Model Based product Definition for improved transfer of tolerance information between digital manufacturing systems (Enhanced use of PMI)

John Bijnens

A thesis submitted in partial fulfilment of the requirements of Staffordshire University for the award of the degree of Doctor of Philosophy

3rd November 2024

Abstract

Model-Based Definition (MBD) implies that the 3D CAD model with 3D annotations is used for all communication between all parties involved in the design and manufacture of a product. MBD relies entirely on the machine readability of the annotation. This means that the type of annotation, its content and the geometric entities to which it refers, the so-called semantic references, can be interpreted by software. This PhD study aims to establish how effective this communication is. The research focuses on geometry, annotations and neutral exchange formats. It considers how accurately geometry can be transferred from one CAD system to another and what can affect this accuracy. For annotations, the focus is on machine readability. This study investigates available options in different CAD systems. The conclusion when exporting to STEP AP242 is that they are not always equivalent and that the designer should be aware of this. Finally, a software package has been developed to help designers and manufacturers avoid as many of the problems identified in this research as possible. This software package analyses, identifies and lists the annotations present in the model and indicates which annotations may cause issues for other manufacturing stakeholders. This could be the lack of semantic references or the loss of data when exporting to STEP. In the latter case, it is possible to automatically convert these annotations to another type where no data is lost. The software has been developed, used and tested by a number of people in the manufacturing environment who have provided valuable feedback on the GUI and functionality. Although the original goal could not be met because of limitations in the library's APIs, the software can be considered a success in the context of this study.

Acknowledgements

In 2014, when considering a PhD, I received advice from Prof David Cheshire to apply to Staffordshire University. I am thankful to Staffordshire University for accepting my application and giving me the opportunity to commence my PhD. I also express gratitude to Prof David Cheshire for his invaluable guidance in starting this program and my supervisors, Prof Abdel-Hamid Soliman and Chris Wayman, and my external supervisor, prof Tony Dodd, for their unwavering support and assistance.

I have come to liken the process of completing a PhD to constructing a building, where I act as both the architect and the builder. However, numerous individuals have contributed to the various stages of construction, whether knowingly or unknowingly, by sharing their expertise. It would be impossible to list and acknowledge everyone who has contributed, yet there are some who I would like to highlight for their direct guidance and support.

Firstly, I would like to express my gratitude towards Innoptus company and its personnel, particularly CEO Xavier Werbrouck and Thomas Sarre. Xavier Werbrouck provided me with the necessary license to develop the Toolkit application for PTC Creo and collaborated in testing of the software. Thomas Sarre tested the software and contributed with valuable suggestions for the development of the GUI.

Next, I would like to extend my gratitude to

- Michael Fridman, the product manager at PTC, who is responsible for MBD, among other things, for his contributions in enhancing my understanding of PTC's MBD functions and for forwarding my inquiries to PTC's development team

- Raphael Nascimento, the product manager at Sigmetrix and Michael Fridman's predecessor at PTC, for providing insights into PTC's MBD functions and MBD in general

- Richard Katerberg at Visiativ for offering valuable responses to my inquiries related to CATIA

- Raf Schepers of Siemens for answering my questions regarding Siemens NX

- Marc Brouwers, senior consultant at CADAC Group, for answering my questions about Autodesk Inventor

Last but not least, I must express my utmost gratitude to my beloved wife, Myriam. Her unwavering patience with me as a petulant, often unresponsive, desk-bound monk was boundless. Throughout numerous hours, she served as a sounding board for a plethora of ideas. Thank you so much for enabling me to pursue my dream.

Contents

List of Figures

List of Tables

1

Literature Review: Part I - Claims of MBD

1.1 Introduction

A new philosophy that originated in the aerospace industry is making inroads in the world of mechanical design. This new philosophy is called MBD, an acronym of "Model-Based Definition", and is believed to replace the so-called "old or traditional way" — the 2D engineering drawings — in the near future (Venne et al. 2010). Ding et al. 2021 define MBD as 'A model-based method for defining product data. Based on the topological structure and geometric data of the model itself, process information such as surface roughness, tolerances and annotations are added'. In concrete terms, this means that tolerances, roughness, GD&T, annotations are applied directly to the 3D model. These are therefore called 3D annotations (Lenne et al. 2009).

Proponents of MBD claim applying MBD will lead to immense (time) benefits and greater accuracy when compared with the "traditional way" (Quintana, Rivest, Pellerin, Venne et al. 2010) not only because of the way data are handled within a CAD system but also because of the implications for PDM and PLM systems (Alemanni et al. 2011). There are four claims related to interpretation by humans that seem to back up this assumption (Garcia et al. 2011).

1. It is much easier to interpret an MBD model compared with a 2D drawing
2. It is much faster to create an MBD model as less dimensions need to be created
3. The 2D drawing is ambiguous where the MBD model is not,
4. As the 3D annotations are placed directly onto the 3D model, the MBD model is always up-to-date whereas with 2D drawings one is never sure whether this is the latest version or not.

Zhong et al. 2021 argue that the work of the quality inspection is currently becoming increasingly difficult because it is very hard to translate the product quality requirements into the actual inspection work, namely to determine what should to be measured and how. They state the main reasons for this are ambiguity in understanding and as such serious dependence on human experience. This can all be solved by automating the whole process by using MBD. Nyffenegger Felix et al. 2020 argue that MBD enables automatic generation of CNC toolpaths.

The four claims mentioned above are examined in this chapter.

1.2 MBD model is easier to interpret

The claim that an MBD model is easier to interpret must be viewed from two angles. The first relates to the geometry, the second to the info added to the model through annotations. As far as geometry is concerned, a 3D model is indeed easier to interpret than 2D drawings (Hudspeth 2006; Unver 2006). Annotations, however, are a completely different story. Annotations are visible dimensions, tolerances, notes, text or symbols (American Society of Mechanical Engineers 2019). Dimensions can have two kinds of tolerances assigned to them. They can have symmetrical tolerances that comply with the ISO 2768 standard. In that case, the tolerances are not explicitly shown together with the nominal value of the dimension, or they may have other tolerances that must be explicitly shown after the nominal value of the dimension to which they are assigned. Within the MBD philosophy only the latter kind of dimensions needs to be created in the model. Tolerances like GD&T (an acronym of Geometrical Dimensioning and Tolerancing) that specify what variations on form, orientation and location (Henzold 2020) are allowed together with notes, text and symbols must be created in the model within the MBD philosophy and in the drawing within the "traditional way". As far as annotations are concerned the only difference between the "traditional way" (see Figure 1.1) and the MBD philosophy (see Figure 1.2) is the omission in the MBD model of dimensions to which no explicit tolerances are assigned. The logical consequence of this is that applying the MBD philosophy does not in itself mean that interpreting the display of the annotations becomes easier. There is a difference between the "traditional way" and the MBD philosophy regarding the metadata that is added to the annotations. This will be discussed in more detail in Chapter 3, which is dedicated to MBD and annotations.

The four claims

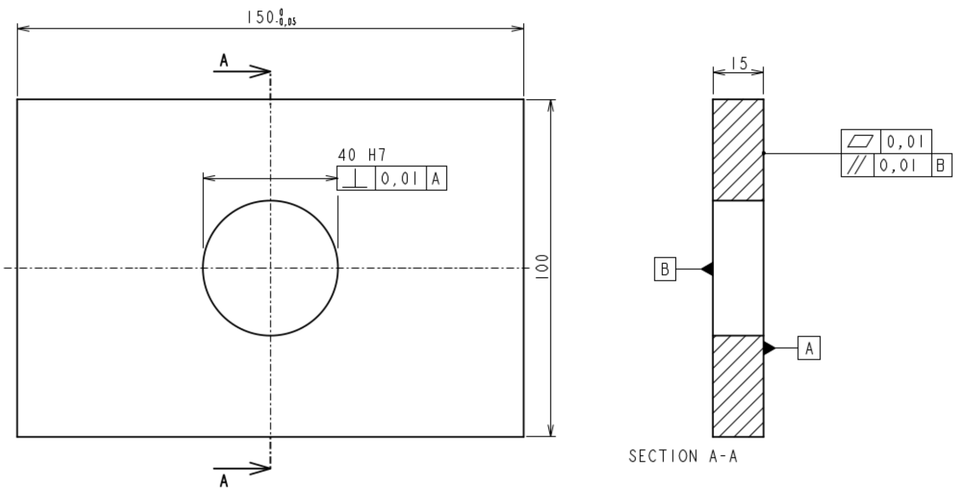


Figure 1.1: model dimensioned in the "traditional way"

1.3 An MBD model is easier to create

It takes less time to create an MBD model compared to traditional 2D drawings. This is mainly due to two reasons. One is the fact that, as mentioned in the previous section, within the MBD philosophy it is not necessary to create dimensions in the model to which no explicit tolerances are assigned. However, the biggest time gain is due to the freedom the designer has in placing the annotations on the model, in contrast to

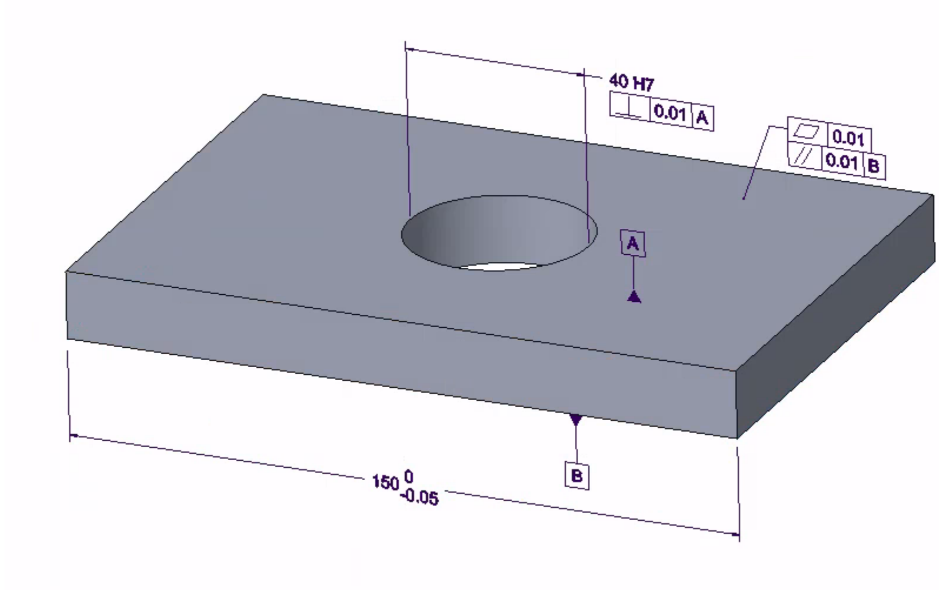


Figure 1.2: model dimensioned according to the MBD philosophy

2D drawings where it is quite a challenge to find the right views and cross-sections to place all the necessary annotations (Sandberg et al. 2019; Venne et al. 2010; Alemanni et al. 2011). The detailing itself, namely choosing the right dimensional tolerances and the correct form tolerances, is a mainly experience-driven process that remains just as difficult in the MBD philosophy as in the "traditional way". Proper detailing is necessary to ensure that the parts will function as expected after manufacture. Incorrect or unnecessarily tight tolerances can lead to high manufacturing costs (Menin et al. 2012).

1.4 An MBD model is not ambiguous

If not enough views and cross-sections are given, a 2D drawing is ambiguous (see Figure 1.3). The 3D MBD model, on the other hand, can be freely rotated by the user to inspect the model from any angle and, as such, avoids any ambiguity.

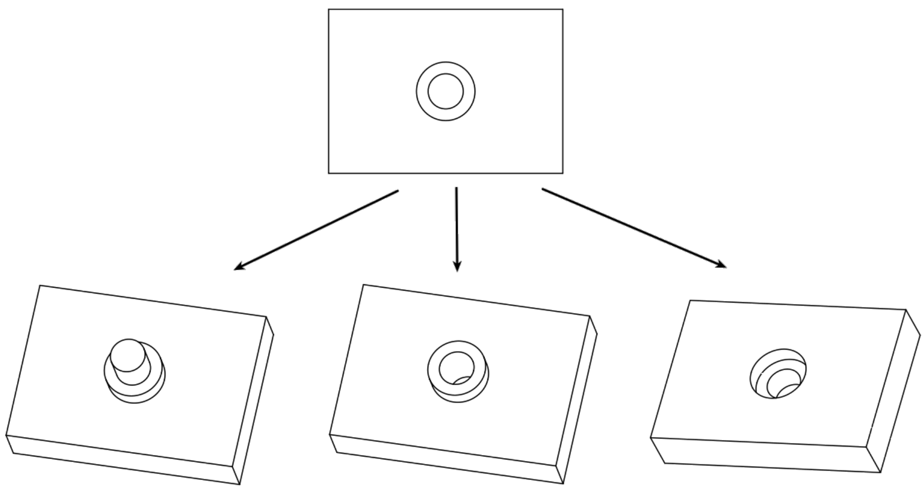


Figure 1.3: Multiple interpretations possible with one view

1.5 An MBD model is always up-to-date

"Up-to-date" can refer to three things:

- product geometry
- product annotations
- availability of the latest version

1.5.1 Product geometry

An MBD model is intended to be more up-to-date than a 2D drawing. However, this is not always the case. As stated before (see **??** on page **??**) there are four different steps for the transition to MBD. In case of the first step, where only 2D drawings are used, this assumption is entirely correct because there is no 3D model. In Table 1.1 this is represented by a minus sign for 2D drawings with no connection to a 3D model. As for steps two and three, it is a more nuanced story. In this case, the views are derived directly from the 3D model. Most CAD systems follow the principle of "concurrent engineering". "Concurrent engineering" means that the different phases of the design process take place almost simultaneously (see Figure 1.4) (Sohlenius 1992). There are two possible situations. The first is that the derivation of the 2D views from the 3D model is something that only happens when the 2D views are created. If a change is then made to the 3D model, the views are not automatically updated. This is not true concurrent engineering. In this case, there is no guarantee that the 2D views are up to date with the current state of the 3D model. In Table 1.1, this is indicated by a minus sign in the row "2D drawing - No true concurrent engineering". A second possibility is that there is a synchronisation of the 2D views with the 3D model from which they are derived. This means that if the 3D model changes, the 2D views automatically change too. This is true concurrent engineering. In this case, however, there is assurance that the 2D views are up-to-date with the current state of the 3D model. What remains unclear is whether the 2D drawing still contains the correct projection views to enable correct interpretation of the geometry after a change to the 3D model. For this reason, the row "2D drawing - True concurrent engineering" is given a +/- sign. In a true MBD model everything is directly linked to the 3D model, as 2D drawings are no longer used. So the product geometry is always up to date. This is why the row "MBD" has a plus sign.

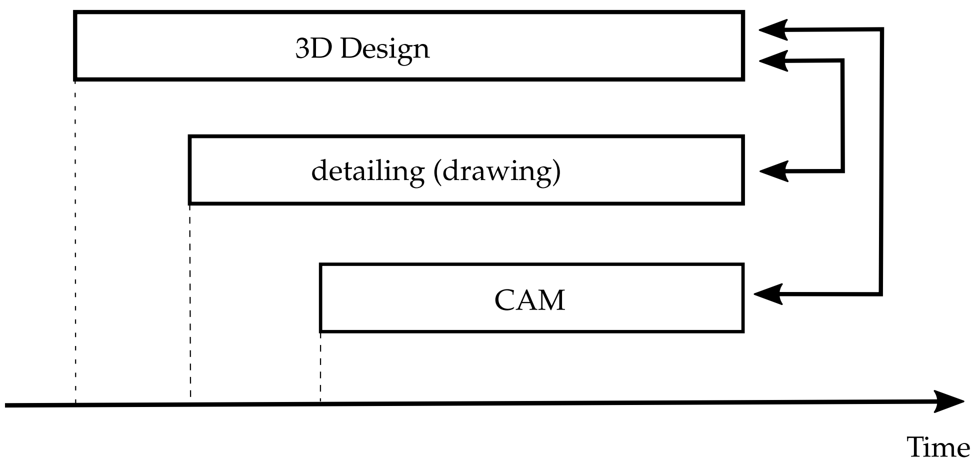


Figure 1.4: Design process in Concurrent Engineering

Table 1.1: When are MBD and 2D drawings always up-to-date in terms of geometry

		Geometry
2D drawings	No connection to a 3D model	-
2D drawing	No true concurrent engineering	-
	True concurrent engineering	+/-
MBD	2D drawings no longer used	+

1.5.2 Product annotations

Before going into the claim that an MBD model is always up-to-date as far as annotations are concerned, a distinction must be made between the different methodologies that can be used to create these annotations, on the one hand, and between the specific types of annotations in question, on the other.

As for the methodologies, there are two different methods that can be applied separately or in combination. In the first method, dimensions are derived directly from the dimensions used to create the features with which the CAD model is built. In the second method, dimensions are added to the model independent of those used to build the model. Many CAD systems allow the designer to specify specific tolerances for a dimension when sketching (Figure 1.5) or creating a feature (Figure 1.6), and to specify a general tolerance that is automatically applied to a dimension if no tolerances other than the general tolerance are assigned to that dimension (Figure 1.7).

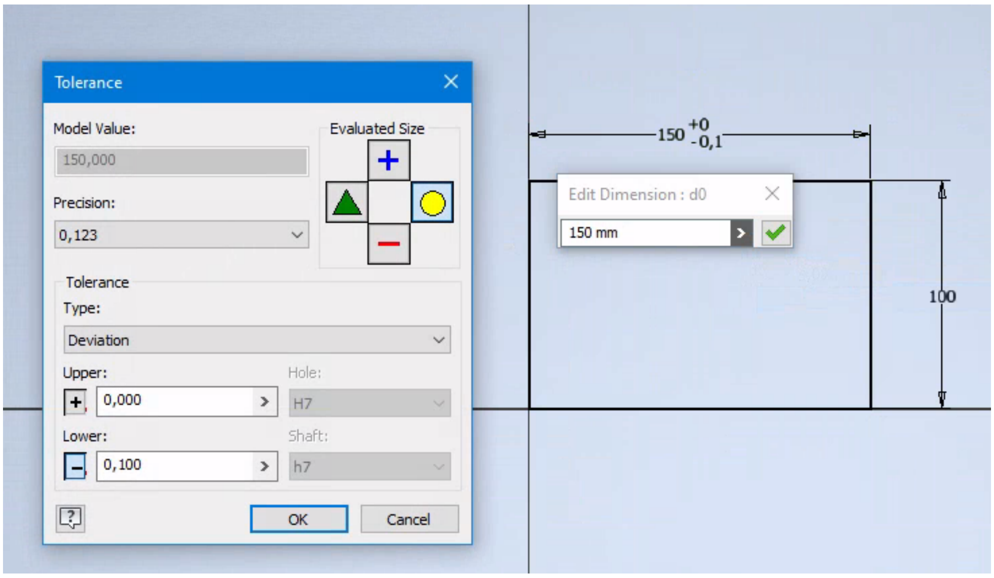


Figure 1.5: Assigning tolerances in sketcher (Inventor 2022)

When dimensional tolerances are incorporated into the creation of the CAD model and stored within the mathematical CAD model there is no need to place additional dimension annotations in the model. If necessary, they can still be derived from the internal dimensions. As the MBD model relies solely on the 3D model it is in this case always up-to-date. For this reason, the cell “MBD - Derived from features” in Table 1.2 contains a plus sign. In the case of 2D drawings, the dimensions must always be added to the projection views by an additional manual action by the designer, even if they can be derived from the feature dimensions. Some CAD systems can derive the dimensions automatically from the feature dimensions and place them on the projection views, but verification by the designer of their correct placement remains necessary.

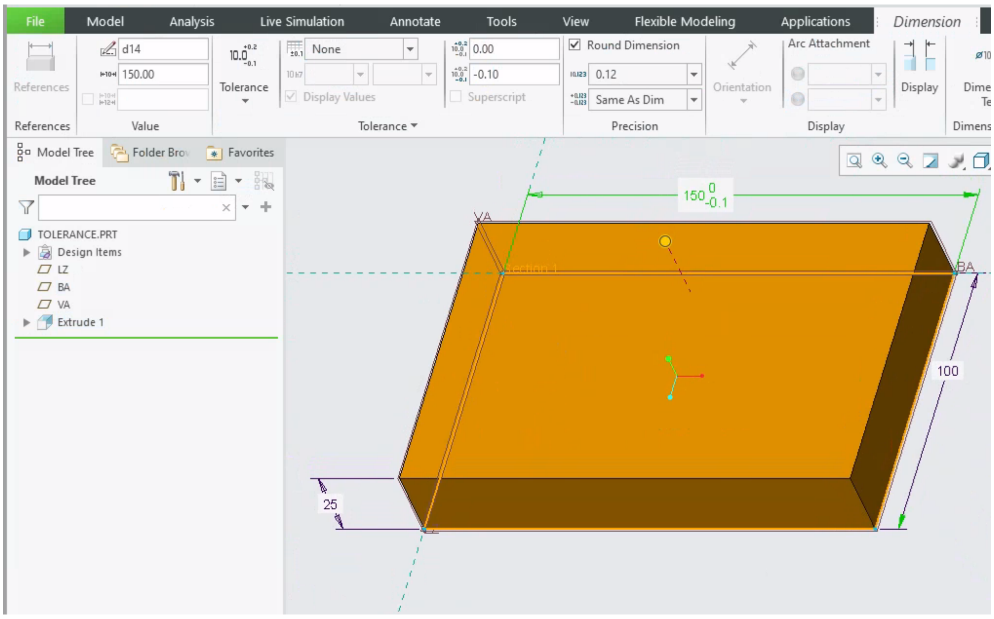


Figure 1.6: Assigning tolerances to feature dimensions (Creo 8)

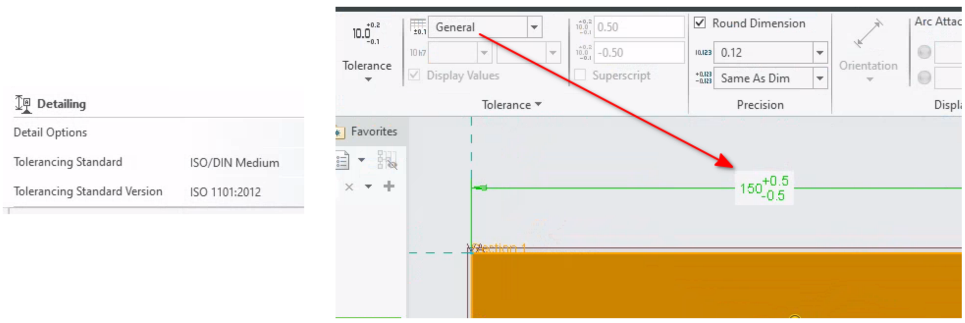


Figure 1.7: Specification of general tolerance in PTC Creo 8

For this reason, the cell “2D Drawing - Derived from features” in Table 1.2 contains a +/- sign.

When the dimensions and its tolerances are independent of how the model was built, namely when they are added afterwards, a stakeholder (designer, manufacturer, etc.) who consults the model can never be sure the MBD model or the 2D drawing is up-to-date. For this reason, the cells “MBD - Created independently from features” and “2D Drawing - Created independently from features” in Table 1.2 contain a minus sign.

As for the specific types of annotations, a distinction must be made between dimensions (linear, angular, radial, etc.) on the one hand and GD&T, symbols, notes on the other hand. In the latter case, these are not dimensions, so they cannot be created by deriving them from dimensions used to constrain sketches and features to build the model. As a result, GD&T, symbols and notes always have to be created independently of the model's construction (Rinos et al. 2021). This means a stakeholder (designer, manufacturer, etc.) is never sure whether the version that can be consulted, be it a 3D MBD model or a 2D drawing, is really up-to-date. For this reason, the cells “MBD - GD&T, Symbols, Notes” and “2D Drawing - GD&T, Symbols, Notes” in Table 1.2 contain a minus sign. Independently is used in the context of the CAD model can be constructed in the CAD system without them. Of course, entities belonging to the CAD model, such as edges and surfaces, can be used within the creation of this type of annotations.

Table 1.2: When are MBD and 2D Drawings always up-to-date in terms of annotations

	Dimensions		GD&T, Symbols, Notes
	Derived from features	Created independently from features	
MBD	+	-	-
2D Drawing	+/-	-	-

1.5.3 Availability of the latest version

Delivering the most up-to-date, the latest version, of a design to the different stakeholders depends on how a company is organised, for example by using PDM/PLM systems (Schleich et al. 2018). PDM and PLM are the acronyms for “Product Data Management” and “Product Life cycle Management” respectively. PDM and PLM are both version control systems, with PLM adding additional functionality such as workflow management (Huhtala et al. 2012) (see Figure 1.8).

Table 1.3: Availability of latest version

	Up-to-date
MBD (use of PDM/PLM)	+
2D Drawing (use of PDM/PLM)	+

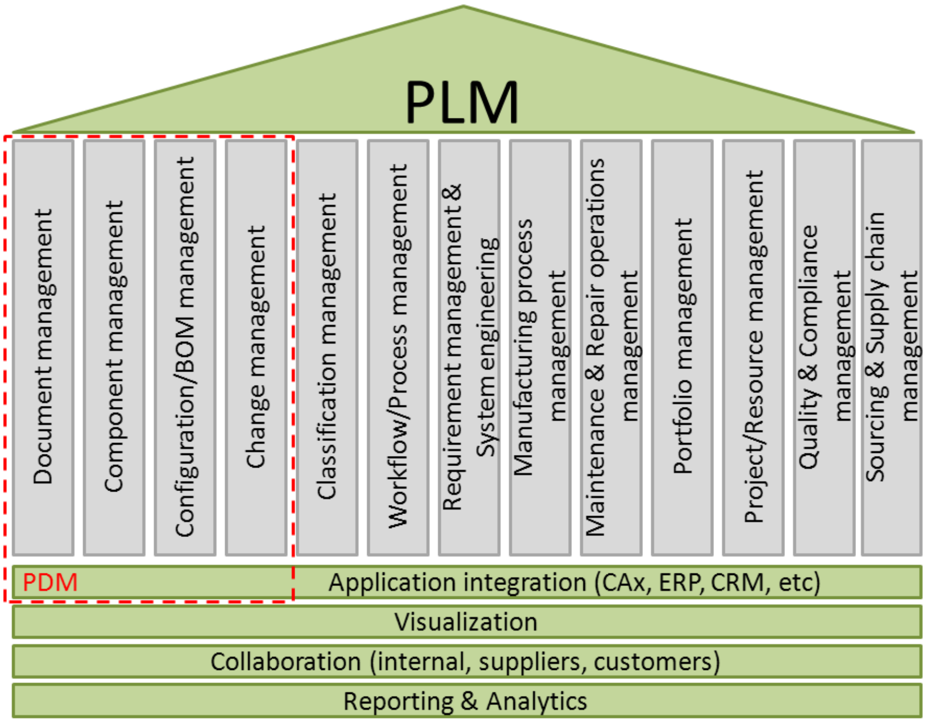


Figure 1.8: What comprises PDM and PLM (Huhtala et al. 2012)

2 Literature Review: Part II - MBD and geometry

2.1 Introduction

In the MBD philosophy, the 3D CAD model is the authority for passing on information to the various stakeholders (Beckers et al. 2016). This means that only the dimensions that deviate from the standard tolerance assigned to the model are explicitly placed on the 3D model (Agovic et al. 2022, p. 3). All other dimensions are completely determined by the geometry of the 3D model. This raises the question of how accurately this geometry can be handled and transferred. Gerbino 2003 states 'The most critical problems in data exchange are the different internal mathematical representation schemes and the internal accuracy of the geometric definitions in the modelling kernel of the various CAD systems'. This chapter discusses the most commonly used types of accuracies in CAD systems and examines their impact on creating geometry. The effect on the transfer of geometry between different CAD systems will be discussed in Chapter 6, which is dedicated to neutral exchange files.

2.2 Accuracy

2.2.1 Absolute accuracy

Absolute accuracy is the smallest distance between two points at which the CAD system can still distinguish the two points as individual points (Gerbino 2003). It is called absolute because the value of the accuracy is independent or absolute with respect to the size of the model. Sometimes absolute accuracy is also referred to as end-point tolerance (Steinbrenner et al. 2001). The following example will clarify this. If the units are mm, an absolute accuracy of 0.01 means that two points that are 0.01 mm apart can still be recognised as two individual points. Two points that are 0.005 mm apart will be considered to coincide. Some CAD systems allow the designer to change the accuracy used to build a model (see Figure 2.1 and Figure 2.2). If a feature creates an edge that is smaller than the set absolute accuracy, this feature will fail and an error message will be displayed. For one of the projects in my former research group, a mould flow analysis had to be carried out on a Panasonic mobile phone casing. There were roundings on this casing with a radius of 0.1 mm. This radius will be used in further examples to demonstrate the effect of the absolute accuracy set. Figure 2.3 shows a model in PTC Creo with an assigned absolute accuracy of 0.2 mm which causes the creation of a round feature with a radius of 0.1 mm to fail. Figure 2.4 shows a model in CATIA v5 with the design limits set to Large range which results in an absolute accuracy of 0.1 mm which causes the creation of a round feature with a radius of 0.1 mm to fail.

Ideally the absolute accuracy used by a CAD system to construct a model internally should be exactly the same as the accuracy used to store the model in an external

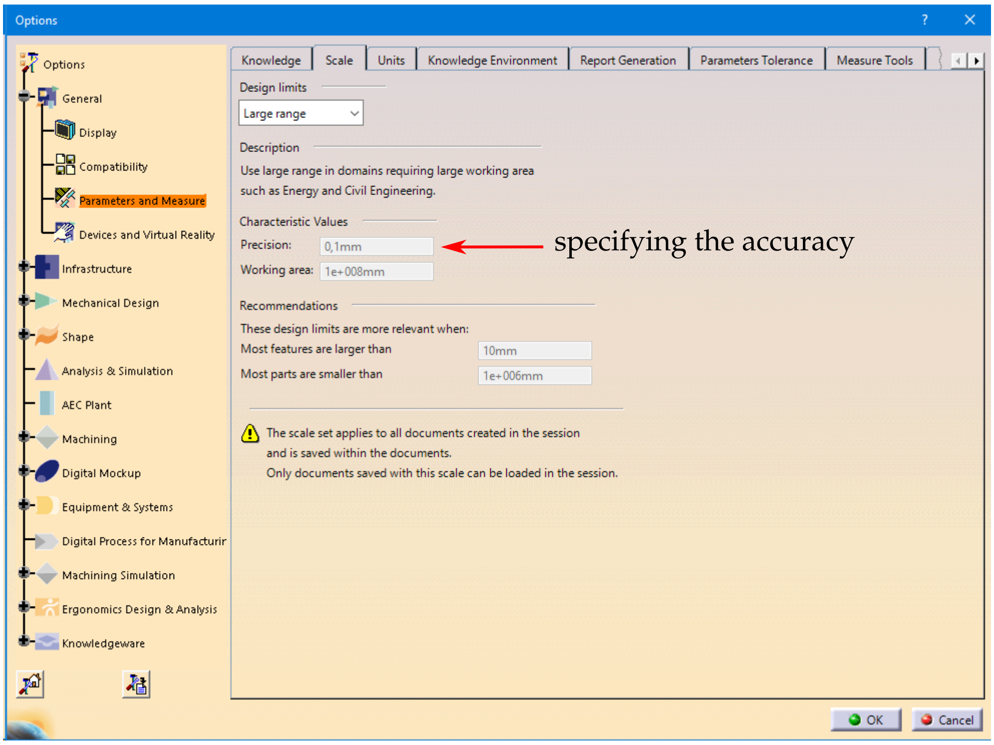


Figure 2.1: Specifying the accuracy in CATIA v5

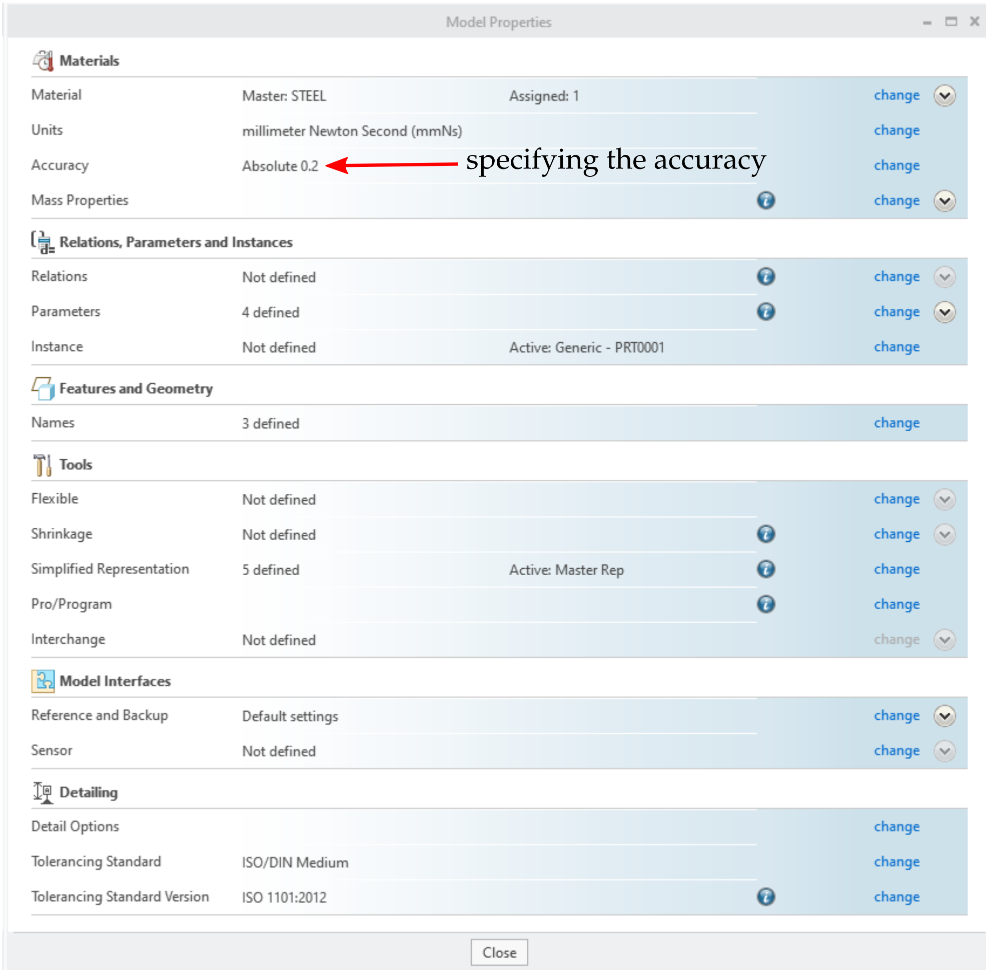


Figure 2.2: Specifying the accuracy in PTC Creo 8

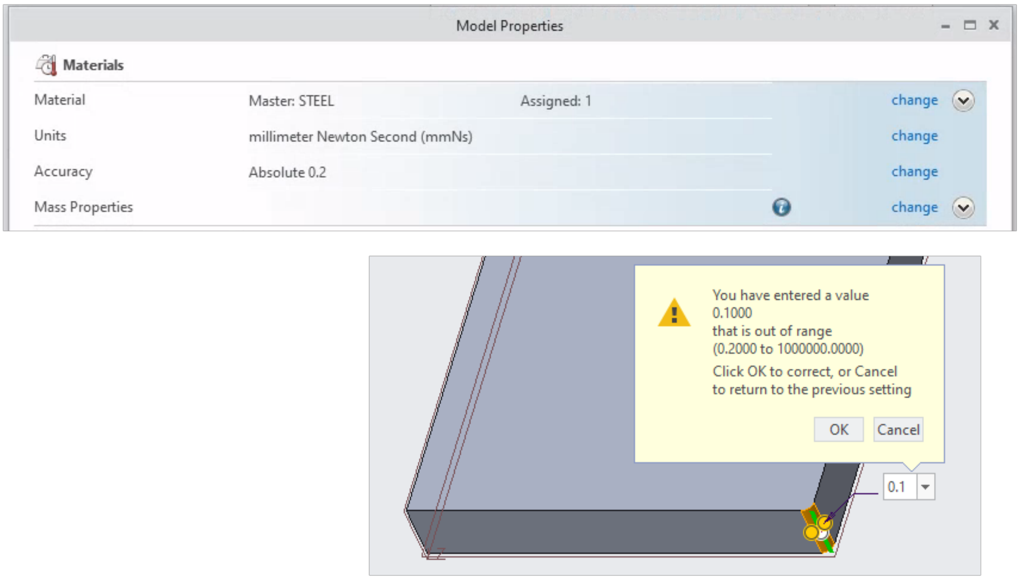


Figure 2.3: feature fails in PTC Creo due to an incorrect absolute accuracy

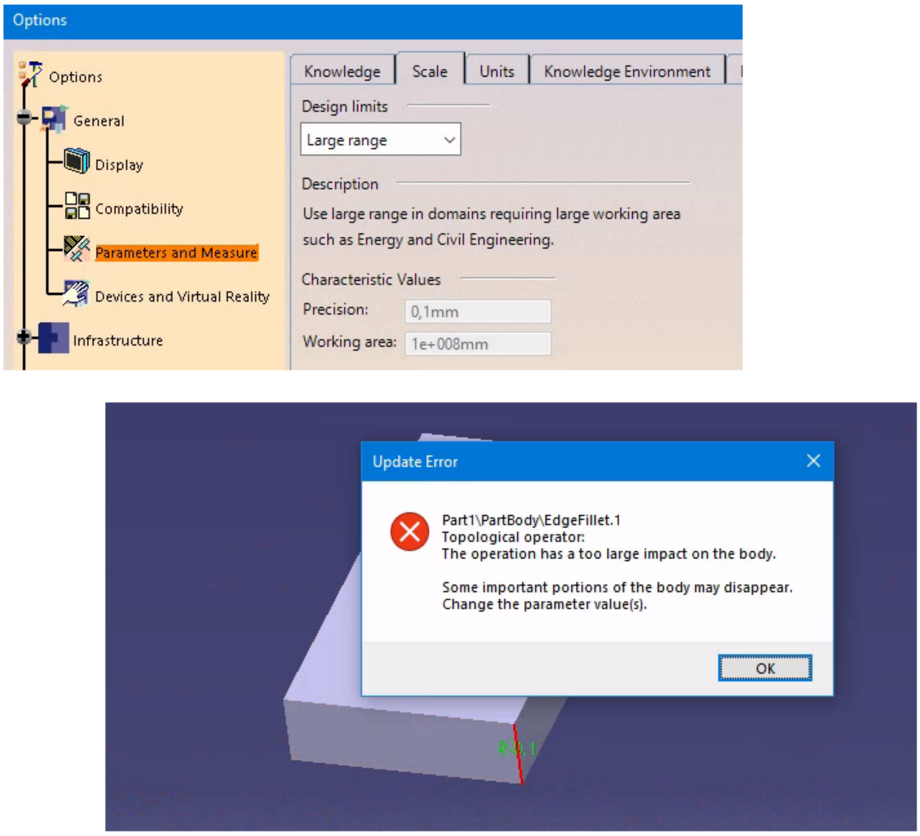


Figure 2.4: feature fails in CATIA v5 due to an incorrect absolute accuracy

format like e.g. IGES or STEP. As will become clear in Chapter 6, which is dedicated to neutral exchange files, this is not always the case. Very few CAD systems have a one to one relationship between the internal accuracy used in the native model and the accuracy of the external format.

Mixing components with a different absolute accuracy in an assembly can lead to problems (Gerbino 2003). Depending on the CAD/CAM package used and its settings, assembling parts with different accuracies can effect the accuracy applied in the assembly model itself. This can cause a cutting operation in the assembly model to fail because, for example, a resulting edge is smaller than the active accuracy. The fact that different absolute accuracies are used often occurs when the parts are designed by different companies where each company uses its own standard with an associated accuracy. The problems that may arise from this can be of various kinds. An example is a hole through two parts mounted on top of each other (Figure 2.5).

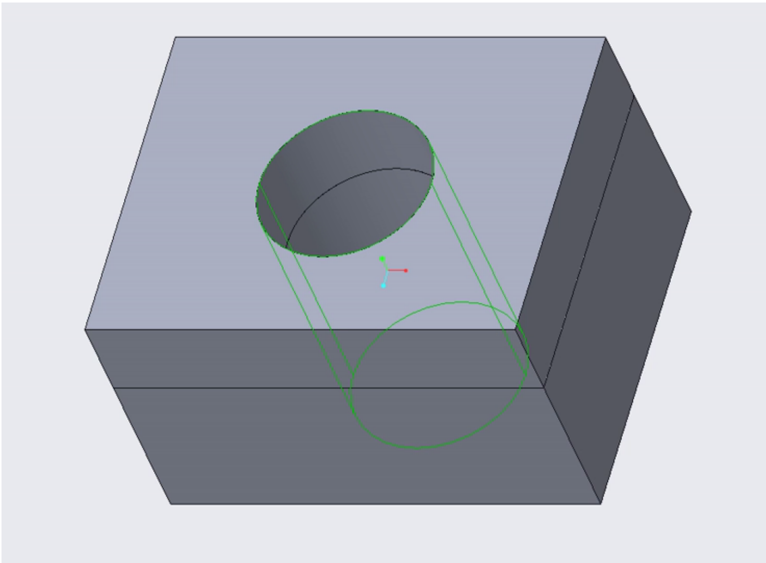


Figure 2.5: hole through two parts, created in PTC Creo 8.0.4.0

It may be the intention of the designer that this hole is made only after the two parts have been assembled. This is often the case in welded assemblies where holes are drilled after the parts are welded together. In this case, the hole may only exist at the assembly level and must be dimensioned at this level. It may also be the intention that this hole is made in one of the parts before and the hole in the other part after assembly. In that case, the hole must exist at the level of the assembly and at the level of the component and must be dimensioned at both levels. However, it may also be intended that this hole is in both parts before they are assembled. In that case, the hole must exist at the level of both parts and must be dimensioned at these levels. If the absolute accuracy is not the same for all the CAD models involved (the assembly and the two parts), this can lead to erroneous results. An assembly of parts with different accuracies is something that sometimes happens when a design involves multiple companies or uses parts from suppliers' CAD libraries. Each company has it own standards and settings for the accuracy of a CAD model. The possibility of erroneous results is illustrated by another example. Consider an assembly consisting of two parts, where the assembly itself and one part have the same absolute accuracy, 0.01 mm, and the second part has a different absolute accuracy, 0.5 mm (Figure 2.6). These accuracies have been specifically chosen to demonstrate the resulting problem as clearly as possible.

