exact values of the

parameters, then the software give the exact contact pressure values of both contacts, and the red and blue bars of Figure 27 change depending on them.

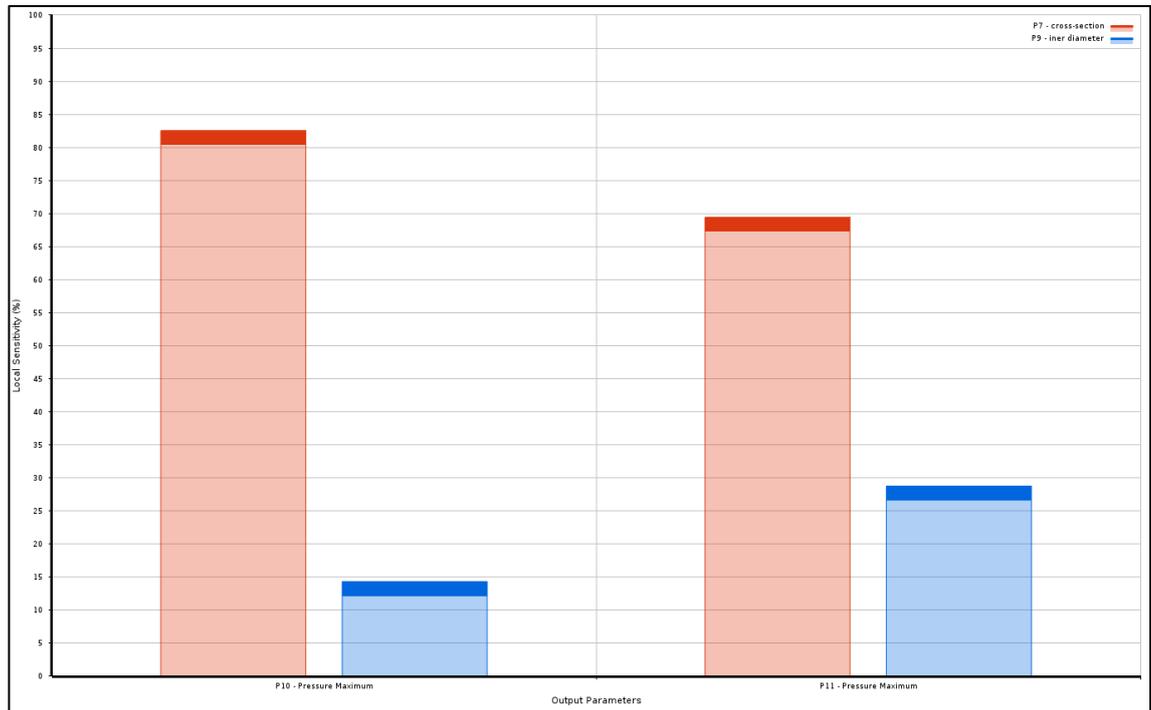


Figure 27. Sensitivity of the different parameters - Local Sensitivity

In this case, the left side of the graph represents the sensitivity of the parameters when the O-ring is contacting the piston, and the other side is the contact between the O-ring and the cylinder. Moreover, the red bars represent the cross-section and the blue one the inner diameter in each contact.

Apart from all these tools, the user has more options to analyse the results such as “Local Sensitivity Curves” and “Spider” graphs. However, the O-ring model because did not need to be studied that deep because the objective was to check the effects of the tolerances and it is already done.

3.5 Six Sigma Analysis

Finally, the last analysis that this procedure need was a very common statistical study called “Six Sigma”. The definition of this method is the following.

Six sigma is an organized and systematic problem-solving method for strategic system improvement and new product and service development that relies on statistical methods and the scientific method to make dramatic reductions in customer defined defect rates and/or improvements in key output variables. (Theodore T.Allen, Introduction to Engineering Statistics... 2007, 27)

Theoretically, this method is thought to be applied when the models are manufactured and their dimensions are checked. Although in this case, the models were not manufactured, it was been working under the hypothesis that our manufacture process has a reliability that belongs to the six sigma process, what is the same, a reliability of 99.73% like it can be seen in the normal distribution bell of Figure 28 (Six Sigma DMAIC Process). This study gives the noise parameter that the doe study could not consider, so after this stage the full robustness could be checked.

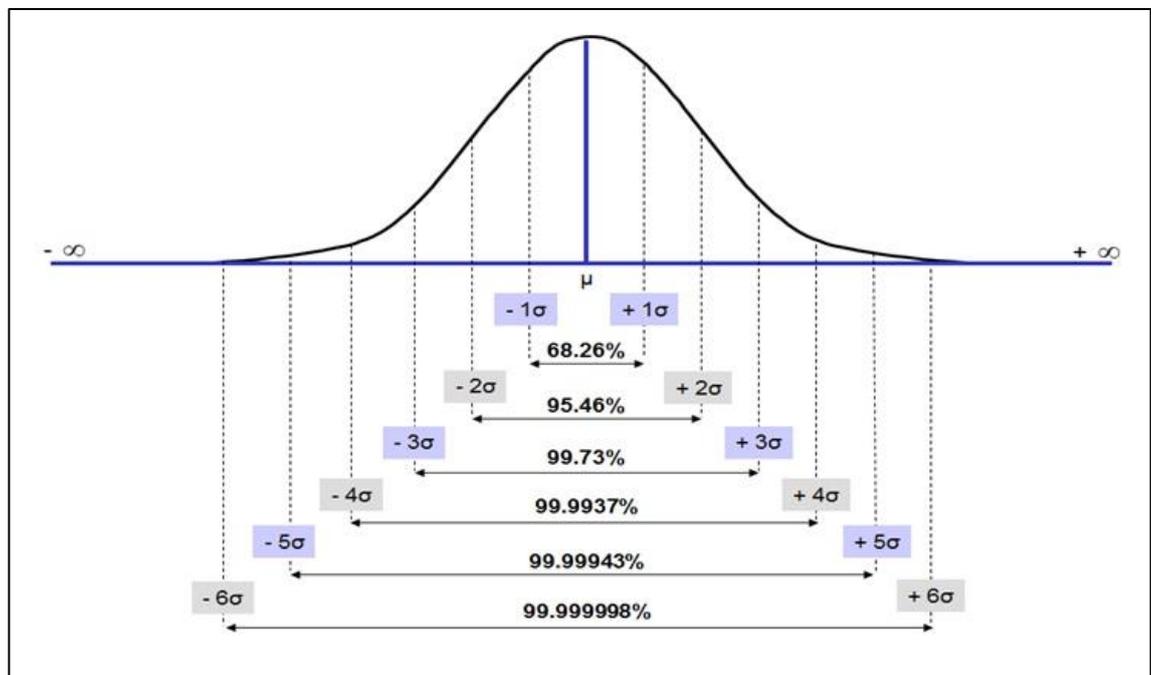


Figure 28. Six sigma – normal distribution – (Six Sigma DMAIC Process)

This statistical method was applied for each parameter of our model, in the next stages is explained how to make the software run in order to calculate the normal distributions of the cross-section as well as the inner diameter.

First, to set up the tool so that it can calculate it is important to define the mean value (m) of the parameter as well as the standard deviation (σ). So, working under the six sigma hypothesis the values are:

- **Cross-section distribution**

$$m_c = 1.765 \text{ mm}$$

$$Z = \frac{HL - m_c}{\sigma_c} = 3 \rightarrow \sigma_c = \frac{HL - m_c}{3} = \frac{1.815 - 1.765}{3} \rightarrow \sigma_c = 0.0167 \text{ mm}$$

- **Inner diameter distribution**

$$m_i = 23.785 \text{ mm}$$

$$Z = \frac{HL - m_i}{\sigma_i} = 3 \rightarrow \sigma_i = \frac{HL - m_i}{3} = \frac{1.815 - 1.765}{3} \rightarrow \sigma_i = 0.0733 \text{ mm}$$

Afterwards, when the normal distribution settings are ready the program starts running and it gave the distributions that can be seen in the Figure 29 and Figure 30.