When a cutting operation is created in the assembly and applied at part level (Figure 2.7), it will only be executed within one of the two parts, namely the part with an absolute accuracy of 0.01 mm. Because of the different absolute accuracy the cutting

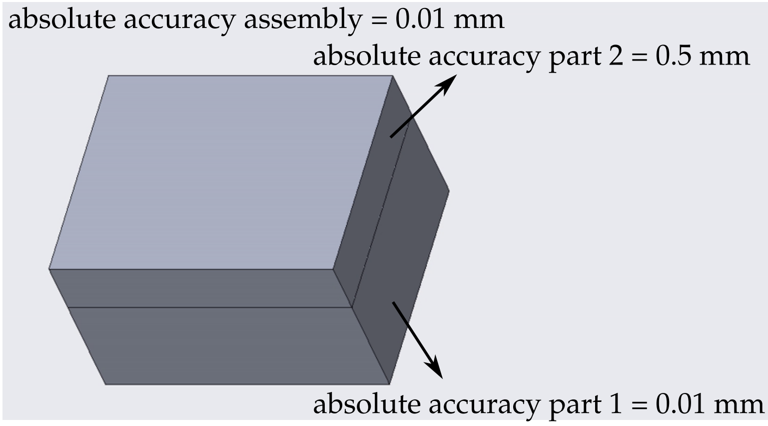


Figure 2.6: Assembly with multiple absolute accuracies,
created in PTC Creo 8.0.4.0

operation results in an edge size that can be detected in the part with an absolute accuracy of 0.01 mm and is too small for the part with an absolute accuracy of 0.5 mm. No warning message or message to indicate an error is issued by the CAD system. It is up to the designer to verify everything is in order.

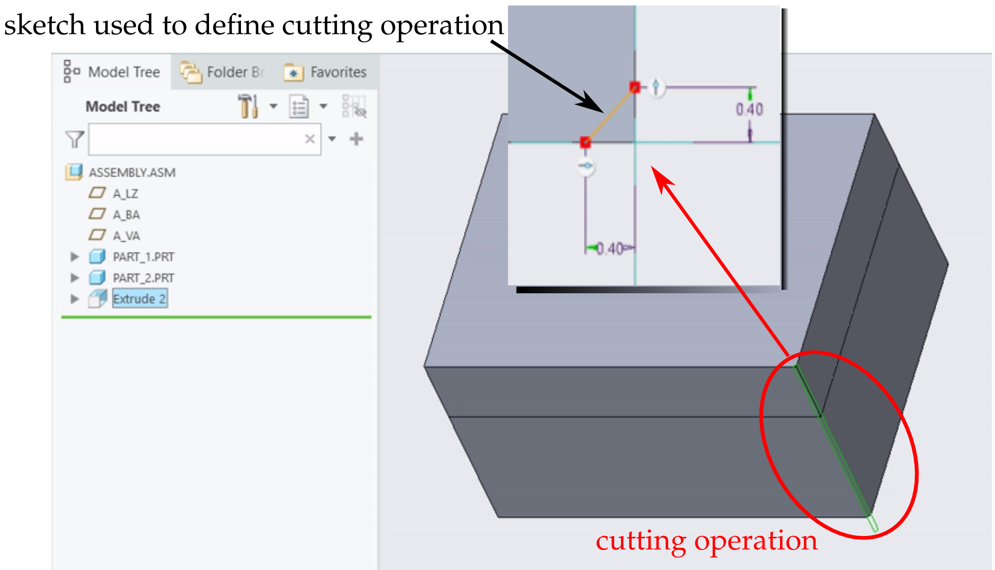


Figure 2.7: Creation of a cut in the assembly with size 0.4 mm

This problem seems to be an artificial, manufactured problem that does not exist in reality. This is not the case. A real-life example of where this type of problem occurs is when working out the cavity split of a plastic mould for injection moulding complex products. This can result in very small edges which, if cut during the assembly of the mould, where mould parts can have different accuracies, can lead to failure of the model.

Some CAD systems like CATIA don't allow the mixing of parts with a different absolute accuracy in an assembly (Dassault Systèmes 2022) while others like PTC Creo Parametric do.

2.2.2 Relative accuracy

The term "relative accuracy" means that the corresponding absolute accuracy is relative to the size of the 3D model. In the past, this method was often used to speed up calculations of complex shapes. Since modern computers are much faster than they were 10 to 20 years ago, it is no longer advisable to use this accuracy, especially when working with mechanical parts and assemblies. Using it anyway could cause CAD models to fail.

Consider the following example. Figure 2.8 shows a mobile phone case[1]. The case was originally designed in CATIA v5. It was then opened in PTC Creo Parametric 8.0.5.0 using a default relative accuracy of 0.0012.

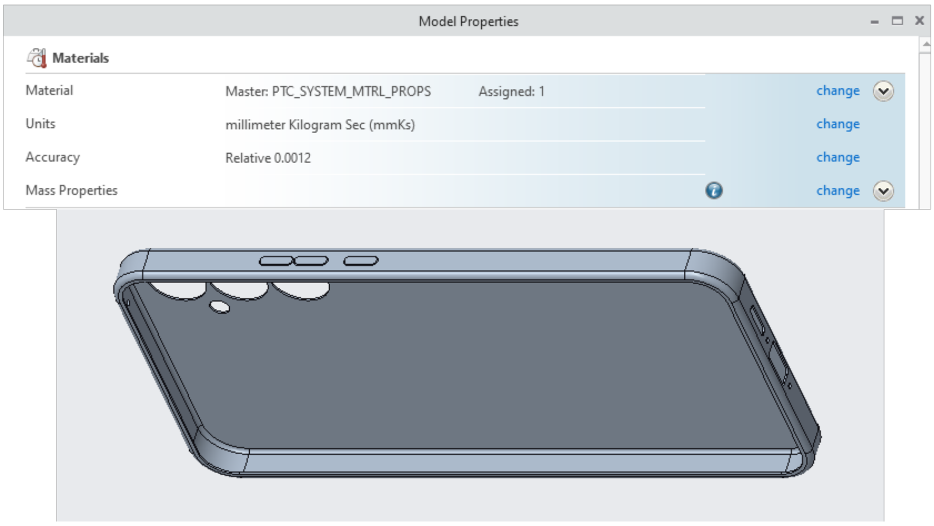


Figure 2.8: Model in PTC Creo with a relative accuracy of 0.0012

To determine the corresponding absolute accuracy, PTC Creo creates a bounding box around the product that is tangent to the product on all sides. Datum entities such as datum planes, datum axes, datum points and datum coordinate systems are also taken into account to calculate the bounding box. It then determines the length of the diagonal from one corner of the box to the other (Figure 2.9).

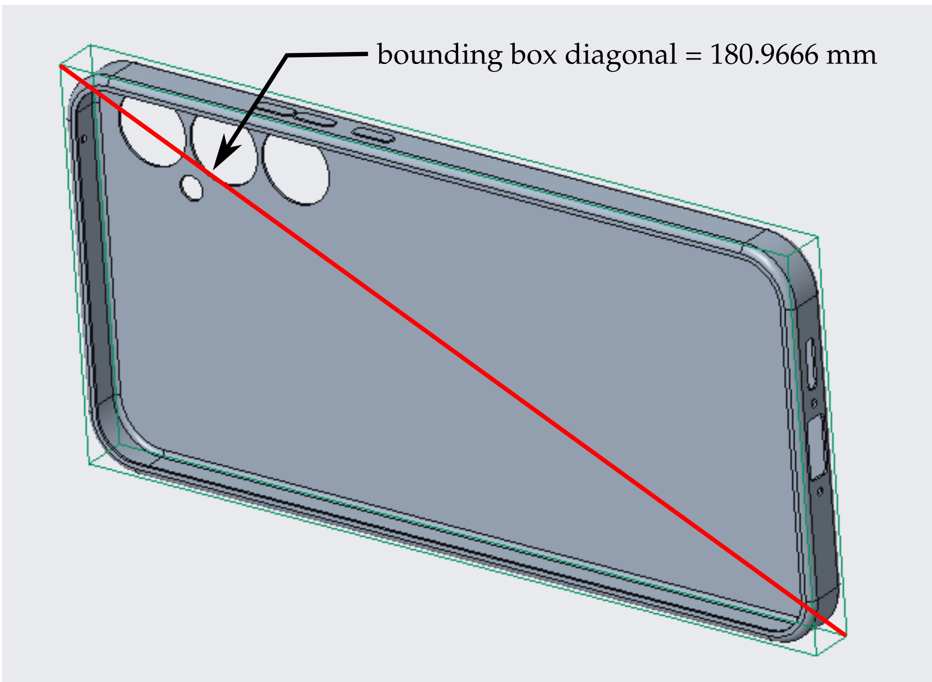


Figure 2.9: Model in PTC Creo with a relative accuracy of 0.0012

[1] Source:GRABCAD (https://grabcad.com/library/phone-case-79)

In this case the length of the bounding box diagonal is 180.9666 mm. According to the documentation[1] provided by PTC the relative accuracy A is calculated using the following formula

$$A < F \times s / D$$

Hereby

A : the recommended relative accuracy

F : a factor depending on the complexity of the geometry. The value can range from 3 (complex products) to 10 (simple shapes) and is determined by the CAD software using an unknown algorithm from PTC

s : minimum distance at which the systems considers entities to be separate or, in other words, the corresponding absolute accuracy.

d : diagonal of box whose sides are parallel to default coordinate system axes and which just encloses the part

The tests carried out during this thesis have shown that the following formula gives a sufficiently accurate approximation of the actual absolute accuracy

$$s = A \times D/10$$

For this particular example this gives

$$s = 0.0012 \times 180.9666/10 = 0.02$$

This can be easily verified by exporting the CAD model to IGES or STEP. Both exchange formats use only absolute accuracy. To verify this CAD model, it is exported to IGES (Figure 2.10).

```
PTC IGES file: phonecase0.igs                                          S      1
1H,,1H;,10HPHONECASE0,14Hphonecase0.igs,27HCreo Parametric by PTC Inc., G      1
7H2022124,32,38,7,38,15,10HPHONECASE0,1.,2,2HMM,32768,0.5,             G      2
15H20240614.162644,0.0180959,180.967,4Hjohn,7HUnknown,10,0,            G      3
15H20240614.162644;                                                    G      4
```

Specification of absolute accuracy in IGES file

Figure 2.10: Corresponding absolute accuracy specified in IGES file is 0.018 mm

Any change to the CAD model that results in a change to the bounding box diagonal will result in a change to the applied absolute accuracy. As a result, assembling parts of different sizes is the same as assembling parts of different absolute accuracies. So all the comments made in the absolute accuracy section apply here too.

2.2.3 Curve tolerance

Besides relative and absolute accuracy another additional type of accuracy is often used in CAD systems, the so-called "curve tolerance". This is the radius of a tube along an edge of two neighbouring surfaces (Cam 2000; Sangole 2000). If the edge of the other surface remains within this tube, the two surfaces are considered to be connected (see Figure 2.11). When curve tolerance is combined with the concept of absolute accuracy, it is possible to define a higher accuracy in addition to the general accuracy (absolute accuracy) at some transitions between surfaces. The use of two distinct terms, namely "accuracy" and "tolerance", can potentially lead to confusion. After all, both terms pertain to the concept of tolerance, which is defined as the permissible deviation.

Some CAD systems such as CATIA and Inventor allow the designer to specify a specific value for the curve tolerance when joining surfaces (Figure 2.12). This value may differ from the absolute value used to build the model. Other CAD systems like PTC Creo assume the curve tolerance is the same as the absolute accuracy when joining surfaces. They give an error when the value of the curve tolerance is greater than that of the active absolute accuracy (Figure 2.13).

[1] TPI 32869 (https://support.ptc.com/appserver/cs/view/solution.jsp?n=32869)

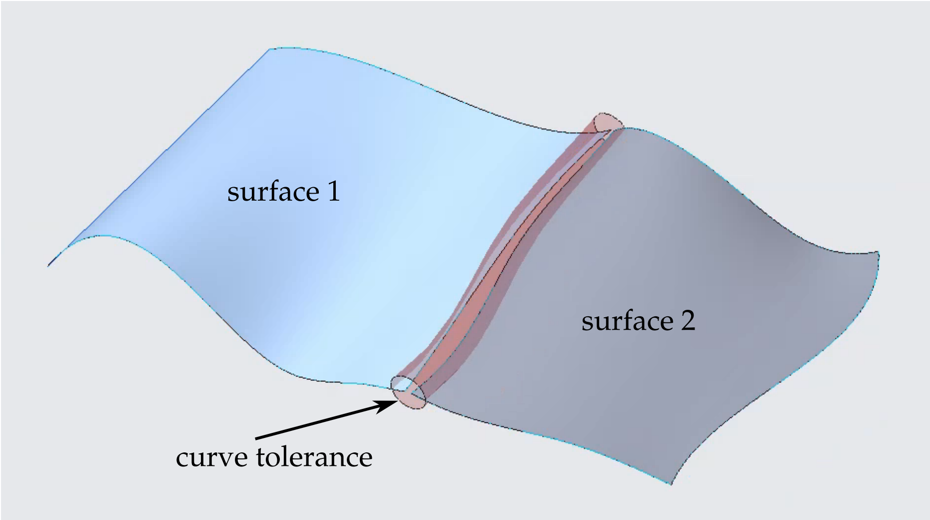
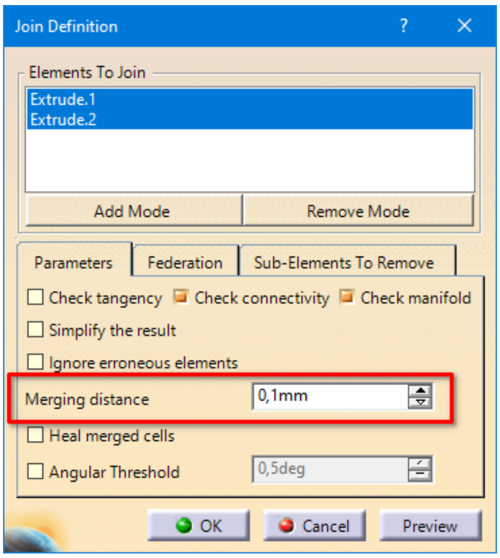
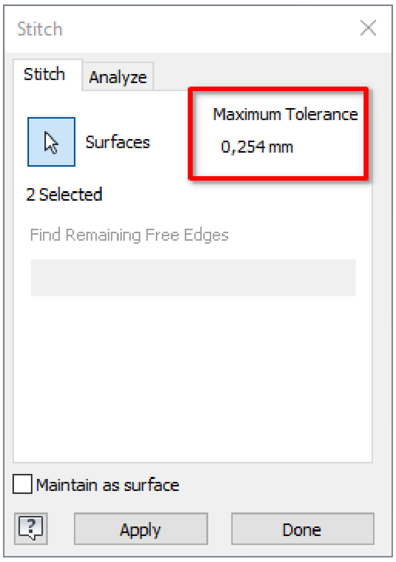


Figure 2.11: Curve tolerance



(a) Tolerance specified for joining surfaces in CATIA v5



(b) Tolerance specified for joining surfaces in Inventor 2022

Figure 2.12: Curve tolerance applied in joining of surfaces

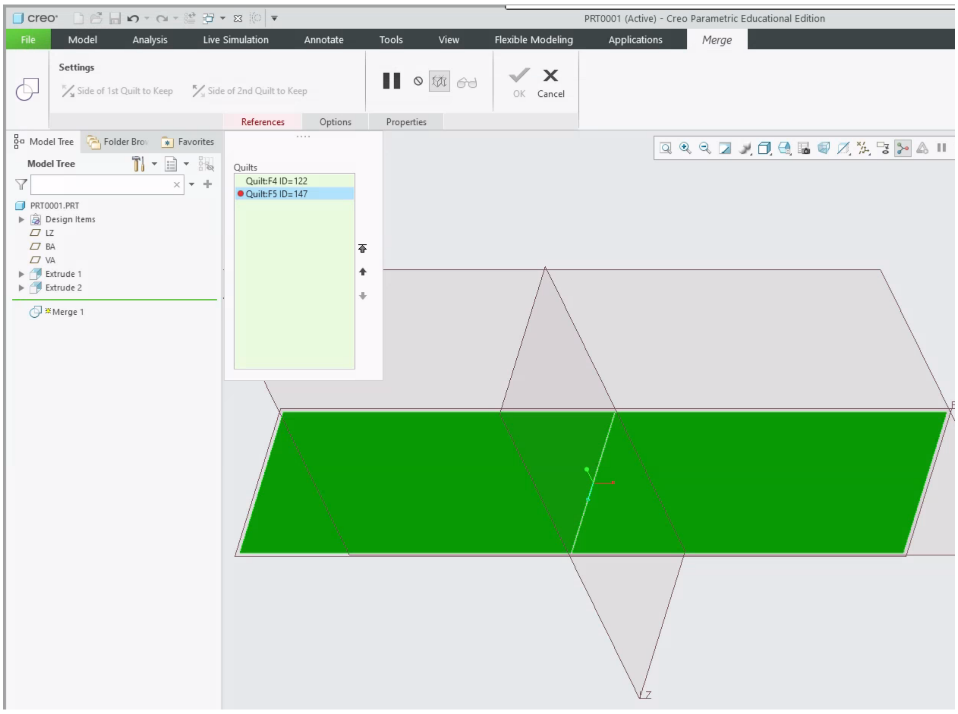


Figure 2.13: Failure in join operation because curve tolerance is greater than the active absolute accuracy

The most logical is that the "curve tolerance" has the same value as the "absolute accuracy" used within the CAD model. If this is not the case, it can cause problems when changes are made to the model or when the model is exported to a neutral format such as STEP. This will be discussed in more detail in Chapter 6, which is dedicated to neutral exchange files.

2.3 Mathematical model

A distinction must be made between geometry and topology. Geometry is the mathematical description of a component relative to a reference in space (Chern 1990). Topology describes the connectivity between components (Saxena et al. 2005). When applied to a beam, it can be defined as *width* $\times$ *height* $\times$ *length* (Figure 2.14). This is the geometric description of the beam. The shape of the beam can be described in several ways. The boundaries can be defined by 6 planes, the 6 planes can be described by 2 triangles each, ... (Figure 2.15) These are different topological descriptions of the beam. Within a CAD system there are currently three ways of defining topology. These are analytical, NURBS-based and based on subdivision modelling (Antonelli et al. 2013). The analytical method and the NURBS-based method are the most widely used (Antonelli et al. 2013).

2.3.1 Analytical

Analytical means that a model is described using analytic geometric entities such as points, lines, circles, planes, cylinders, etc. These are exact descriptions. This gives the impression that a form can only be defined in one unique way. This is not the case. As each CAD system uses its own kernel, different mathematical methods are applied to create the geometry. For example, a cylindrical form can be described by one CAD system using a single surface with a splitting edge (Figure 2.16), a single surface with no splitting edge (Figure 2.17), while another CAD system may do this by dividing the cylinder into two halves (see Figure 2.18) (Gerbino 2003). In each

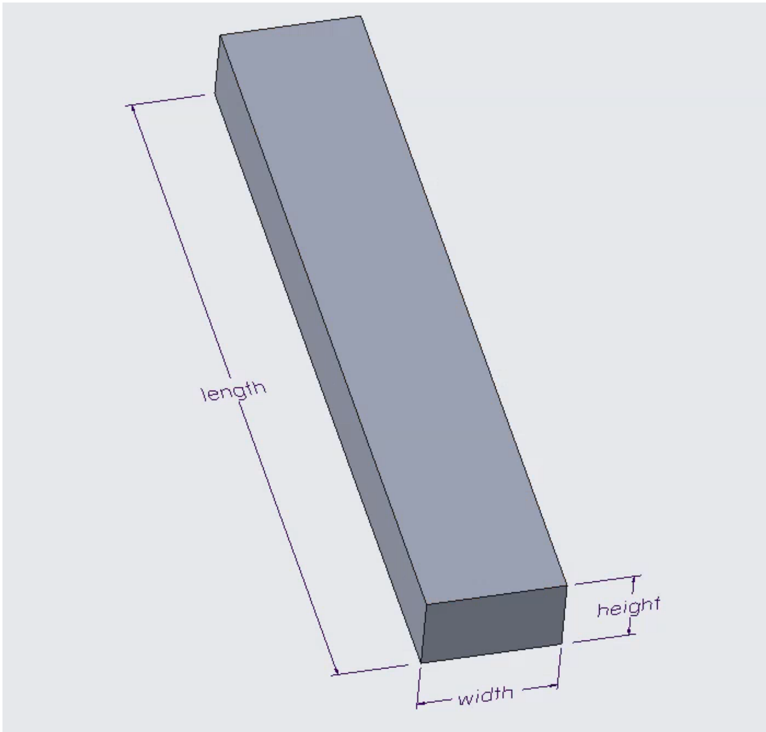
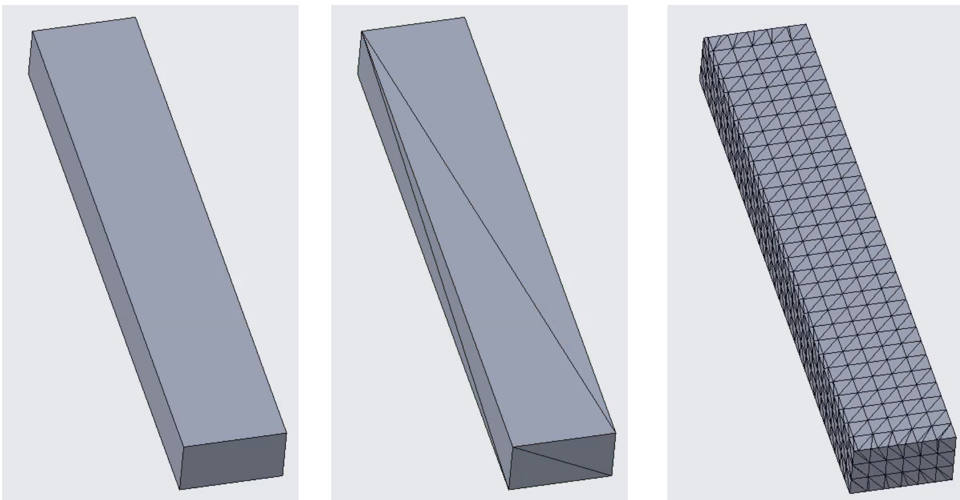


Figure 2.14: A geometric representation of a beam:
width × *height* × *length*



(a) beam defined by 6 faces

(b) beam defined by 12 triangles

(c) beam defined by a triangular mesh

Figure 2.15: Different topological representations of a beam

example, the cylinder was created by the extrusion of a circle. The intersection of two analytic geometric entities or the projection of one entity onto another results in a new analytic geometric entity if possible[1].

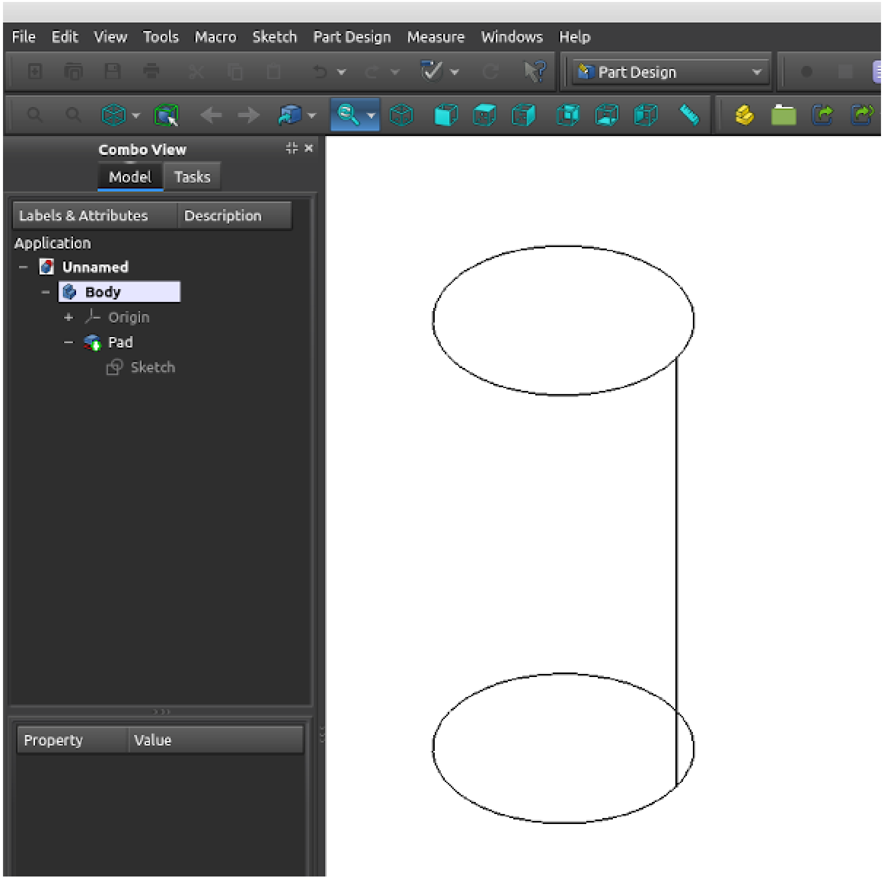
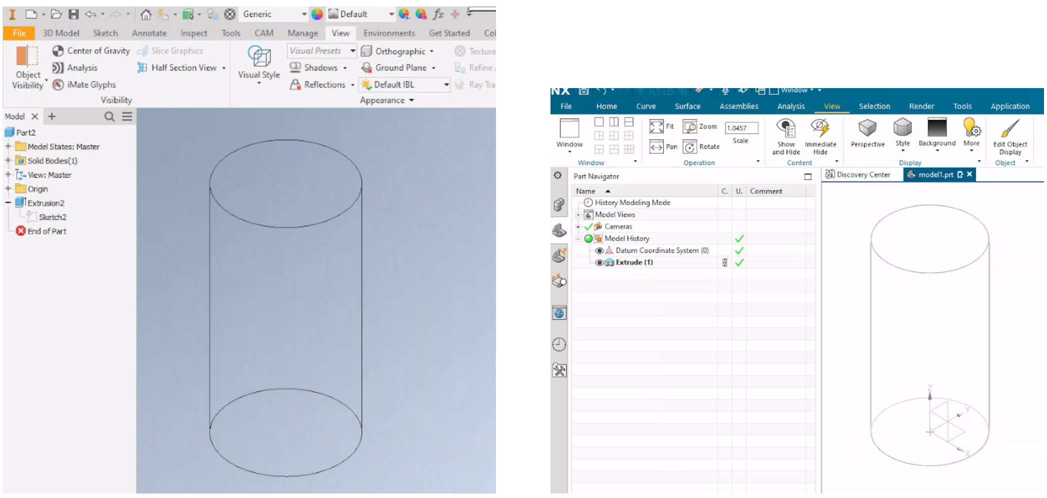


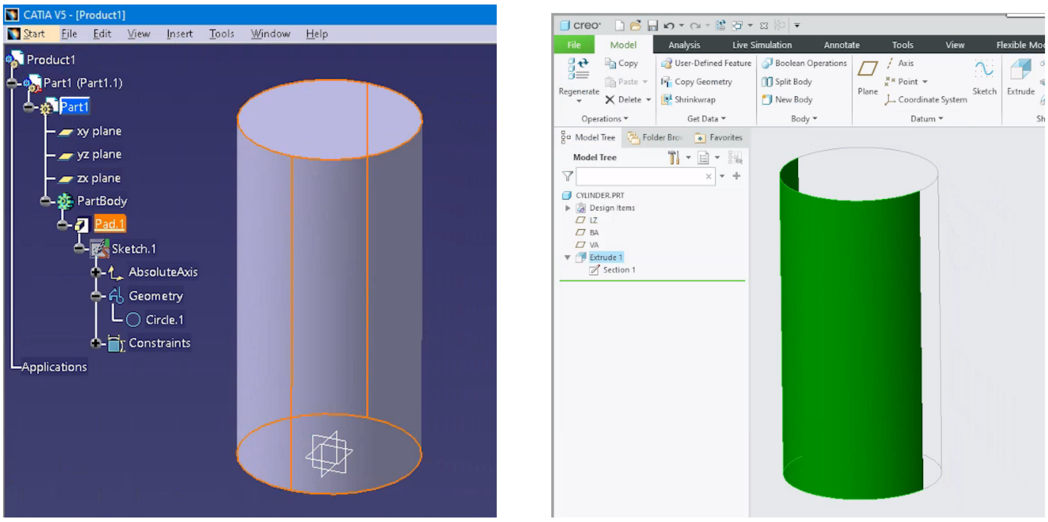
Figure 2.16: Cylinder created in FreeCAD
Cylinder is a continuous surface with a splitting edge



(a) Cylinder created in Inventor (b) Cylinder created in Siemens NX

Figure 2.17: Cylinder shell is a continuous surface with no splitting edge

[1] This was verified by creating an intersection curve between a plane and a cylinder in PTC Creo, CATIA (exported to STEP AP242), Inventor (exported to STEP AP242), Siemens NX (exported to STEP AP242). In PTC Creo the edge type of the intersection curve was checked to verify this was of type "arc" or type "spline".



(a) Cylinder created in CATIA v5 (b) Cylinder created in PTC Creo 8

Figure 2.18: Cylinder shell consists of two halves

2.3.2 NURBS

Based on NURBS, the geometry is described by curves and surfaces defined by splines. However, the splines are not always NURBS. They can also be B-splines and in some cases Bézier splines. In some CAD systems the degree of the spline polynomials is determined automatically, other CAD systems allow the user to determine the degree (see Figure 2.19, Figure 2.20 and Figure 2.21). The calculation of the intersection of two NURBS based entities or the projection of two entities where one of them is NURBS based is done iteratively whereby the accuracy is determined by the active absolute accuracy of the CAD model. The result is a non analytic geometric entity[1]. This can have a major impact on the accuracy with which a 3D model can be transferred from one CAD system to another (La Course 2001).

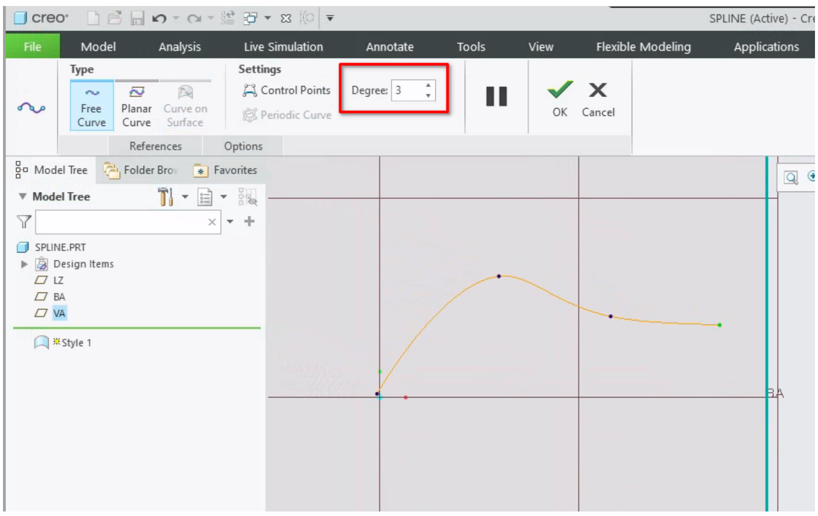


Figure 2.19: spline creation in PTC Creo

[1] This was verified by creating an intersection curve between a plane and a solid created by revolving a spline around an axis in PTC Creo, CATIA (exported to STEP AP242), Inventor (exported to STEP AP242), Siemens NX (exported to STEP AP242). In PTC Creo the edge type of the intersection curve was checked to verify this was of type "arc" or type "spline".

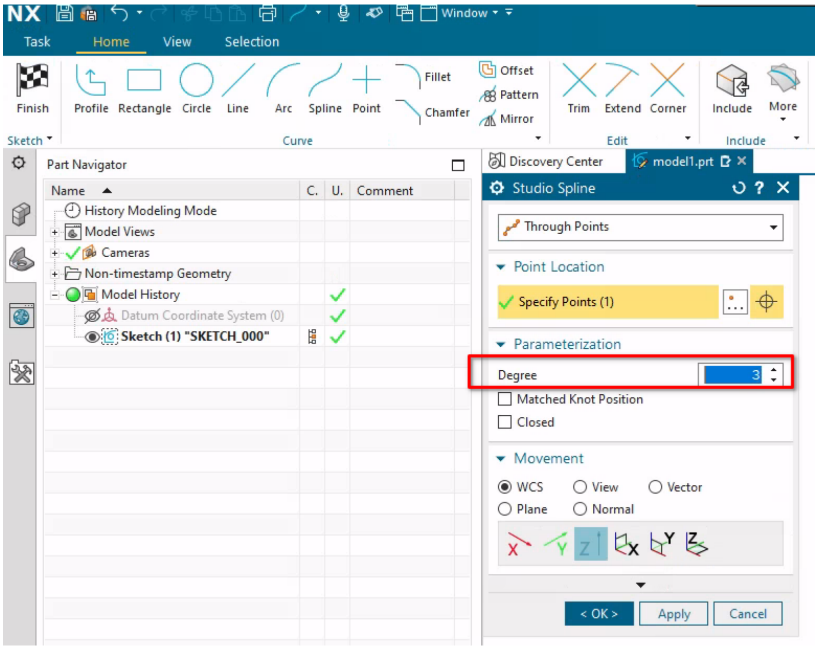


Figure 2.20: spline creation in Siemens NX

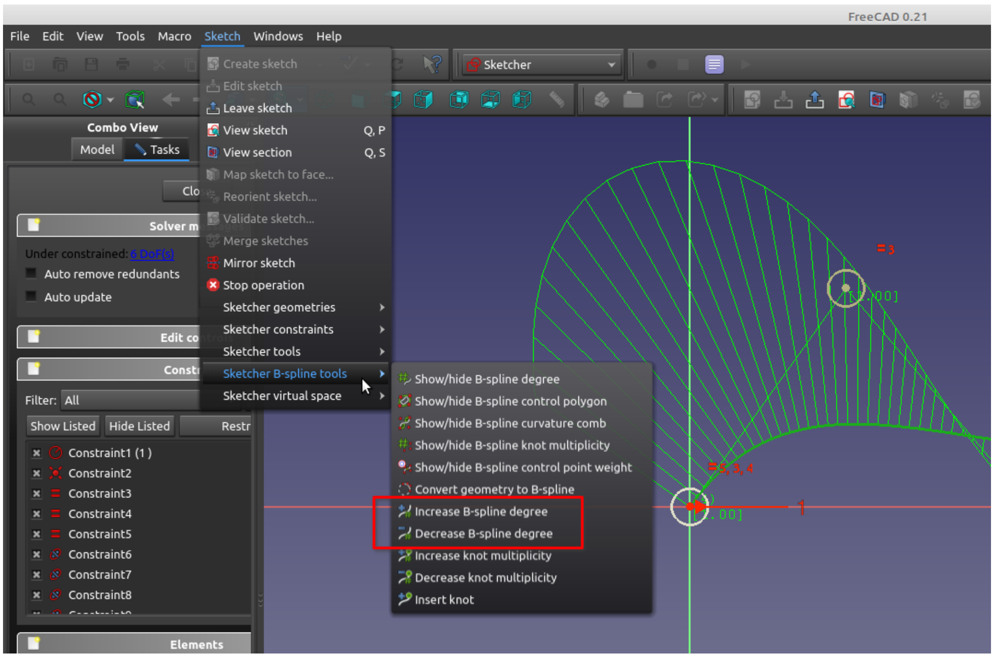


Figure 2.21: spline creation in FreeCAD

2.3.3 Subdivision modelling

Subdivision modelling is a special method for creating surfaces that is used for so-called freestyle or freeform modelling (Figure 2.22). It has been in existence for more than 30 years (Antonelli et al. 2013) but only recently has this method gained traction in the CAD world. It is mainly used for product design and reverse engineering (W. Ma et al. 2000).

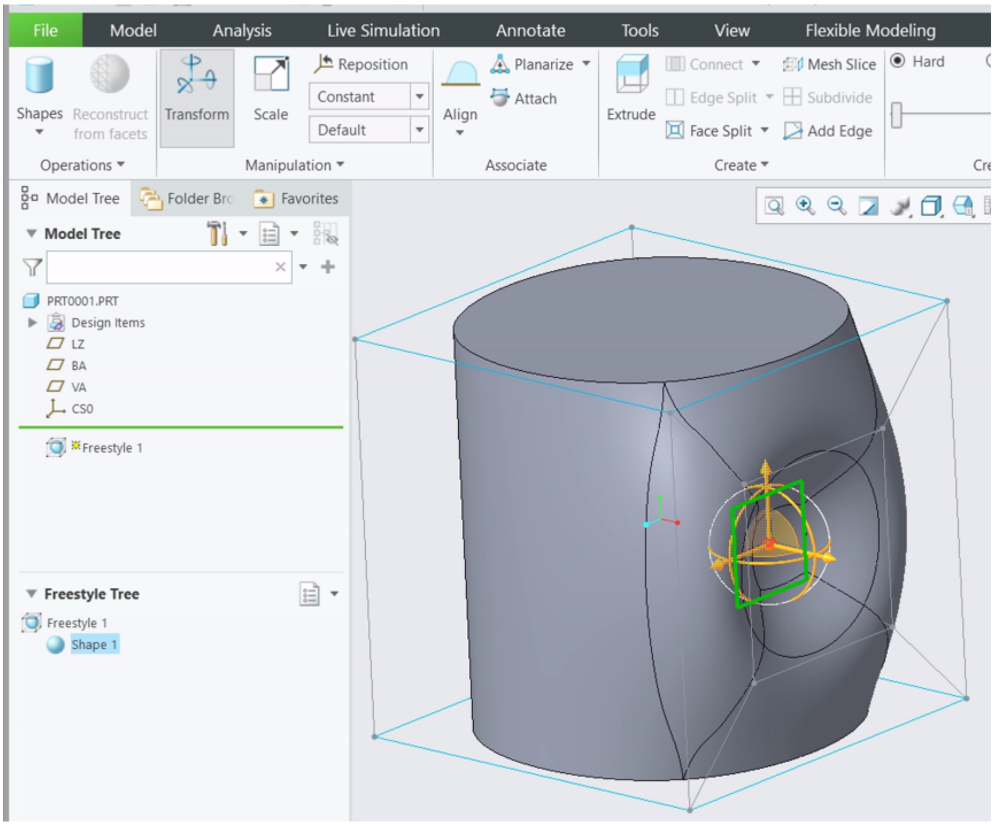


Figure 2.22: Freestyle module in PTC Creo 8.0.4.0

Each CAD system uses it own methods to implement subdivision modelling. Some systems use B-spline surfaces while others use modified NURBS surfaces (Antonelli et al. 2013). Together with the applied absolute accuracy, this can have a major impact on the accuracy with which a 3D model can be transferred from one CAD system to another.

2.4 Conclusion

There are few problems when every stakeholder involved in a project uses the same CAD system. However, this is rarely the case. Collaboration and interoperability are hampered by a multitude of CAD systems and associated CAD formats (Nnaji et al. 2004). In an article from 2004, Gallaher et al. mention a cost of $15.8 billion for US companies because of this. This has motivated the development of vendor neutral data exchange formats such as IGES, STEP (Lachenmaier et al. 2015). Gerbino 2003 argues that the different internal accuracies and internal mathematical representations are the cause of the most critical data exchange problems. Consequently, in order to minimise problems with these exchange formats, the designer must be well aware of the settings, properties such as applied accuracies and consequences of the mathematical formats used, of the CAD system (Gu et al. 2001). This means that the statement "the 3D model is the authority" isn't something that is achieved automatically just by using a 3D CAD system.

3

Literature Review: Part III - MBD and annotations

3.1 PMI

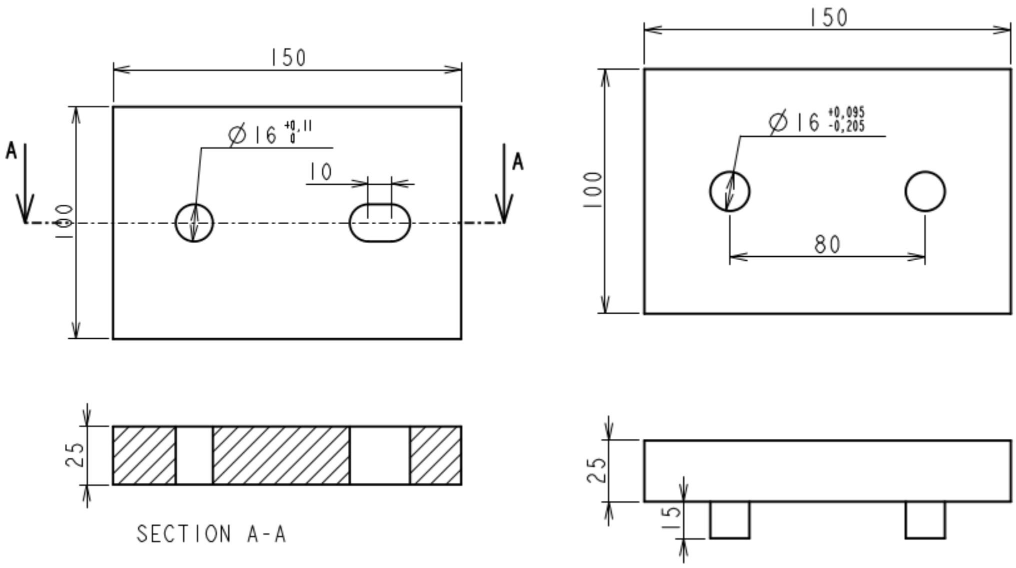
MBD and PMI (Product and Manufacturing Information) are two terms that are frequently mentioned in the same breath. PMI then often refers to annotations such as GD&T and dimensional tolerances applied to the 3D model (Feeney et al. 2015). However, PMI includes much more than just these annotations. Through research for this thesis, it has become apparent that when discussing PMI, one should distinguish between two aspects of PMI. The first aspect are the annotations, but also the references of these annotations and the metadata, which are the machine-readable parameters behind these annotations. The second aspect is the method used to create this PMI and which determines which information can be retrieved from this PMI.

3.1.1 Annotations

The most important aspect of annotations is the way they are displayed and stored in the CAD model. On first inspection, there seems to be no difference between the display of annotations in the different CAD systems. They can all be read and interpreted by a human user. However, this does not automatically mean that they can also be recognised and read by software. If they are just a graphical presentation where each annotation is some kind of symbol comprised of individual lines, they are called “presentation PMI” or “graphical PMI” (Lipman and Lubell 2015). If they can be read by software also described as machine-readable, they are called “representation PMI” (Hedberg et al. 2016). Machine-readability of an annotation relates to four points, namely the ability to determine the type, the content, the semantic references and special additional properties.

The annotation type

It is important to be able to determine the annotation type, because there is a hierarchy of types. Not all annotations are equally important. For example, there are dimensions that are necessary to manufacture a part (Figure 3.1) and there are for example dimensions that are only added as an additional means to verify an assembly, but which are not strictly necessary to build that assembly (Figure 3.2). Another example is a tolerated dimension in combination with a GD&T annotation (Figure 3.3). In MBD, dimensions that follow the general tolerance, which has symmetric tolerance fields, ar not explicitly annotated in the model. The dimension $20^{\,0}_{-0.2}$ has an asymmetric tolerance field that is different from the general tolerance. This makes it the highest priority dimension in the model. The tolerance zone assigned to this dimension is shown in Figure 3.4. The GD&T annotation can be seen as a superposition on top of the tolerated dimension, fine-tuning the dimensional requirements. The assigned tolerance zone is now reduced to a subzone with a height of 0.01 mm that must be within the zone defined by the tolerated dimension (Figure 3.5).



(a) Production dimensions on part 1 (b) Production dimensions on part 2

Figure 3.1: Dimensions necessary for production

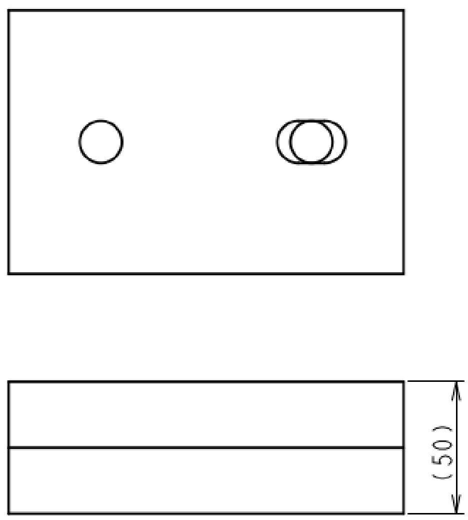
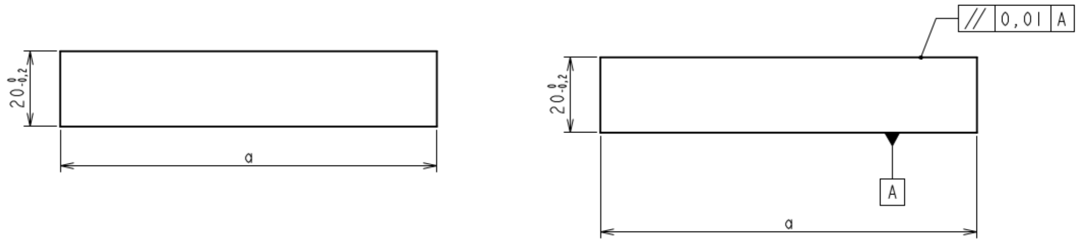


Figure 3.2: Reference dimension in an assembly

The annotation content

In order to make use of the annotations for all kinds of applications, such as the automatic generation of First Article Inspection Reports (Capvidia 2016) or the generation of programs for CNC coordinate measuring machines (CMM) (Fang et al. 2016), it is absolutely necessary to be able to read the content of the annotations and to determine which geometrical references (points, axes, edges, surfaces) they refer to. These



(a) Tolerated dimension (b) Production dimensions on part 2

Figure 3.3: Impact of hierarchy within annotations

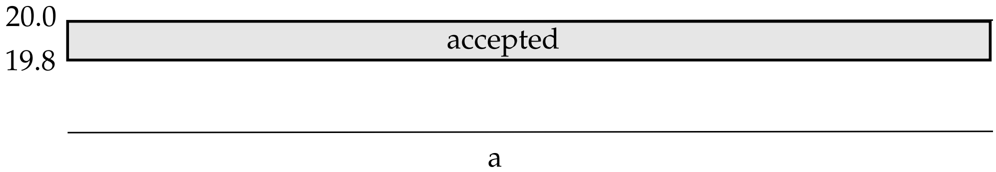


Figure 3.4: Assigned tolerance zone for the main dimension

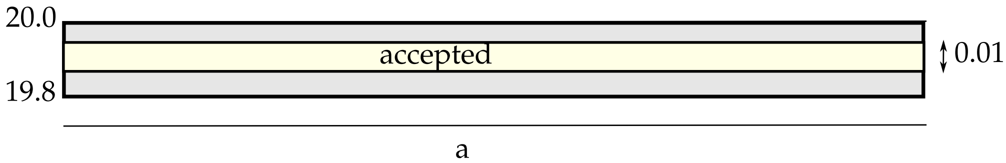


Figure 3.5: Accepted tolerance zone for the main dimension with GD&T added

references are referred to as "semantic references", where "semantic" means that they specify exactly what the annotation is referring to (Jian et al. 2023).

Since not all applications require the same information, the following section examines what exactly is needed for a particular type of application and, more specifically, for the creation of a "First Article Inspection Report" and for the generation of a measurement programme.

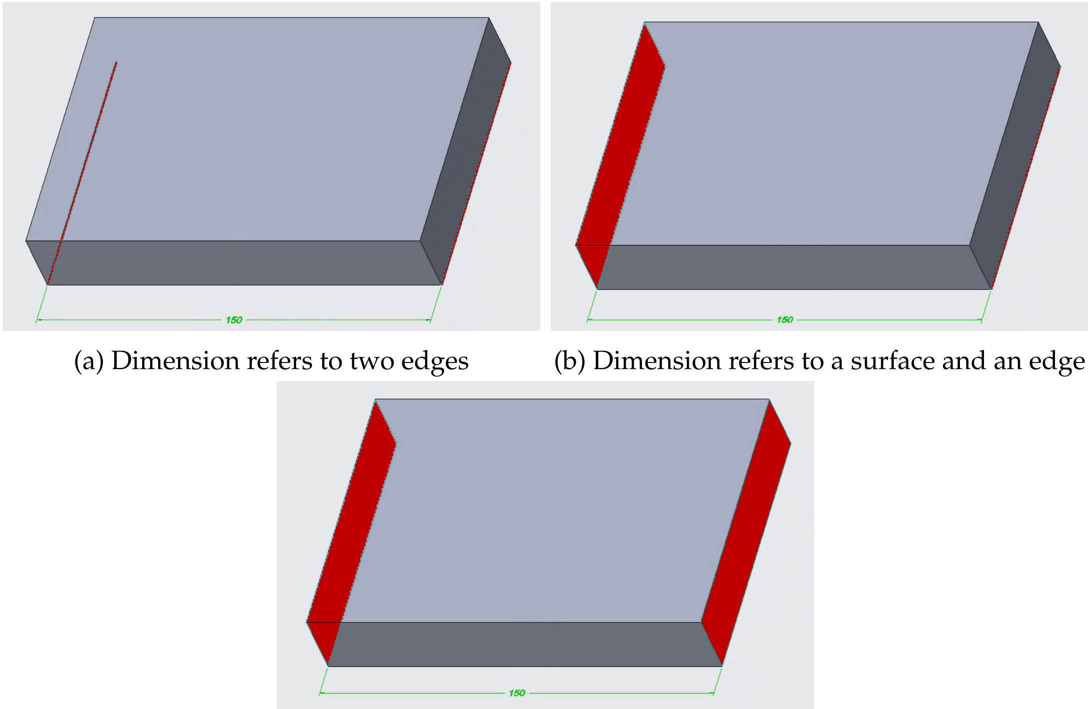
What is a "First Article Inspection Document"? Guthrie CAD/GIS Software 2022 describes a "First Article" as 'the first item that is manufactured using the same production processes as mass production by any subcontracted factory or supplier. This is carried out so that the client can validate how the supplier is capable of producing parts, and assemblies that meet all engineering and design requirements on mass scale.'

A "First Article Inspection Report", often abbreviated as FAIR, is a document listing all dimensions, tolerances, GD&T that a product must meet to be approved for the application it is designed for (Guthrie CAD/GIS Software 2022). Insight Team 2020 states that "First Article Inspections" usually cover specific industries such as aerospace and defence, automotive, electrical and electronics, and medical devices.

Over time, most companies in these sectors have developed their own methods for creating FAIRs. This made it difficult for these companies to exchange information and therefore work together. Therefore, in 2000, the SAE International organisation created a new standard, AS9102, which aims to standardise the FAIRs (Morris 2007; AIF 2010). Around 2004, a survey was conducted in the USA among suppliers to the aerospace industry to determine how much time they needed to prepare a FAIR in accordance with the AS9102 standard. Morris 2007 states that the median supplier size was fewer than 100 employees and also that 57% of the suppliers reported devoting at least one full-time person to first articles, while some 40% have more than one dedicated person. Preparing a FAIR is a very intensive manual activity, which includes listing all critical product dimensions, GD&T, notes and specifications, each of which must be given a unique identification number. If one is overlooked (what is called a "quality escape"), it can lead to defective products and assembly processes, thus contributing to high costs for resolving these and damage claims. Morris 2007 states that the price for a single major escape can easily exceed $1 million. As the aerospace supply chain consists of thousands of suppliers, compliance with AS9102 takes millions of working hours (Morris 2007). This means that anything that can help reduce the number of working hours required to prepare a FAIR can contribute significantly to reducing production costs. A whole range of software packages was developed to automate the drafting of FAIRs as much as possible. The software was able to recognise dimensions, GD&T, notes in a 2D drawing and their location within the drawing using OCR (an acronym of Optical Character Recognition). Examples of such software packages are QA-CAD, First Article Inspection, DISCUS Desktop, PDF Auto-Ballooning & First Article Inspection. Morris 2007 states 'The quality engineers who have applied these tools report productivity gains as high as 70%.' Because these tools use OCR for annotation recognition, it cannot be guaranteed that the recognition success rate is 100%. For example, Abuhaiba 2006 indicates a success rate of 98% or more for a state of the art OCR algorithm. It is here that MBD can provide added value, as annotation recognition is not based on OCR. Instead, direct use is made of the data structure in the MBD model. If the type and content can be determined by reading the data structure that contains the annotation within the MBD model, this enables recognition with a 100% success rate. This rules out the use of "presentation PMI" as this is only a graphical representation that does not allow the type and content of the annotation to be determined directly, namely without the use of some kind of OCR. If a module or an application is written that runs in or on top of a CAD package, not only can a list of all annotations in the MBD model be generated, but the annotations can also be shown or highlighted in the CAD model. There are a number of neutral exchange formats specifically for MBD. This means that these include support for representation PMI. Examples of such exchange formats are STEP AP242 and the new QIF, which has been developed specifically for quality control. This is reflected in its name. QIF stands for Quality Information Framework. When such an exchange format is used, it is possible to develop an application that can generate a list of all 3D annotations assigned in the CAD model exported to that format (Michaloski 2016; Lipman 2017). In combination with a viewer for QIF or STEP AP242 these annotations can also be shown or highlighted in the 3D model display (CAPVidia 2022; TransMagic 2022). The use of neutral exchange formats such as QIF and STEP AP242, presents specific problems. These, together with the formats themselves, are discussed in Chapter 6. In conclusion, applications such as those for creating FAIRs require CAD software and neutral exchange file formats to support "representation PMI", which allows for the retrieval of the type and content of 3D annotations. This does not necessitate the capacity to read the semantic references (i.e. the geometric entities to which the annotations are linked) or to make changes to the CAD model based on the dimension tolerances and GD&T applied.

Semantic references assigned to an annotation

Applications such as packages for the automatic generation of measurement programs for CNC-controlled coordinate measuring machines need more information than just the type and content of the annotation. They also need to know which point(s), which axis or axes, which edge(s), which plane(s), which surface(s) the annotation refers to. These topological entities are the previously mentioned "semantic references". Without these "semantic references", automatic processing of the annotations is not possible and human interpretation is necessary (Figure 3.6).



(a) Dimension refers to two edges (b) Dimension refers to a surface and an edge

(c) Dimension refers to two surfaces

Figure 3.6: Without semantic references, it can be unclear what a dimension refers to

It is not enough just to have a list of which references are linked to which annotations. An example. In a linear dimension between two surfaces, there are two semantic references. Clearly, one reference belongs at one end of the dimension and the other at the other end. In a linear dimension with more than two references (see Figure 3.7), it must be possible to determine which references belong together.

The aforementioned paragraphs discuss the retrieval of both the type and content of 3D annotations, as well as the determination of the semantic references to which an annotation refers. This is not solely related to the information present in the CAD model; it also concerns the possibility of retrieving that information by means of a computer program. In order for this information to be retrieved by a computer program, there must be function calls in the programming libraries of the CAD system itself or in external libraries that allow this (Ramnath et al. 2020). These function calls are part of what are known as APIs. API stands for Application Programming Interface and is the collection of methods and data formats that can be used to communicate with an application. The function calls must allow a computer program to retrieve the semantic references and determine their interrelationship. This is not always the case, as was observed during the software development of this PhD. In order to make a clear distinction between what is part of the literature study, of which this chapter is a part, and what is part of the research findings, this will be discussed in more detail in chapter 8, which deals with the software development undertaken during this PhD study.

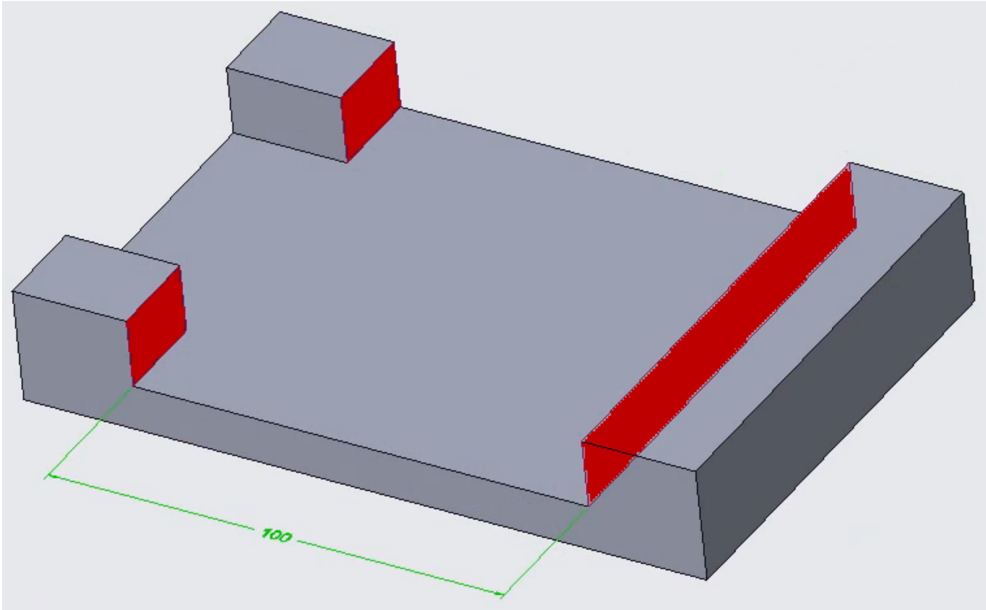


Figure 3.7: Linear dimension with three semantic surfaces

3.1.2 Annotation standards

When a company starts with MBD, there is a risk of straying into overly imaginative territory with the way 3D annotations are applied to the CAD model. In practice, this means that many companies have their own way of working, which makes cooperation with other companies difficult (J. Herron et al. 2019). Those difficulties relate to both human interpretation (Figure 3.8 and Figure 3.9) and so-called machine-readability (Fischer et al. 2015). A consistent data format is necessary to enable a good and smooth exchange of information between the different stakeholders involved, such as designers, manufacturing, quality control (Conover et al. 2006; Quintana, Rivest, Pellerin and Kheddouci 2012). This means that standards are absolutely necessary (J. Herron et al. 2019). These standards must be open so that they can be implemented and used by everyone (Hedberg 2017).

The ASME Y14.41 and the ISO 16792 standards were developed to meet these requirements. Both are based on earlier standards such as the ASME Y14.5 which defines how annotations such as dimensional and geometrical tolerances should be applied in a 2D drawing. ASME Y14.41 and ISO 16792 define how this must be done in a 3D model. To this end, new concepts are introduced such as “annotation plane” and “saved views”.

An “annotation plane” is a plane in the 3D model on which the annotation lies. The orientation of the plane determines the orientation of the annotation (Conover et al. 2006) (Figure 3.10 and Figure 3.11).

ISO states ‘Saved views of a design model may be defined to facilitate presentation of the model and its annotation’. CAD systems often use their own terminology to designate “saved views”. In PTC Creo they are called “combined views”, in Siemens NX “model views”, in Inventor “view representations”, in SolidWorks “annotation views”. A “saved view” can be considered the MBD equivalent of a view in a 2D drawing. Zhou et al. 2022 states ‘Annotated 3D models are a valid alternative to traditional drawings to effectively communicate product information, but their primary value may not be in terms of human interpretation, but in the automation mechanisms that can be enabled by the format.’ A term that is very often mentioned when talking about MBD and automation possibilities is “machine-readability”. It is extremely important to define exactly what this term means. “Machine-readability” in the restricted definition means that the content of an annotation can be read. An example of

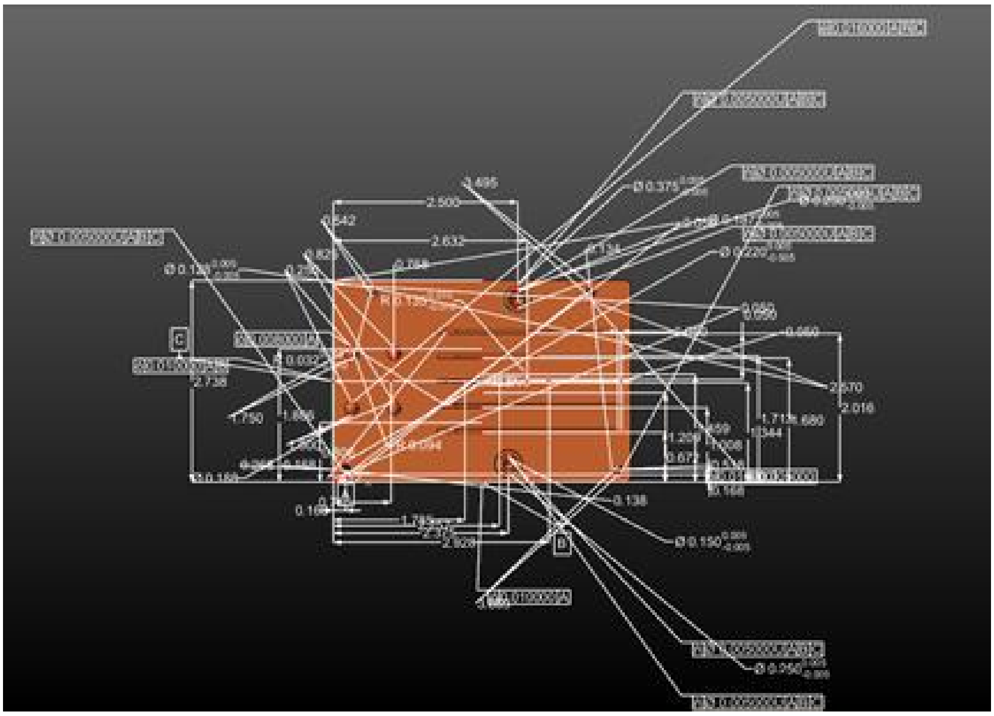
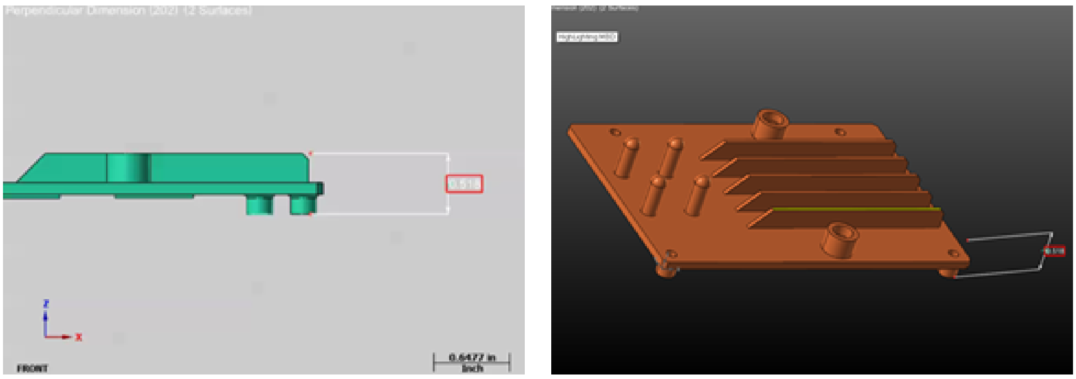


Figure 3.8: PMI spaghetti (CAPVidia 2014) after import in CAM system (Fischer et al. 2015)



(a) PMI in original view

(b) PMI in 3D view after import in CAM system

Figure 3.9: PMI difficult to interpret after import in CAM system (Fischer et al. 2015)

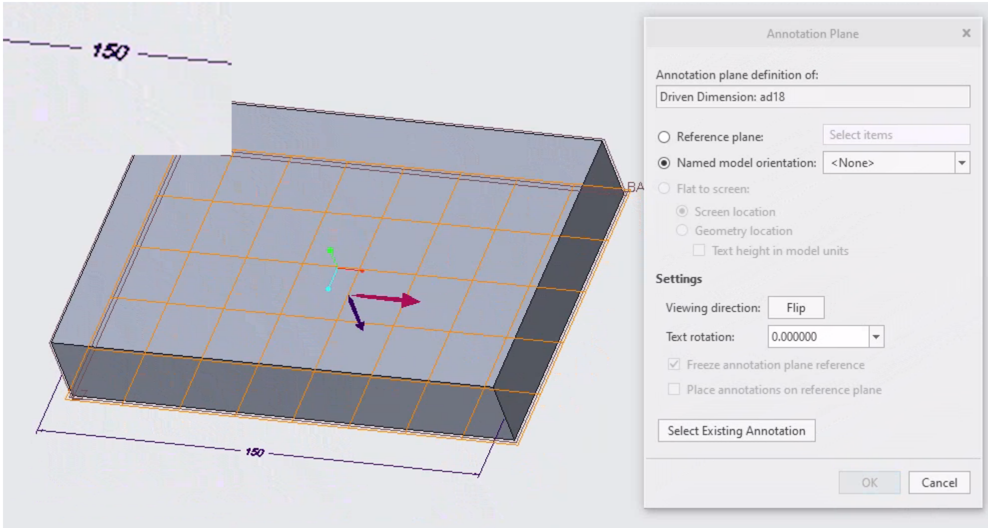


Figure 3.10: PTC Creo 8 - Annotation plane with first orientation option

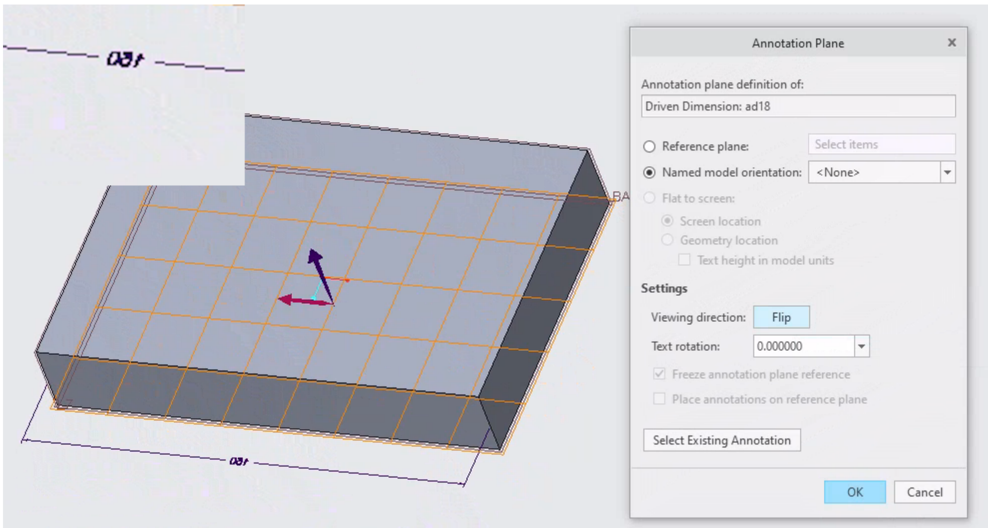
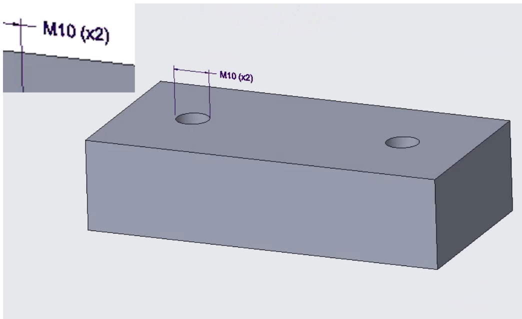


Figure 3.11: PTC Creo 8 - Annotation plane with second orientation option (text is mirrored)

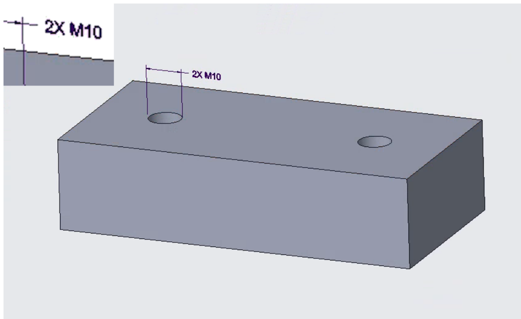
this is reading the numerical value of a diameter dimension. “Machine-readability” in the extended definition refers to “representation PMI” which means that the annotation has a data structure that allows another software application to determine what type of annotation it is (note, dimension, GD&T), what the contents of this annotation is and what this annotation refers to, the so-called semantic references (Feeney et al. 2015). It is a challenge to create annotations that are both “human-readable” and “machine-readable” (Fischer et al. 2015).

To enable these "automation mechanisms", machine-readability as defined in the restricted definition is not sufficient. Even if this were the case, standards such as ASME Y14.41 and ISO 16792, among others, must be meticulously followed in order to be able to interpret the contents of an annotation correctly.

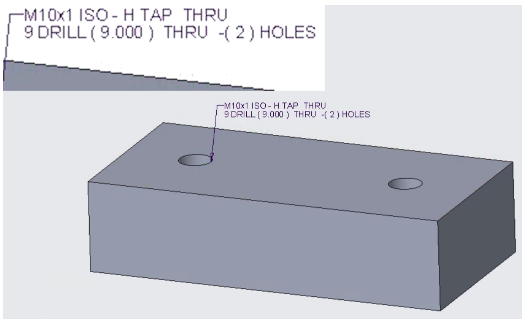
Figure 3.12 shows three different ways the same two holes can be dimensioned. It is not sufficient to only be able to read the text of the dimension in order to interpret it correctly. Figure 3.12a shows a dimension as found in a company design. This way of dimensioning was part of the company standard. Within the company everyone understands what it means. When the model is used by a stakeholder outside the company, this can be confusing. Figure 3.12b is correctly dimensioned according to the ASME standard but can cause problems when neutral exchange formats are used as information about the depth of the thread can be lost. Chapter 6 is dedicated to neutral exchange formats and discusses these issues in detail. Figure 3.12c shows a dimension scheme that is (almost) fully compliant with the ASME standard. It is the only scheme whereby the complete information of the holes, meaning the depth of the hole and the thread, can be transferred between different systems under all circumstances.



(a) dimension scheme 1



(b) dimension scheme 2



(c) dimension scheme 3

Figure 3.12: Machine readable, but difficult to interpret

“Under all circumstances” means here that this is not only the case when this transfer is made to another computer running the same CAD software as that what was used to create the design, but also when neutral exchange formats are used. In order to support the designer in applying dimensions in accordance with the standard, many CAD manufacturers have introduced so-called advisors. This is a software module that checks whether the entered dimensions and GD&T comply with the standard (Morey 2020).

In order for a software application to make full use of an MBD model, being able to read and interpret the contents of an annotation is not enough. This becomes clear with the following example. A software application that can make use of MBD is a software package for generating measuring programmes for a CMM (an acronym of Coordinate Measuring Machine). If this application only knows whether a linear measurement, a diameter (the annotation type), needs to be measured and which condition it must satisfy (the content of the annotation), but does not know where to measure, the annotation info is not useful. In order to know where to measure, it is necessary to be able to determine to what points, curves, surfaces an annotation refers. These are the "semantic references" (Figure 3.13).

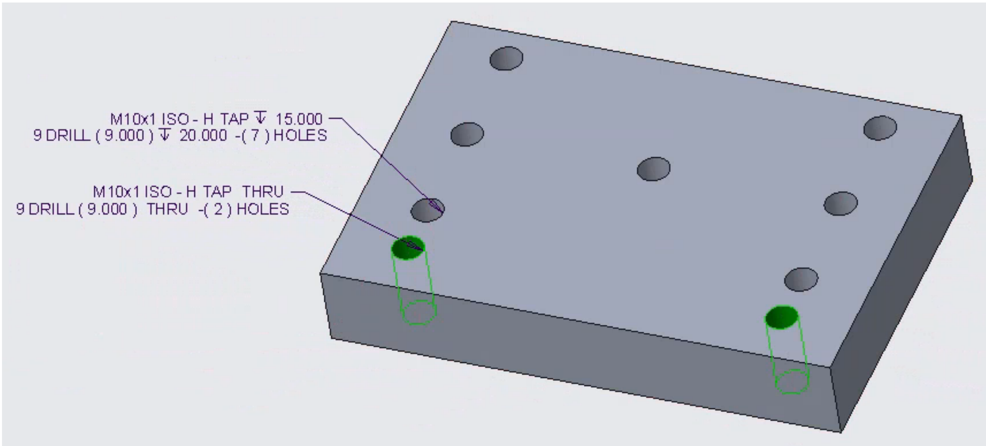


Figure 3.13: Semantic references (surfaces coloured green) indicating to what holes the annotation refers

3.1.3 Creation method

CAD systems offer their users various methods for making annotations. On first inspection, all these methods seem equivalent and seem to lead to the same result. However, the tests carried out during this PhD study have shown that the different methods lead to a difference in which information can be retrieved afterwards, how it is retrieved and which information is retained when exporting to a neutral exchange format such as STEP AP242. This will now be demonstrated with concrete examples. To facilitate comparison between different CAD systems and different methods, the same MBD model is always created.

Siemens NX

In Siemens NX there are two different methods to create 3D annotations. A first method is via the "Drafting" module. This will be discussed in Example 1 below. A second method is via the "PMI" module. This will be discussed in Examples 2 and 3 below.

Example 1

Figure 3.14 shows a screenshot of a model created in Siemens NX Version 2019, Build 2501. Using the "Drafting" application available in Siemens NX, two linear dimensions with an asymmetric tolerance were created in this model.

This model is then exported to a STEP AP242 file, ensuring that the option to export PMI data is activated (Figure 3.15).

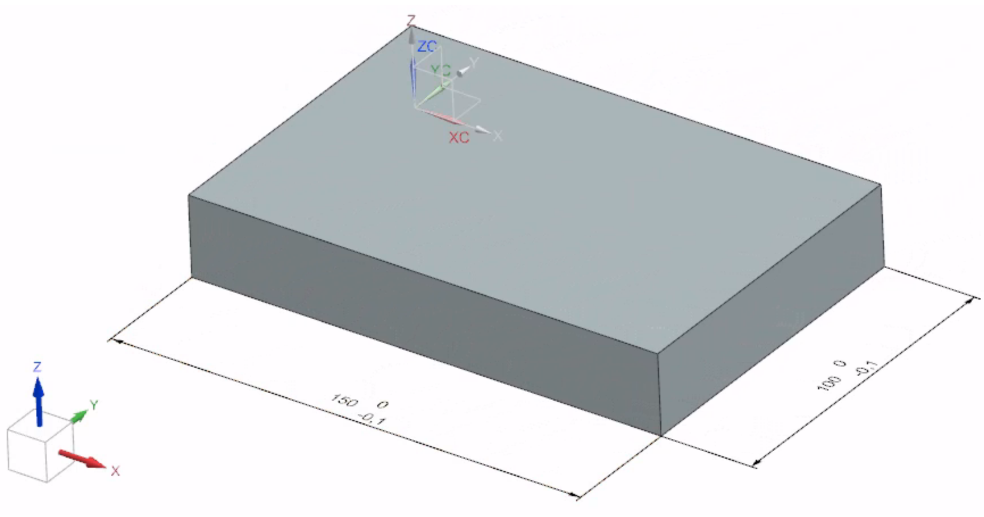


Figure 3.14: Siemens NX - linear dimensions created using the Drafting module

This STEP file is imported into PTC Creo Parametric 8.0.4.0 and Autodesk Inventor 2022. When using Inventor it is important to also enable the detection of PMI data within STEP files (Figure 3.16). The result of the import into PTC Creo is shown in (Figure 3.17a) and the result in Inventor is shown in (Figure 3.17b). It can be seen that the linear dimensions are visible in both Creo and Inventor. On first inspection, everything seems to be in order. When an attempt is made to retrieve the content of these dimensions in Creo (Figure 3.18a) and Inventor (Figure 3.18b), it appears that this is not possible. It can be concluded that these linear dimensions are stored in the STEP file as "presentation PMI". This means that they can be read and interpreted by humans, but not by software.

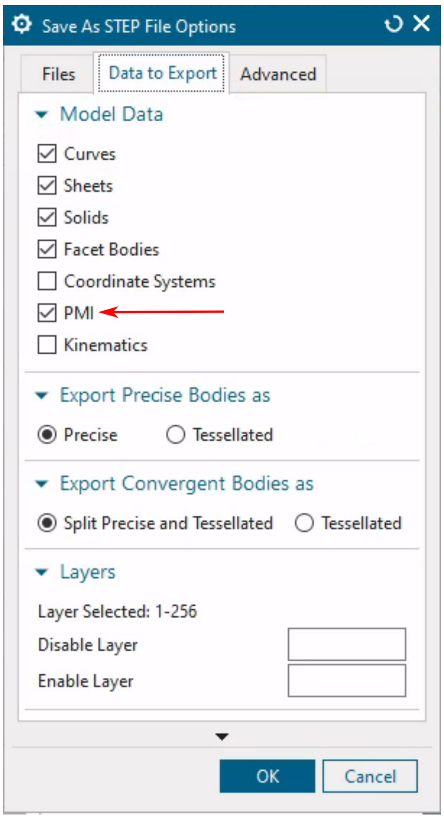


Figure 3.15: Siemens NX - PMI export enabled

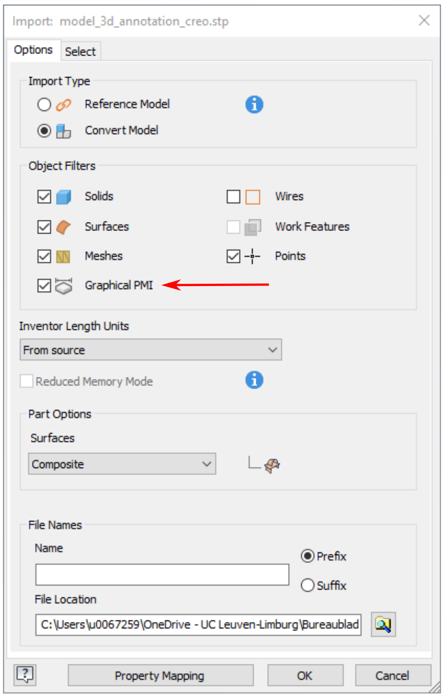
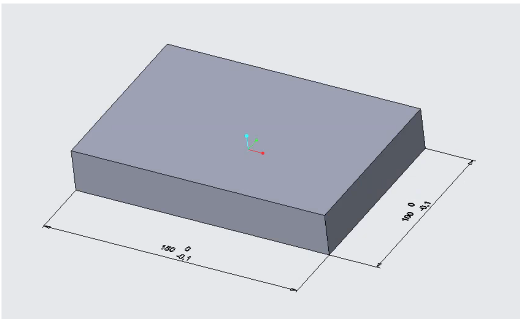
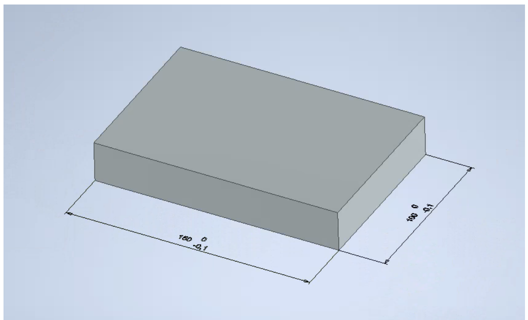


Figure 3.16: Inventor - PMI import enabled

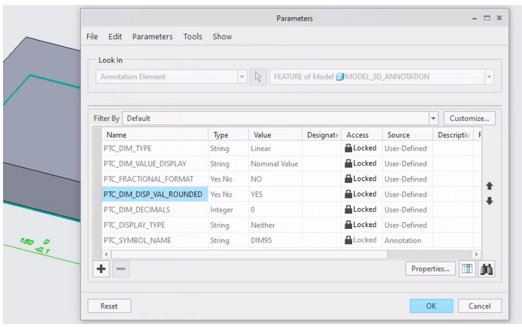


(a) STEP file imported in PTC Creo 8

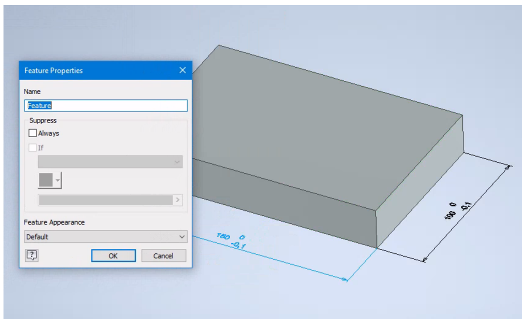


(b) STEP file imported in Inventor 2022

Figure 3.17: STEP file generated by NX imported in Creo and Inventor



(a) Dimension parameters in PTC Creo 8



(b) Dimension properties in Inventor 2022

Figure 3.18: The dimension content cannot be retrieved in Creo and Inventor

Example 2

Figure 3.19 shows a screenshot of a second model which looks identical to the one in Example 1. However, instead of using the "Drafting" module in Siemens NX version 2019, build 2501, this time the "PMI" module was used to create two linear dimensions with an asymmetric tolerance in the CAD model.

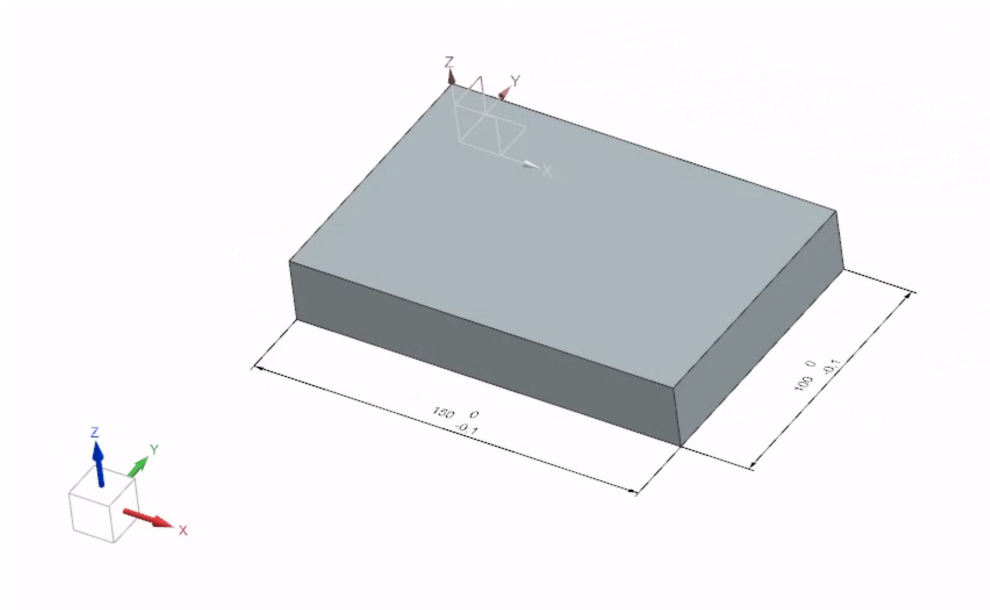


Figure 3.19: Siemens NX - linear dimensions created using the PMI module

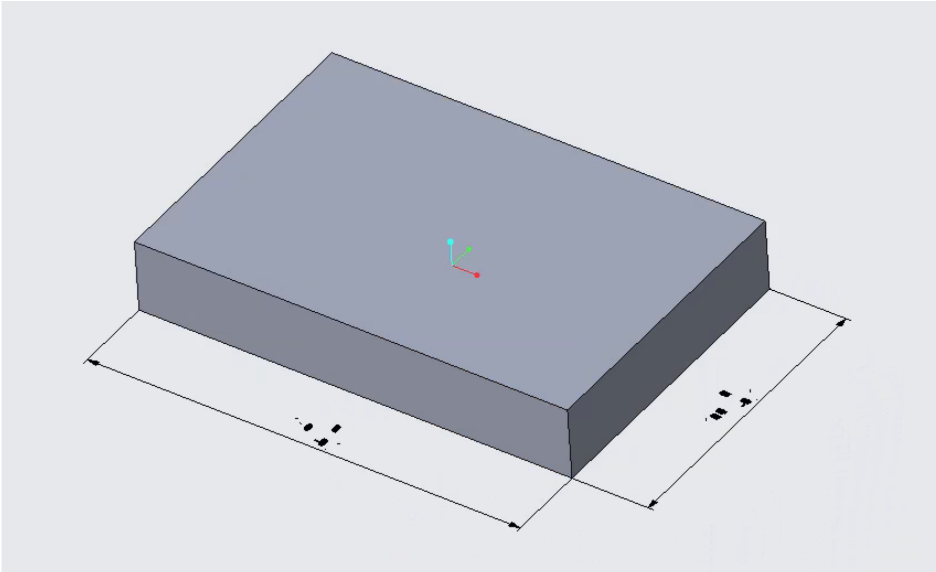
This second model is then exported to a STEP AP242 file, ensuring that the option to export PMI data is activated (Figure 3.15).