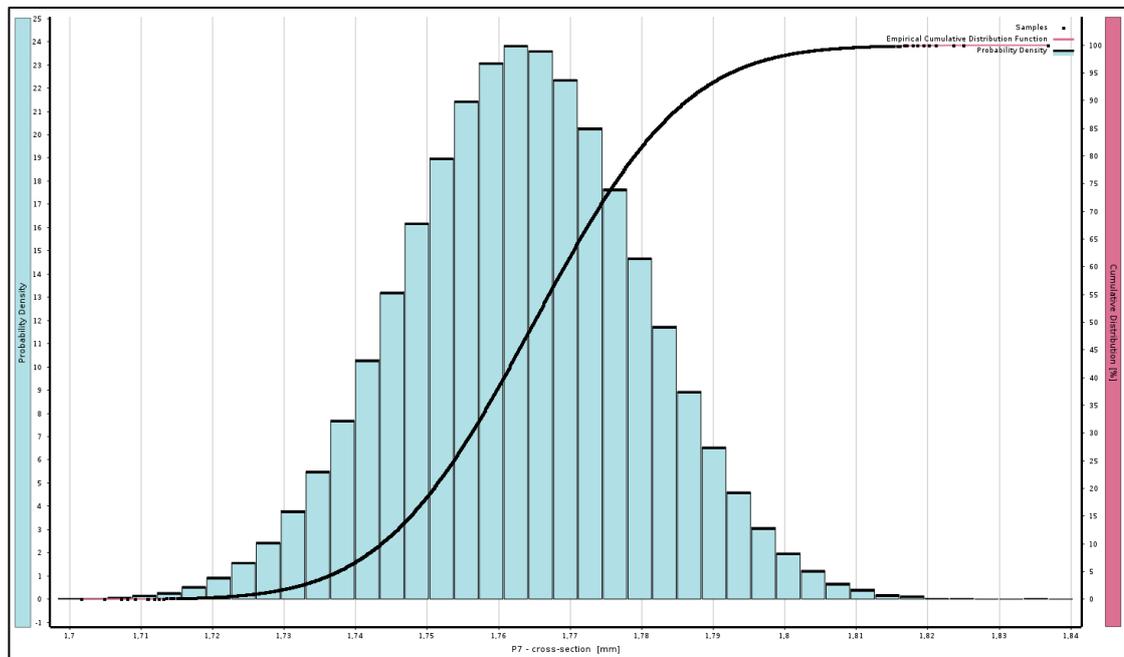


Figure 29. Cross-section normal distribution – Six Sigma Analysis

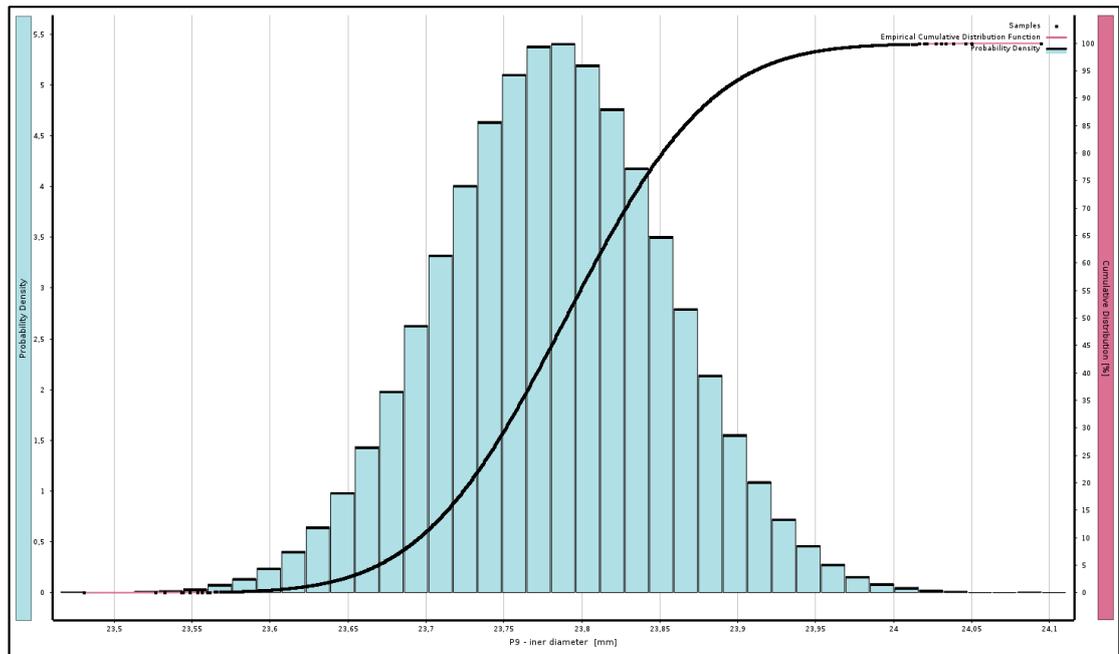


Figure 30. Inner diameter normal distribution – Six Sigma Analysis

Regarding the graphs, it is seen that the normal distributions have symmetry from the mean value as a proper normal distribution has to be. Moreover, there are some parts that would not be inside the tolerances limits and the manufacturer should scrap them, to be exact, the probability that a model manufactured would be out of one parameter tolerances is a 0.3% like it was explained before. Comparing the two bells that the software give, the only differences are the mean value, which means that the bell was moved, and that the thickness of the inner diameter bell is bigger that the cross-section one.

As a result, the program also provided the probability distribution of the contacts pressure depending on the distributions of the design parameters, which have been analysed before. Therefore, what comes next are the probability distributions that the contacts pressure would have.

- O-ring contacting piston. the results of this contact can be checked in Figure 31
- O-ring contacting cylinder. the results of this contact can be checked in Figure 32

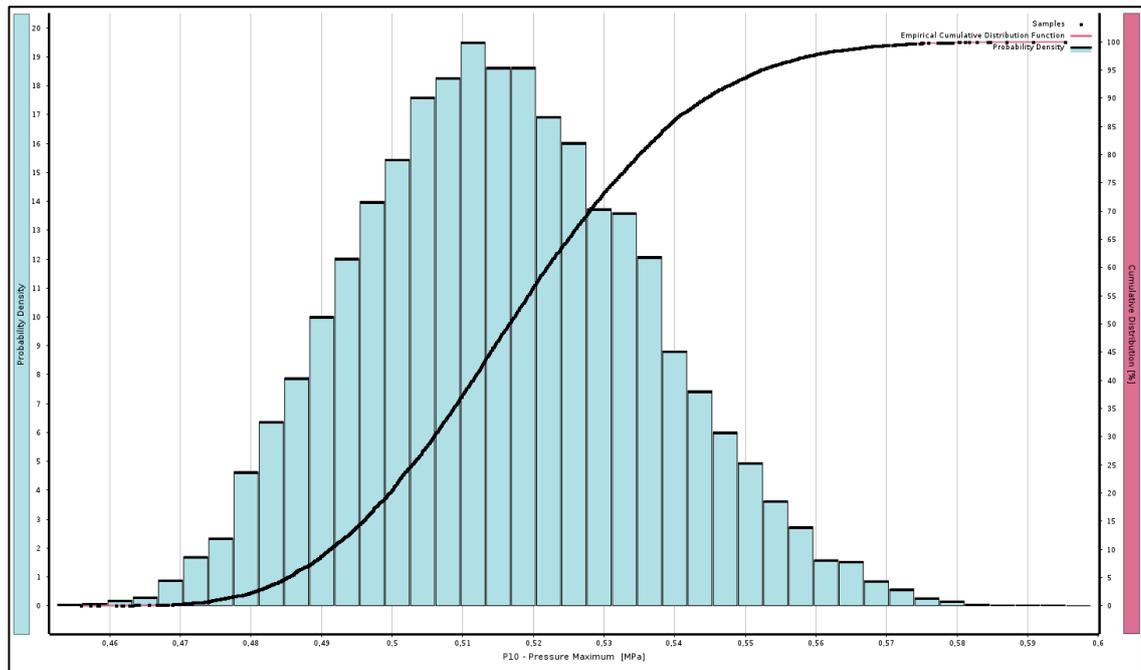


Figure 31. Probability distribution, O-ring vs the piston – Six Sigma Analysis

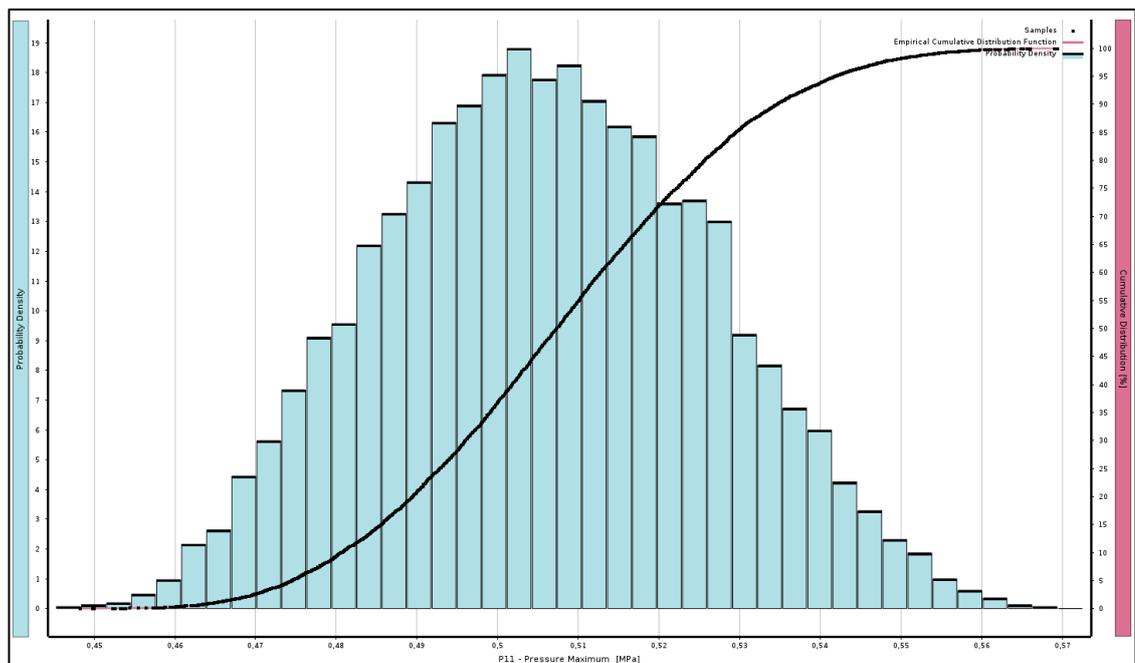


Figure 32. Probability distribution, O-ring vs the cylinder – Six Sigma Analysis

What is seen in both graphs is that they are not symmetric as the design parameters were, this means that the design parameters had correlation between each other and that the final response depends not only of the values of those parameters, it depends on how they are combined as well.