When this STEP file is imported into PTC Creo Parametric 8, it can be seen that the linear dimensions are present in the imported model. The visual representation of the dimensions is not legible (Figure 3.20a). However, the parameters assigned to the "representation PMI" data of these linear dimensions are present in the model (Figure 3.20b).

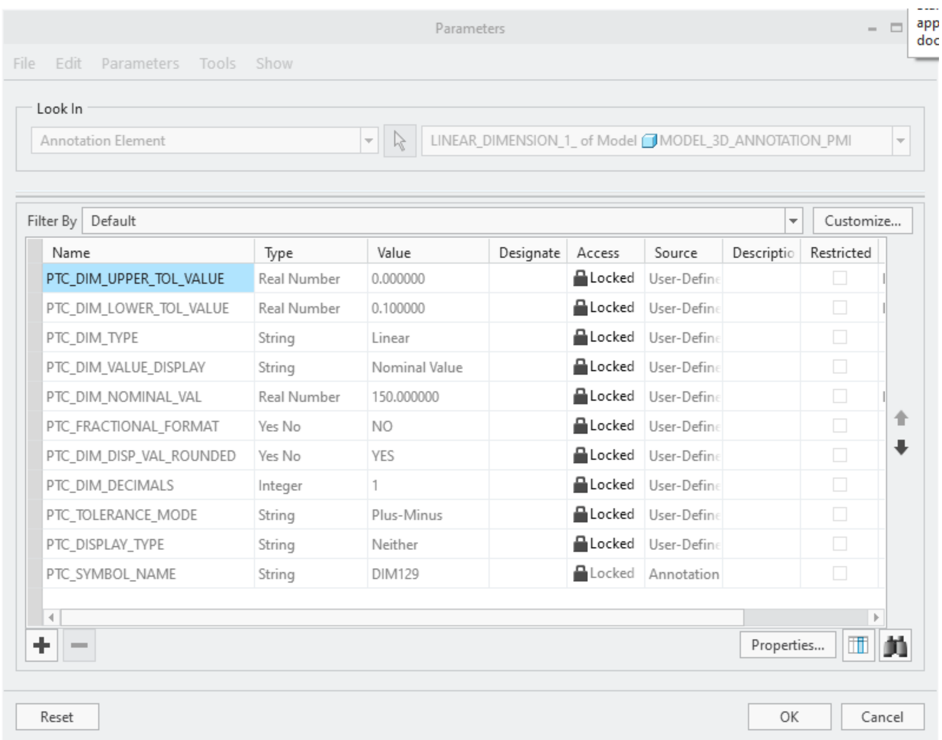
When this STEP file is imported into Autodesk Inventor 2022, it can be seen that the linear dimensions are visible. Contrary to PTC Creo 8, the visual representation is legible (Figure 3.21a). However, contrary to PTC Creo 8, the parameters assigned to the "representation PMI" data of these linear dimensions are not present in the model (Figure 3.21b).

Example 3

Figure 3.22 shows a screenshot of a model created in Siemens NX version 2019, build 2501, which looks identical to the one in Example 1 and Example 2. As in Example 2, the "PMI" module has been used to create the two linear dimensions with an asymmetric tolerance in the CAD model. The difference from Example 2 is that a different method has been used to specify the view plane (Figure 3.23) in which the dimensions are created. In this case the bottom face of the part was selected. This face has a vector perpendicular to this face and pointing outwards, away from the volume. This means that when the dimension text is created in this face, this vector points against the viewing direction. The result of this can be seen in Figure 3.24. The dimension text is flipped when looked upon the top view (XY plane seen against the Z axis). In Siemens NX this is not the case as the software ensures that the dimension text is always correctly oriented.

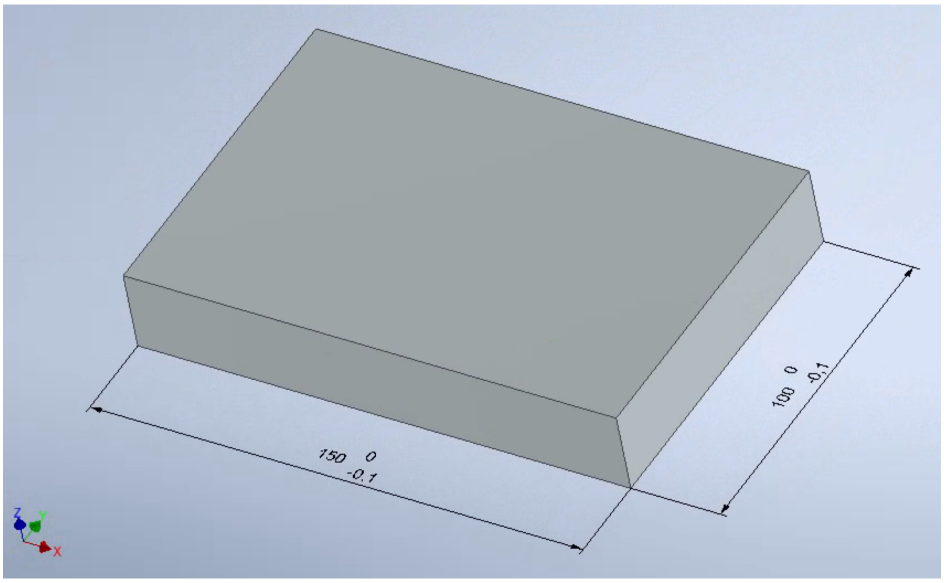


(a) Dimension not legible after import in PTC Creo 8.0.4

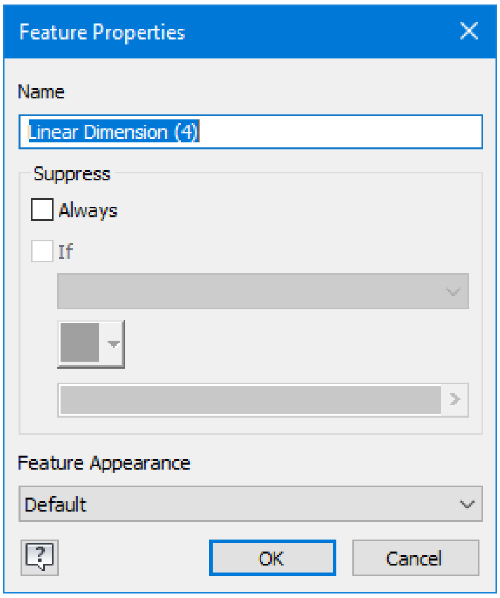


(b) Representation PMI data preserved after import

Figure 3.20: STEP file generated by NX imported in Creo



(a) Dimension legible after import in Inventor 2022



(b) Representation PMI data not preserved after import

Figure 3.21: STEP file generated by NX imported in Inventor

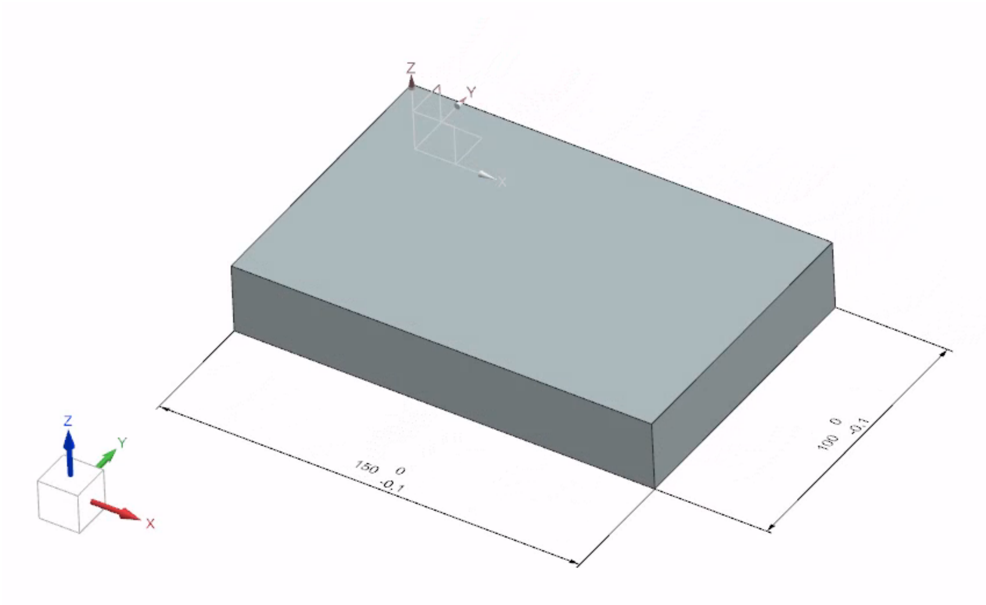


Figure 3.22: Siemens NX - linear dimensions created using the PMI module

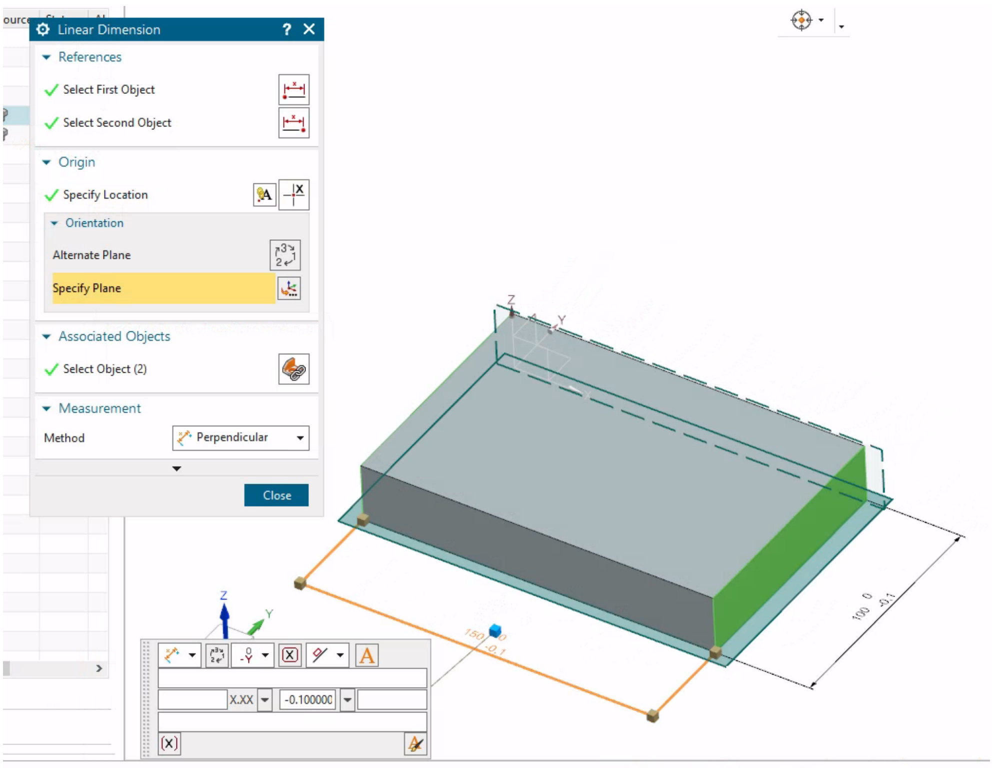


Figure 3.23: Siemens NX - specifying annotation plane for linear dimension

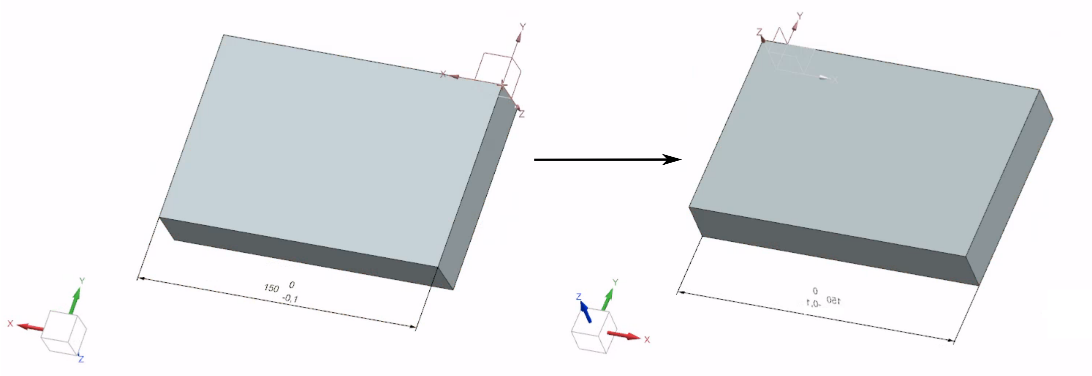


Figure 3.24: Resulting text orientation based on the vector of the selected annotation plane

This third model is exported to a STEP AP242 file, ensuring that the option to export PMI data is activated (Figure 3.15).

When this STEP file is imported into PTC Creo 8, it can be seen that the linear dimensions are present in the imported model. The visual representation of the dimensions is not legible. However, the parameters assigned to the “representation PMI” data of these linear dimensions ar present in the model. The result is thus the same as is shown in Figure 3.20.

When this STEP file is imported into Autodesk Inventor 2022, it can be seen that that the linear dimensions are visible. Contrary to PTC Creo 8, the visual representation is legible. However, because of the chosen annotation plane, the display of the dimensions is flipped (Figure 3.25). As was the case in previous examples, the parameters assigned to the “representation PMI” data of the linear dimensions are not available.

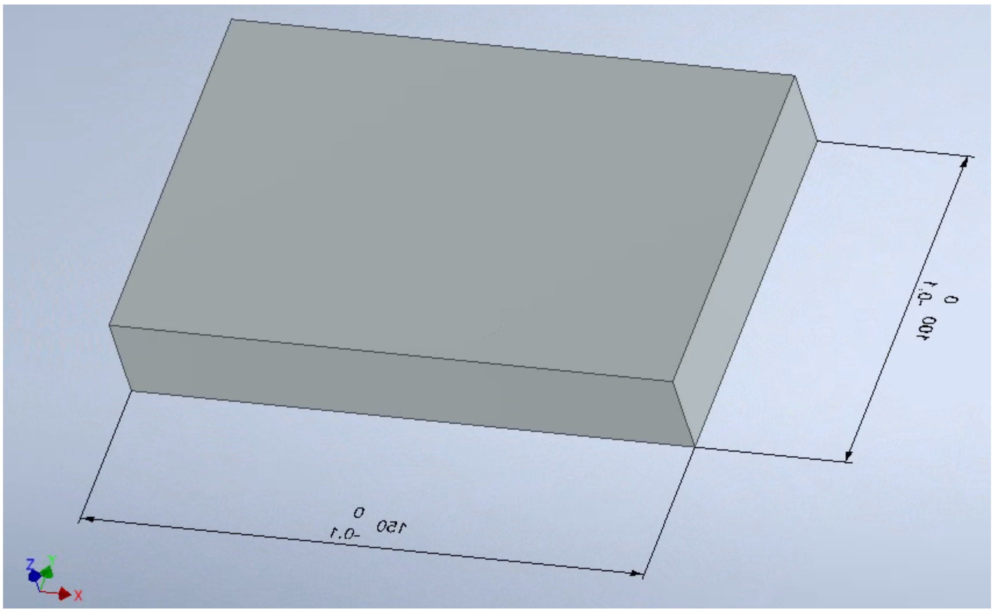


Figure 3.25: dimension text is flipped after import in Inventor 2022

PTC Creo Parametric

In PTC Creo Parametric there are three different methods to create 3D annotations. A first method is to place "driving dimensions". This will be discussed in Example 1 below. A second method is to place so-called "driven dimensions". This will be discussed in Example 2 below. A third method is to place "annotation features". This will be discussed in Example 3 later in the text. As in the Siemens NX tests, the same MBD model is used for the three different methods, with only the method of assigning dimensions differing.

Example 1: Driving dimensions

Figure 3.26 shows a screenshot of the model created in PTC Creo Parametric 8.0.4.0. The linear dimensions are created using the function "Show Annotations". This means that dimensions used to create a feature are directly adopted as 3D annotations in the 3D model. These dimensions are called "driving dimensions" as modifying their values changes the model. This kind of annotations is also described as annotation elements owned by the feature the dimensions belong to (Fridman 2019) (Figure 3.27).

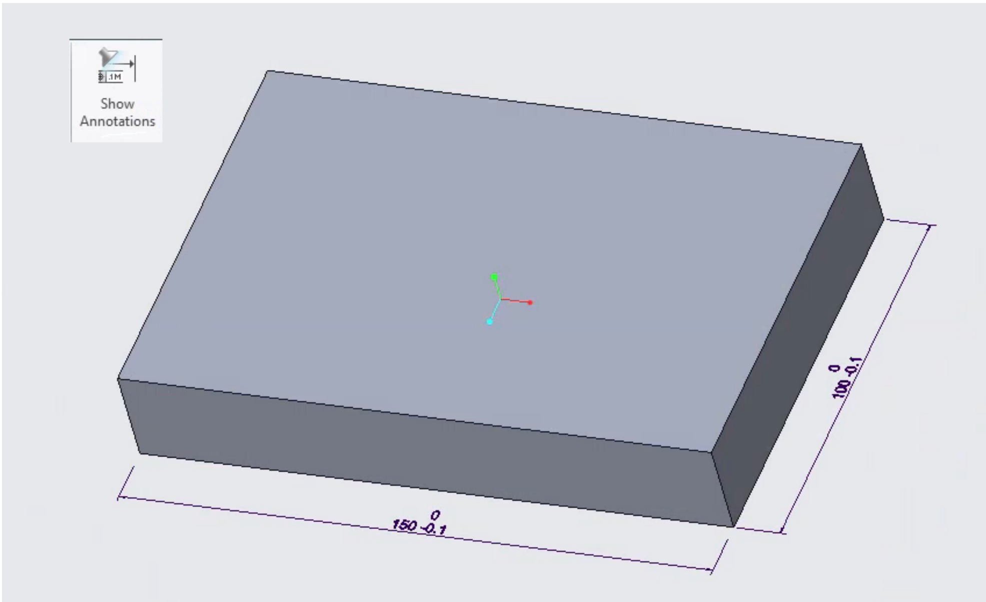


Figure 3.26: Creo 8 - linear dimensions created as "driving dimensions"

This model is exported to a STEP AP242 file (edition 1), ensuring that the option to export PMI data is activated (Figure 3.28). What exactly "edition 1" means is described in Chapter 6, which is dedicated to neutral exchange files.

When this STEP file is imported into Siemens NX 2019, Build 2501, it is important to also enable the detection of PMI data within STEP files (Figure 3.29). The linear dimensions that were present in the native Creo model are not visible in Siemens NX (Figure 3.30). However, the dimensions are present within the imported model (Figure 3.31) but no data is attached to these dimension objects (Figure 3.32). The dimensions are not visible because their display depends on the view they are linked to. In Creo only one combination view with the name "Default All" was active (Figure 3.33). This view contained the complete 3D model with the 3D annotations. Within Siemens NX this view must be activated first (Figure 3.34) in order to enable the display of the dimensions (Figure 3.35). It can be concluded that when 3D annotations are created in PTC Creo 8 as "driving dimensions", information is lost because they are only stored as "presentation PMI" within the STEP file.

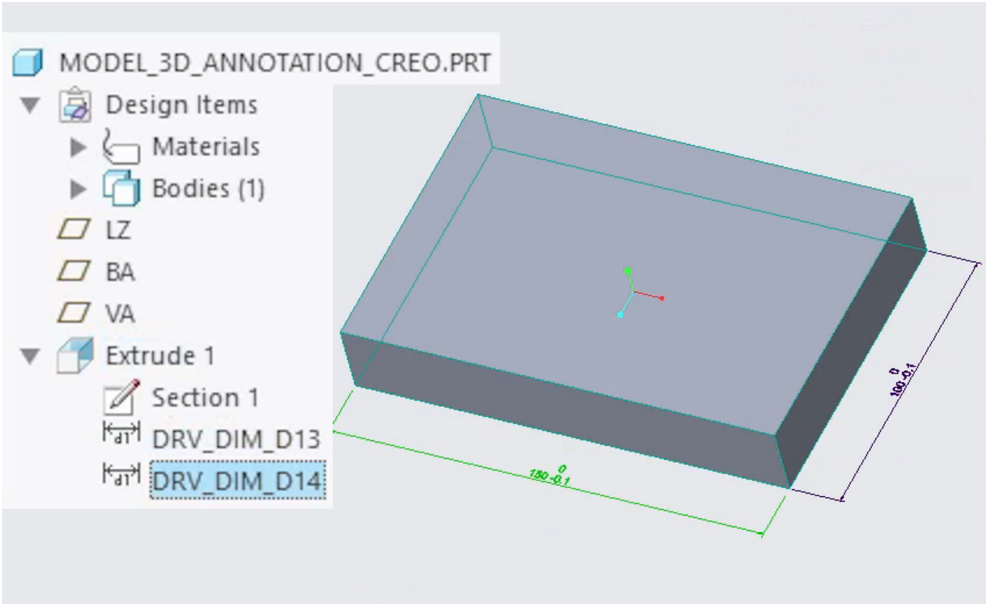


Figure 3.27: Creo 8 - annotation elements owned by the extrude feature

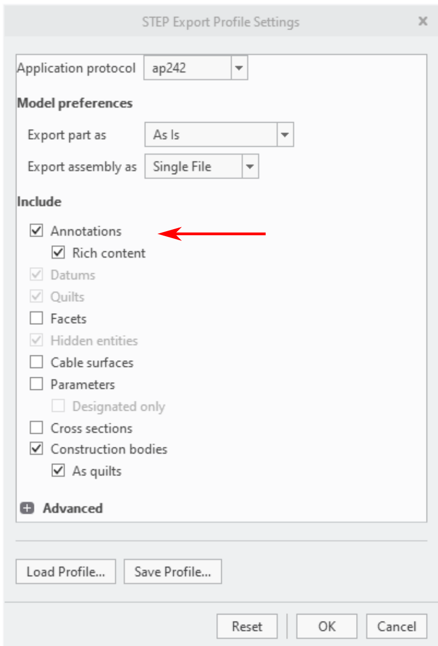


Figure 3.28: Creo - PMI export enabled

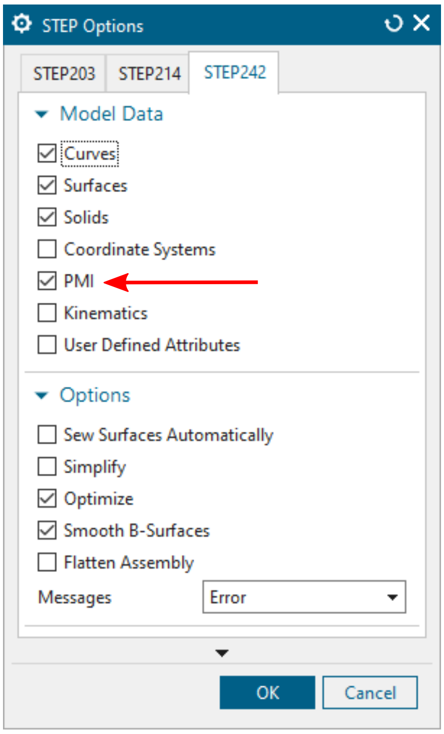


Figure 3.29: NX - PMI import enabled

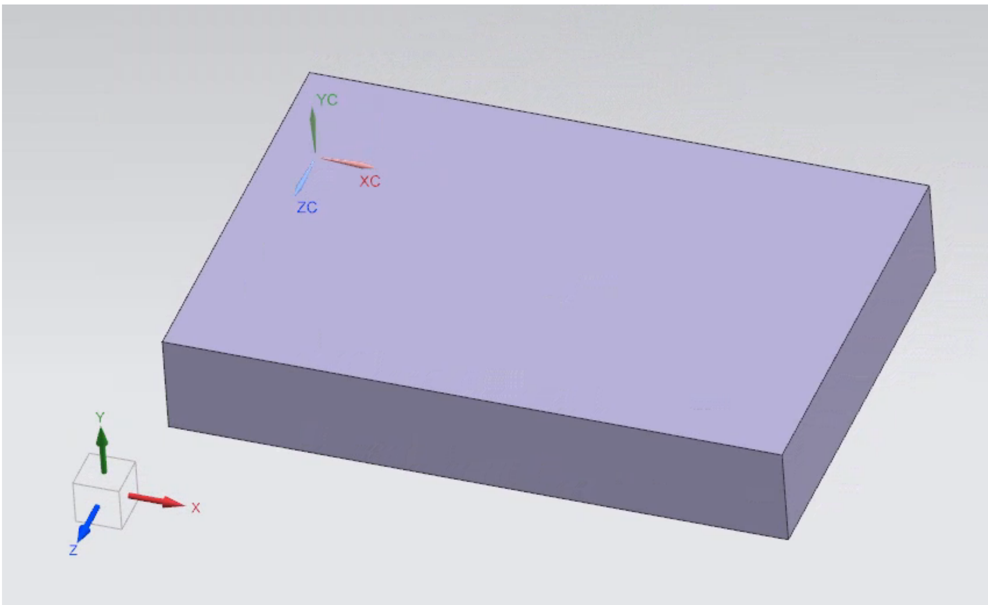


Figure 3.30: NX - linear dimensions are not visible after import

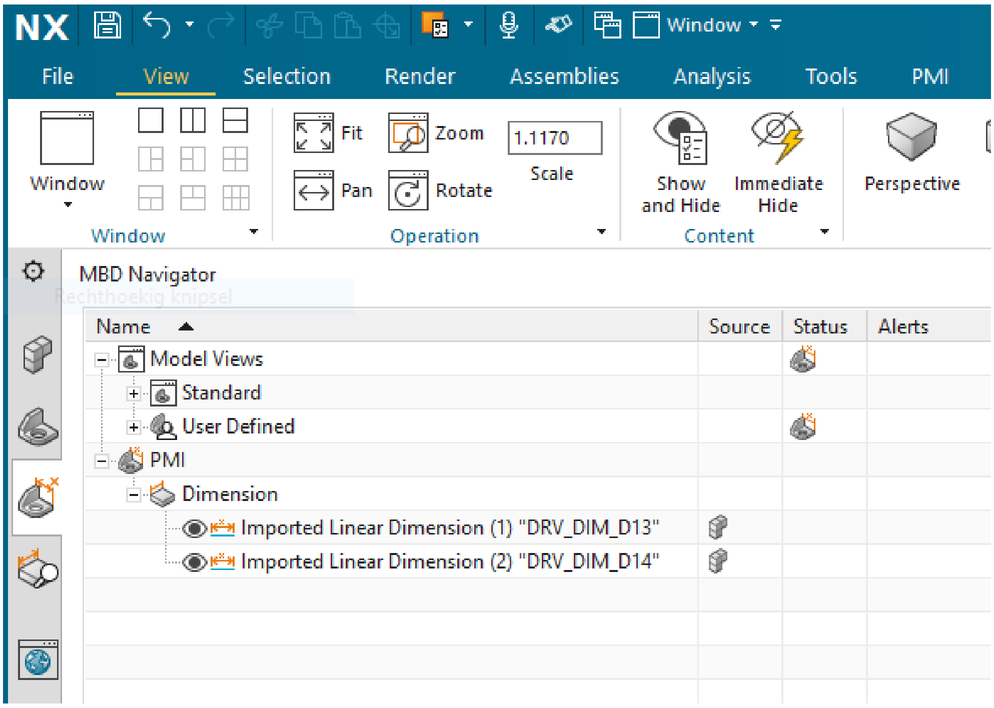


Figure 3.31: NX - linear dimension objects present in NX model

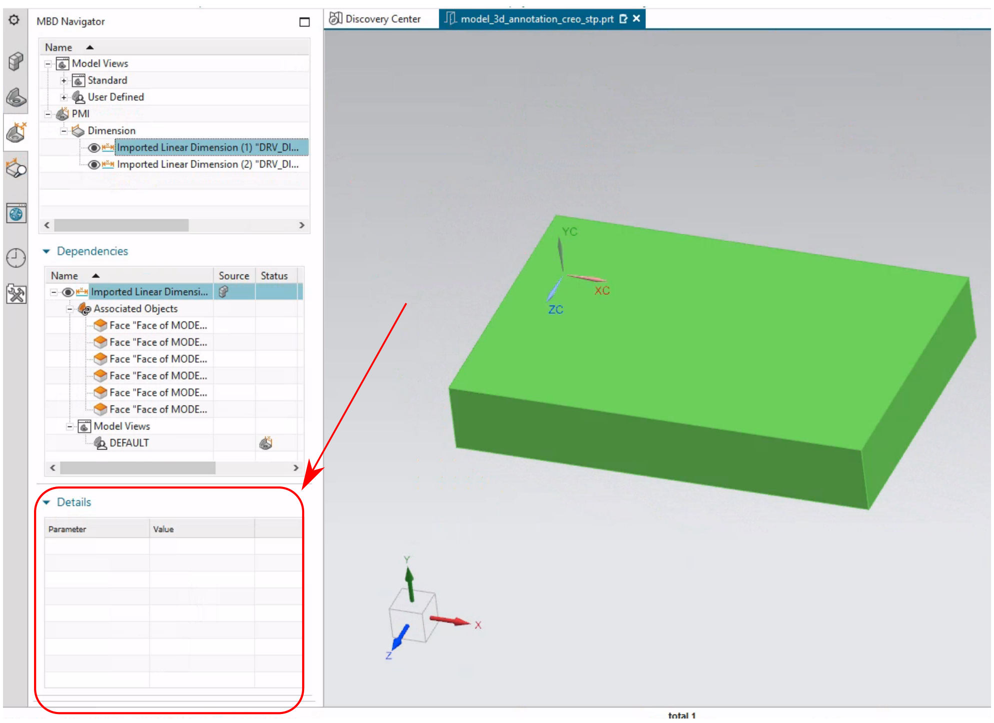


Figure 3.32: NX - linear dimension objects present in NX model with no data

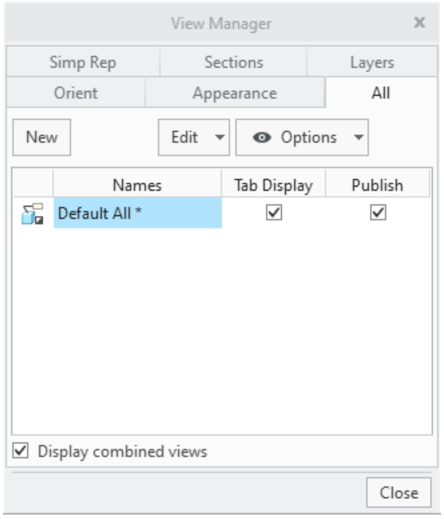


Figure 3.33: Creo - 3D model and 3D annotations in one combination view

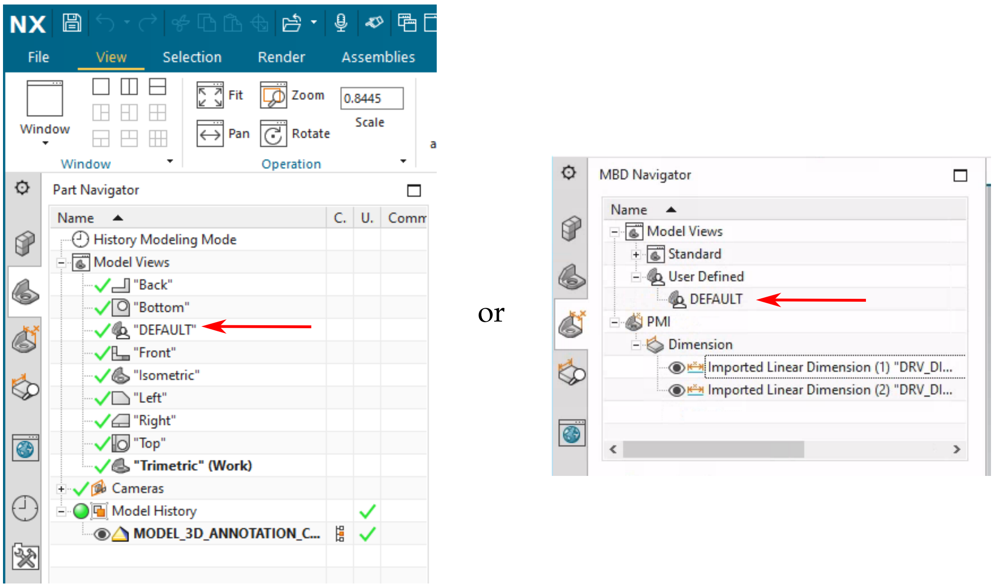


Figure 3.34: NX - Correct combination view must be activated

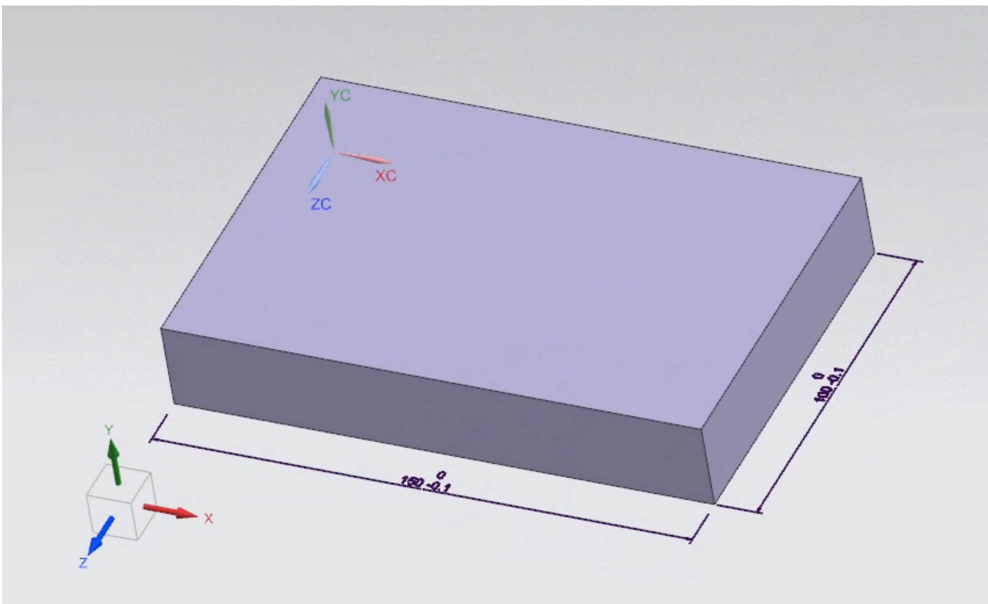


Figure 3.35: NX - linear dimensions are visible when the correct view is activated

When the STEP file that is exported by PTC Creo Parametric 8.0.4.0 is imported into Autodesk Inventor 2022, it is important to also enable the detection of PMI data within STEP files (Figure 3.16). The name "graphical PMI" suggests that the PMI data are imported only as "presentation PMI". In contrast to Siemens NX, the linear dimensions that were present in the native Creo model are visible in Inventor without any additional intervention by the user (Figure 3.36). However, there is no data attached to these dimension objects (Figure 3.37). From this it can be concluded that indeed only "presentation PMI" are retained.

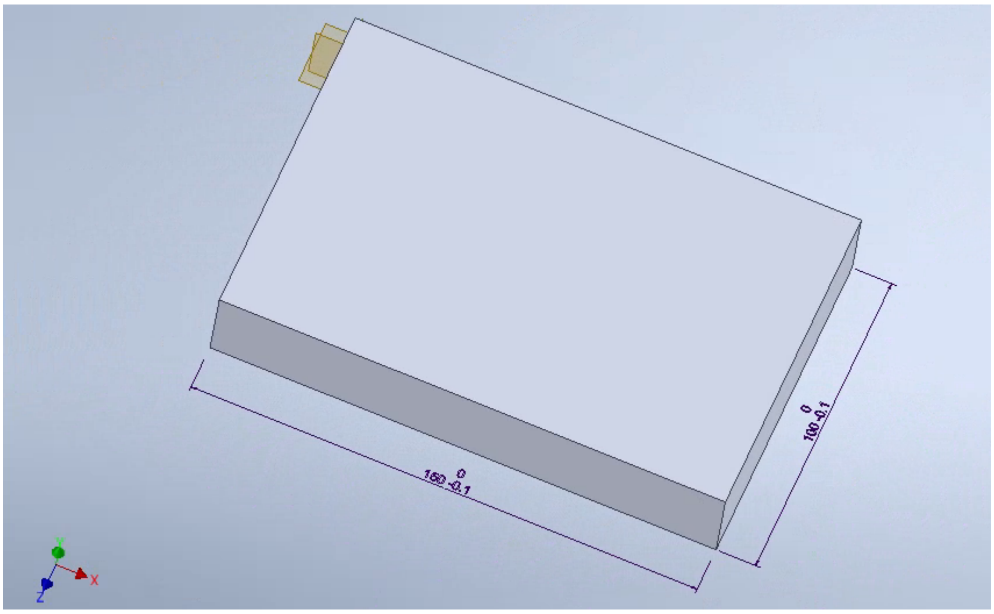


Figure 3.36: Inventor - linear dimensions are visible after import

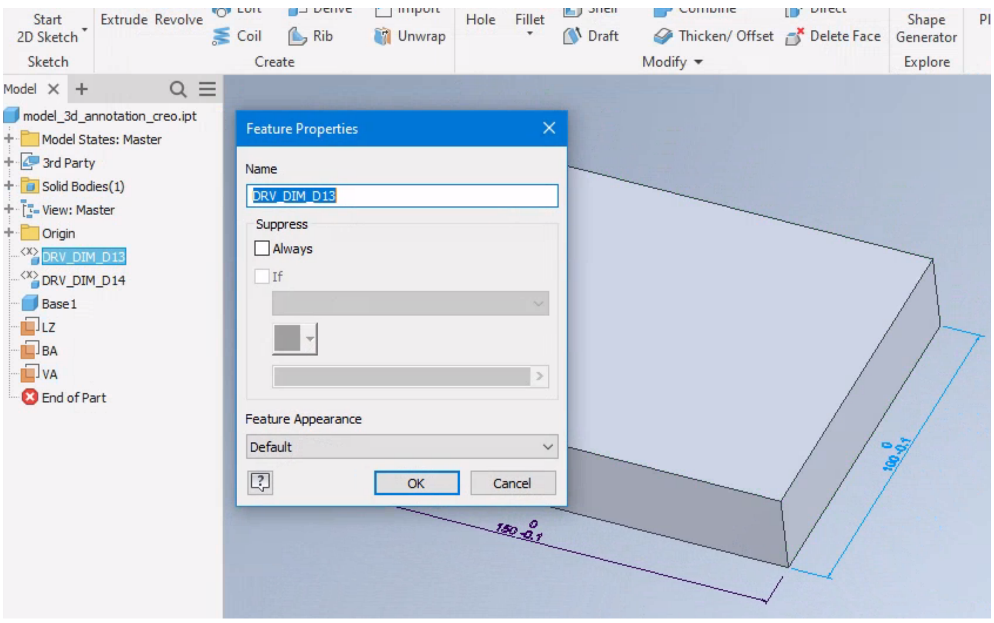


Figure 3.37: Inventor - linear dimensions are visible but no data attached to it

Example 2: Driven dimensions

Figure 3.38 shows a screenshot of the model created in PTC Creo Parametric 8.0.4.0. Instead of using "driving" dimensions, this time the linear dimensions are created using the "Dimension" function. This means that the dimensions are not derived from dimensions used in the sketches that form the basis of the features used to build the CAD model (as is the case with "driving" dimensions). Rather, they are separate dimensions that have no correlation with the construction of the model and are manually added on top of the existing CAD geometry. They are called "driven dimensions" as their values cannot be modified directly. They adapt when the dimensions of the model are changed by specifying new numerical values in the features used to build the model. Hence the name "driven dimensions". This type of annotations is also referred to as "stand-alone annotations" (Fridman 2019) (Figure 3.39).

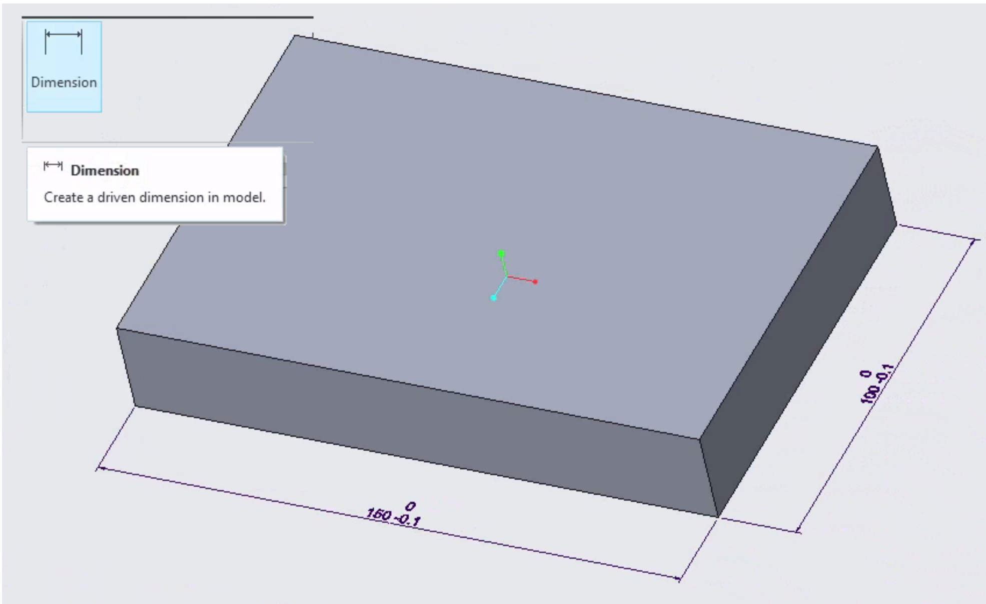


Figure 3.38: Creo 8 - linear dimensions created as "driven dimensions"

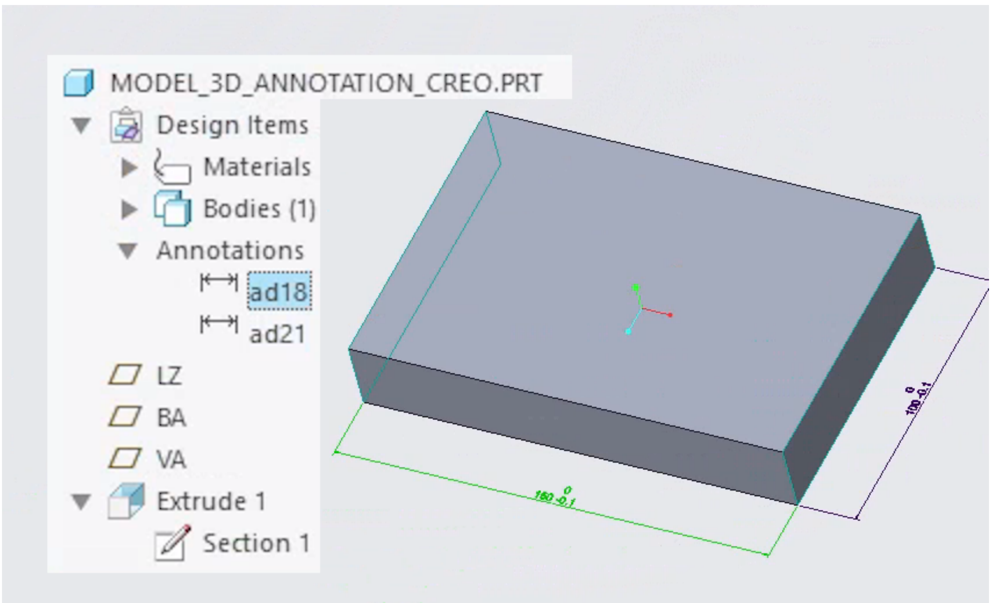


Figure 3.39: Creo 8 - stand-alone annotations

This model is exported to a STEP AP242 file (edition 1), ensuring that the option to export PMI data is activated (Figure 3.28). What exactly “edition 1” means is described in Chapter 6, which is dedicated to neutral exchange files.