If this studied would be made with the exact material that the manufactures would use and if the materials limits were known, it would be easy to check if there would be models

that even if their parameters values would be inside the tolerances defined, they could be out of the material limits. That is the reason why these graphs was created. However, in this case it is impossible to check this issue because a default material, which its limit pressure is not known, is been used.

4 DISCUSSION

It is very common to see companies where their engineers work in order to create a simulation of a model that will be manufactured, these simulations let the company know how the behaviour of the part will be and depending on the results they take decisions such as changing tolerances or even dimensions. Although most of these simulations are set up only considering the nominal values of the model bound, taking in account the tolerances of the bounds and even all the Product and Manufacturing Information that a model could have will provide more precise results as well as more useful information to check the robustness and take better decisions. Actually, that was the main purpose of the thesis, to include and consider PMI in a model in order to check its robustness.

First, before setting up any simulations the software needed a model to work on. So, the purpose was to create a 3D CAD model in which the Product and Manufacturing Information was included in it. The model created is an O-ring because it was thought that it would be a good model to follow all the process and its geometry would not give any problems. On the other hand, it was realized that it is not so common in the sector to include PMI in a 3D model for manufacturing purposes. Therefore, what it was done is to create two tutorials that explains step by step how to set up the PMI annotations in a 3D model to try to normalize this procedure. In this case, the tutorials are thought to be followed when using Solidworks 2018 and Siemens NX 11.0.

The next stage was to investigate how to share files between CAD programs and downstream ones so that the model could be used with no missing information. As a result, the only type of file that stores all the PMI data included in 3D models and that has all the specifications required is STEP AP242.

After that, the goal was to import the STEP AP242 file in the ANSYS workbench 19.1 in order to simulate it. However, the software does not have any tool that considered the PMI annotations in the model so it was decide to take different steps to get to the same results. Due to the model is axisymmetric; every equidistant point of the model has the same behaviour, so a 2D surface model would give the same information that the 3D one does. Therefore, that is what it was done because it helped to the Finite Element software to run the simulation faster. At this point, the model was set up using the software tools

in order to simulate the contact pressure of the O-ring under the nominal values of the bounds, so the software provided the first results. However, these results were not the final ones because it was needed to consider the PMI data somehow.

Therefore, the next objective was to create simulations that generate a response surface of the contact pressure that take in account the tolerances of the model. To achieve this goal the Design of Experiments tool in the same ANSYS workbench 19.1 was configured with the design parameters, the O-ring cross-section and the O-ring inner diameter, and their respective tolerances. After running the simulation, the software gave the values of different contact pressure situations inside the tolerances range. Afterwards, the final response surfaces were generated so that the user could check the model robustness. Actually, the results show that it is important to consider the parameters tolerances because the behaviour can suffer changes that if the user only simulates the nominal value model it could miss a lot of relevant information. The results also show that the contact pressure is quite more sensible to cross-sections variation than inner diameter variations, this kind of results are the ones that help to take decisions

Finally, to fully check the robustness of the O-ring the noise parameter should be known in order to define the standard deviation of the process. Although it is impossible to get the noise parameters without manufacturing the model, a six-sigma study was realised under the hypothesis that the manufacturing process has a total deviation of 6σ so it has a reliability of 99.7%. Another simulation ran again so that the probability distribution of the contact pressure was calculated. Regarding the results, it is seen how the probability distribution does not keep the symmetry of the normal distribution, which means that there are correlations between the design parameters. Moreover, if the material used was the exact manufacturing material and its pressure limits were known the last distribution results would show if there was any combination of parameters that even if they were inside the tolerance range they could be out of the limit pressure.

At this point, I would say that this study is a very generic study so it does not make sense to fully analyse the result values. However, it helps to reaffirm that it is so important to consider the tolerances in every simulation in order to not miss any information and take good decisions in the manufacturing process. At the same time, the fact that it is so generic helps to recreate the same procedure with any other kind of model, which its robustness needs to be checked as the O-ring was.

On the other hand, this thesis has some points that could be completed in order to get a more accurate procedure with no need to work under hypothesis. The first thing that could be made is define the exact material that the real world model would have, when this point was completed a “Real World Study” could be made so that the noise parameters could be defined and the six sigma study would have much more sense. Moreover, if any Finite Element software would have a tool to consider PMI data so that the Design of Experiment step was automatically calculated, this procedure would be shorter as well as much more effective.

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Calculation Results Concentric Position Eccentric Position Housing fill: Recommendations for FFKM O-Rings Cross-sectional Compression: 12-18%

APPENDICES

Appendix 1. PMI/MBD data in Solidworks 2018 (Tutorial)

To create PMI annotations in Solidworks designs we must follow the next steps.

- ✓ Select **Tools** in the top left of the screen.



Figure 33. Solidworks drop-down options

- ✓ In the **Tools** drop-down select **Options** 

In the top left of the dialogue box there are two different tabs, **System Options** and the **Document Properties**, for our purposes we are going to focus in the document properties, concretely in the **DimXpert**.

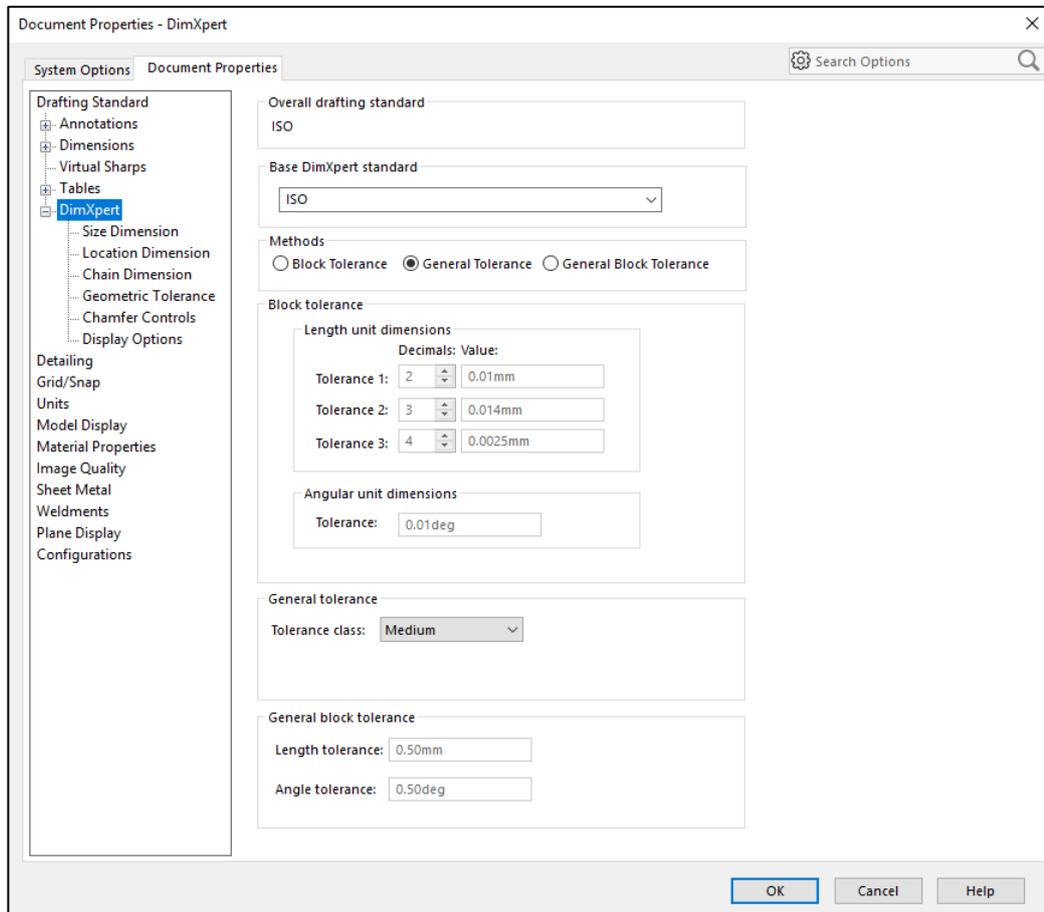


Figure 34. Document Properties – DimXpert dialogue box

✓ Select **DimXpert**.

- Define the “**Base DimXpert standard**”, it is the standard that the document will follow to create the annotations.

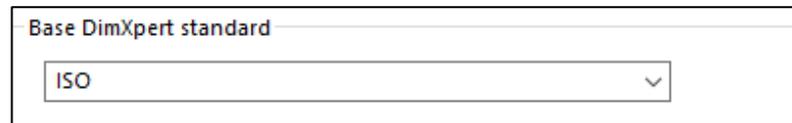


Figure 35. Base DimXpert standard

- Chose a method. Depending on your choice, the MBD will be created by different ways.

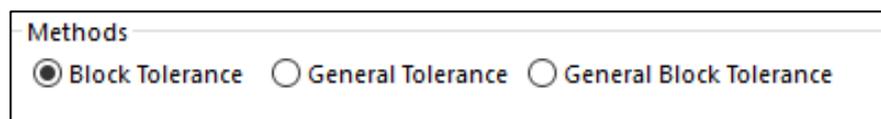


Figure 36. MBD method dialogue box

You just have to follow the step of your chosen method, Block Tolerance (1) or General Tolerance (2).

1. Block Tolerance

- ✓ In the same dialogue box, define **the Length unit dimensions** and **Angular unit dimensions**. Here you decide how many decimals will appear in your tolerances.

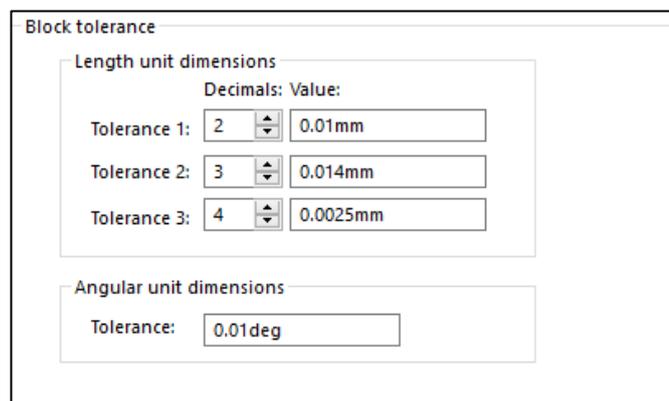


Figure 37. Block tolerance dialogue box

- ✓ In the DimXpert drop-down select **Size Dimension**. Define the default tolerances depending on what kind of dimensioning is going to be used, you can also chose what default style is the best for each case.
- Bilateral, different tolerance for each side of the nominal value
 - Symmetric, same tolerance for both sides of the nominal value

- Block, the default value is set as a tolerance

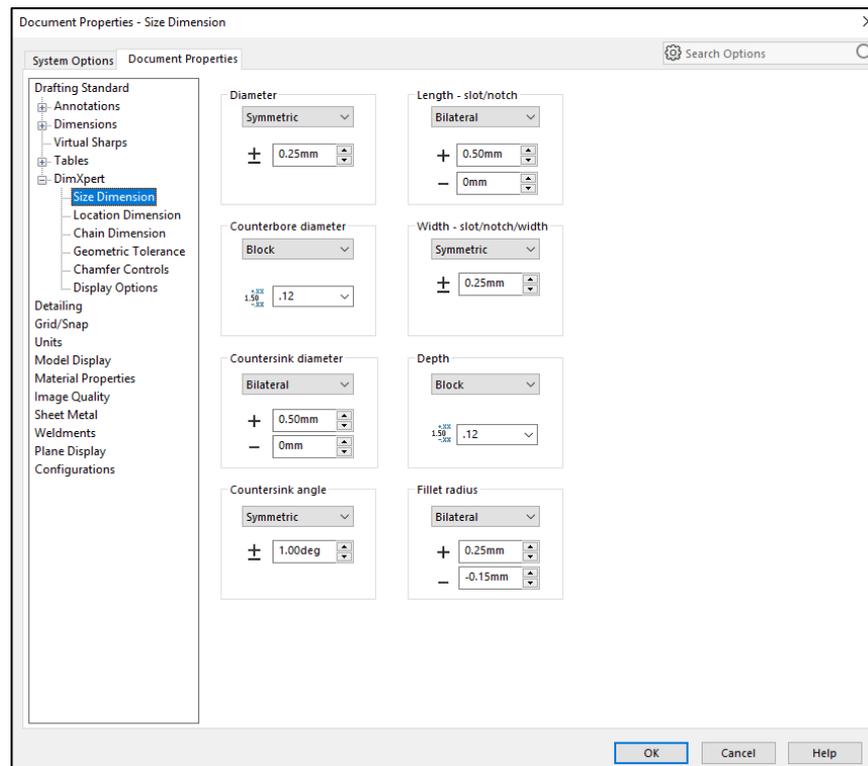


Figure 38. Size Dimension dialogue box

- ✓ Select **Location Dimensions**. Chose the **Inclined Plane Dimension Scheme** that makes the model more understandable and fits better with the rest of the annotations.

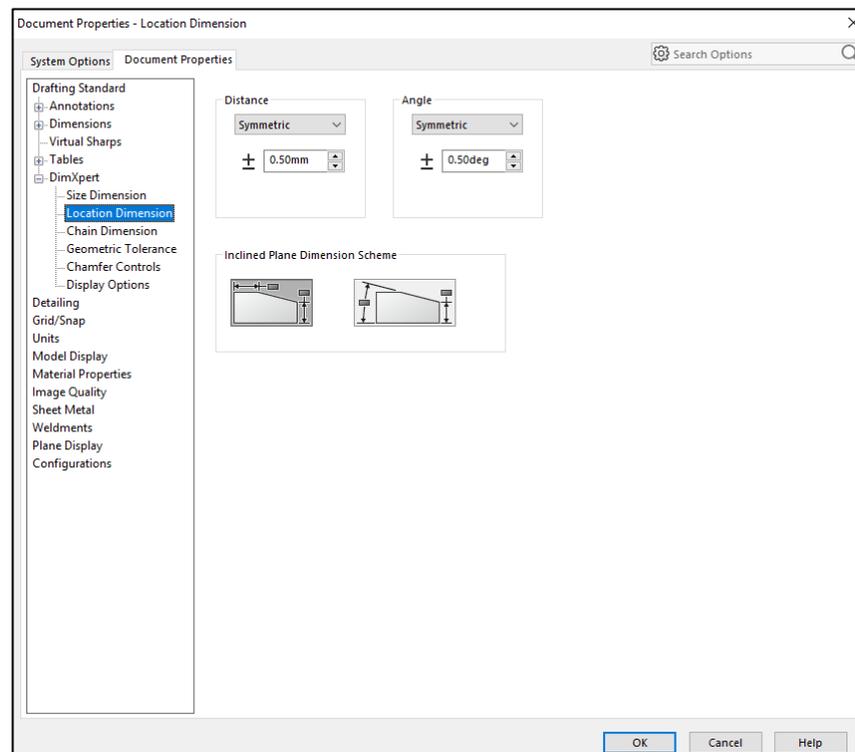


Figure 39. Location Dimension dialogue box

- ✓ Select **Chain Dimension** in the **DimXpert** drop-down.
 - In **Hole dimension** and **Pocket dimension** areas chose what dimension method will be better to be applied, **Chain** or **Baseline/Datum**.



Figure 40. Document Properties - Chain Dimension - Dimension method

- If there's a hole pattern, chose how you want the **Pattern location** and the **Distance between features** tolerance to be, choosing the type of tolerance you prefer.

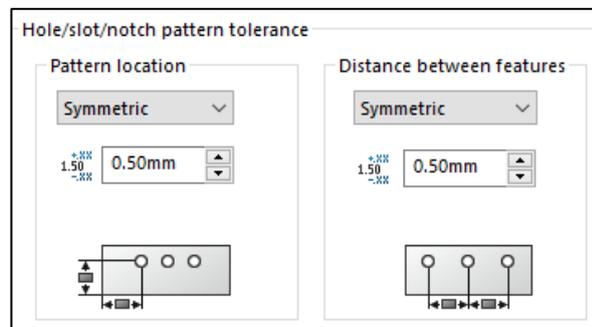


Figure 41. Chain Dimension - Hole/slot/notch pattern tolerance

- ✓ Select **Geometric Tolerance** in the **DimXpert** drop-down. There you can set up how the geometric tolerances have to appear in the model annotations.

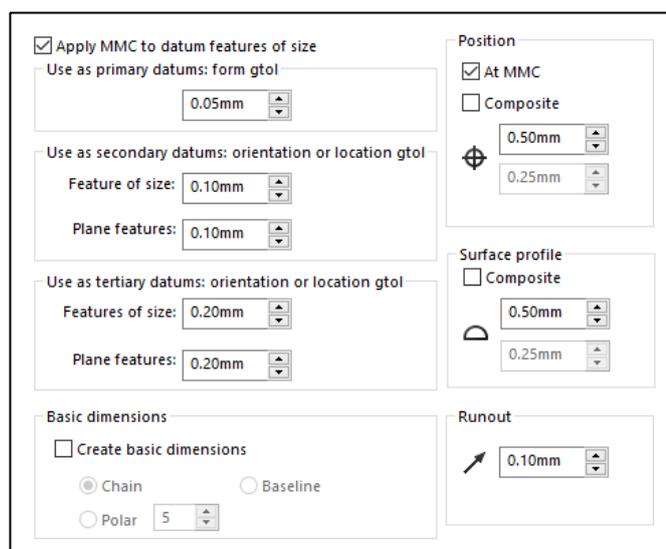


Figure 42. Geometric Tolerance dialogue box

- ✓ Select **Chamfer Controls** in the **DimXpert** drop-down. Define the Chamfer width ratio and the Chamfer maximum width.

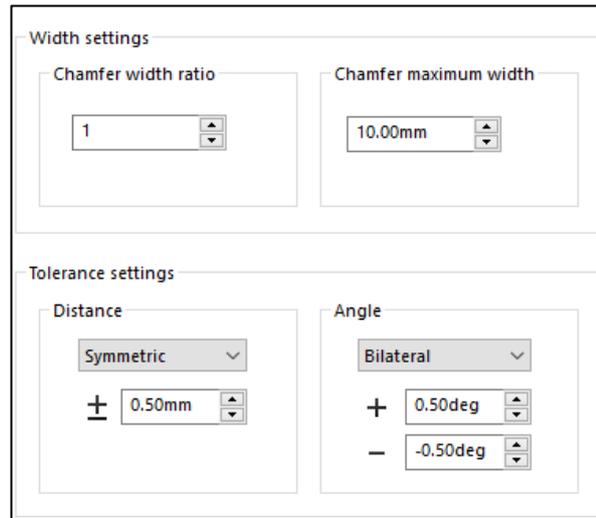


Figure 43. Chamfer Controls dialogue box

- ✓ Select **Display Options** in the **DimXpert** drop-down, and select which of the available schemes you prefer for each item. In the following figure you can see the difference in the annotation schemes.