When this STEP file is imported into Siemens NX 2019, Build 2501, it is important to also enable the detection of PMI data within STEP files (Figure 3.29). The linear dimensions that were present in the native Creo model are not visible in Siemens NX (Figure 3.40). However, the dimensions are present within the imported model (Figure 3.41) and contrary to the previous example all the data (values and references) attached to these dimension objects is retained (Figure 3.42). The dimensions are not visible because their display depends on the view they are linked to. In Creo only one combination view with the name "Default All" was active (Figure 3.33). This view contained the complete 3D model with the 3D annotations. Within Siemens NX this view must be activated first (Figure 3.34) in order to enable the display of the dimensions (Figure 3.43). From this it can be concluded that when 3D annotations are created in PTC Creo 8 as “driven dimensions”, the information of the annotations is preserved as they are stored as “representation PMI” within the STEP file.

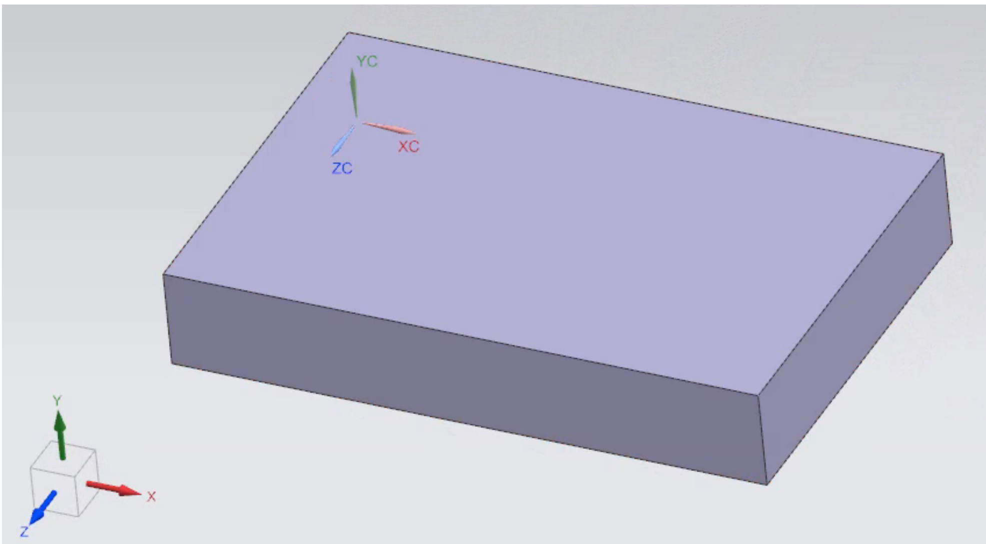


Figure 3.40: NX - linear dimensions are not visible after import

When the STEP file that is exported by PTC Creo Parametric 8.0.4.0 is imported into Autodesk Inventor 2022, it is important to also enable the detection of PMI data within STEP files (Figure 3.16). The name “graphical PMI” suggests that the PMI data is imported only as “presentation PMI”. In contrast to Siemens NX, the linear dimensions that were present in the native Creo model are visible in Inventor without any additional intervention by the user (Figure 3.44). However, there is no data attached to these dimension objects (Figure 3.45). From this it can be concluded that indeed the 3D annotations can only be retrieved as “presentation PMI”.

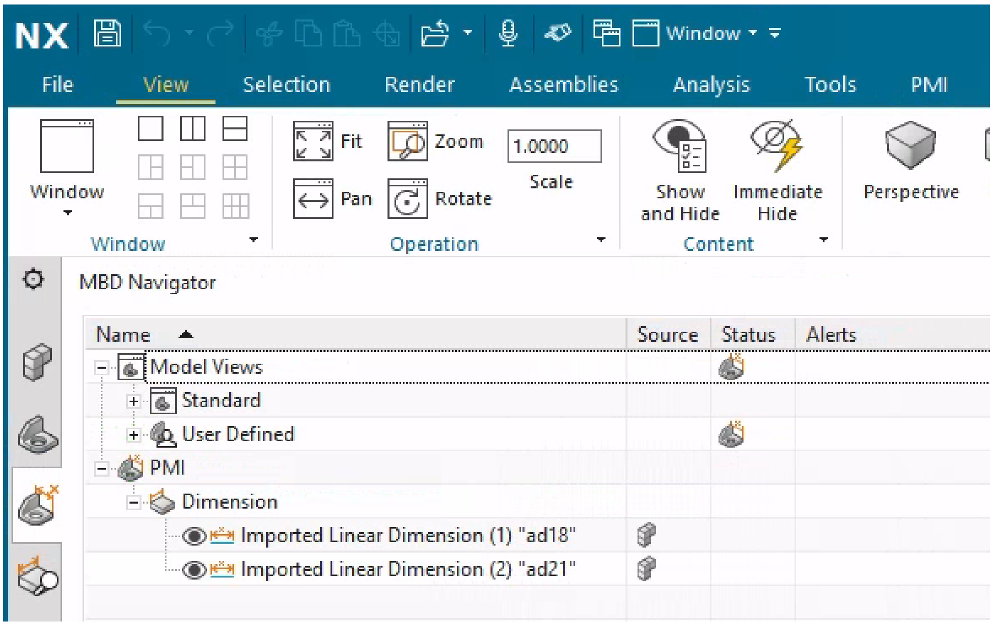


Figure 3.41: NX - linear dimension objects present in NX model

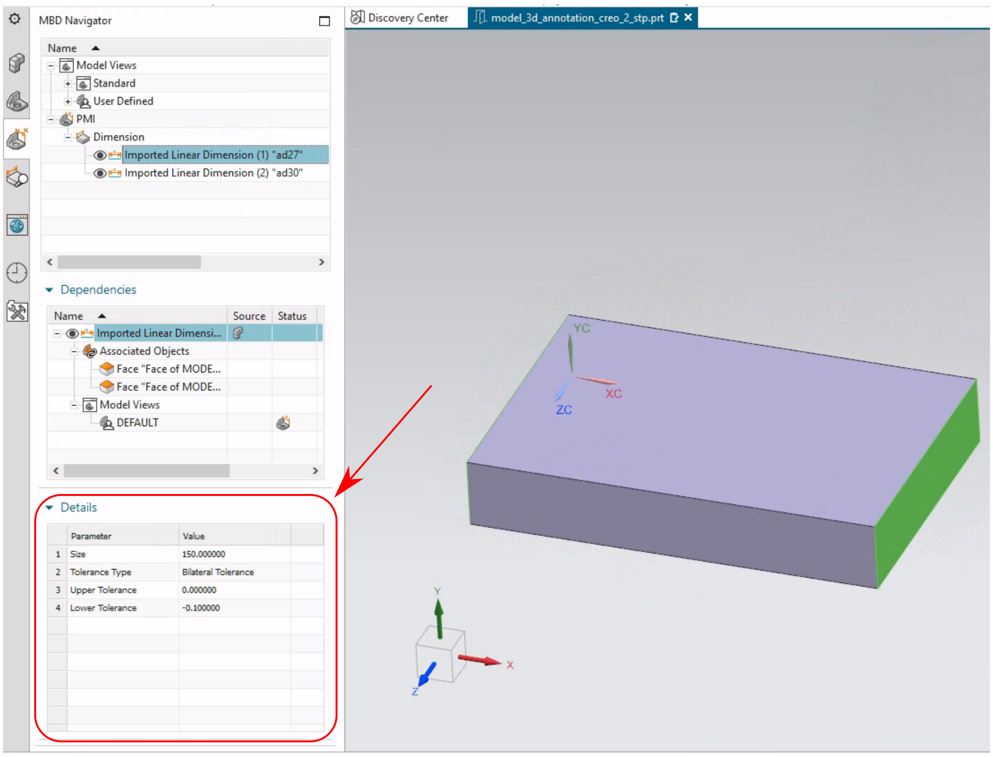


Figure 3.42: NX - linear dimension objects present in NX model with data

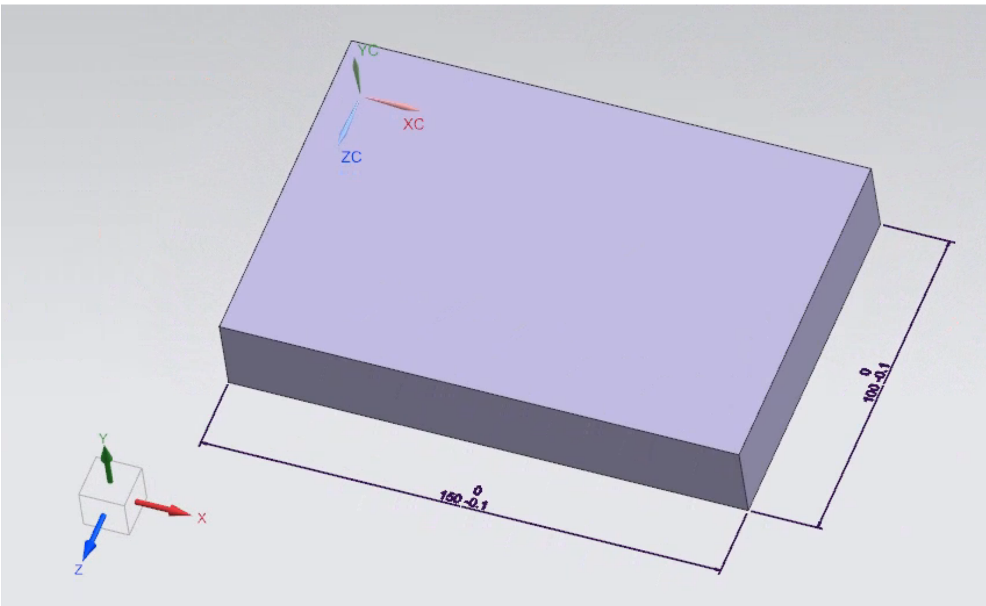


Figure 3.43: NX - linear dimensions are visible when the correct view is activated

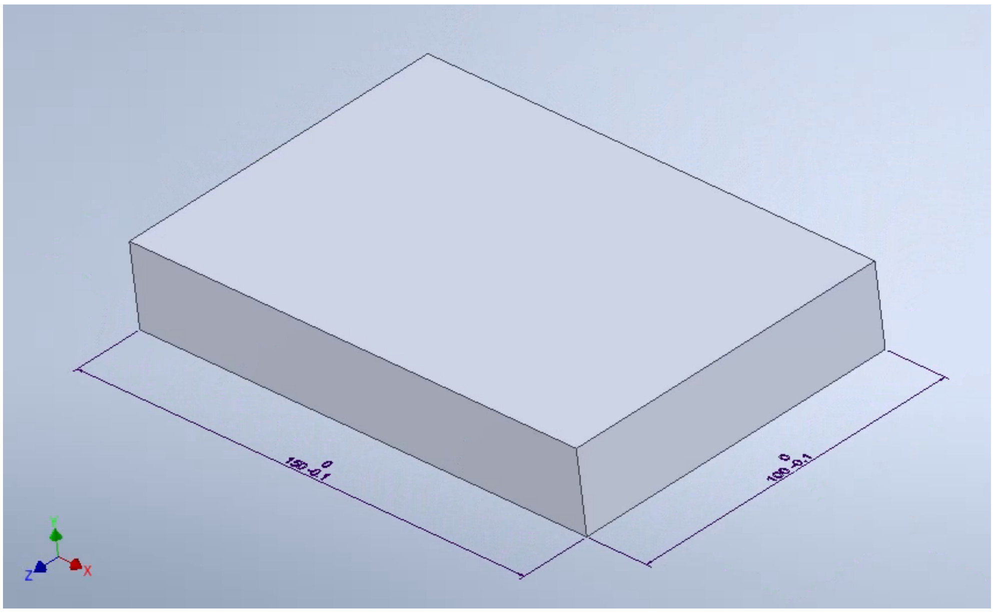


Figure 3.44: Inventor - linear dimensions are visible after import

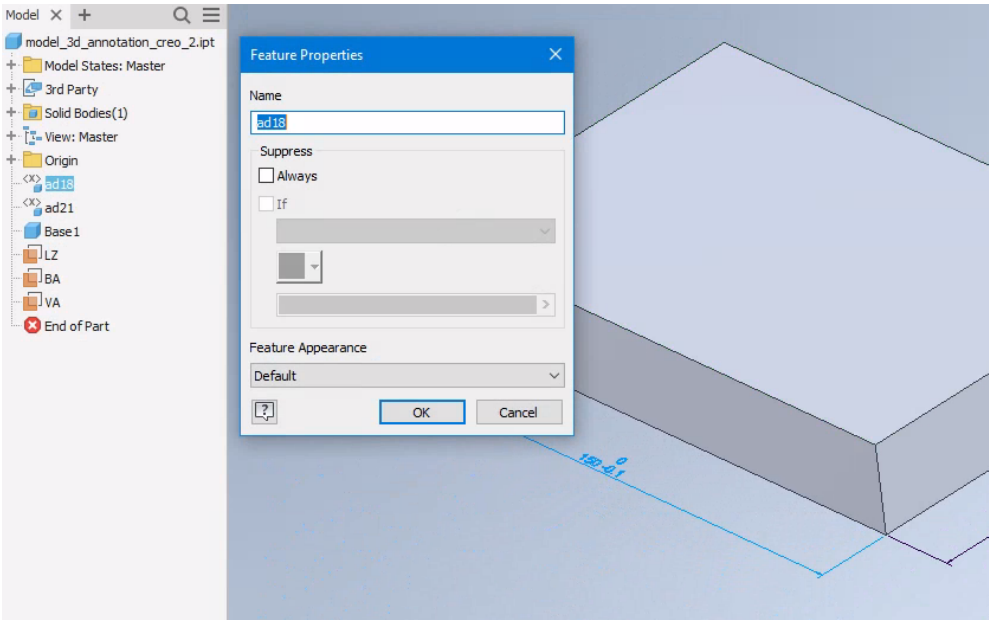


Figure 3.45: Inventor - linear dimensions are visible but no data attached to it

Example 3: Annotation features

Figure 3.46 shows a screenshot of the model created in PTC Creo Parametric 8.0.4.0. Instead of using "driving" dimensions as in example 1 or "driven" dimensions as in example 2, this time the linear dimensions are created using the third option available in PTC Creo, the so-called "annotation features". "Annotation features" are features that can contain annotations like driven dimensions, GD&T and others. These annotations are also described as annotation elements owned by the annotation feature (Fridman 2019) (Figure 3.47).

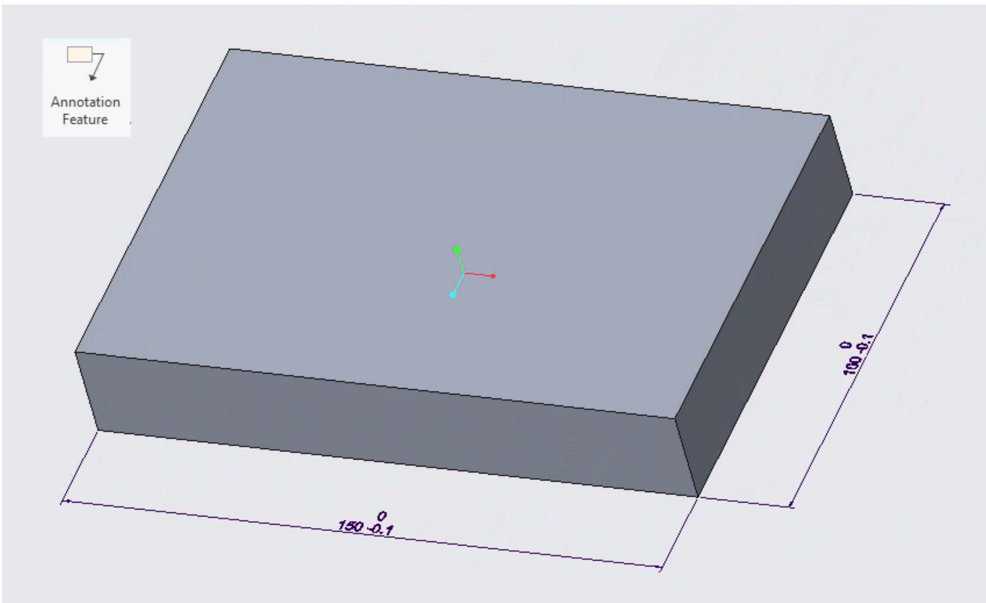


Figure 3.46: Creo 8 - linear dimensions created embedded in an "annotation feature"

This model is exported to a STEP AP242 file (edition 1), ensuring that the option to export PMI data is activated (Figure 3.28). What exactly "edition 1" means is described in Chapter 6, which is dedicated to neutral exchange files.

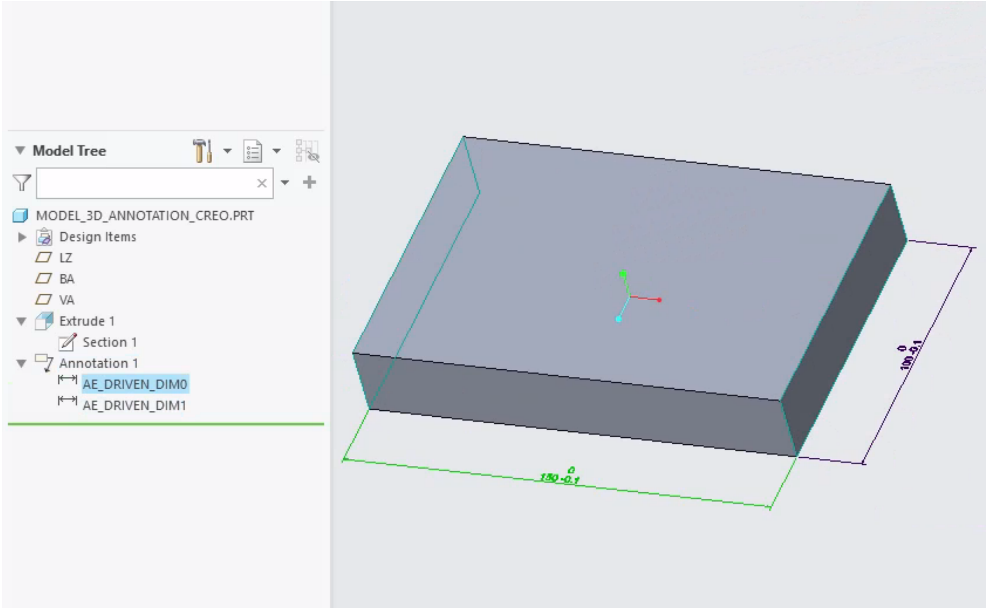


Figure 3.47: Creo 8 - annotation elements owned by the annotation feature

When this STEP file is imported into Siemens NX 2019, Build 2501, it is important to also enable the detection of PMI data within STEP files (Figure 3.29). The linear dimensions that were present in the native Creo model are not visible in Siemens NX (Figure 3.48). However, the dimensions are present within the imported model (Figure 3.49) and all the data (values and references) attached to these dimension objects is retained (Figure 3.50). The dimensions are not visible because their display depends on the view they are linked to. In Creo only one combination view with the name "Default All" was active (Figure 3.33). This view contained the complete 3D model with the 3D annotations. Within Siemens NX this view must be activated first (Figure 3.34) in order to enable the display of the dimensions (Figure 3.51). From this it can be concluded that when 3D annotations are created in PTC Creo 8 as "annotation features", the information of the annotations is preserved as they are stored as "representation PMI" within the STEP file.

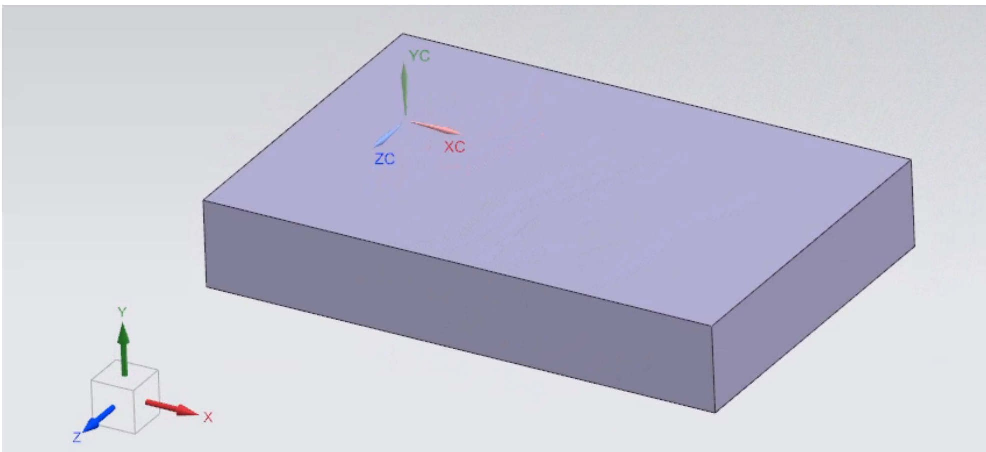


Figure 3.48: NX - linear dimensions are not visible after import

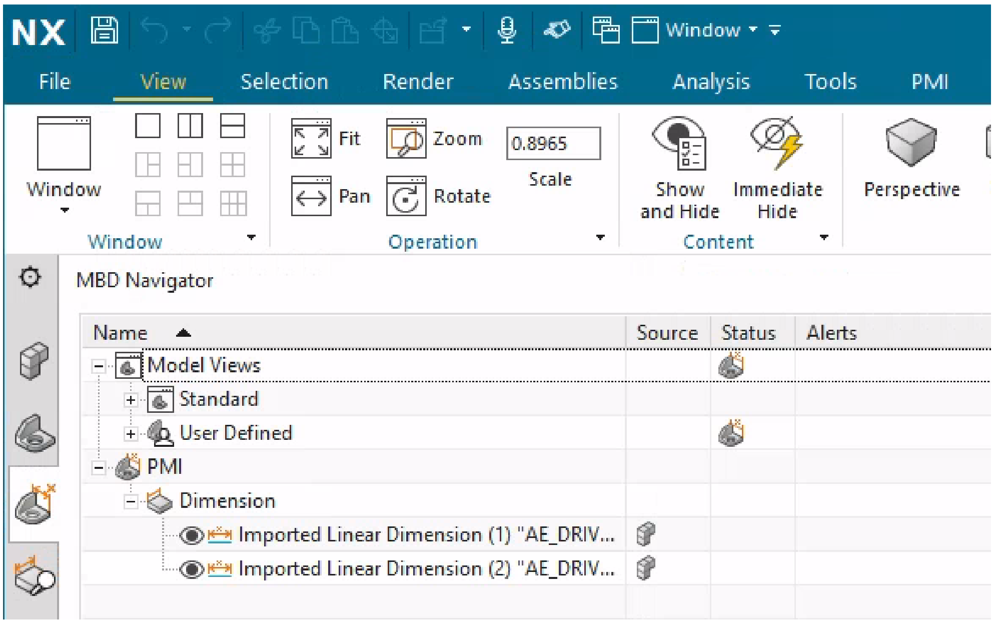


Figure 3.49: NX - linear dimension objects present in NX model

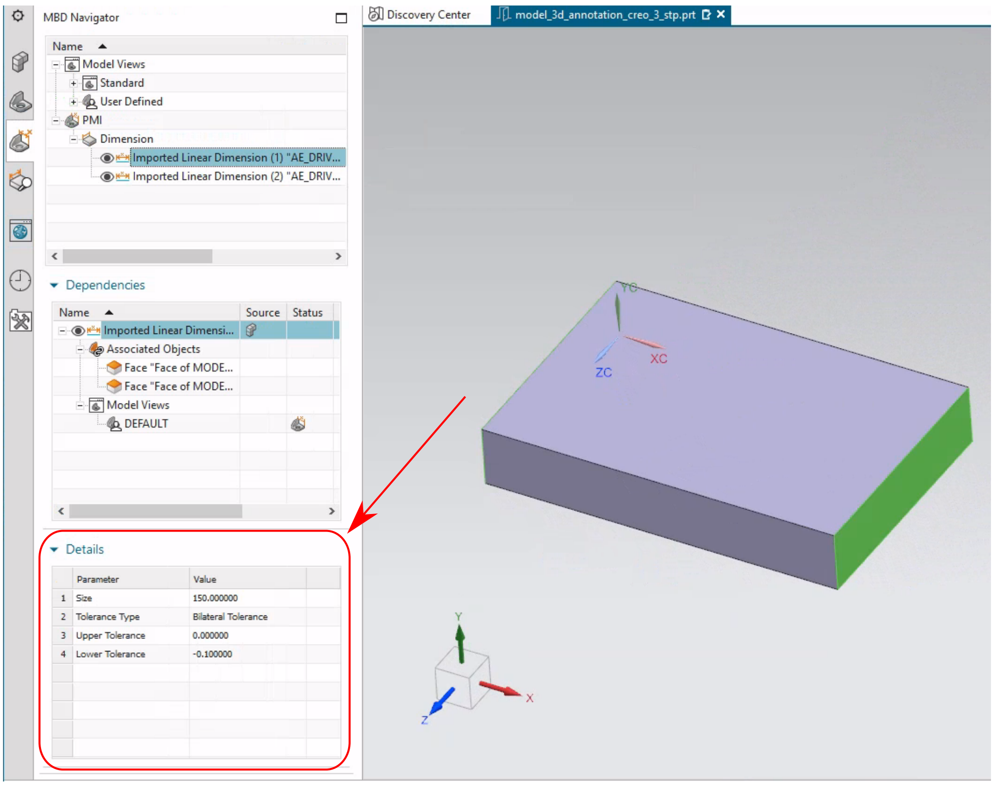


Figure 3.50: NX - linear dimension objects present in NX model with data

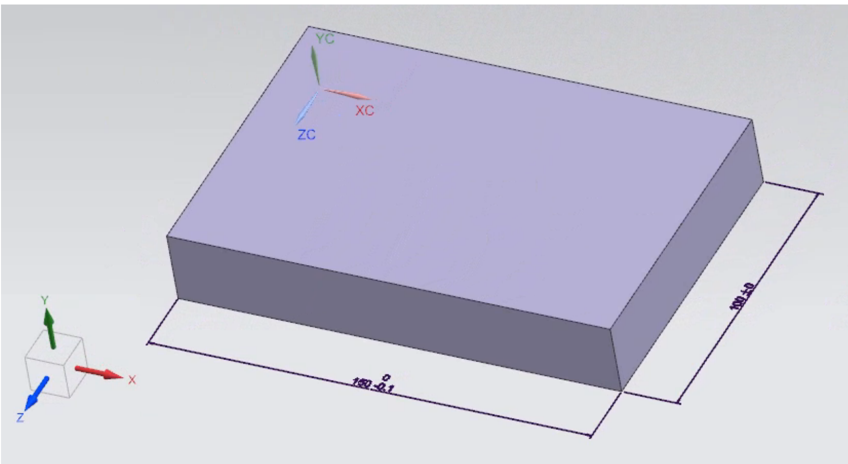


Figure 3.51: NX - linear dimensions are visible when the correct view is activated

When the STEP file that is exported by PTC Creo Parametric 8.0.4.0 is imported into Autodesk Inventor 2022, it is important to also enable the detection of PMI data within STEP files (Figure 3.16). The name "graphical PMI" suggests that the PMI data is imported only as "presentation PMI". In contrast to Siemens NX, the linear dimensions that were present in the native Creo model are visible in Inventor without any additional intervention by the user (Figure 3.52). However, there is no data attached to these dimensions objects (Figure 3.53). From this it can be concluded that indeed the 3D annotations can only be retrieved as "presentation PMI".

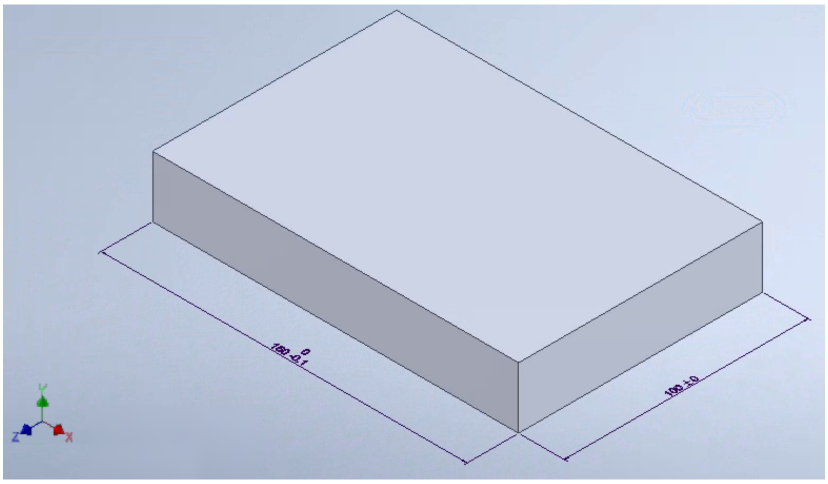


Figure 3.52: Inventor - linear dimensions are visible after import

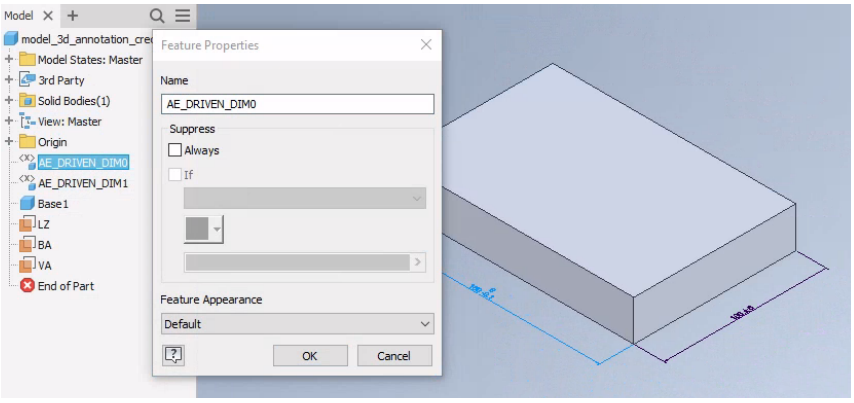
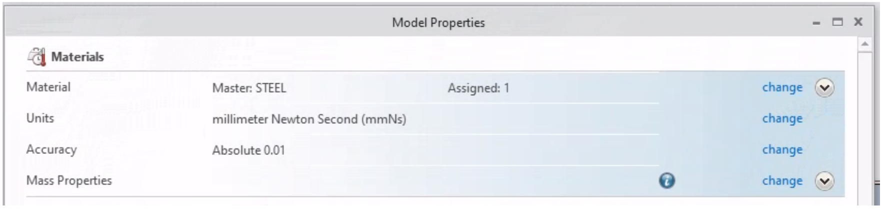


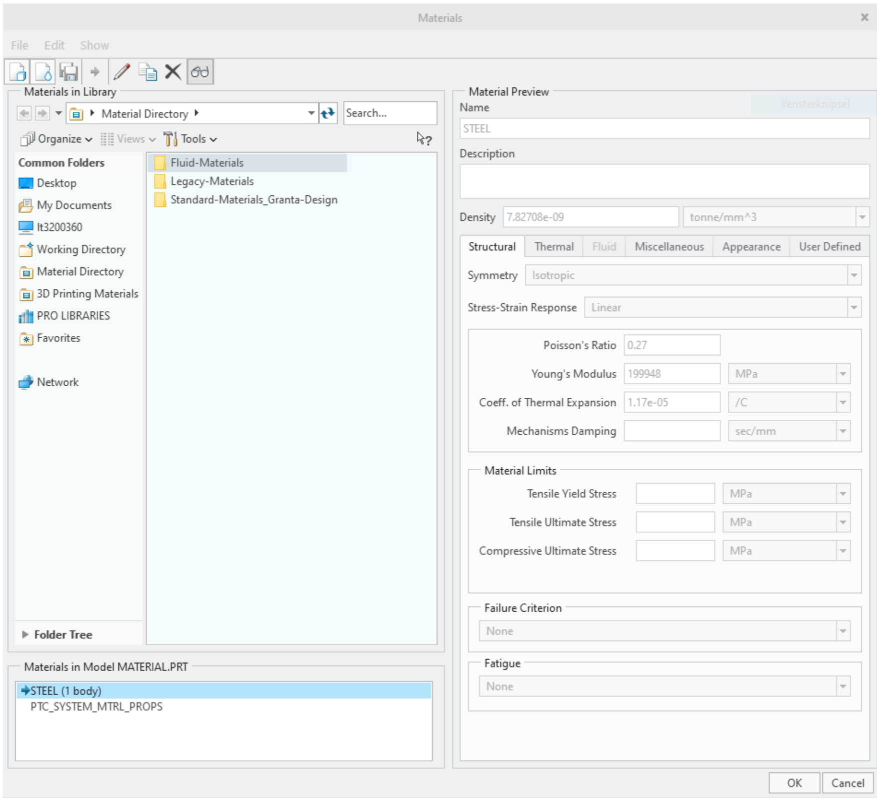
Figure 3.53: Inventor - linear dimensions are visible but no data attached to it

3.2 Metadata

Metadata is all data that is not visible on the CAD model, but is stored internally in the model's data structure. All the representation PMI is represented as metadata (Ramnath et al. 2020). An example of such metadata are the properties of the material from which the product will be manufactured (Gopalakrishnan et al. 2020). Figure 3.54 shows how this is implemented within PTC Creo. These material data can be automatically applied in the calculation of mass properties and in finite element calculations.



(a) Material configuration



(b) Material details

Figure 3.54: Creo 8 - product material stored in metadata

Other examples of metadata are the dimension type, the tolerance type (Jing et al. 2020a), product identification data (Alemanni et al. 2011) and semantic references. Each CAD system has its own set of metadata and its own system to implement them (Peng et al. 2020). This means that not every variable of a metadata structure of a certain CAD system has an equivalent in the data structure of another CAD system. This can have consequences for the smooth exchange of data. An example of this is the way in which threads are implemented in the various CAD systems. In PTC Creo 8.0.4.0, a hole feature with a metric thread internally consists of two other features . The first feature is the hole that has to be pre-drilled. A second feature is a cylindrical surface that is used to correctly represent the thread in a 2D drawing (see Figure 3.55).

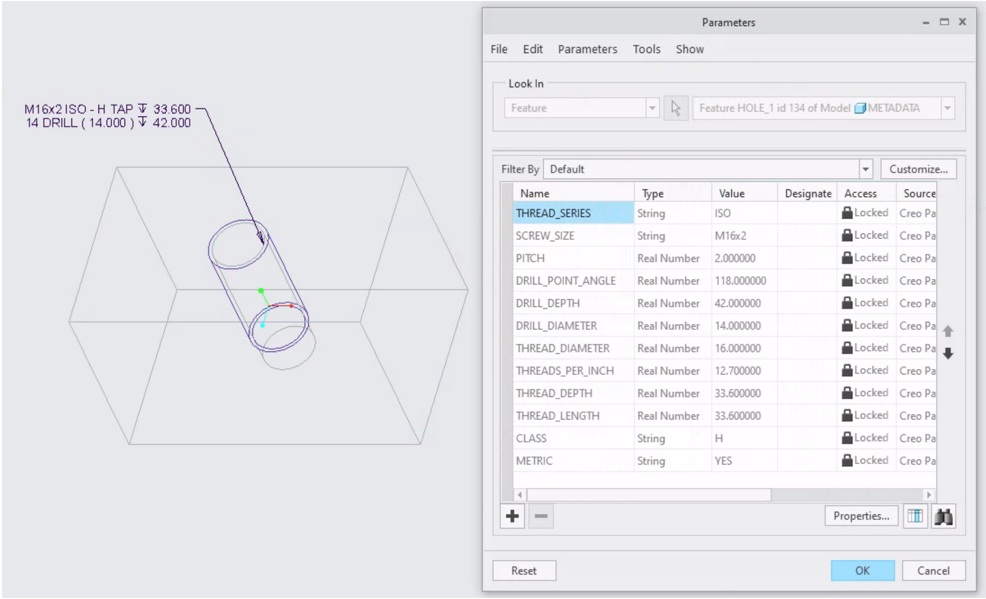


Figure 3.55: Creo 8 - Hole feature with thread (surface is shown in magenta)

In Siemens NX 2019 Build 2501 there is only one feature with its own set of parameters (Figure 3.56).

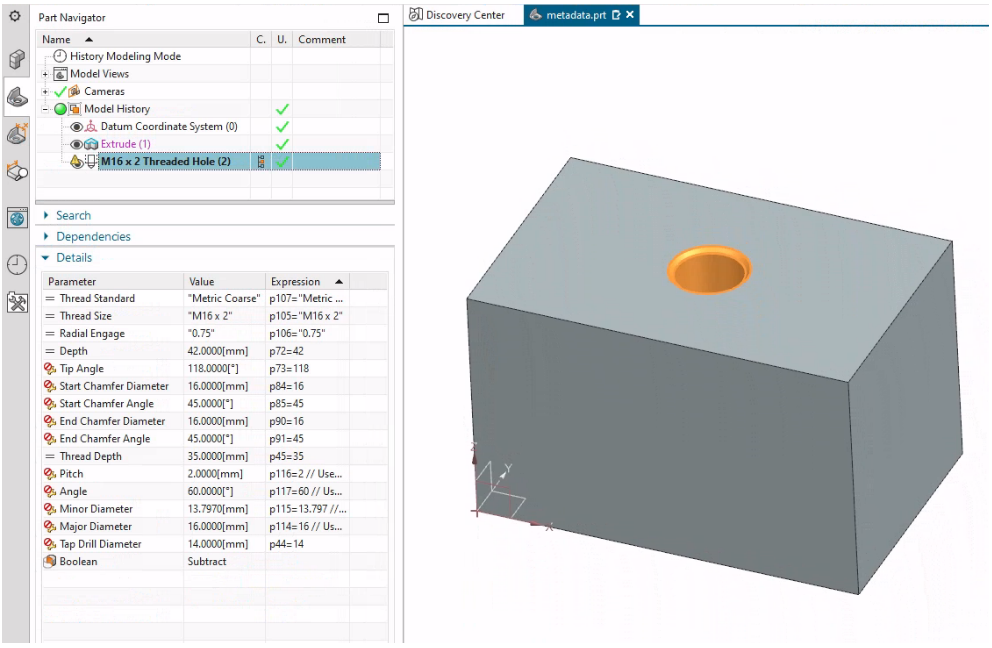


Figure 3.56: NX 2019 - Hole feature with thread

3.3 Conclusion

As previously stated in chapter 2, there are few problems if everyone involved in a project uses the same CAD system. Just as problems occur when geometry is exchanged between different CAD systems, they also occur when annotations are exchanged (H. Ma et al. 2006). A primary cause is the different implementation of annotations in different CAD/CAM systems. A second cause is that a given annotation can be created in different ways in the same CAD/CAM system. However, each of these ways is not equivalent and will behave differently when the model is exchanged with another CAD system.

4 Summary of Literature Review and Rationale for the PhD Study

Proponents of MBD claim that it makes product development easier and less time-consuming. This is claimed to be achieved by eliminating the need for 2D drawings (Liu et al. 2012) and the need to regenerate data over and over again. The latter refers to

1. Automatic generation of First Article Inspection Documents (abbreviated FAID) (Buscei 2018)
2. Automatic generation of measurement programmes for CNC coordinate measuring machines (CMMs) (Zhong et al. 2021)
3. No need to recreate 3D models from 2D drawings for use in CAM systems (Trainer et al. 2016)
4. Using PMI to automatically generate toolpaths in CAM systems (Morey 2020).

4.1 Automatic generation of FAID

Based on discussions with representatives of companies in the US during this PhD study, the claim *Automatic generation of FAID* seems to be true. It was stated that these documents were previously mostly generated manually by examining the 2D drawings. This was a time-consuming and error-prone process. If representation PMI is used in the MBD model, this can be executed automatically. Representation PMI here means that the type of annotation (linear dimension, diameter dimension, angular dimension, Geometrical Dimensioning & Tolerancing) can be determined. It is not necessary to assign semantic references to it. For example, it is not necessary to be able to determine the entities between which a linear dimension indicates distance. It is sufficient to note that the dimension is linear, together with the nominal value and the tolerance assigned to it.

4.2 Automatic generation of measurement programmes

In order to enable the automatic generation of measuring programmes for CMMs, the annotations must not only be machine-readable, but must also be assigned semantic references. It is imperative that these semantic references correctly and unambiguously reflect what the linked dimensions refer to. The designer must therefore not only ensure that the MBD model has the correct dimensions and tolerances, but also that they have all been assigned the correct semantic references. So while the designer's task may be made easier by having more options for dimension placement and no longer having the sometimes difficult task of ensuring the correct views and cross-sections, it is made more difficult again by the mandatory assignment of semantic references.

4.3 No need to recreate 3D models from 2D drawings

The claim that manufacturing no longer needs to recreate the 3D model from 2D drawings is false. For almost 30 years the 3D model (the native model or a STEP model) is used as the basis for CNC programming, with the 2D drawing serving as additional information to know what specific tolerances, ... that deviate from the general tolerance, should be applied in which places (Pratt 2001). The only difference is that MBD does not require additional 2D drawings to know which dimensions have tolerances that deviate from the general tolerance. These are now applied to the 3D model itself.