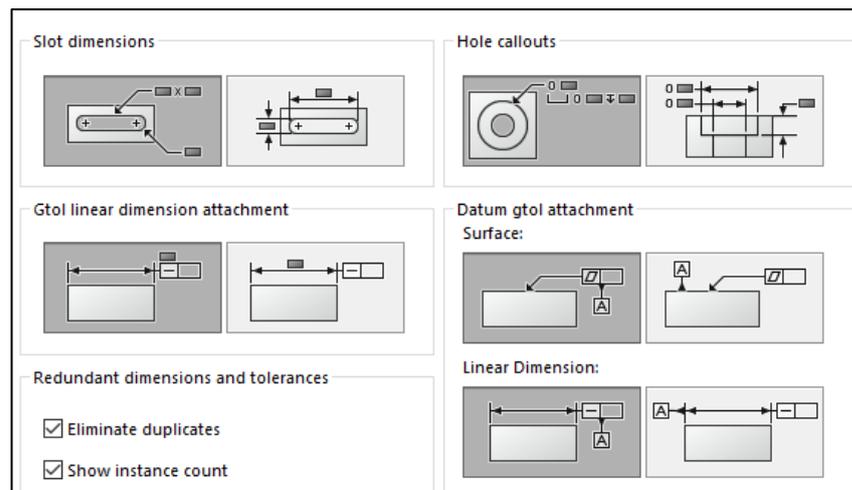


Figure 44. Display Options dialogue box

When all the points have been set up, click **OK** at the right down corner of the dialogue box to start placing the dimensions and annotations in the 3D model interface.

- ✓ Select **Datum** in DimXpert working bar (Figure 13), set up the different options that have appeared in the left side of the screen. Choose the **Label Settings**, the **Leader** and customize **the Datum**, select the surface where the Datum Feature goes.

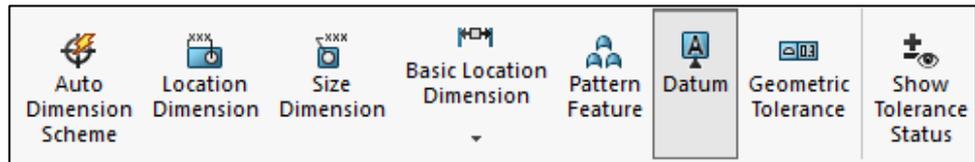


Figure 45. DimXpert working bar

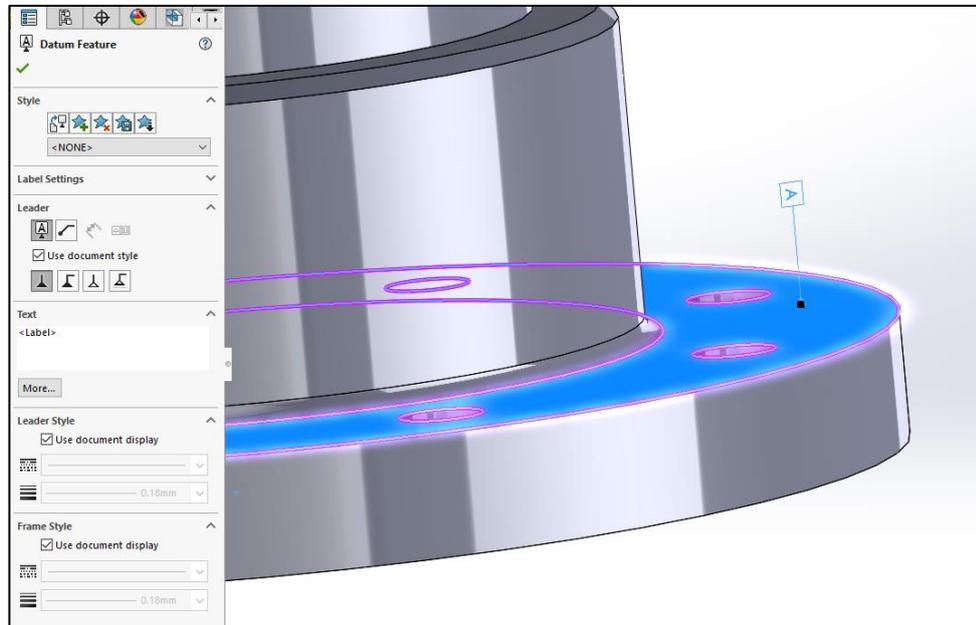


Figure 46. Model interface – Datum Feature set up

- ✓ Select **Auto Dimension Scheme**, define the different options in the settings area and select the reference features in the 3D model interface. Press on the tick and the annotations with the default tolerance will appear and the model surfaces will be in green and yellow.

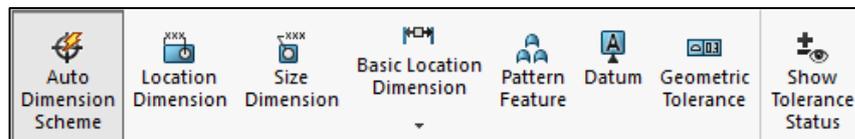


Figure 47. DimXpert working bar – Auto Dimension Scheme

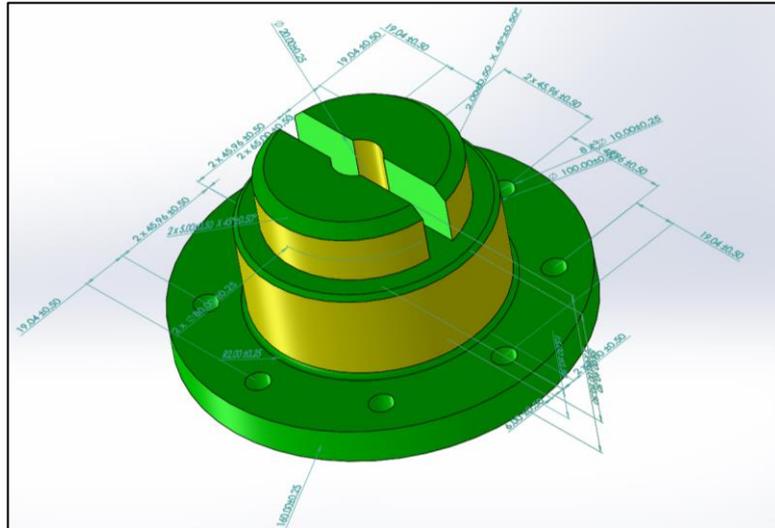


Figure 48. Auto Dimension Scheme result

The yellow means that the model is not full defined, that some information is missing to full understand the model. Anytime you want to check if the model is well defined or not you just need to press in the **Show Tolerance Status** (Figure 49) and you will be able to check the colours.



Figure 49. DimXpert working bar – Show Tolerance Status

The procedure to add a dimension that is missing for the well-definition of the model is the same as the one that's explained later in the **General Tolerance** method. At this point, you can also decide to change some tolerances and dimensions just double-clicking on them.

2. General Tolerance

To start placing the annotations in the model we have to first know what icons use in the DimXpert working bar.

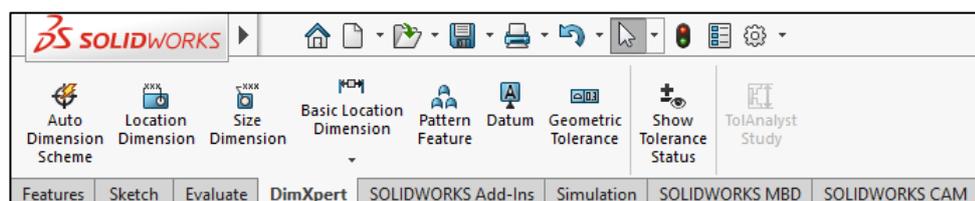


Figure 50. DimXpert working bar

What general tolerance method consists on is to define the dimensions individually. The next steps explain how to set up and place those dimensions.

- ✓ Select **Location Dimension** in the **DimXpert** working bar. Select the geometric elements (surfaces, edges...) you want to define and place the dimension. The default tolerance is already defined. In case the tolerance shown is not the one you expect, it's easy to change in the drop-down that you have in the left part of the screen.

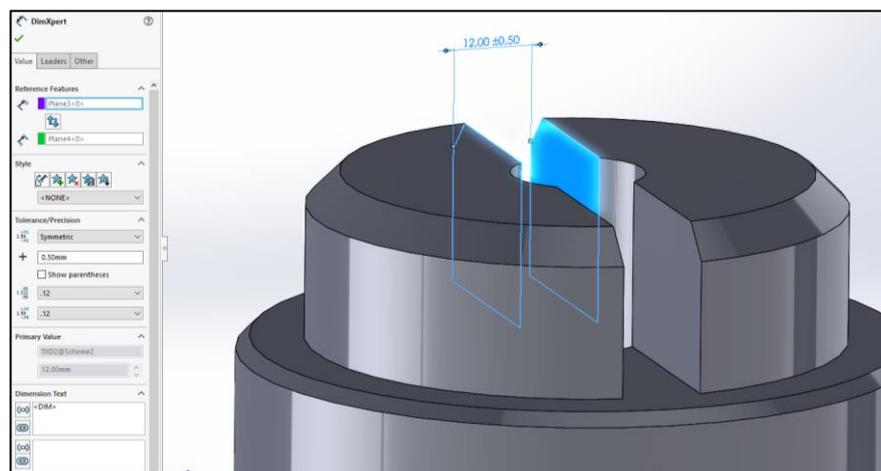


Figure 51. Setting up Location Dimensions

- ✓ Select **Size Dimension** in **DimXpert** working bar, here you can define holes, radius, diameters, etc. Select the element in the model and the dimension will appear with the default tolerance. A similar drop-down as the **Location Dimension** one appears where you can edit different points of that annotation, including the tolerance.

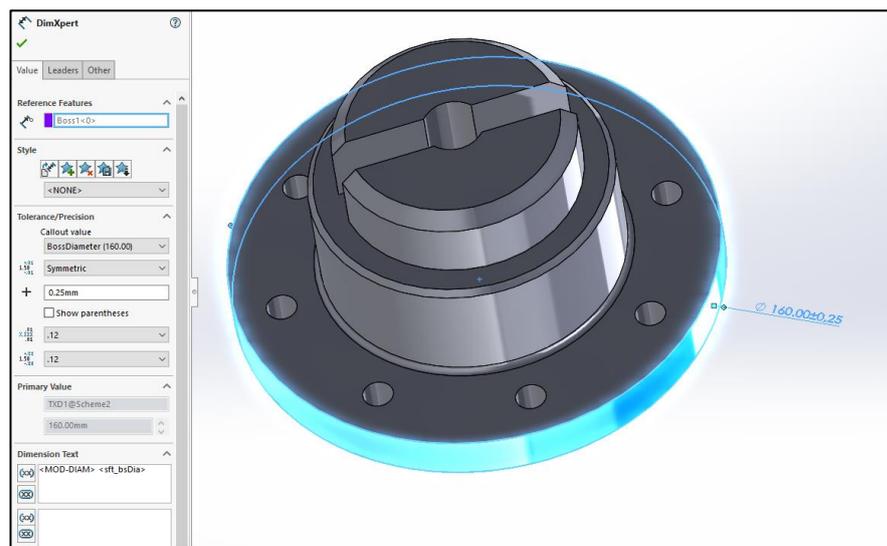


Figure 52. Setting up Size Dimension

Appendix 2. PMI/MBD data in Siemens NX 11.0 (Tutorial)

To create PMI annotations in Siemens NX 11.0 models we must follow the next steps.

- ✓ Add the PMI tab in the working bar. Select **Application** and then select **PMI**, the PMI working options will be added automatically.

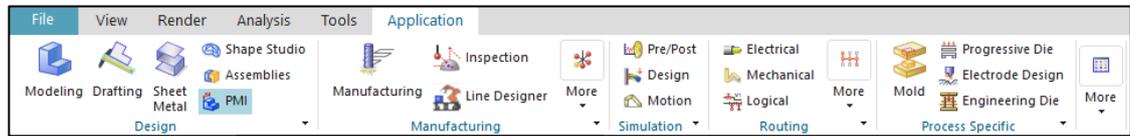


Figure 54. NX 11.0 working bar - Application

- ✓ In the drop-down **Part Navigator** open **Model Views**, chose the working model view to start defining the PMI with a double-click.

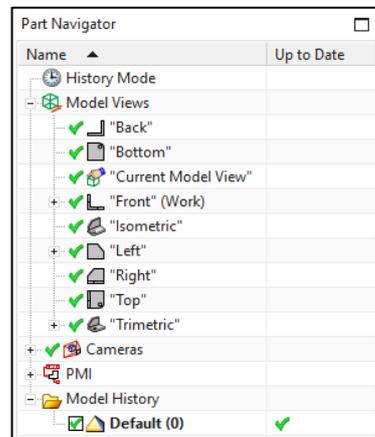


Figure 55. Model View drop-down

To start defining the dimensions:

- ✓ Go in the **Dimension** area, in **PMI** working bar, the different icons show what kind of dimensioning generate each one, so select the icon according to the dimension you want to set and follow the dropdown steps.



Figure 56. PMI dimensioning options

- ✓ When you have selected one of those icons you just have to follow the steps that the drop-down options show, like selecting one or two different surfaces depending of the kind of dimensioning you have chosen.

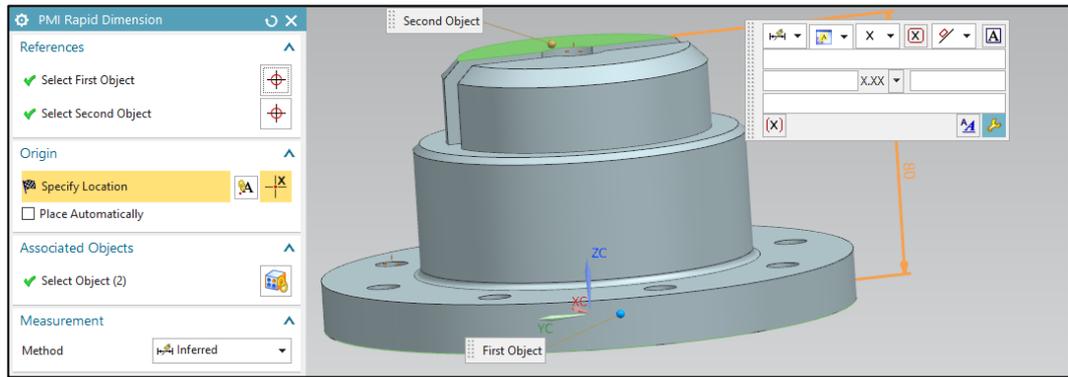


Figure 57. Model interface setting up the Rapid Dimension

- ✓ When you have placed the dimension in the right location of the model interface a dialog square appear to set the dimension up (type of text, type of tolerances, inspection dimension, text over dimension line, number of decimals, etc.)

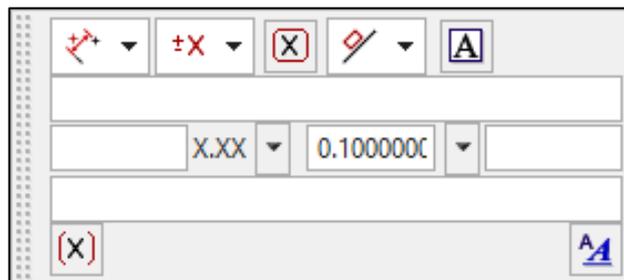


Figure 58. Dialog square to set up the dimension

To set the annotation you have different options in the **Annotation** working bar. (Beforehand, to place any kind of annotation you just have to press the left button of the mouse in the location, with the button pressed just move the mouse and an arrow will be generated automatically to place the text in a more understandable place.)

- ✓ **Note.** Select Note symbol, specify where you want to place the text and write down the text in the available space.

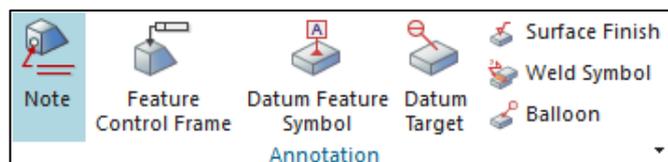


Figure 59. PMI – Annotation working bar - Note

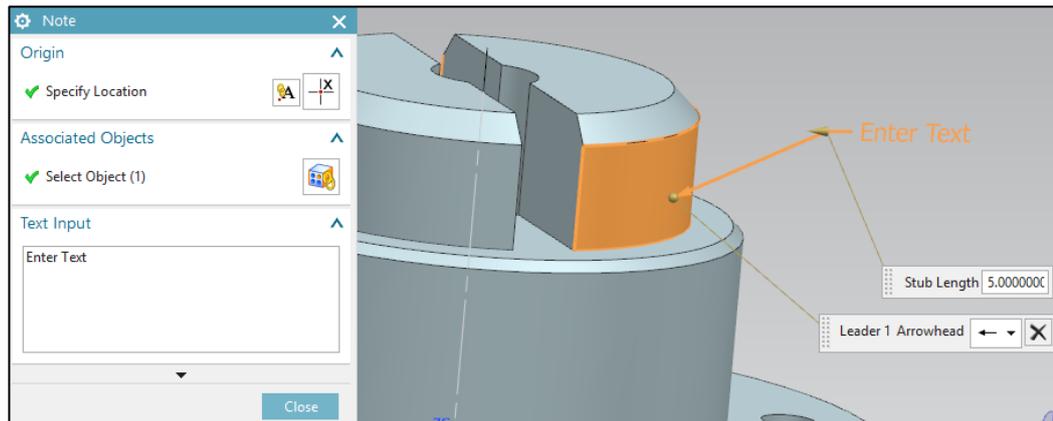


Figure 60. Setting up Notes

- ✓ **Feature Control Frame.** Select Feature Control Frame, define all the geometric tolerance to place in the Frame dropdown, and then specify the location to place it.