4.4 Automatic toolpath generation

Nyffenegger Felix et al. 2020 states "Automatic toolpath generation is a distinct but repeatedly gained benefit enabled by MBD". However, with regard to the claim of automatic toolpath generation in CAM systems, the research and interviews conducted during this PhD study did not provide any evidence to substantiate this claim.

4.5 Rationale for the PhD Study

4.5.1 Implications of CAD Model Dimensioning vs. Model Based Definition

The fact that 2D drawings are no longer needed does not make the job of the manufacturing department any easier. If asymmetric tolerances are applied in the design (a common practice to ensure that mechanical components can be assembled correctly (Maghsoodloo et al. 2000)), it may be necessary to adjust the 3D model to position the nominal value of a dimension at the centre of the assigned tolerance range. This is necessary to generate the correct CNC toolpaths. A CAM package always uses the nominal dimensions of the CAD model when calculating toolpaths. The ease of achieving this adjustment to the centre of the assigned tolerance range using the native CAD model will depend on how the designer has constructed the model. If the designer has considered both the principles of functional dimensioning and design intent when creating the features the CAD model is built with, the task of the manufacturing personnel will be made easier. This consideration is largely overlooked in the available literature on MBD, where any method of creating a CAD model is considered equivalent. Consequently, one of the research questions of this thesis is: how do the principles of functional dimensioning and design intent in the creation of CAD models influence the ease with which 3D models can be adjusted to accommodate asymmetric tolerances in the context of MBD? To investigate this, the methods available for creating dimensions in some commonly used CAD systems (PTC Creo, CATIA, Siemens NX, Autodesk Inventor) are examined. A distinction will be made between dimensions derived from functional dimensioning and independent dimensions. This will be discussed in more detail in Chapter 5. Some MBD advocates are calling for a ban on the use of asymmetric tolerances. However, this has consequences which will be examined. The prohibition of asymmetric tolerances is discussed in Chapter 7.

4.5.2 Neutral exchange formats

If the CAD model that needs to be adjusted to position the nominal value of a dimension at the centre of the assigned asymmetric tolerance range is not the native model, but has been transferred via a neutral exchange format such as STEP, the process becomes more complex. In the MBD philosophy, the 3D model is the authority (Gregorio et al. 2023). This means that only dimensions with assigned tolerance ranges that deviate from the general tolerance are explicitly assigned in the model (Mohammed 2023). The logical consequence of this is that one must be able to rely completely on the accuracy of the CAD geometry. The literature review shows that this is almost always taken for granted. Not all CAD/CAM systems use the same mathematical libraries (the so-called kernels such as Parasolid, ACIS, CGM, Granit, ...) to describe the geometry of the model. Not all the stakeholders involved in the creation of a product (designers, manufacturers, quality control, ...) use the same CAD/CAM system. This can lead to problems when exchanging models between them. This leads to the following question: how do differences in the mathematical kernels used by different CAD/CAM systems affect the exchange of models between stakeholders involved in the product realisation process, and what potential problems can arise when neutral exchange files are used? To investigate this, a method will be developed to check how much a model exported from one CAD system to IGES and STEP and imported back into another system differs from the original. At the same time, it will be investigated whether this transfer involves data loss in annotations and features such as threaded holes in the MBD model. This will be discussed in more detail in Chapter 6.

4.5.3 Software development

The general conclusion is that the main beneficiary of MBD is the quality department. For the manufacturing department the benefits are much less. The demands on a designer for an MBD model are much higher than for traditional 2D drawings. It is no longer enough to know how to correctly dimension a model and apply tolerances, but now it is also necessary to know all the intricacies of the CAD system used (which method of dimensioning preserves data when exported to a neutral exchange format) and meticulously assign the correct semantic references. In order to assist the designer with this and to help the manufacturer with making the CAD model suitable to generate the correct CNC toolpaths to produce the part, a software package will be developed. To ensure that th software package is actually usable, feedback is sought from test users in local companies. This software package is discussed in Chapter 8.

5 Implications of CAD Model Dimensioning vs. Model Based Definition

5.1 Introduction

"Re-use your CAD data" is one of the most touted benefits of MBD (J. B. Herron 2013). There are four cases where this re-use may be applicable.

The first case is the creation of First Article Inspection Documents. Here the use of MBD models is indeed very convenient and time-saving. When representation PMI is supported by the CAD system and applied by the designer, the contents of the annotations can be read out and listed by a software package. This saves the time-consuming manual preparation of these lists and prevents errors that can occur due to incorrect retyping of data. This is indeed real re-use of the CAD data. This use case does not make high demands on the designer. It is sufficient to simply annotate the 3D model using representation PMI.

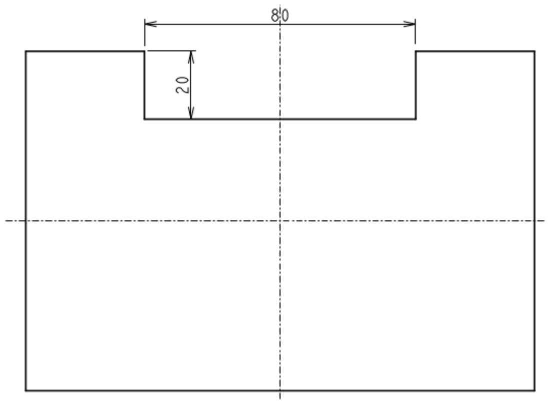
The second case is the generation of measurement programmes. In order to do this, being able to just read the contents on an annotation is not enough. The content on an annotation, such as the dimension type, the value of a dimension and the associated tolerances, must be interpretable and the semantic references to which the annotation refers must be retrievable. This use case demands more from the designer, because the designer must check whether the correct references have been assigned to the annotation. As the discussion of the examples in subsection 5.3.1 shows, this requires a thorough understanding of the capabilities and limitations of the CAD system used. Whether there is a discrepancy between the dimensioning scheme used to build the model and the one created in the MBD model has no influence. What exactly is meant by a dimensioning scheme is discussed in section 5.2.

The third case is the generation of CNC programmes. As with the previous use case, the content of the annotation must be fully interpretable and the designer must ensure that the references to which the annotation refers are assigned to the annotation. However, this is not sufficient for all use cases. A first case where higher demands are placed on the MBD model is where asymmetrical tolerances are applied. It is then necessary for the CNC programmer to be able to make changes to the model. toolpaths are created at the nominal size of the CAD model (Xu et al. 2015). If an asymmetrical tolerance is assigned to a dimension, the nominal value of that dimension must be brought to the centre of the tolerance field. Whether it is easy for the CAM programmer to do this depends on two things. The first is the CAD/CAM system used. If this is the same CAD/CAM system as the designer's or a CAD/CAM system that can read the designer's native CAD format while retaining the features used, this will make it easier for the CAM programmer but not always easy. Why this is the case will be explained later in this chapter. A second issue is how the CAD model was constructed by the designer. If there is no discrepancy between the annotations made in the MBD model and the dimensioning schemes used to build the model, it is relatively straightforward for the CAM programmer to change the model. If this is not the case, however, then it becomes a very different story and the use of techniques such as "direct modelling" may be the only solution. "Direct modelling" allows the modification of a CAD model independent of how the model was build. Whether this modification can be performed accurately enough depends on the nature of the mathematical model of the geometry (accuracy, analytical, NURBS, subdivision modelling) to which it is applied. A second case where higher demands are placed on the MBD model is when it comes to holes, ordinary holes and tapped holes. In order to achieve significant time savings, it is crucial that the CAM system is able to identify the type of hole and the depth at which it should be drilled and tapped. As discussed in the examples in subsection 5.3.1 the way a hole is annotated can not always be interpreted correctly by software. Proponents of MBD argue that this problem is solved by applying DFM, which stands for "Design For Manufacturing", in the MBD model. This is not entirely correct and is discussed in chapter 7.

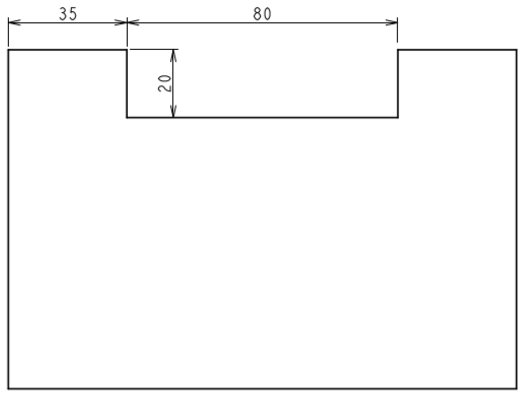
The fourth case is the derivation of new products based on the current model. The requirements already discussed in cases two and three also apply here. Whether it is possible to make adjustments to the model in a relatively simple manner depends entirely on the way in which the original designer built the model. This structure should be logical and there should be no discrepancy between the annotations in the MBD model and the dimensioning scheme used to build the features. In this way, it is easier for others to understand the "design intent" of the original designer. If it is too difficult to make the desired changes to the model, "direct modelling" is also suggested as a solution, which can lead to the same problems as described in the previous case.

5.2 Dimensioning scheme

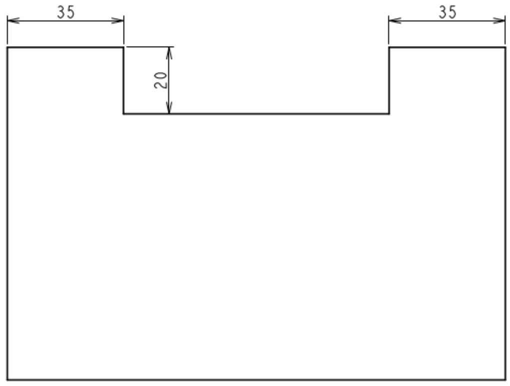
To create a CAD model, the designer uses features like extrude, revolve, sweep, hole, ... To define these features, dimensioning schemes are applied. A dimensioning scheme refers to how and which dimensions are applied. Figure 5.1 shows three different dimensioning schemes for the same sketch. Within MBD, the 3D model is the authority (Pfouga et al. 2018). This means that all nominal dimensions, which must satisfy the general tolerances applicable to the whole model, do not have to be created explicitly in the 3D model. They are determined solely by the geometry of the 3D model. Only dimensions with tolerances that deviate from the generally applicable tolerances need to be created explicitly. Figure 5.2 shows an MBD model with the dimensioning scheme for the same shape as in Figure 5.1. The dimensioning scheme used in the MBD model does not have to match the one used in the CAD system that defined the features used to create the slot. If that is the case, this means that there is a mismatch between the dimensional tolerances resulting from the way the model was created in the CAD system, on the one hand, and the dimensional tolerances defined by the MBD annotations, on the other. At first glance, this may seem odd, especially when the designer has apparently not specified any tolerances when applying dimensions to define the features used to build the CAD model. However, tolerances are always necessary in a mechanical design to ensure that the various parts fit together and function correctly (Childs 2021). To avoid having to assign tolerances to each individual dimension, an ISO standard has been created. It allows the designer to specify which tolerance should be assigned to a particular nominal size without having to explicitly assign it to the dimension. This ISO standard is ISO 2768 (ISO 1989). Some CAD systems, such as PTC Creo, automatically apply this standard to each dimension that is used to create the CAD model. Table 5.1 shows the resulting tolerances for the different dimensioning schemes when a general tolerance, ISO 2768m, is applied.



(a) Dimensioning scheme 1



(b) Dimensioning scheme 2



(c) Dimensioning scheme 3

Figure 5.1: Three different dimensioning schemes for the same shape

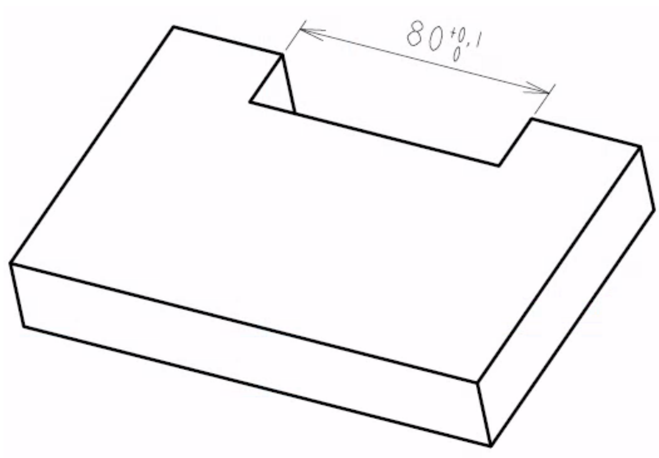


Figure 5.2: Dimensioning scheme in MBD model

Table 5.1: Resulting tolerances for the respective dimensioning schemes

	35 on the left	**35 on the right**	**80**	**total length 150**
Dimensioning scheme 1	34.6 - 35.4	34.6 - 35.4	79.7 - 80.3	149.5 - 150.5
Dimensioning scheme 2	34.7 - 35.3	33.9 - 36.1	79.7 - 80.3	149.5 - 150.5
Dimensioning scheme 3	34.7 - 35.3	34.7 - 35.3	78.9 - 81.1	149.5 - 150.5
Dimensioning scheme MBD	34.70 - 35.25	34.70 - 35.25	80.0 - 80.1	149.5 - 150.5

A CAM system generally uses nominal values to create toolpaths (Hardwick et al. 2013). When symmetrical tolerances are applied, the nominal value corresponds to the middle of the tolerance range. When asymmetric tolerances are applied, this is not the case. In the case of the slot, this means changing the nominal value of 80 to the middle of the tolerance range, i.e. 80.05. This chapter addresses two topics and their close relationship. A first is the importance of carefully considering how and which dimensions and tolerances are applied. This will be discussed in section 5.3. A second is the impact of choosing a particular dimensioning scheme on modifying the model to accommodate the generation of toolpaths. Many publications can be found about the creation and the use of annotations in an MBD model, but none could be found during this PhD study that discusses the relation between the dimensions used in feature creation and the dimensions that are created as MBD annotations.

5.3 Design intent and functional dimensioning

5.3.1 Functional dimensioning

Consider the assembly in Figure 5.3. This assembly consists of two parts A and B bolted together. The bottom part A is a component with four M3 bolt holes (Figure 5.4). A stop is provided for positioning the top part and four tapped holes for fastening. The tolerances that apply to the position of the holes must be aligned for both parts. If this is not the case, the holes may not be correctly aligned during assembly. The parts cannot then be bolted together with the four bolts.

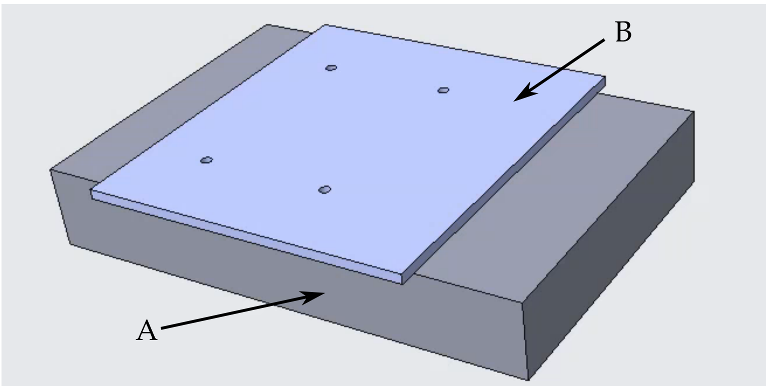


Figure 5.3: Assembly of two parts A and B bolted together

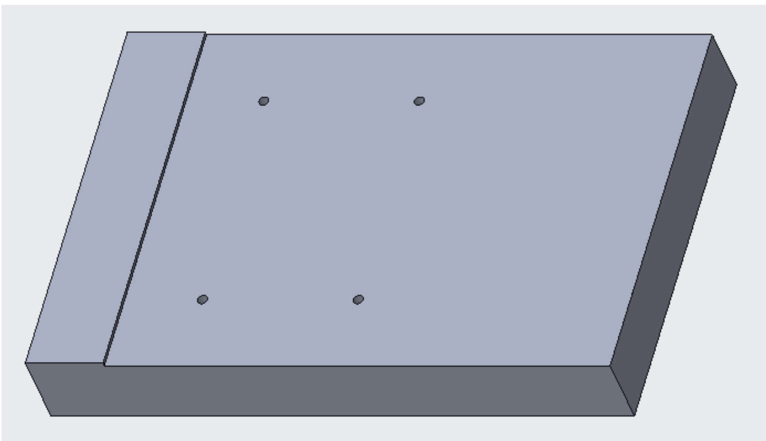


Figure 5.4: Part A with a stop and four mounting holes

There are several possible ways to dimension the four holes. One possible way for the tapped holes in part A is shown in Figure 5.5.

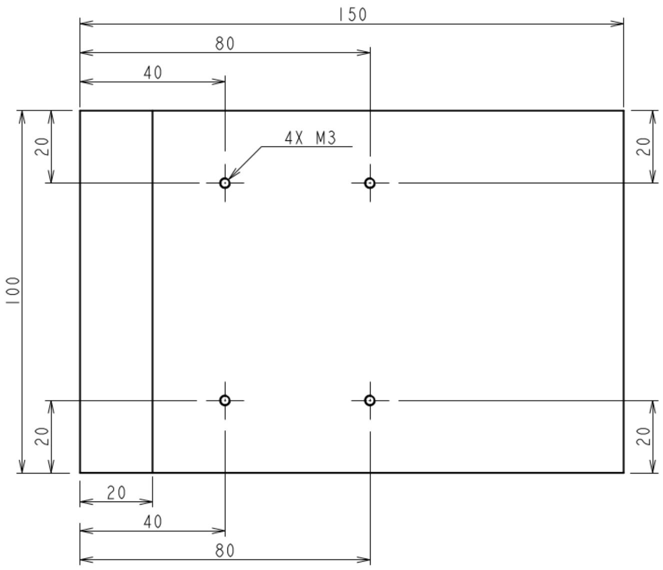


Figure 5.5: One possible way of dimensioning the tapped holes in part A

The four bolt holes are for attaching part B to part A using DIN 912/ISO 4762 socket head cap bolts M3×0.5 . As a result, the mutual position of the four holes is important in order to be able to put the four bolts through part B into the tapped holes of part A. If the component B has four precisely positioned clearance holes, the permissible tolerance on the position of the tapped holes is determined solely by the difference between the diameter of the clearance hole and that of the tapped hole. For an M3, the diameter of the clearance hole medium fit according to DIN EN 20273 is 3.4 mm. The resulting tolerance on the position of the bolt hole is thus $\frac{3.4-3}{2} = \pm 0.2$ mm (Figure 5.6).

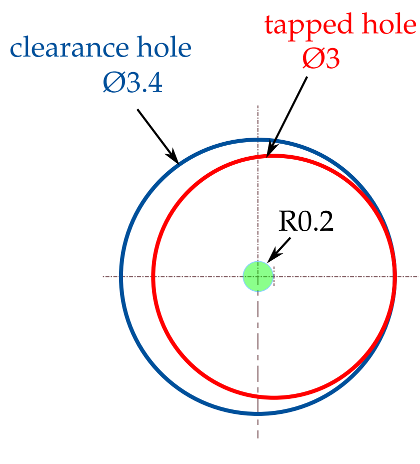
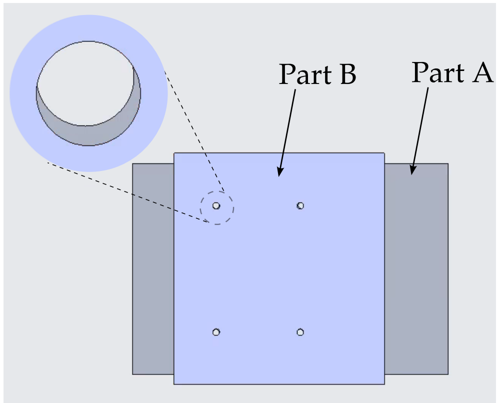


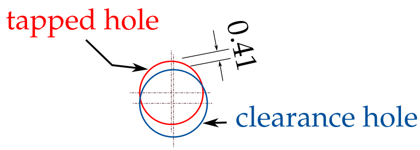
Figure 5.6: The permissible tolerance represented as the radius of the green circle in which the centre of the tapped hole should lie

According to DIN ISO-2768-1 the medium tolerance for a length of 40 mm, 80 mm and 100 mm is ±0.3 mm. For a length of 20 mm this is ±0.2 mm. If the tapped holes are created in the extremes of these position tolerances and the clearance holes are created precisely on the nominal values and part B is placed against the stop on part

A and positioned symmetrically, the four bolts can not be put into the tapped holes as the tapped holes in part A are partially covered by the clearance holes in part B (see Figure 5.7). The maximum value of the overlap in this scenario is 0.41 mm. For another scenario where one tapped hole is aligned with the corresponding clearance hole, the maximum value of the overlap increases to 0.50 mm and 0.82 mm.



(a) View on the positioning of part B on part A



(b) Tapped hole of part A is partially covered by the clearance hole of part B

Figure 5.7: When part B is aligned against the stop on part A and placed symmetrically the position of the holes does not allow the placing of the bolts

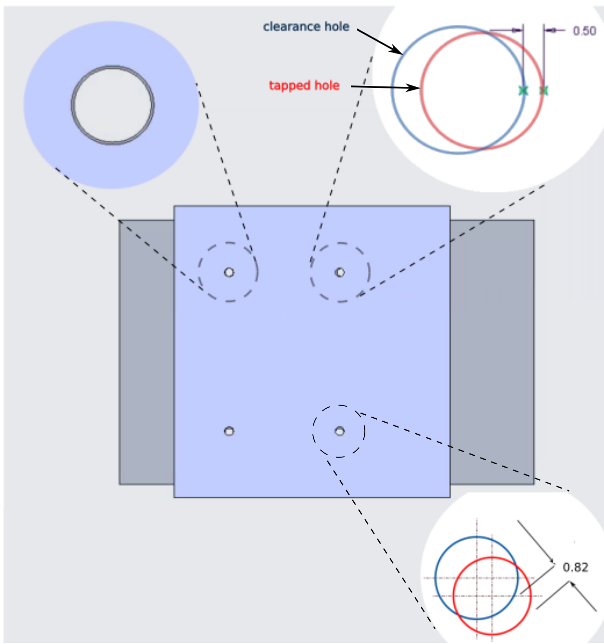


Figure 5.8: When one clearance hole of part B is aligned with a tapped hole of part A the overlap of the other tapped holes increases to 0.5 mm and 0.82 mm

Based on the fact that the overlap referred to in the above example makes it impossible to screw the four bolts into the corresponding tapped holes, the conclusion is that the specified tolerance is not suitable for this application. A possible solution with proper tolerancing for the tapped holes in part A is shown in Figure 5.9. As mentioned earlier, it is conveniently assumed that the clearance holes in part B are precisely positioned.

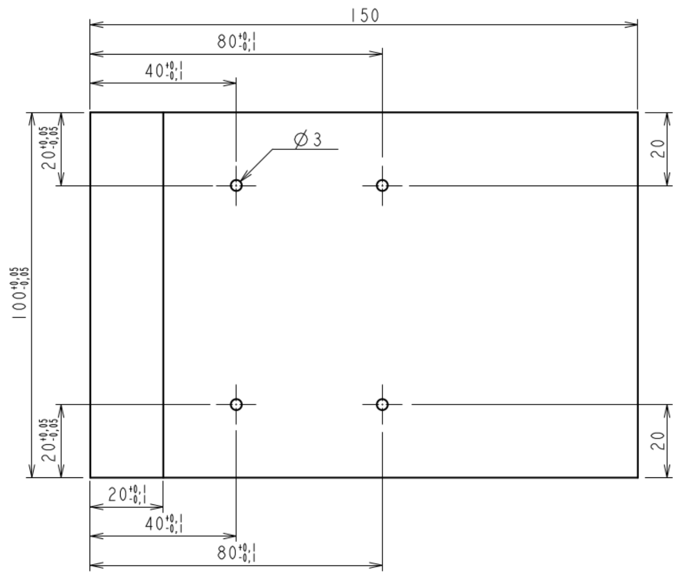


Figure 5.9: One possible attempt to solve the tolerance problem

In this solution, eight measurements are assigned a specific tolerance. The way this is done guarantees a maximum deviation of the centre of the tap holes of part A from the clearance holes of part B of 0.2 mm in each direction. This corresponds to permissible tolerance shown in Figure 5.6. However, several dimensions with lower tolerances (±0.1) are needed to achieve this tolerance. Omar et al. 2011 argues that this leads to higher manufacturing costs.

A better solution is to dimension the holes according to their function (Weill 1988) (see Figure 5.10). This is called "functional dimensioning" (Islam 2004). "Functional dimensioning" here leads to fewer dimensions with tolerances needed to ensure the bolts always fit in the tapped holes. This solution is optimal for both quality control costs and for manufacturing costs (Weill 1988, p. 43).

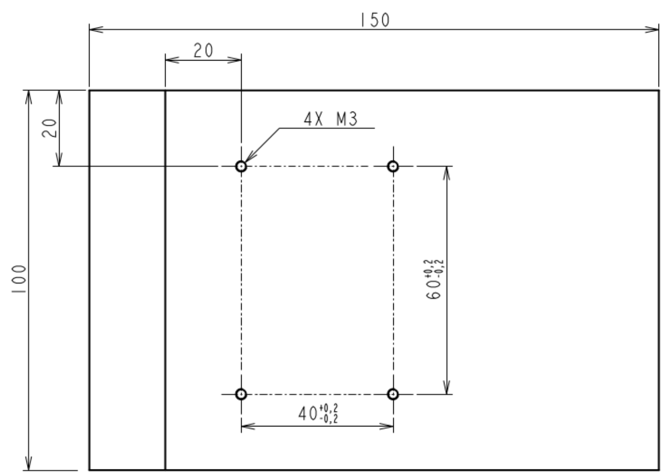


Figure 5.10: The tapped holes of part A dimensioned according to "functional dimensioning"

Modern CAD systems give the designer the freedom to create the holes within the MBD model using features and dimensioning schemes as the designer sees fit (Saal et al. 2021). This means the holes can be created using the dimensioning scheme that can be seen in Figure 5.9. As Jing et al. states 'the problem is that it cannot reflect the design intents and the manufacturing information of the process model'. In this case, it means that the dimensioning scheme used to create the holes does not match the dimensioning scheme that is most optimal for creating the holes. To circumvent this problem the final dimensioning for production is then applied on top of the 3D model (see Figure 5.11).

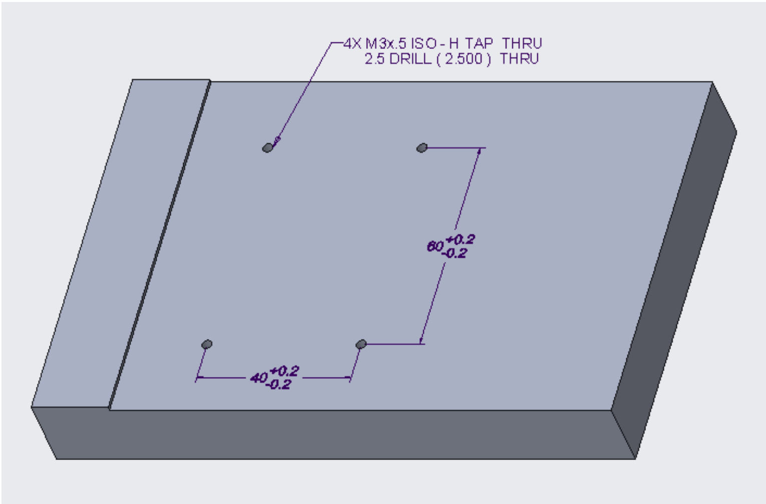


Figure 5.11: Dimensioning of the holes in the MBD model optimised for production

The result is that there is a discrepancy between the dimensioning scheme used to create the hole features (Figure 5.5) and the dimensioning scheme relevant to production (Figure 5.11). This means that changing the positions of the holes by using the dimensions used to make the holes may cause the dimensioning in the MBD model to become invalid.

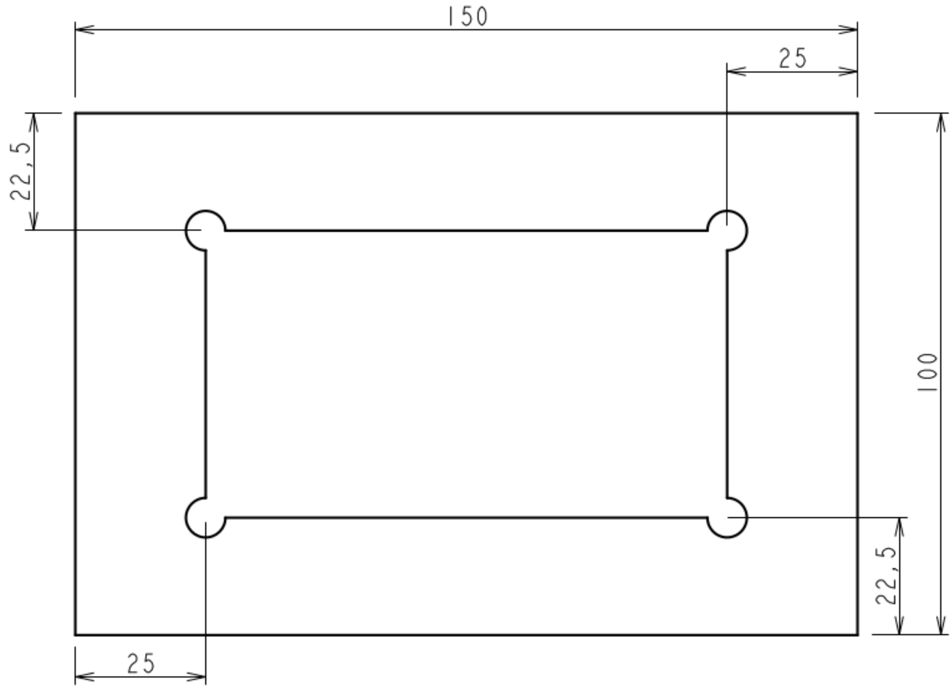
Consequences for the application of MBD

One of MBD's slogans is "re-use your data" (J. B. Herron 2013). This means that data can be passed between all the people involved in the realisation of a product without having to re-create data. Morey 2020, p. 1 says this would make it possible, for example, to use the MBD model directly for quality control and for generating CNC toolpaths. At several meetings organised by CAD vendors and attended during this PhD research, this statement was repeated. "Re-use your data" thanks to MBD was cited as the enabler to automate not only quality control but also CNC programming. When asked for examples of the latter, the speakers always referred to the generation of CNC programmes for drilling and tapping holes. Other applications, they said, were also possible. However, they could not give any concrete examples. The speakers admitted that generating CNC programmes for drilling and tapping holes would only work if both the designer and the CNC programmer used the same CAD/CAM package. It became clear that the speakers' opinions were based on the assumption that both quality control and production use the 3D CAD model in the same way. This assumption implies that both only consume the 3D model, meaning they simply use the model as is. Automatic generation of First Article Inspection Documents and programming of coordinate-measuring machines (CMMs) only require that the PMI (dimensions, assigned tolerances, GD&T) with the associated semantic references are machine-readable (Husted 2019). It may therefore be concluded that with regard to quality control, this assumption holds true. With regards to manufacturing, this is not the case. It is assumed that manufacturing personnel re-create the 3D model they need for programming the CNC in their CAD/CAM system based on the 2D drawings of the product to be made (Rowe 2017). It is also believed that this sometimes happens because a company considers the 3D CAD model to be the holder of its intellectual property and for that reason gives a 2D drawing to another stakeholder instead of the 3D CAD model (CADimensions 2022). In most cases, however, the "problem" of not wanting to pass on the original CAD model for intellectual property reasons is irrelevant, as the model is passed on as STEP because the other stakeholder does not have the same CAD system. For almost 30 years the STEP model is used as the basis for CNC programming, with the 2D drawing serving as additional information to know what specific tolerances, ... should be applied in which places (Pratt 2001). As some of these tolerances are asymmetric, manufacturers must adjust the model to match the nominal values of the CAD geometry to the centre of these asymmetric tolerance bands (Maghsoodloo et al. 2000). "Re-use your data" therefore means more than just consuming the 3D CAD model. It also means that production should be able to modify the model easily. Depending on whether the 3D CAD model is the original CAD model or a STEP model, different techniques often have to be used for this. In the case of the original CAD model, this means that if the dimensioning scheme used to create the features does not match the dimensioning scheme assigned to the CAD model as 3D PMI (see Figure 5.12), it may become more difficult to make the necessary changes and still meet all specified tolerance requirements.

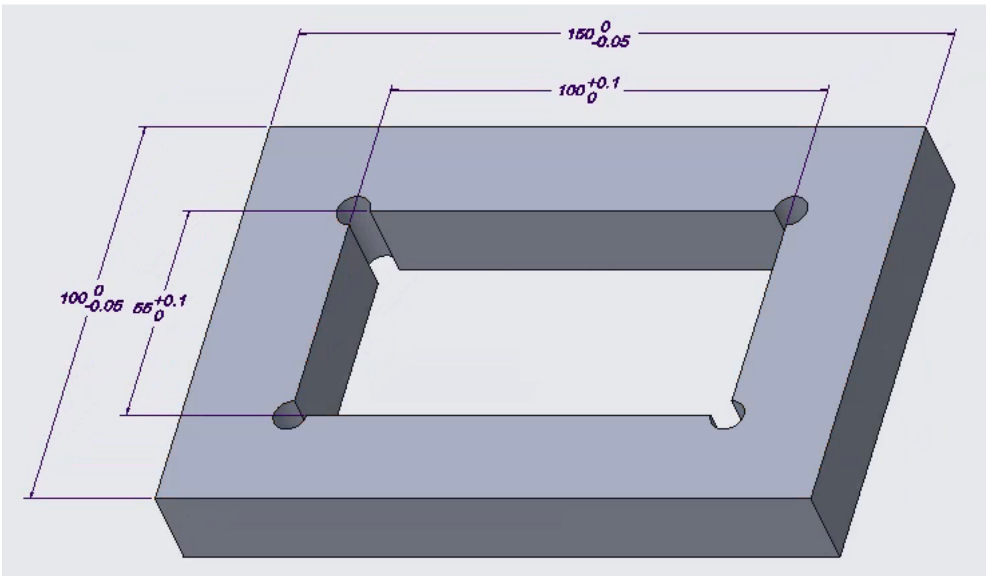
If the dimensioning scheme used to create the CAD model matches that in the MBD view, it is more straightforward to adapt the model to the specific needs of the manufacturing. This would also make it possible to quickly and easily make changes to an existing model to create a new design (Barbero et al. 2017). An example of such a change is the altering of a dimension value. The model should then still regenerate without a feature or an assembly structure crashing. To make this possible, the designer has to think very carefully about how to build the model taking into account possible future modifications. This is called "design intent". Otey et al. 2014 define this as 'Design Intent is a term commonly defined as a model's anticipated behavior once it undergoes alteration'. They also state there is no standard to communicate this. They consider "design intent " even a tacit transfer of knowledge. Alducin-Quintero et al. 2012 suggest that 3D model annotations could be used as a tool for improving this communication. However, Alducin-Quintero et al. seem to ignore the fact that "consuming" information does not have the same meaning for every stakeholder as shown earlier. Ideally "design intent" should be combined with "functional dimensioning" (Otey et al. 2014). To make this possible, the designer must not only understand design methodologies and manufacturing methods, but also the intricacies of the CAD package used (Barbero et al. 2017). However, there are issues that may prevent the designer from achieving this in all cases. To illustrate this the holes in Figure 5.10 will be created in PTC Creo Parametric 8.0.5.0. Creo Parametric gives the designer a number of ways to do this, as is the case in other CAD systems. Which one is chosen depends on the designer's preference and on which method is easiest to create the holes. Some of the options are:

1. the creation of four individual threaded holes using the standard hole feature
2. the creation of four threaded holes by creating a pattern from a threaded hole feature
3. the creation of four threaded holes using the sketched hole feature

As discussed earlier, machine readability of annotations includes the ability to read the data of the annotations and to query the semantic references. A distinction is made



(a) Dimensioning scheme used to create the pocket feature in the CAD system



(b) Dimensioning scheme used to dimension the pocket feature in the MBD model

Figure 5.12: The dimensioning scheme used to create a CAD model may differ from that applied in MBD

between presentation PMI (annotations without any intelligence that are simply displayed) and representation PMI (annotations whose data can all be read as parameters and have semantic references indicating what they refer to). However, no consideration is given to the implications of using particular features of a specific CAD system. It is assumed that all features are equivalent and interchangeable. This is not correct as will be shown in the following subsections where the dimensioning scheme of the three options mentioned above will be compared with the functional dimensioning scheme applied in an MBD view (Figure 5.13).

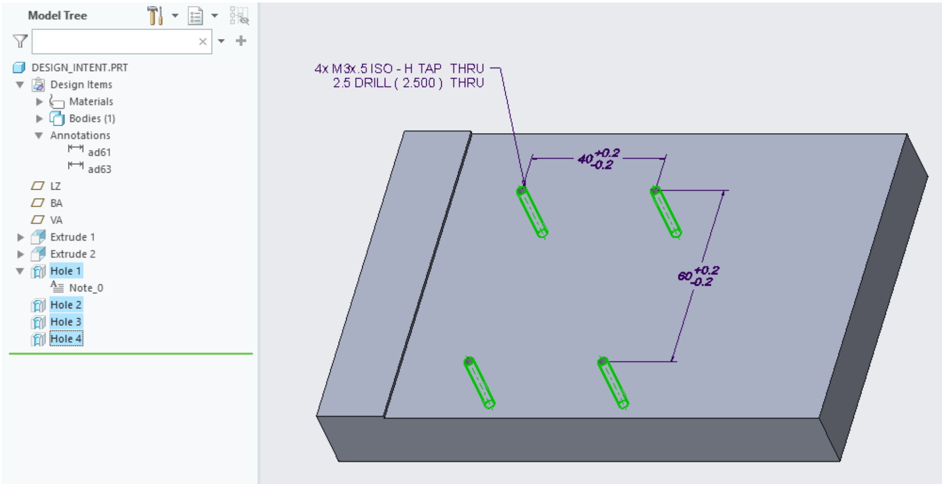


Figure 5.13: functional dimensioning scheme applied in an MBD view

Option 1: The creation of four individual holes using the standard hole feature

The underlying dimensioning scheme used to create the holes by using 4 features can be seen in Figure 5.14. Hole 2 and 4 are dimensioned with respect to hole 1. Hole 3 is dimensioned with respect to hole 2 [1].

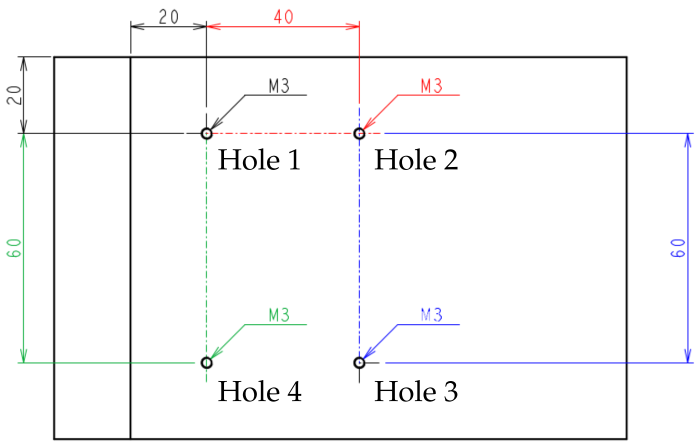


Figure 5.14: Dimension scheme used to create the 4 holes in the CAD system (option 1)

On first inspection, the dimensioning scheme of the MBD view in Figure 5.13 looks very similar to the dimensioning scheme of the holes in Figure 5.14. However, the

[1] When in PTC Creo a linear dimensioning scheme is used to create a new hole feature, it is not possible to dimension the hole with respect to two other holes when using axes. If the axis of a hole is selected as a reference, the two linear dimensions specifying the position of the new hole can only refer to the selected axis or a plane.

latter does not meet the "design intent" of the "functional dimensioning" of the MBD view. This becomes clear when the nominal value of a dimension used to position a hole is altered. It is the designer's responsibility to change the other nominal values in such a way that the correct grouping of the four holes is maintained with respect to their function. Thus, there is no correct "design intent" that ensures the change remains consistent with the functional dimensioning in the MBD view, as the grouping of the holes is broken (Figure 5.15).

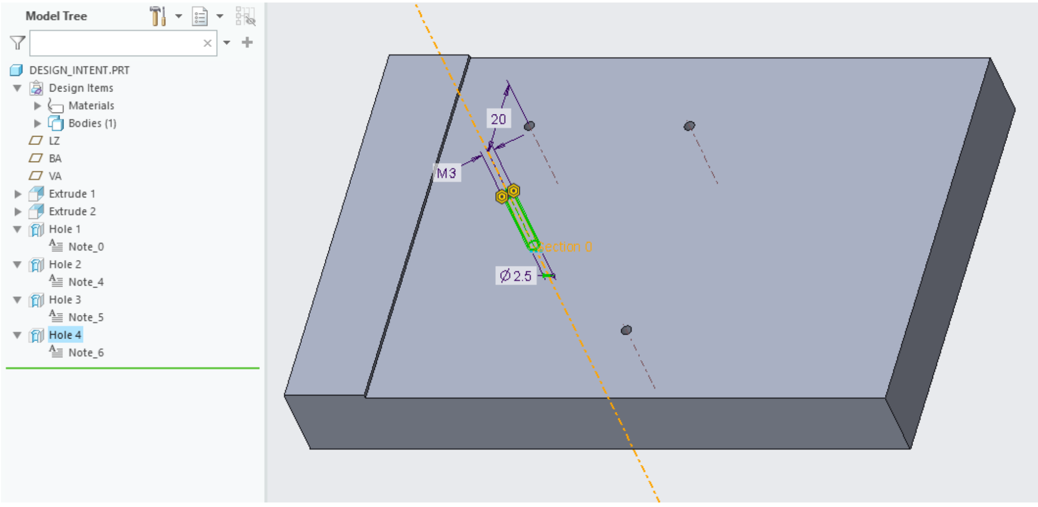


Figure 5.15: Altering the value of the vertical dimension of hole 4 places the hole outside the rectangular grouping and breaks the functional dimensioning scheme of the MBD view

As previously discussed, the "machine readability" of the annotations includes the ability to read out the semantic references. In PTC Creo Parametric, a distinction must be made between two types of annotations. The reason for this is that there is a difference in

1. how they are generated
2. how they are updated
3. how much information is retained when exporting to an external format such as STEP AP242.

These two types of annotations are:

1. annotations generated within or during the creation of a hole feature. An example of this is the note "4X M3x.5 ISO. . . "
2. annotations based on the dimensioning scheme of a feature, these are created in a separate step after the creation of the feature:
   - taken directly from the dimensioning scheme used to create the feature, so-called driving dimension (see subsection PTC Creo Parametric)
   - derived from the model, independent of the dimensioning scheme used to create the feature, so-called driven dimension (see subsection PTC Creo Parametric)

Other CAD systems may have similar features or functionality.

In the case of the annotation "4X M3x.5. . . " in the MBD view, this annotation refers to the cylindrical surfaces of the four holes. However, in the creation of the hole features, the four holes are created as four individual features. This means that without designer intervention, this results in four individual automatically generated annotations as shown in Figure 5.16.

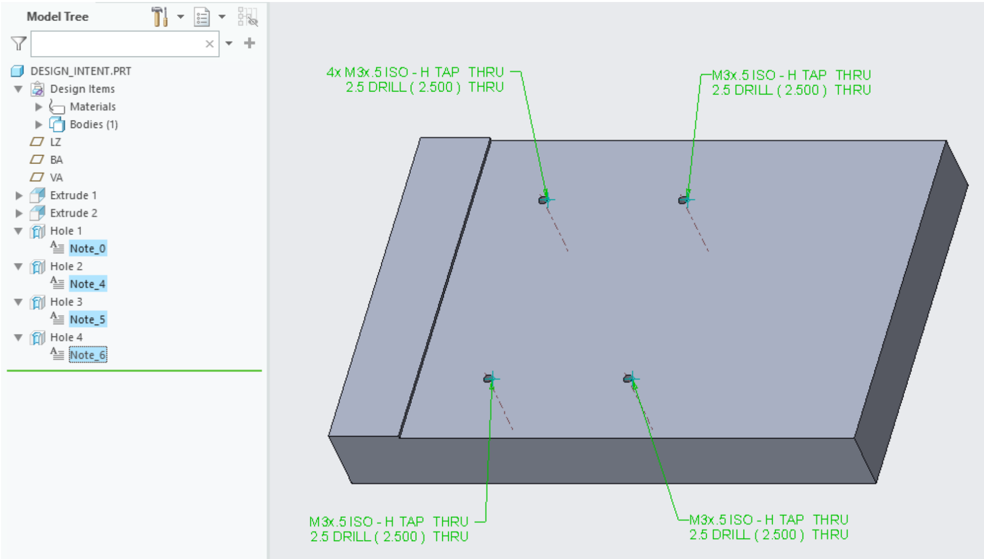


Figure 5.16: Four independent hole features each with their own automatically generated annotation indicated in green and highlighted in the feature tree on the left-hand side

Based on this, each hole feature can be expected to have an annotation with its own set of semantic references. Since there are four holes, four sets of semantic references are expected. However, running the "semantic query" function, which is a function in PTC Creo to display semantic references assigned to an annotation, shows no references have been assigned. This is because the annotation is a note embedded in the hole feature (see Figure 5.17).

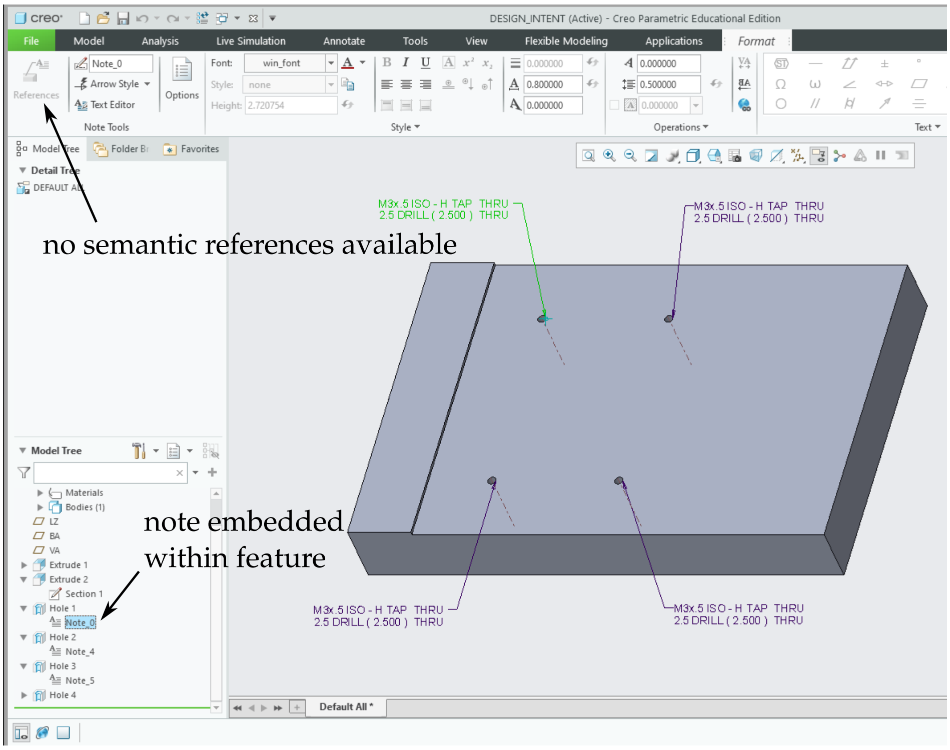


Figure 5.17: Hole annotation is an embedded note with no semantic references

The note itself makes use of parameters that are only available within the feature (see Figure 5.18). To be able to assign semantic references, the annotation must be an “annotation element” or belong to an “annotation feature” (see subsection PTC Creo Parametric). “Annotation elements” and “annotation features” are a kind of stand-alone annotations. Since the annotation in this case is a note embedded in the hole feature, it is fully associated with this feature. This makes it impossible for the designer to assign anything to the annotation itself, such as semantic references. To assign semantic references, it must be decoupled from the hole feature. This can be done by using the “Change parent” function and attaching the annotation to the CAD model instead of the hole feature (see Figure 5.19).

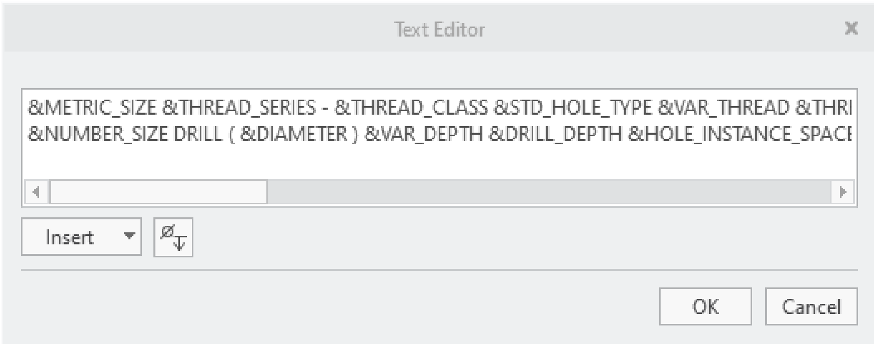


Figure 5.18: The note describing the hole uses internal feature parameters

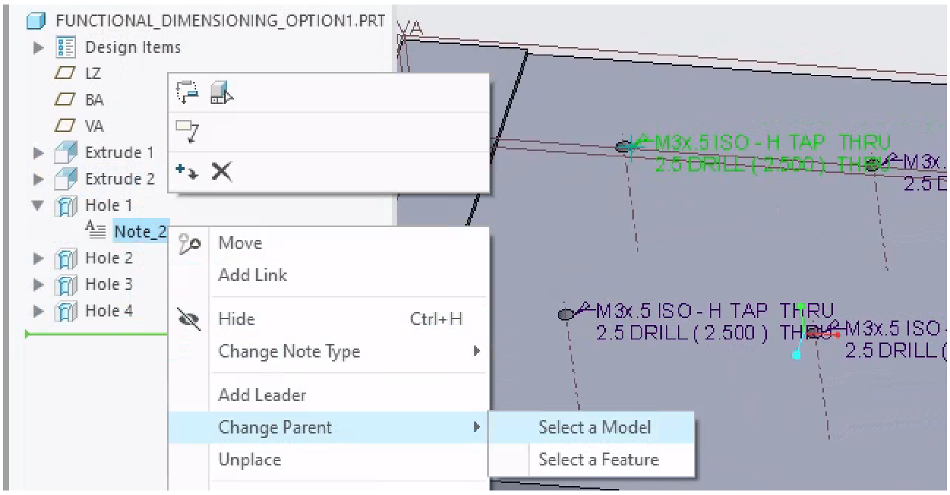


Figure 5.19: Changing the parent of the embedded note from the hole feature to the model

Semantic references (the surfaces the note refers to) can then be assigned to the note using the command “References” (Figure 5.20). The surfaces of the other holes can also be assigned to this note as semantic references (Figure 5.21).

There is another possibility to assign semantic references to the note. It is possible to convert the annotation to an “annotation element” by explicitly assigning it to a “combination state” (see Figure 5.22). A “combination state” is a special function in Creo Parametric that allows the designer to view the model in different states like simplified representations, orientations, exploded states, cross section, ... This can be displayed in a special tab view (Ramesh 2017). This is often used for MBD.

It is still not possible to directly assign semantic references to this annotation element, but now it can be converted to an “annotation feature”. The surfaces defining the four holes can then be assigned as semantic references by specifying four sets of

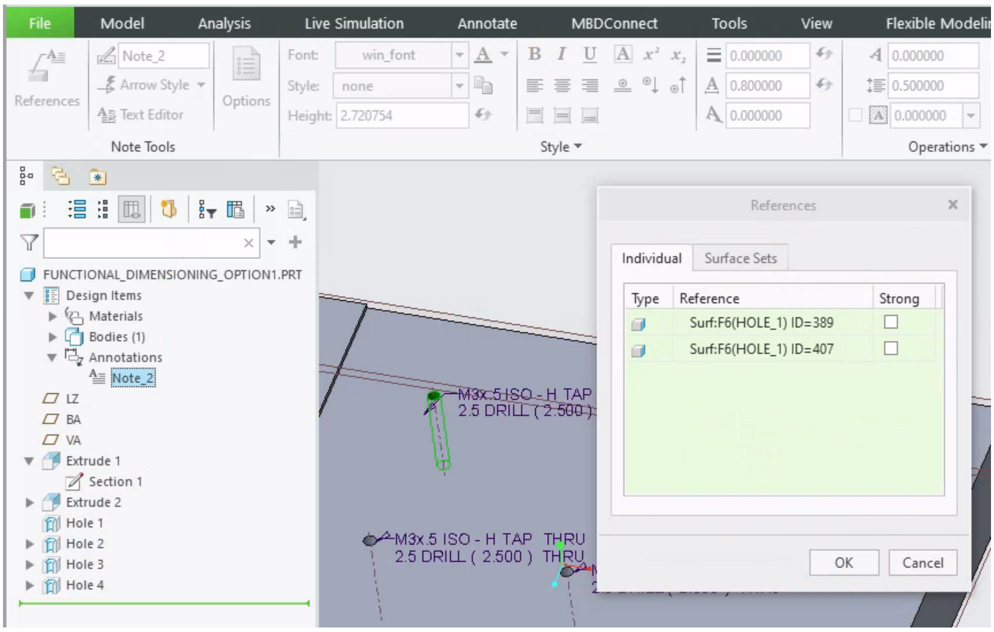


Figure 5.20: Surfaces assigned as semantic references to the note using the "References" command

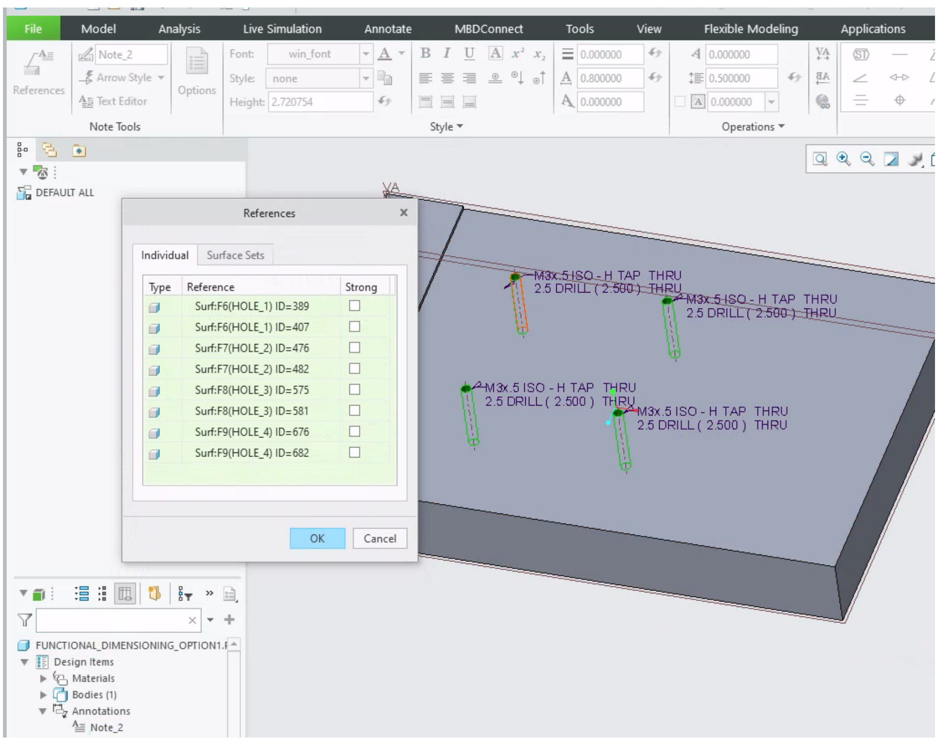


Figure 5.21: The surfaces of the other holes are assigned as semantic references to the note using the "References" command

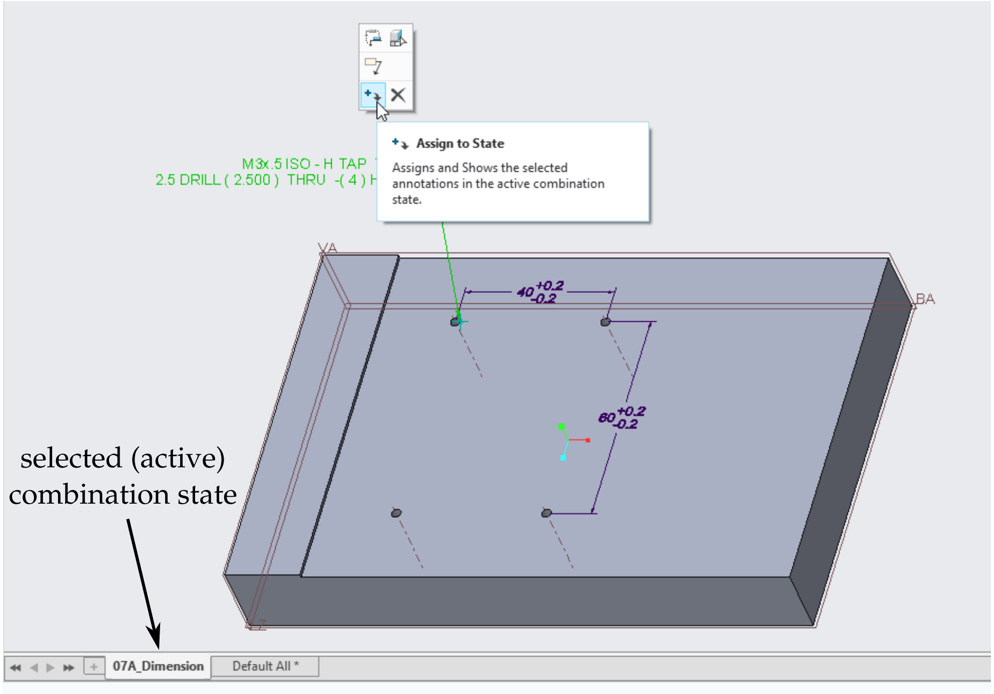


Figure 5.22: Explicitly assign annotation to the active "combination state"

loop surfaces[1], each containing the two surfaces that make up a hole (see Figure 5.23) or by assigning eight individual surfaces (see Figure 5.24). In Creo Parametric a cylindrical surface consists of two holes because that is how the CAD kernel used by Creo Parametric defines a cylinder.

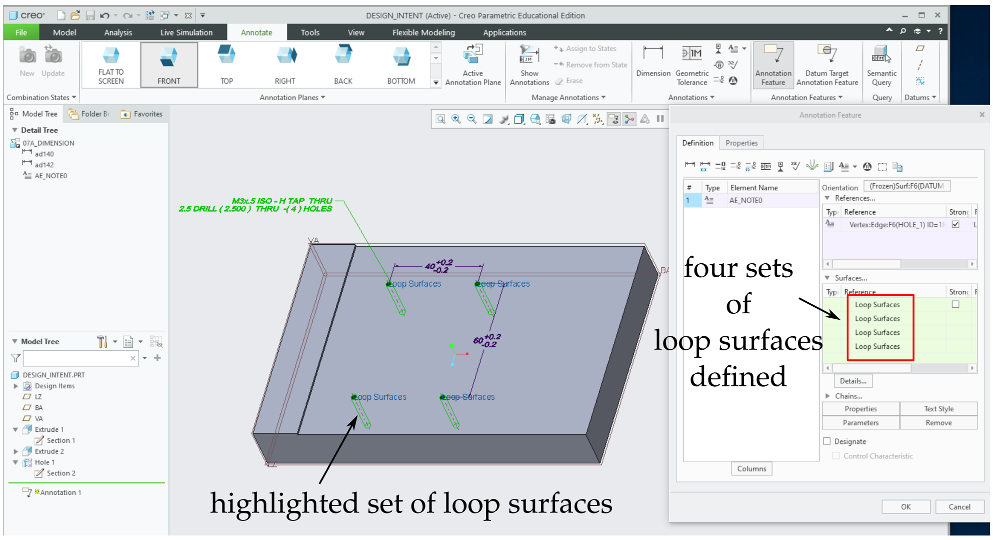


Figure 5.23: Four sets of loop surfaces assigned as semantic references to the annotation

Figure 5.24 shows the list of the cylindrical surfaces of the holes that are manually assigned to the note. Whether four sets of loop surfaces are defined or eight individual surfaces are chosen depends on the philosophy the designer wishes to follow.

In this way, the lack of semantic references in the original note can be resolved. However, to obtain the same indication "4X M3x.5 ISO . . . " as in the MBD view, the following changes are needed:

[1] A loop surface is a collection of surfaces that connect to an imposed boundary, in this case the top circle of a hole

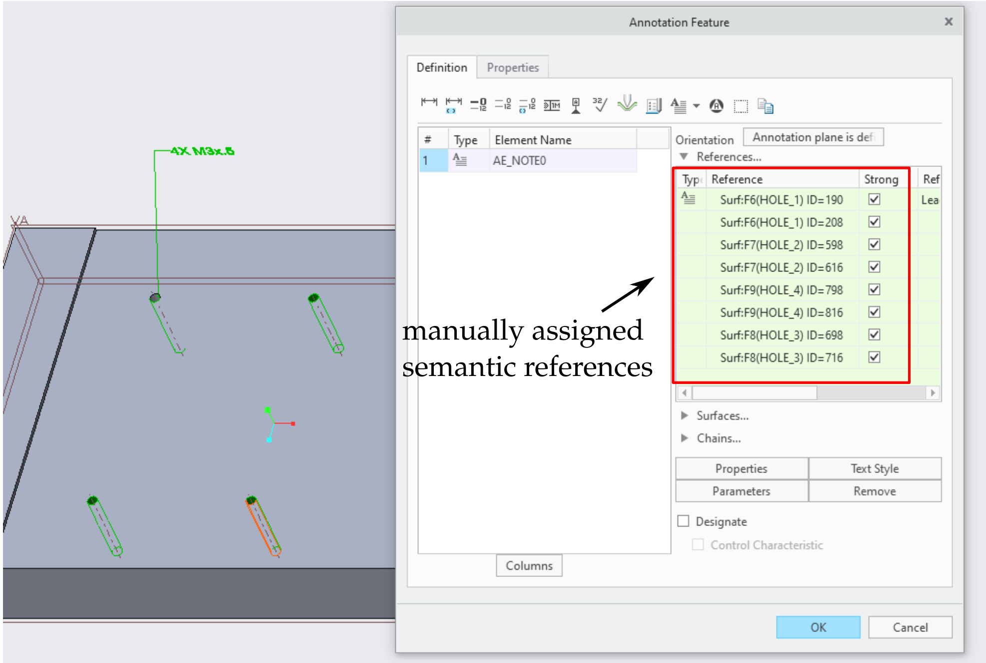


Figure 5.24: The surfaces that define the four holes are manually assigned as semantic references to the note created as an "annotation feature"

- The notes of the other three holes must be removed manually, leaving only the note whose parent has been changed.
- The indication "4X ", indicating that the note refers to four holes, must be added manually at the beginning of this note. This cannot be done by changing the text of the note. Its value is "Locked" (Figure 5.25). The only way to adjust it is to remove the original text and recreate it via a text editor (Figure 5.26). The public feature parameters can be accessed outside the hole feature (Figure 5.27) and can be used within the note text (Figure 5.28). These are linked to a specific feature id. As a result, if the hole whose public parameters is used is deleted, all parameters used in a note become invalid. The four holes are four separate features. Consequently, if the number of holes changes during the product development, the indication "4X" must be changed manually to reflect the new number of holes.
- The leader of the note is attached incorrectly to the model (Figure 5.29). This must be corrected manually and the leader must be attached to all the surfaces of the other holes (Figure 5.30).

Another way to make the note match the one in the MBD view is to recreate it completely as an "annotation feature".

As ISO and ASME standards require that the number of features to which the annotation applies to be of the form 4X, where 4 is the number of features, this may be one of the reasons why machine-readability is not achieved. A designer may be careless when entering the annotation or his or her company may have its own standards that deviate from the ISO and ASME standards. If the designer enters 4 X or 4x instead of 4X, the annotation is still legible for a human. This is not necessarily the case for a computer application. To minimise the occurrence of errors, though this does not completely exclude errors, the annotation text with parameters can be retrieved from previously saved files that contain the note definition. Figure 5.31 shows a manually created note that contains a public feature parameter. The number of occurrences must still be adjusted by the designer.

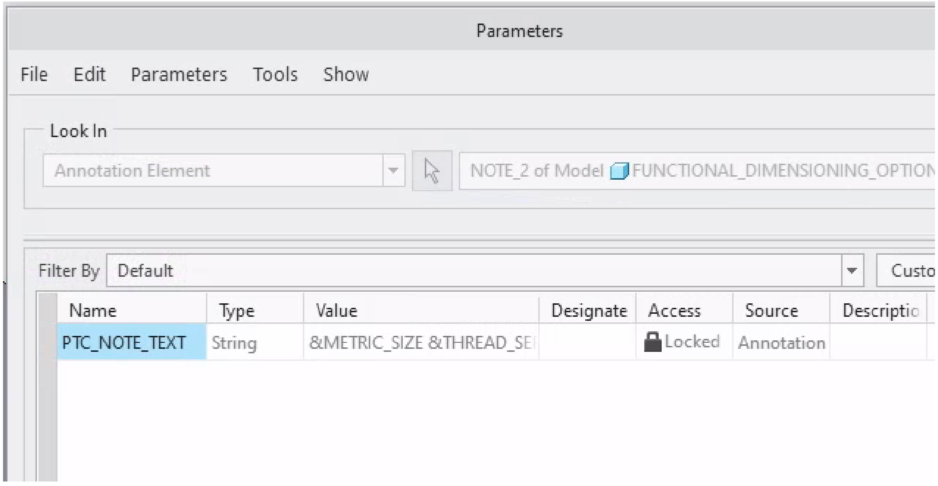


Figure 5.25: Querying the value of the note shows its value is locked. The value of the note is determined by variables (&METRIC_SIZE, ...) whose value is determined by the hole feature through which it was generated.

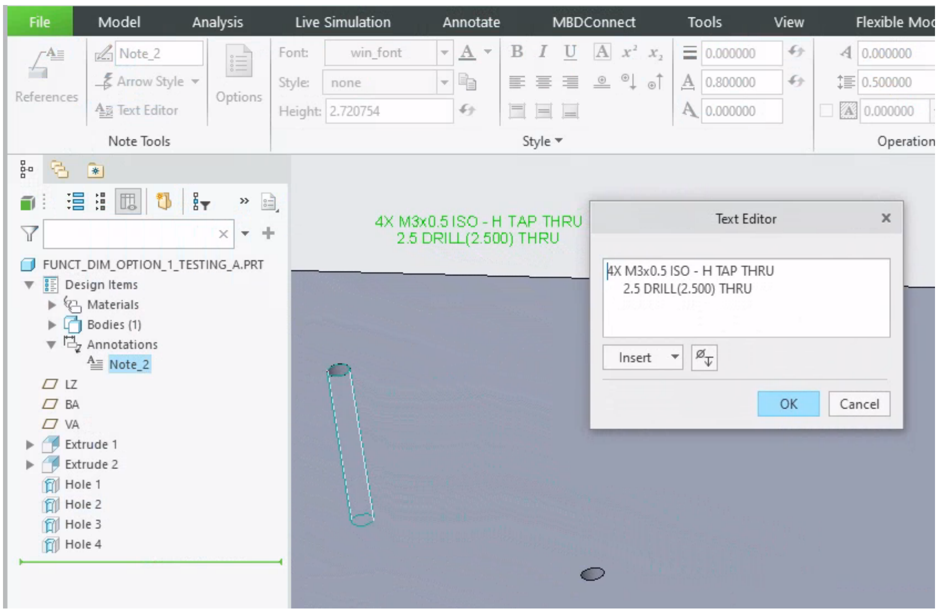


Figure 5.26: The note text is recreated using the note text editor

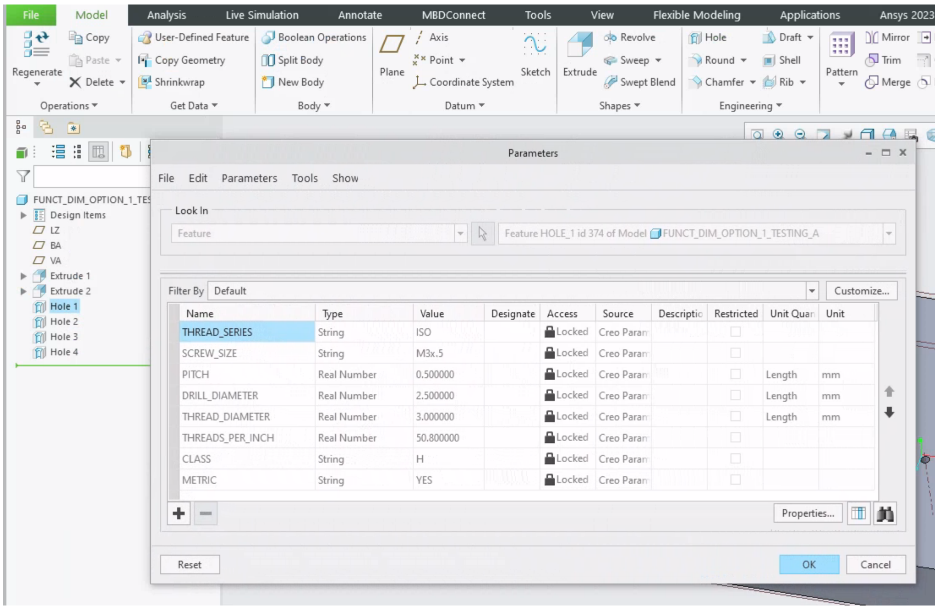


Figure 5.27: The public feature parameters of hole 1 that can be queried

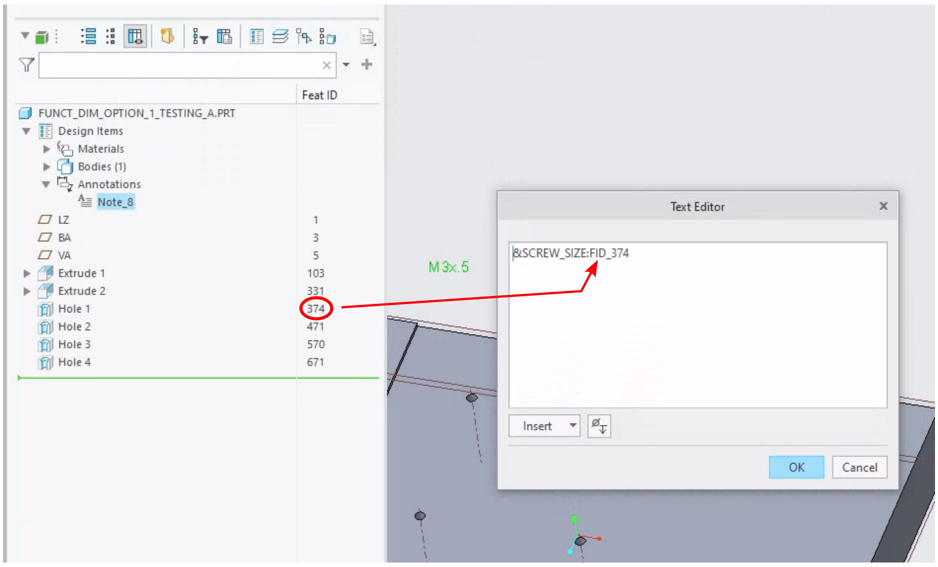


Figure 5.28: Public feature parameters are used in the note text by referencing the hole feature id

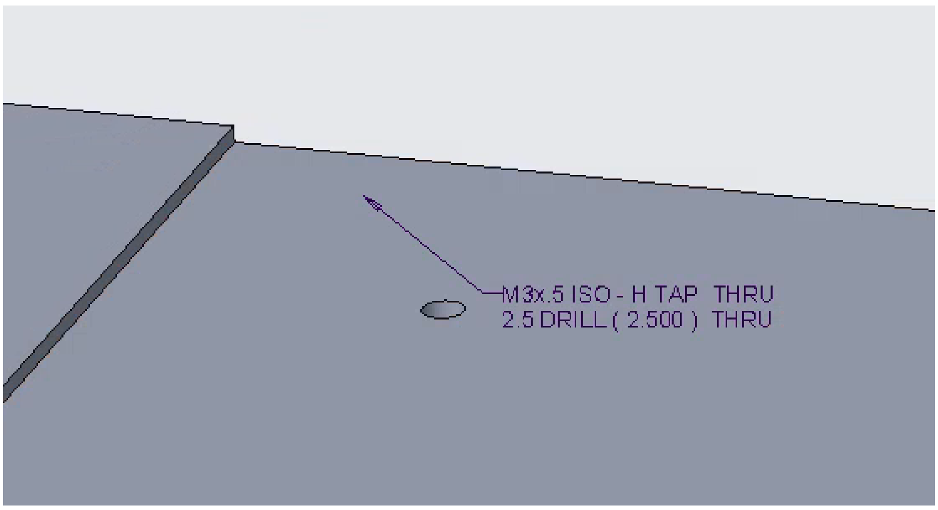


Figure 5.29: The leader of the note is attached incorrectly to the model

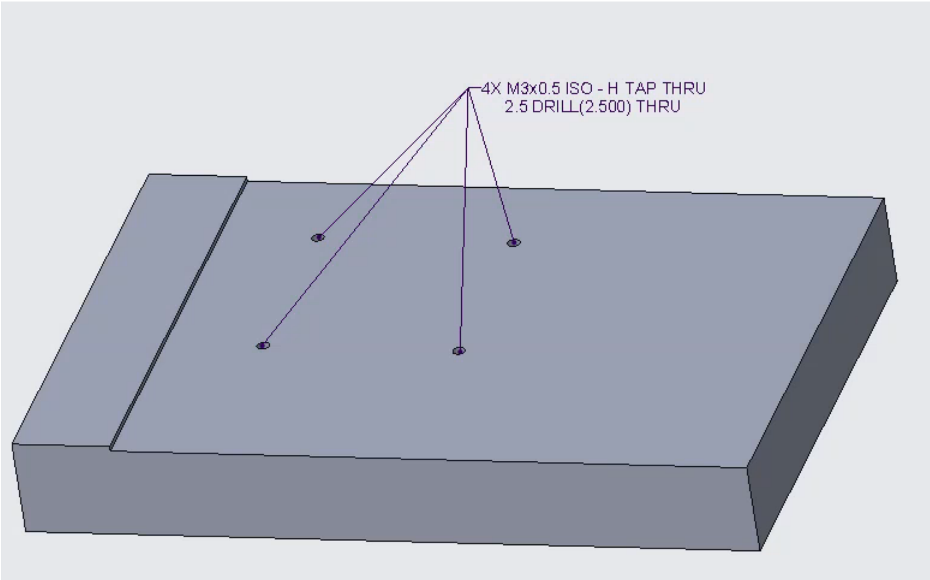


Figure 5.30: The leader of the note is corrected and attached to the surfaces of the other holes

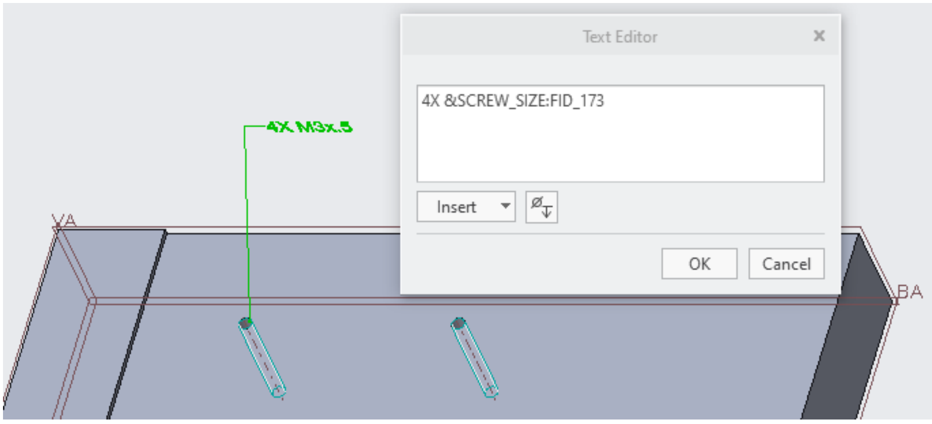


Figure 5.31: Example of a manually created note that contains a public feature parameter. The note is formatted according to ISO and ASME standards

The end result, a stand-alone note manually created as an annotation feature, can be seen in Figure 5.32.

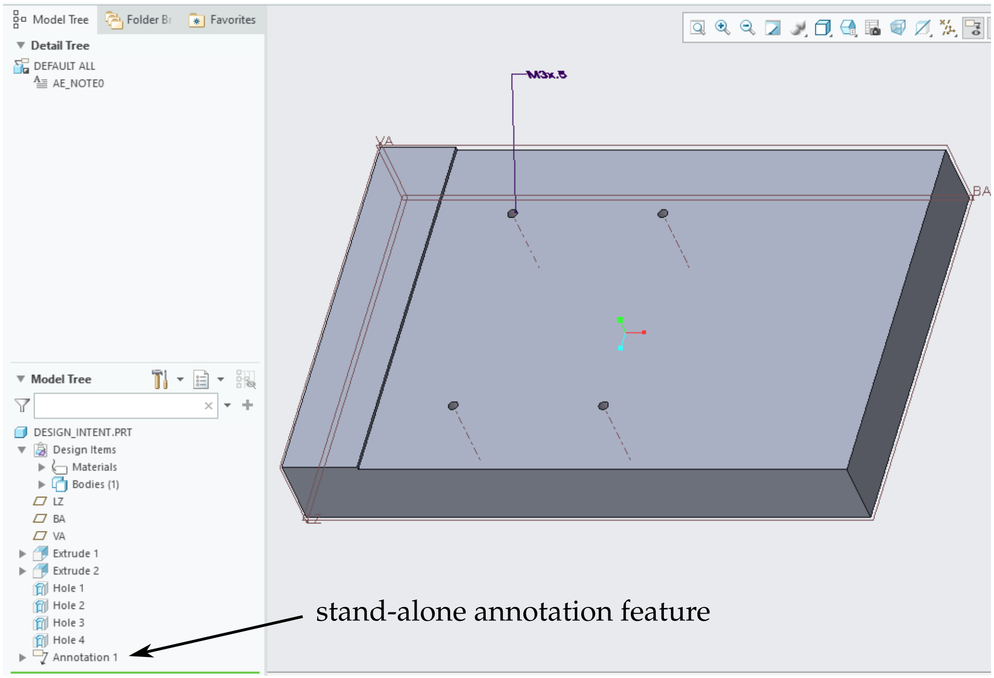


Figure 5.32: Manually created annotation feature is shown as a stand-alone annotation in the Model Tree

In this example, the horizontal and vertical dimension 20 (see Figure 5.33) is not shown in the MBD model (see Figure 5.13) because the dimension 20 must satisfy the general tolerance that applies to the whole model. Therefore, according to the MBD philosophy, it must not be shown explicitly in the 3D model. This value must be derived from the 3D model itself, which acts as the sole authority. The statement "the 3D model acts as the sole authority" implies that all tolerances are determined by the 3D model. This includes tolerances on a dimension value as well as tolerances on the shape. Figure 5.34 shows the tolerance fields for the horizontal dimension 20 as defined by the 3D model. Because the 3D model as a whole is the authority, there is no confusion as to what the dimension values and the shape should be.

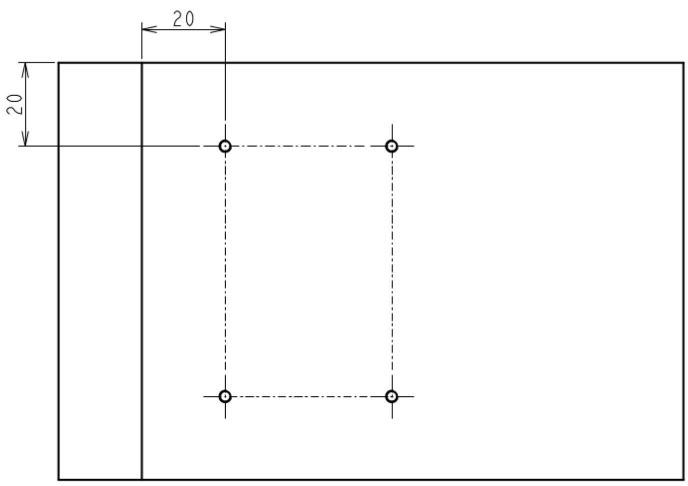


Figure 5.33: Determining position of first hole

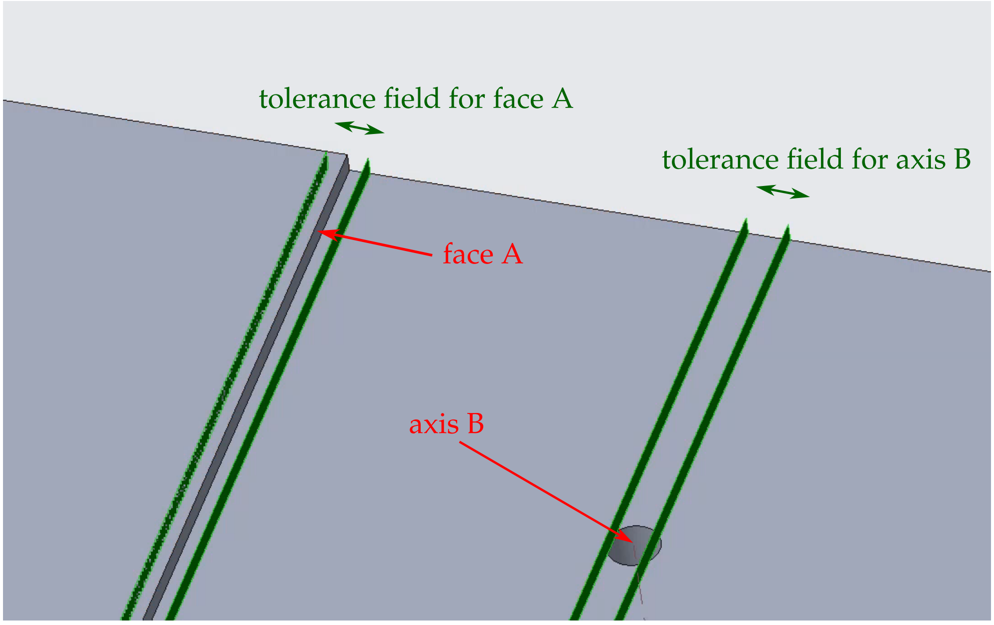


Figure 5.34: Single-direction tolerance fields defining the position of one hole as defined by the 3D mode

When this value is given a tolerance that differs from the general tolerance (see Figure 5.35), the concept of machine-readability comes into play. It is clear to a human what the designer means, but it is not immediately clear to a computer what exactly this dimension refers to. The latter is illustrated in the following example. If a computer has only the image shown in Figure 5.35, there are several possible interpretations of what the dimension $20^{+0.1}_{-0.1}$ refers to exactly. From a human point of view the reasoning will seem absurd, but from a purely software point of view it is not.