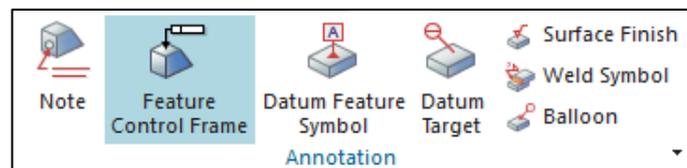


Figure 61. PMI – Annotation working bar – Feature Control Frame

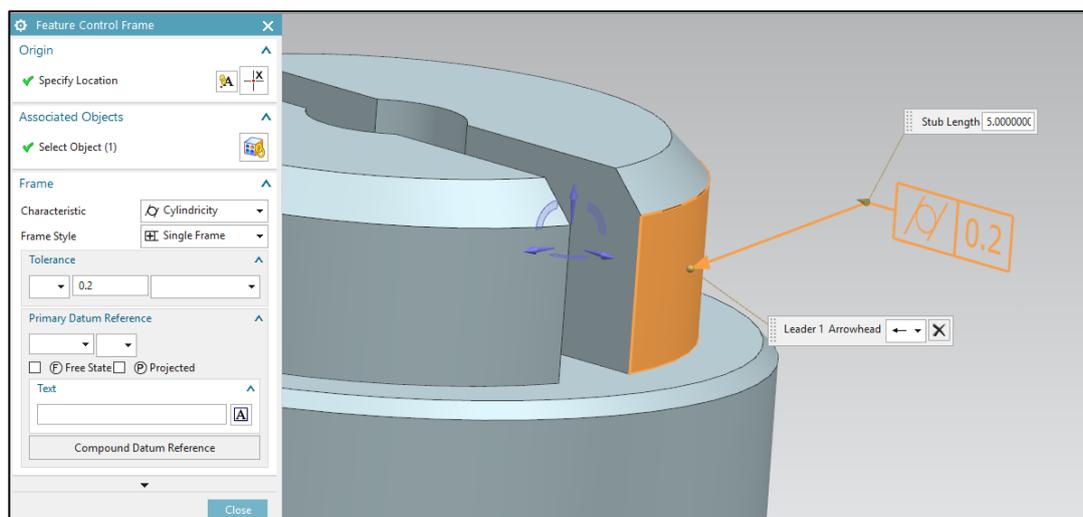


Figure 62. Model interface setting up geometric tolerances

- ✓ **Datum Feature Symbol.** Select Datum Feature Symbol, define a Datum Identifier in the dropdown options. Choose the surface to place the identifier.

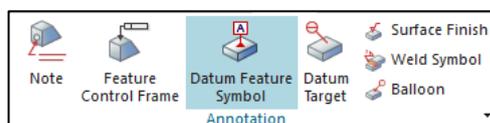


Figure 63. PMI – Annotation working bar – Datum Feature Symbol

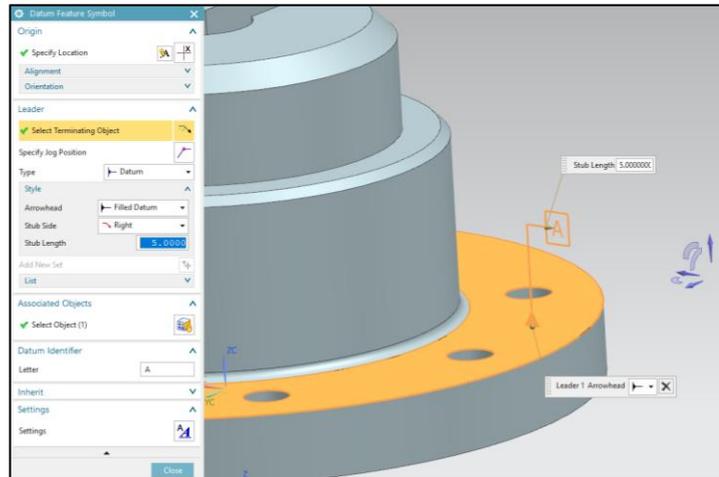


Figure 64. Model interface setting up the Datum Feature

- ✓ **Datum Target.** Select Datum Finish, define the Type to use, and define the different topics of the target: Label, Index and the Terminator.

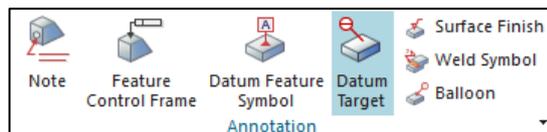


Figure 65. PMI – Annotation working bar – Datum Target

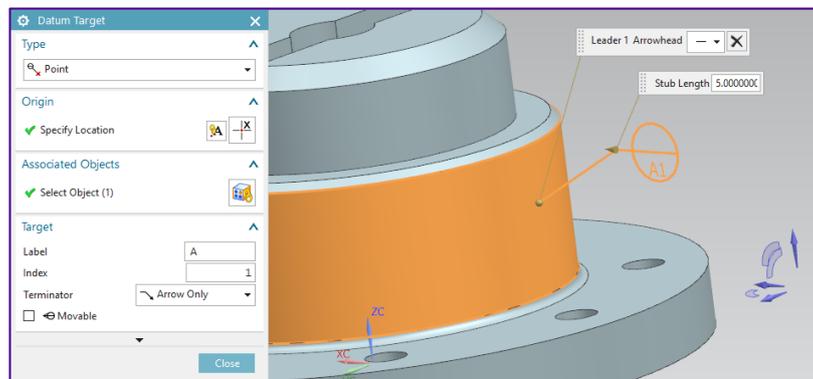


Figure 66. Model interface setting up the Datum Target

- ✓ **Surface Finish.** Select Surface Finish, define the standard to follow, the material Removal and set all the characteristics, then place the symbol in the respective surface.

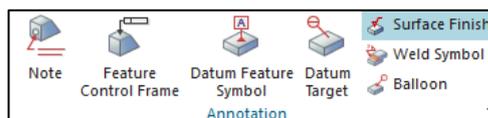


Figure 67. PMI – Annotation working bar – Surface Finish

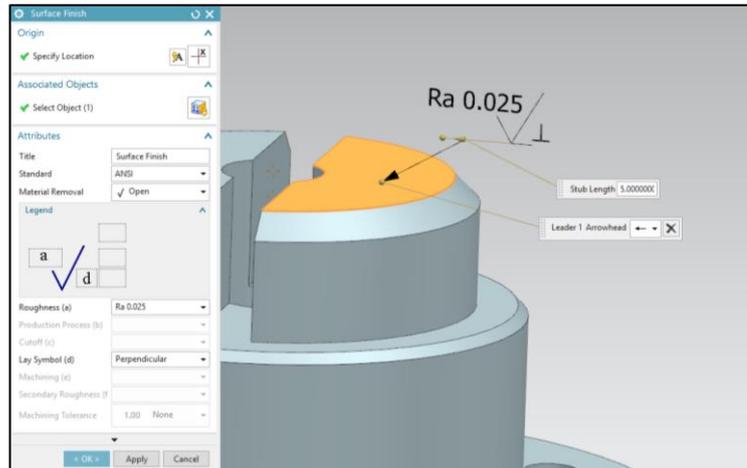


Figure 68. Model interface setting up the Surface Finish

- ✓ **Weld Symbol.** This is useful when you want to specify that to build the model some parts must be welded. The procedure is the same, set up the drop-down options, chose the symbol that shows what kind of welding you want, and place the annotation in the right location.

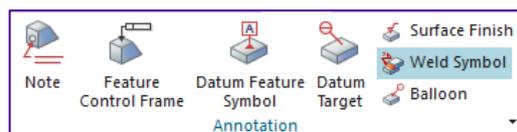


Figure 69. PMI – Annotation working bar – Weld Symbol

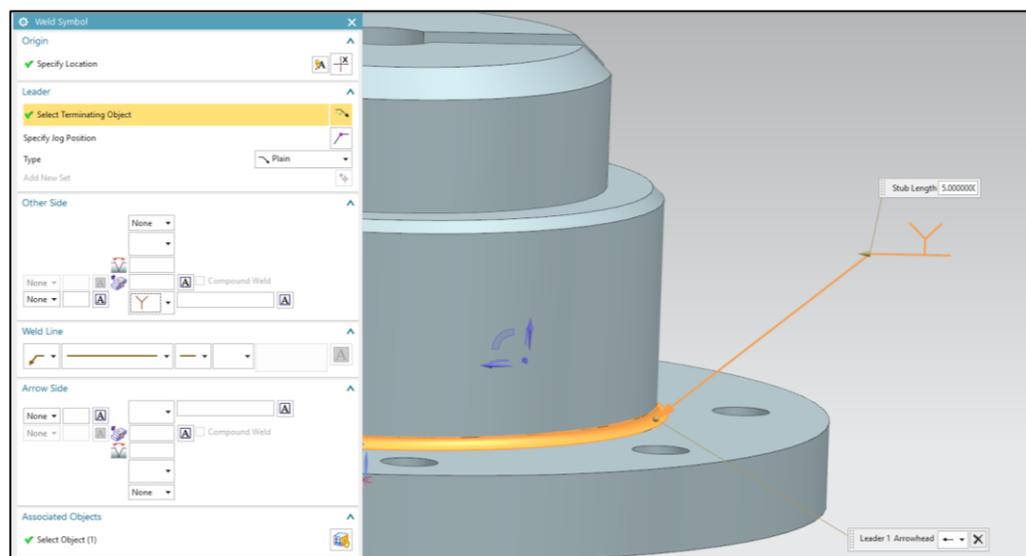


Figure 70. Model interface setting up the Weld Symbol

- ✓ **Balloon.** To identify an upgrade of a part in the model we use the balloon notes. Select Balloon, define all the attributes to show and place the balloon in the interface model.

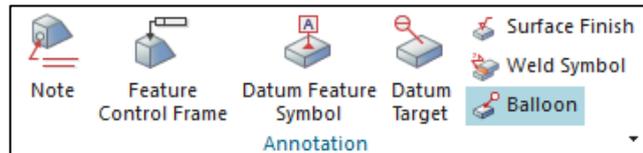


Figure 71. PMI – Annotation working bar – Balloon

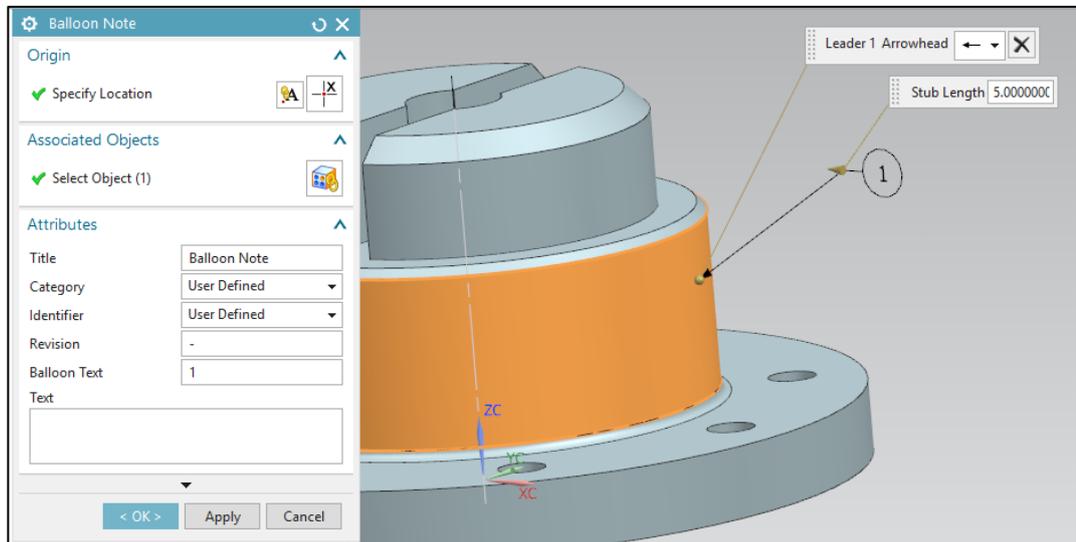


Figure 72. Model interface setting up a Balloon

PMI also includes customer's annotations to make sure that the provider is informed of some important items that the model must have, usually most of these annotations are about how to manufacture the model, because it belongs to a bigger group of pieces that have to fit together in the final assembly.

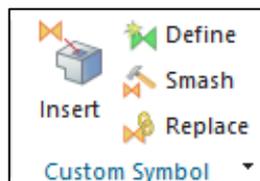


Figure 73. PMI – Custom Symbol

- ✓ To include these annotations select **Insert** in the **Custom Symbol** options. A drop-down options appear in which you can look for the correct symbol in the **Folder View** area.

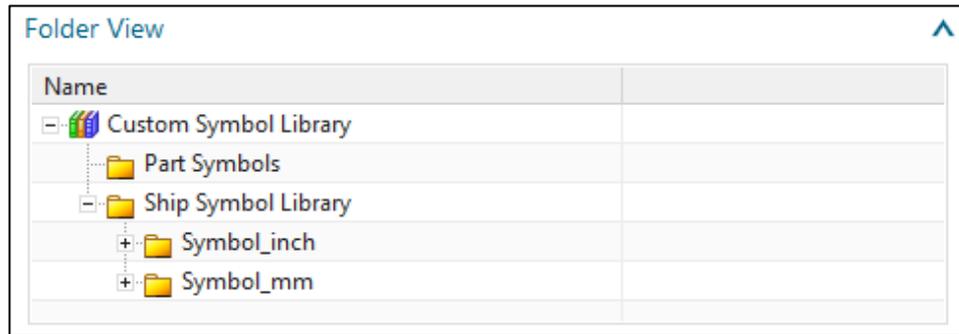


Figure 74. Custom Symbol's Folder View

- ✓ When the symbol has been chosen just place it in the right position. (Sometimes the size of the symbol is not right proportionated with the model, to solve this problem just change the scale in the **Symbol Attributes** area).

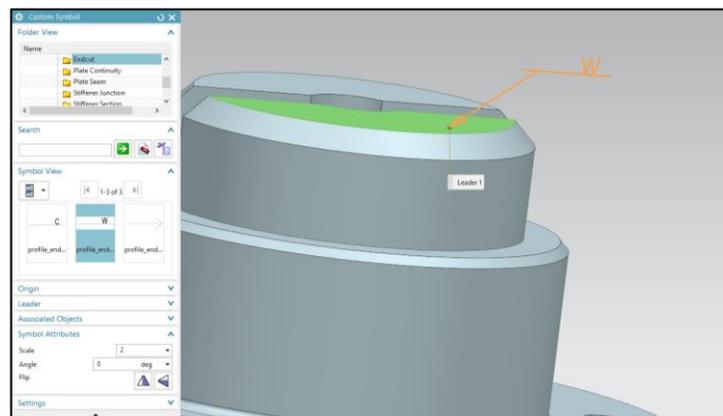


Figure 75. Model interface setting up the Custom Symbols

Moreover, you can also include geometrical marks that make the model more understandable.

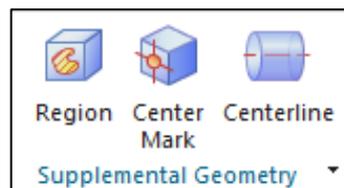


Figure 76. PMI – Supplemental Geometry

- ✓ Select some of the options in the **Supplemental Geometry** area in the working bar and place them in the right position just clicking in the corresponding geometry surface or edge

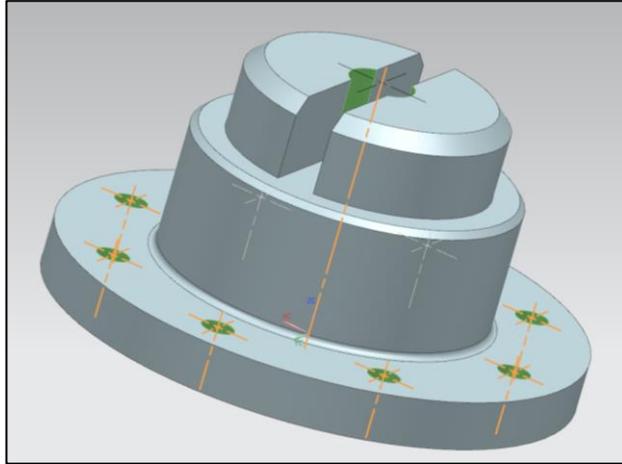


Figure 77. Model interface setting up geometric marks

Appendix 3. How to generate a STEP AP242 file

In case of Solidworks, when the model is ready and full defined with all the PMI data included, the user just has to go in the **Solidworks MBD** working bar and select the Publish STEP 242 File icon. In Figure 10 it can be checked how this icon looks like. Afterwards, a dialogue box appears asking where to store the file in the system and the STEP file is generated automatically.



Figure 78. Solidworks MBD working bar – Solidworks 2018

On the other hand, to export the Siemens NX 11.0 3D model, the procedure is quite different. To generate a STEP file the user has to click on **File** in the top left of the screen, place the mouse on **Save** and select **Save as** from the dropdown recently appeared. A dialogue box pops asking where to store the document, the file name, and the type of extension. In this last point select **STEP242 XML Files (*.stpx)** like it is shown in Figure 79.

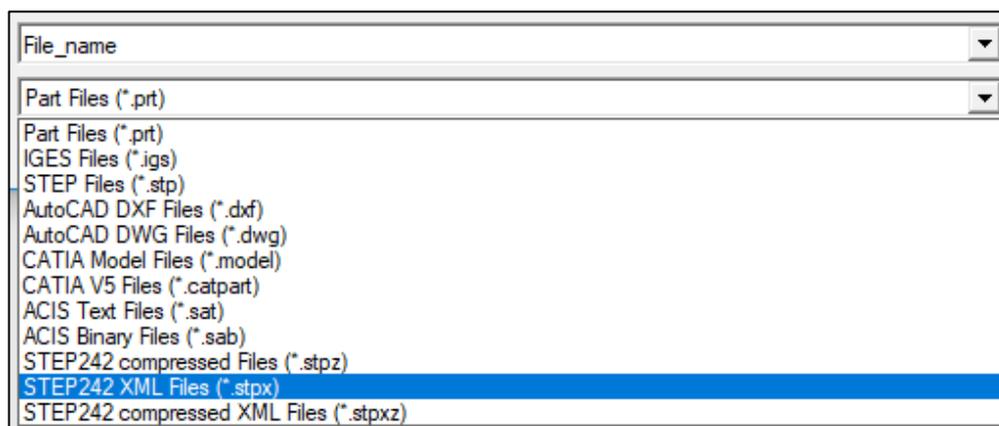


Figure 79. Extensions of Save as dialogue box – Siemens NX 11.0

Appendix 4. ANSYS details

- **Connections settings**

The topics modified of **Details of “Contact Region 1”** are the following:

- Including some more edges in the contact target.
- Switch the type of contact to **Frictional** with a frictional coefficient of 0.2.
- Switch the contact behaviour to **Asymmetric**.
- Turn the Trim Contact **Off**.
- Switch the Normal Stiffness to **Factor** of **0.01**.
- Switch the **Update Stiffness** to **Each Iteration**.
- Switch Pinball Region to **Radius** of **1, mm**.
- Switch the Interface Treatment to **Add Offset, Ramped Effects**.

When this first contact was defined, it was needed to go for the second one, the piston contacting the O-ring. The O-ring edge was selected to define the **Contact Bodies** and the piston edges to define the **Target Bodies**, the other modifications that had to be done in **Details of “Contact Region 2”** were the same as the ones in Contact Region 1.

- **Mesh settings**

Element size of each mesh:

- O-ring Element Size was 0.1 mm
- Cylinder Element Size was Default (2.2508 mm)
- Piston Element Size was Default (2.2508 mm)

- **Statistic Structural settings**

The next list shows the modifications that had to be done in **Analysis Settings**:

- Turn the **Auto Time Stepping** on.
- Change the Substeps values
 - Initial = 1000
 - Minimum = 100
 - Maximum = 5000
- Turn on the **Large Deflection**.