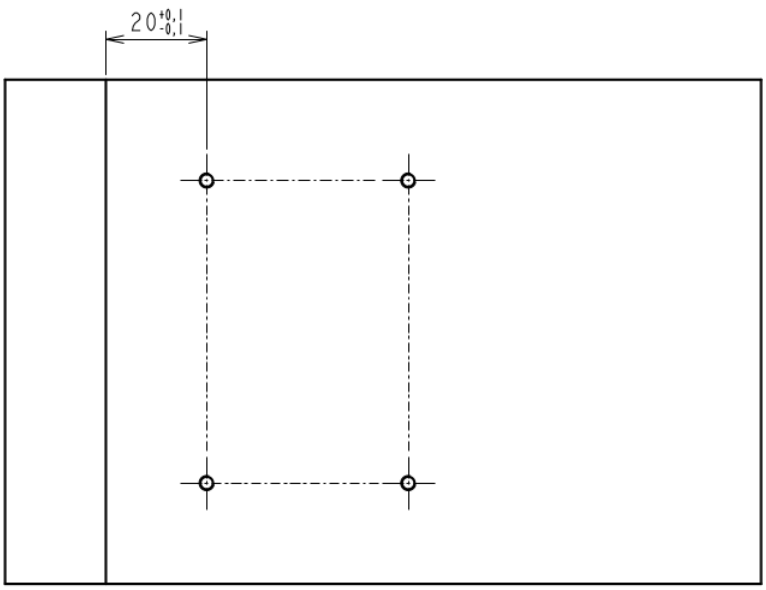


Figure 5.35: A dimension with a tolerance different from the general tolerance is added to the 3D model

The different entities to which dimension $20^{+0.1}_{-0.1}$ might refer are shown in Figure 5.36. There are nine different possible combinations:

1	a upper vertex and j hole axis	5	e side surface and j hole axis
2	b top vertical edge and j hole axis	6	f bottom edge and j hole axis
3	c lower vertex and j hole axis	7	g upper vertex and j hole axis
4	d top edge and j hole axis	8	h bottom vertical edge and j hole axis
		9	i lower vertex and j hole axis

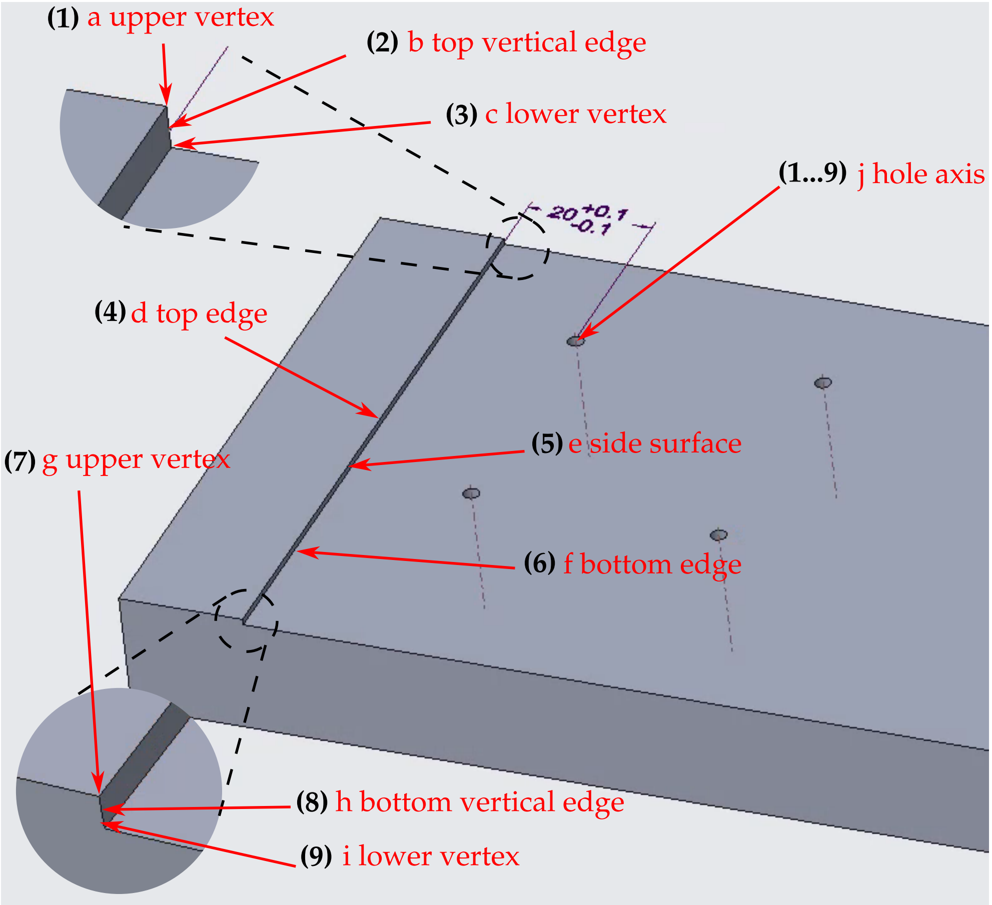


Figure 5.36: Possible entities the dimension 20 is referring to

From a manufacturing point of view, where the combination of “e side face” with “j hole axis” makes the most sense, the other combinations seem irrelevant and do not occur. This is not the case. In most cases, the designer will create the dimension 20 by clicking on “some geometrical entities”, which leads to the desired display of the dimension 20. The effect of this is that the CAD system chooses two arbitrary semantic references, with the designer unaware of what the CAD system has actually chosen. If other stakeholders (manufacturing, quality control) want to make full use of the semantic references (such as automatic generation of measurement programmes), then it is the responsibility of the designer to ensure that meaningful semantic references are assigned to the dimension.

There is one aspect about semantic references that has not been mentioned so far which is there are semantic references that have different functions. The type of semantic reference dealt with so far refers to the vertices, axes, edges, surfaces to which the annotation refers. For example, the surfaces of the holes to which the annotation "M3x.5 ISO . . . " refers or, in the case of the dimension $20^{+0.1}_{-0.1}$, the references between which this dimension is to be measured. However, there are two more reference types. One type defines the plane in which the annotation is displayed and the viewing direction on this plane. The other type is the so-called “leader attachment reference”.

Some CAD systems, such as Siemens NX, ensure that the value of the dimension always faces the user. This means that it remains legible no matter how the user rotates the model. This can give the impression that the semantic reference and the view vector, which controls the direction of view on the plane where the annotation is displayed, are always correct. However, this is not the case. This is particularly apparent when the model is exported to a neutral exchange format such as STEP AP242. It may happen that the dimension value is displayed upside down and mirrored when the STEP file is imported into another CAD system (Figure 5.37).

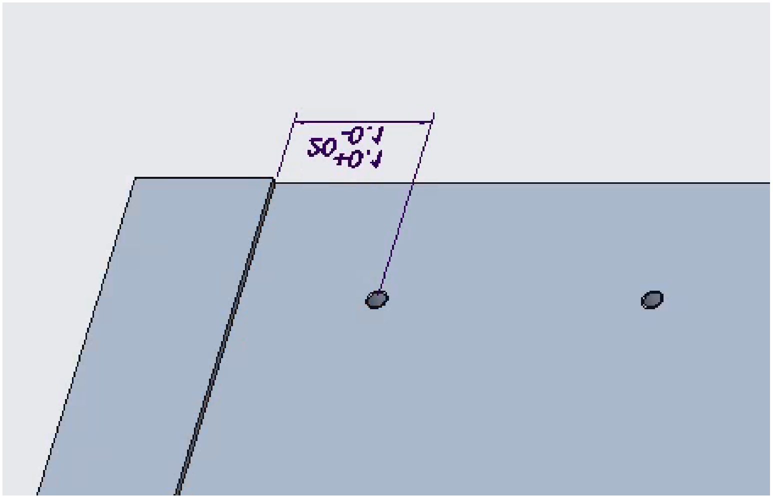


Figure 5.37: Result of a non-optimal view vector on the display plane after importing a STEP AP242 file generated by Siemens NX into PTC Creo Parametric

In the case of the annotation "M3x.5 ISO . . . ", the "leader attachment reference" is the top arc of the hole to which the leader is pointing. In the case of the dimension $20^{+0.1}_{-0.1}$ these are the references to which the extension lines connect. Different types of problems may occur:

- the same references may have different functions, which means that they belong to two types of references at the same time
- it is not possible or difficult to recognise the type of reference

If the same references serve both to connect a leader or an extension line and to indicate that the annotation refers to them, is it sufficient to mention these references only once or must they be mentioned twice, once for each purpose? To answer this question, it must be considered together with the second problem. This is that it is not always possible to recognise the reference type. To understand this better, consider the case where the $20^{+0.1}_{-0.1}$ is created as a driven dimension. The designer is allowed to select any reference (vertex, point, edge, axis, surface) he or she wants. However, only two references may be selected. Suppose the upper edge and the axis of hole 1 is selected (see Figure 5.38).

When the function to query the assigned references is executed, the function indicates the upper edge as the "First dimension reference" and the axis as the "Second dimension reference" (see Figure 5.39). So the answer to the question of whether semantic references are assigned to the annotation is yes.

The question that remains unanswered is to which of the three previously discussed types these references belong. To answer that question, the driven dimension must first be converted to an annotation feature. Figure 5.40 shows one of the possible ways in which this can be done.

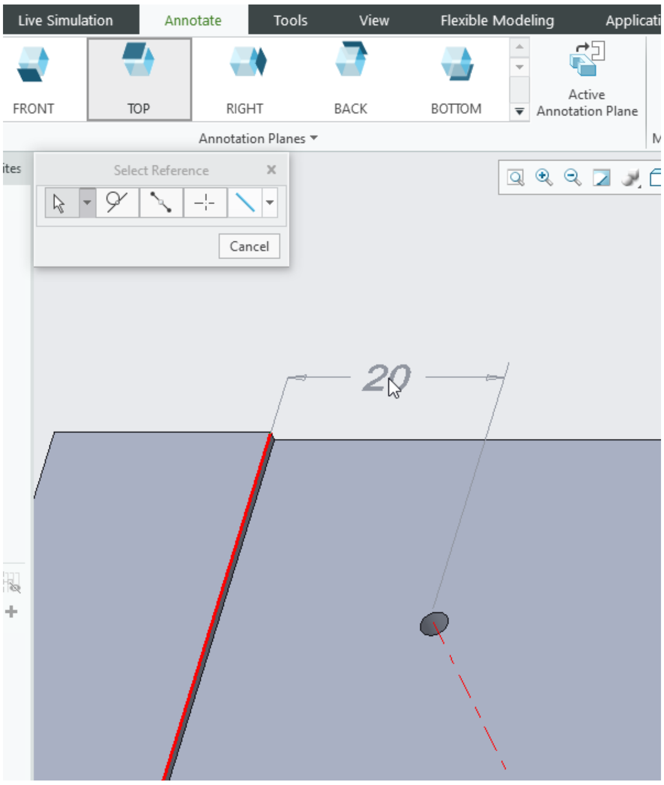


Figure 5.38: Upper edge and axis (coloured red) selected as references during the creation of the driven dimension $20^{+0.1}_{-0.1}$

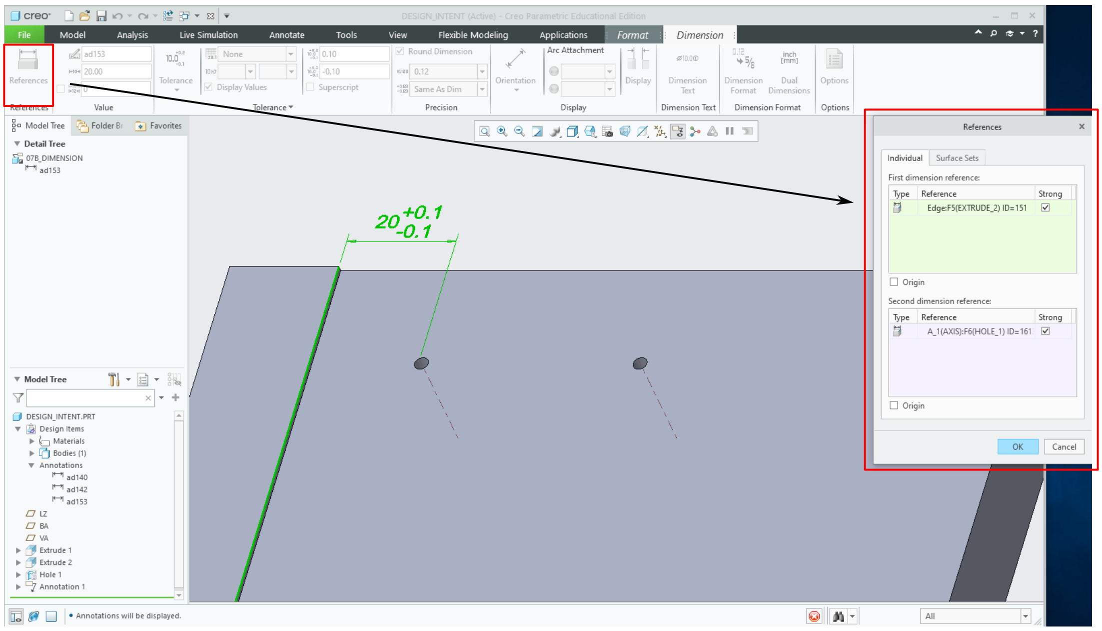


Figure 5.39: Querying the references show two references are assigned to the annotation

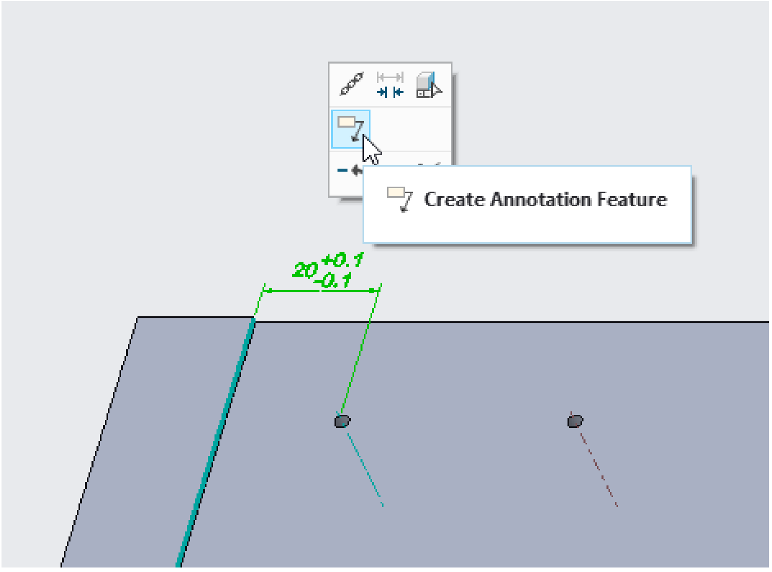


Figure 5.40: One of the possible ways of converting an annotation to an annotation feature

The result can be seen in Figure 5.41. It shows that the semantic references are only used to connect the extension lines of the dimension. The additional references that are needed to facilitate correct measuring must be added manually.

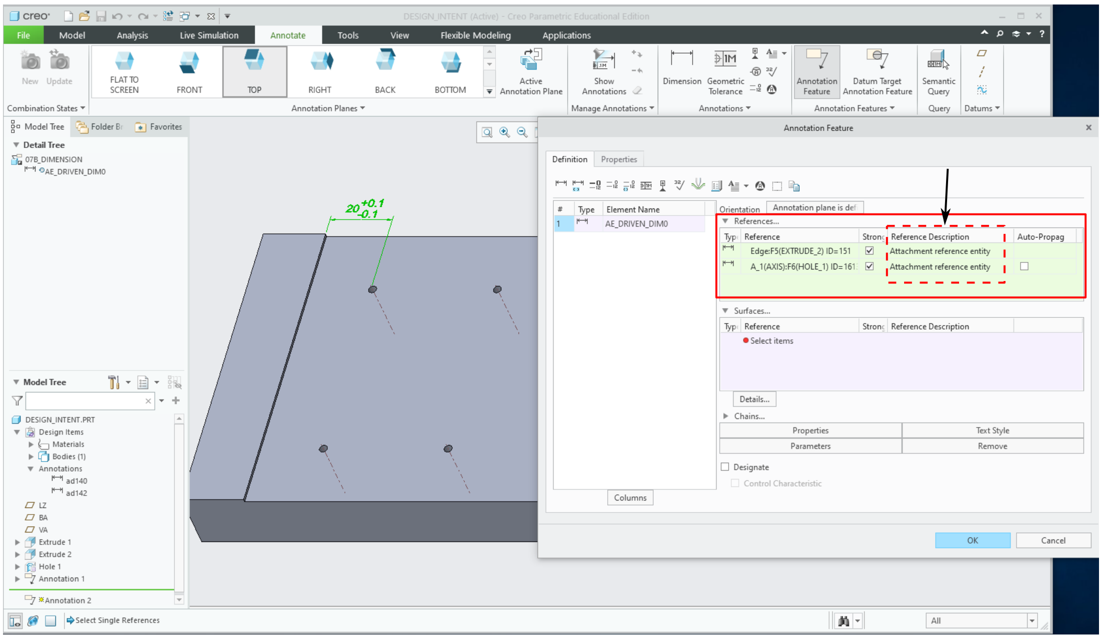


Figure 5.41: After conversion to an annotation feature it becomes clear the two semantic references only serve as attachment references

If the sole purpose is to generate First Article Inspection Documents (Capvidia 2016), these additional references are not necessary. In this case, it is sufficient to be able to determine the type of annotation and read its values. With this information, the annotations can then be listed.

Option 2: The creation of four holes by creating a pattern

As mentioned, there are several ways to create the four tapped holes in PTC Creo Parametric, the CAD system used to demonstrate some of the problems that can occur. Some of the options available are

1. Create four separate tapped holes using the standard hole function
2. Create four tapped holes by creating a pattern from a threaded hole feature
3. Create four threaded holes using the sketched hole feature.

The first option was covered in the previous subsection. This subsection covers the second option. The holes are created in two steps. The first step is to create the first threaded hole. This is done using a linear dimensioning scheme (Figure 5.42).

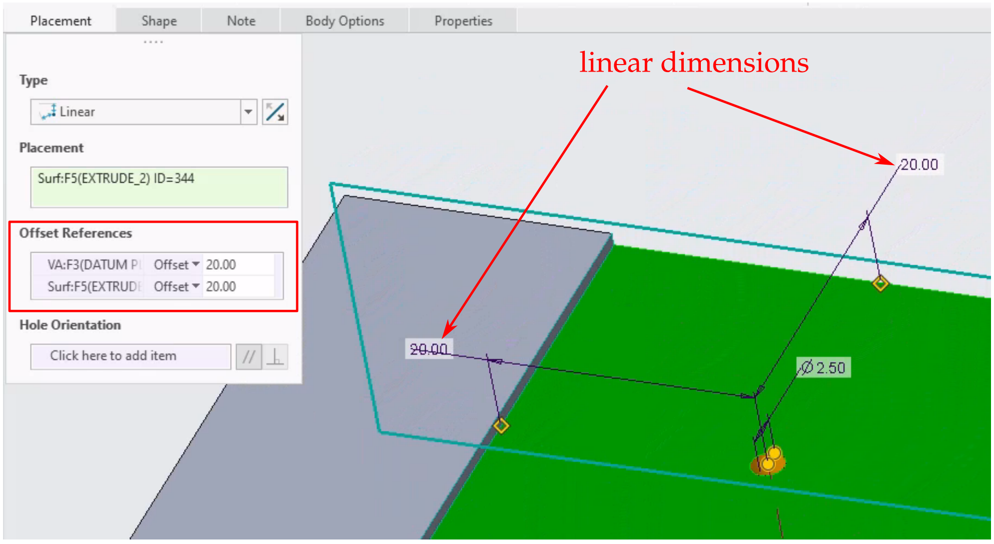
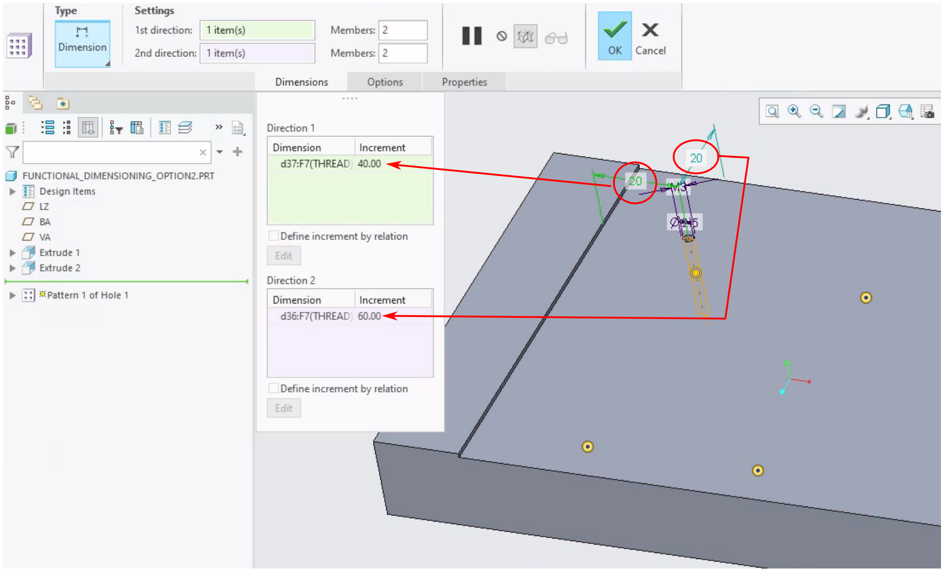


Figure 5.42: Step 1, creation of the first hole using a linear dimensioning scheme

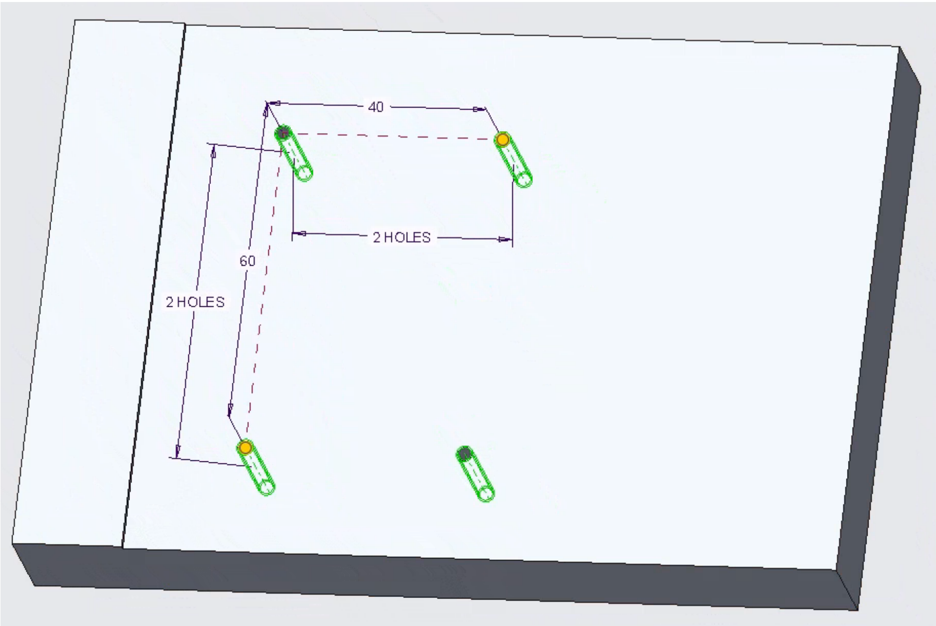
The second step is to create a pattern based on this first hole. The two linear dimensions defined when creating the first hole are used here to define the offset and direction of the pattern (Figure 5.43). Comparing the dimensioning scheme of the pattern in Figure 5.43b with the one in the MBD model in Figure 5.44, it can be concluded that they are the same. The dimensioning scheme also conforms to "functional dimensioning", which is now the result of a proper implementation of "design intent". In the case of option 1, where the four holes are created as individual holes, changing a dimension value can result in the desired grouping of the holes being broken. Changing the dimension from 60 to 40 means that the four holes no longer lie on a rectangle (Figure 5.45a). In the case of option 2, where the four holes are defined as a pattern, the four holes will remain on a rectangle even if the dimension is changed from 60 to 40 (Figure 5.45b).

As already mentioned, the "machine readability" of the annotation includes the ability to read out the semantic references. The problems with retrieving the semantic references are similar to those described in section 5.3.1. In the CAD system Creo Parametric, annotations can be created in the MBD model in three different ways.

As discussed in subsection 3.1.3 and briefly mentioned in Option 1 (p. 71), the dimensions 40 and 60 in the MBD model in Creo Parametric can be created in different ways. They can be catalogued in different ways.



(a) The linear dimensions used to create the first hole are used to define the pattern directions



(b) The resulting pattern

Figure 5.43: Step 2, creating a hole pattern using the dimensions of the first hole to define the pattern directions

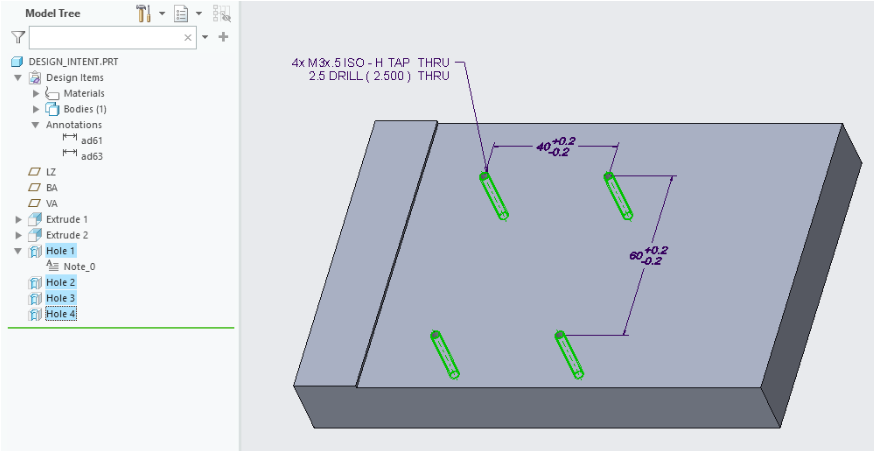
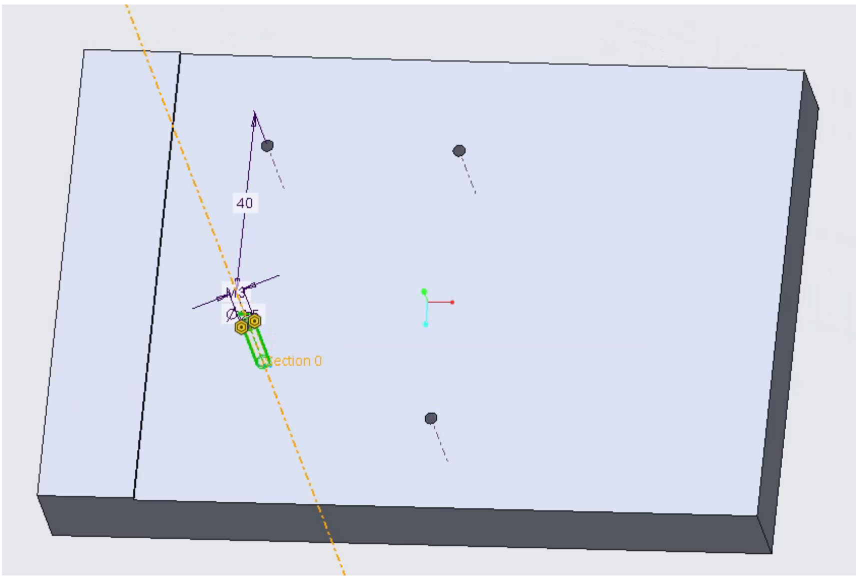
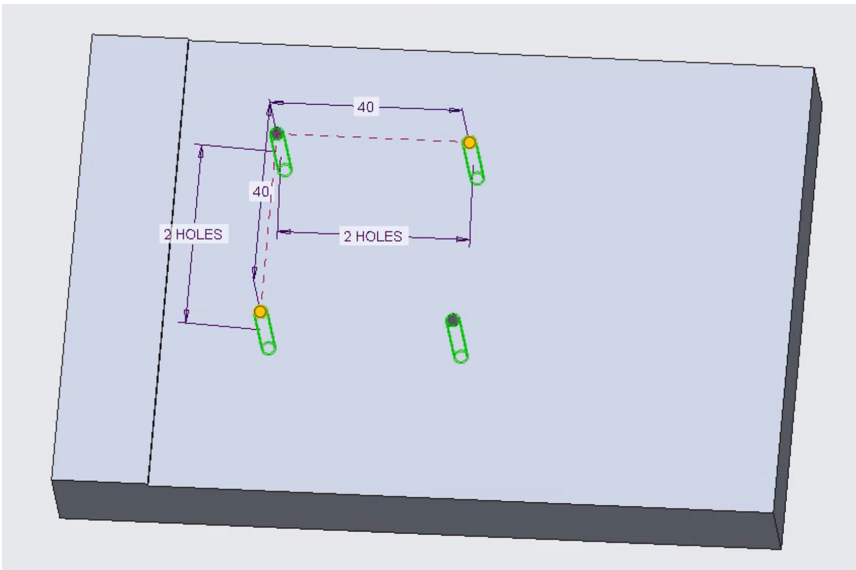


Figure 5.44: functional dimensioning scheme applied in an MBD view



(a) Option 1 (the four holes are created as individual holes): after changing the dimension from 60 to 40 the four holes no longer lie on a rectangle



(b) Option 2 (the four holes are defined as a pattern): after changing the dimension from 60 to 40 the four holes remain on a rectangle

Figure 5.45: The result of options 1 and 2 changing the dimension from 60 to 40

One way is to distinguish between

1. Driving dimensions — Dimensions that are derived directly from dimensions used to create a feature
2. Driven dimensions — Dimensions that are created manually and have nothing to do with the dimensions used to create features
3. Annotation features — These can be thought of as containers that can contain multiple driven dimensions

Another way is to distinguish between

1. Annotation elements — Annotations that belong to another feature like a driving dimension or a driving or a driven annotation that is embedded in an annotation feature
2. Stand-alone annotation — An annotation that stands on its own, a driven dimension that is not embedded in an annotation feature

This subsection will use the first way of cataloguing.

Driving dimensions

As the dimensions resulting from the pattern creation are the same as those in the MBD view, the designer could choose the first method and use driving dimensions (Figure 5.46). This means that the annotation is derived directly from the internal dimensions of the pattern and can be used to modify the pattern. Hence, the name "driving". On first inspection this is a very convenient type of annotation. The values are derived from the pattern dimensions and can be queried (Figure 5.47).

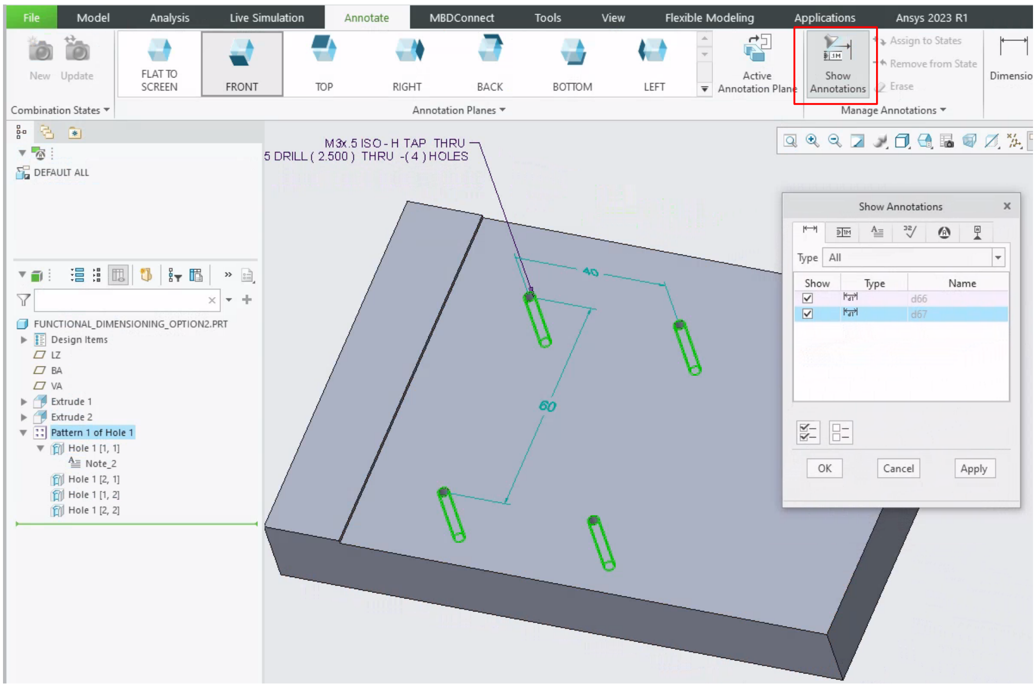


Figure 5.46: The linear dimensions in the MBD view are created by deriving them directly from the hole pattern using the "Show Annotations" option

When querying the semantic references, it appears that no references are assigned to the annotation (Figure 5.48). It is not possible to convert this annotation to another type. If the model is exported to STEP AP242, the values will no longer be available as parameters that can be queried. This will be discussed in chapter 6. These two problems, the lack of automatically assigned semantic references and possible problems

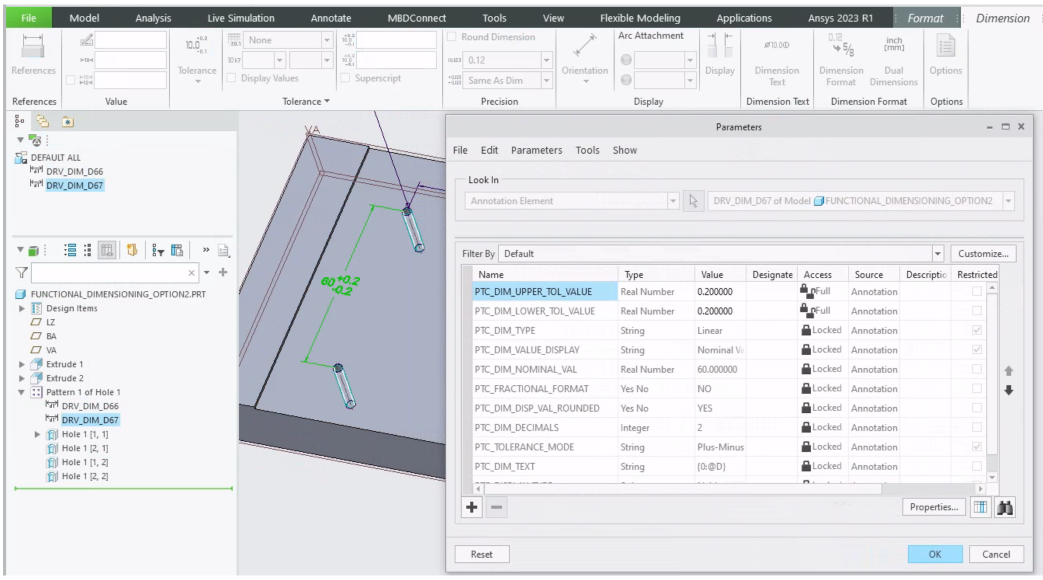


Figure 5.47: The values of the driving dimension can be queried as parameters

with the parameters, mean that the use of this type of annotation should be discouraged. If the designer wishes to annotate the MBD model with semantic references whose values can be queried even after export to STEP AP242, other methods must be used which, as will be shown below, require more work and increase the likelihood of errors.

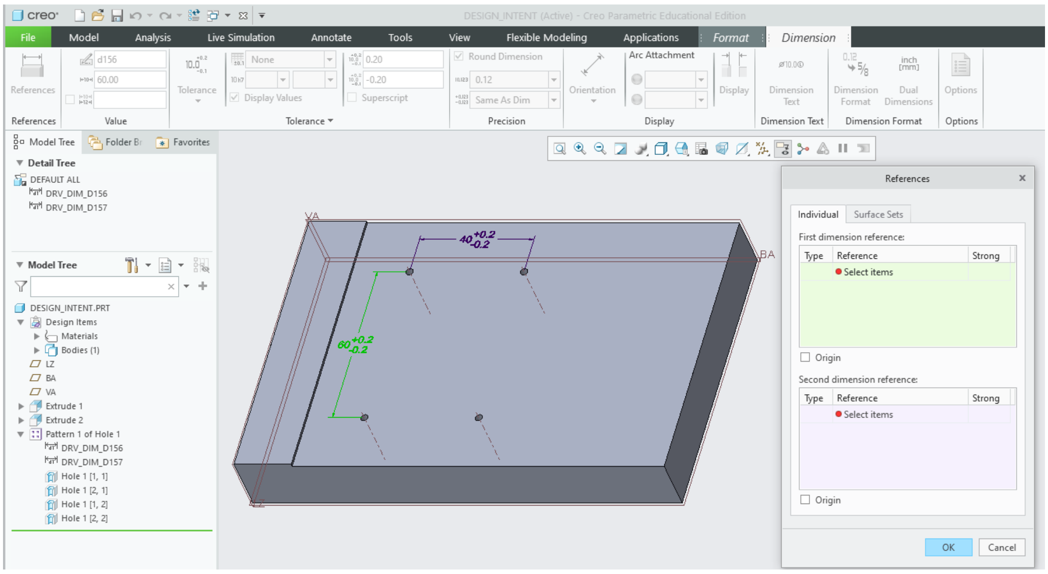


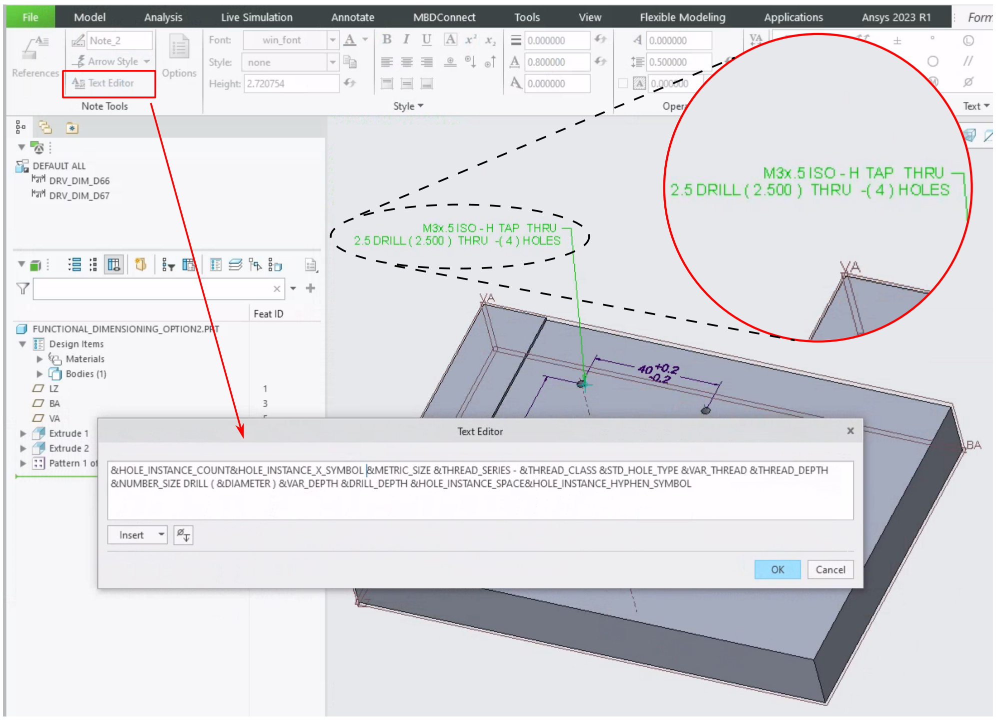
Figure 5.48: After creation, no semantic references are attached to the driving dimension

With respect to the note "M3x.5 ISO ...", as established in Option 1: The creation of four individual holes using the standard hole feature, the note uses parameters that are only available within the feature (Figure 5.49).

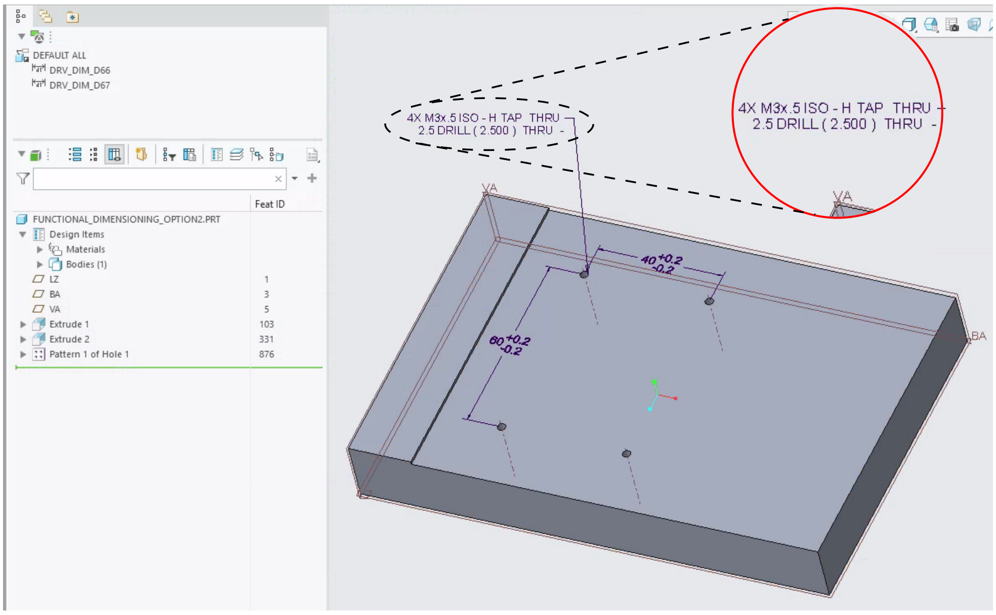
The note automatically generated by the hole feature does not fully comply with the hole specification as described in ASME Y14.5. If the designer wishes to bring it in line with the standard, the easiest way to do this is to make direct changes to the original note generated by the hole feature using the note text editor (Figure 5.50). Conforming to the standard makes it easier to identify the hole pattern parameters in the note text, thus improving machine-readability.

```
&METRIC_SIZE &THREAD_SERIES - &THREAD_CLASS &STD_HOLE_TYPE &VAR& THREAD
&THREAD_DEPTH
&NUMBER_SIZE DRILL ( &DIAMETER ) &VAR_DEPTH &DRILL_DEPTH
&HOLE_INSTANCE_SPACE&HOLE_INSTANCE_HYPHEN_SYMBOL&HOLE_INSTANCE_LEFT_BRACKET
&HOLE_INSTANCE_SPACE&HOLE_INSTANCE_COUNT&HOLE_INSTANCE_SPACE&HOLE_INSTANCE_
RIGHT_BRACKET&HOLE_INSTANCE_SPACE&HOLE_INSTANCE_LABEL
```

Figure 5.49: The note specifying the thread uses internal feature parameters



(a) Modifying note text directly in the note text editor to meet ASME Y14.5 specifications



(b) The modified hole annotation is now in compliance with the ANSI Y14.5 standard

Figure 5.50: Making changes to the hole annotation using the note text editor

Driven dimensions

As mentioned previously, the dimensions 40 and 60 in the MBD model in Creo Parametric can be created in different ways. They can be catalogued in different ways. The one that is used in this subsection is

1. Driving dimensions (discussed in the previous subsection)
2. Driven dimensions (will be discussed in this subsection)
3. Annotation features (will be discussed in the next subsection)

If the designer creates dimensions 40 and 60 as driven dimensions (Figure 5.51), there is no relationship between them and the dimensions used to create the function. Driven dimensions are derived from the model, independent of the dimensioning scheme used to create the features of the model.

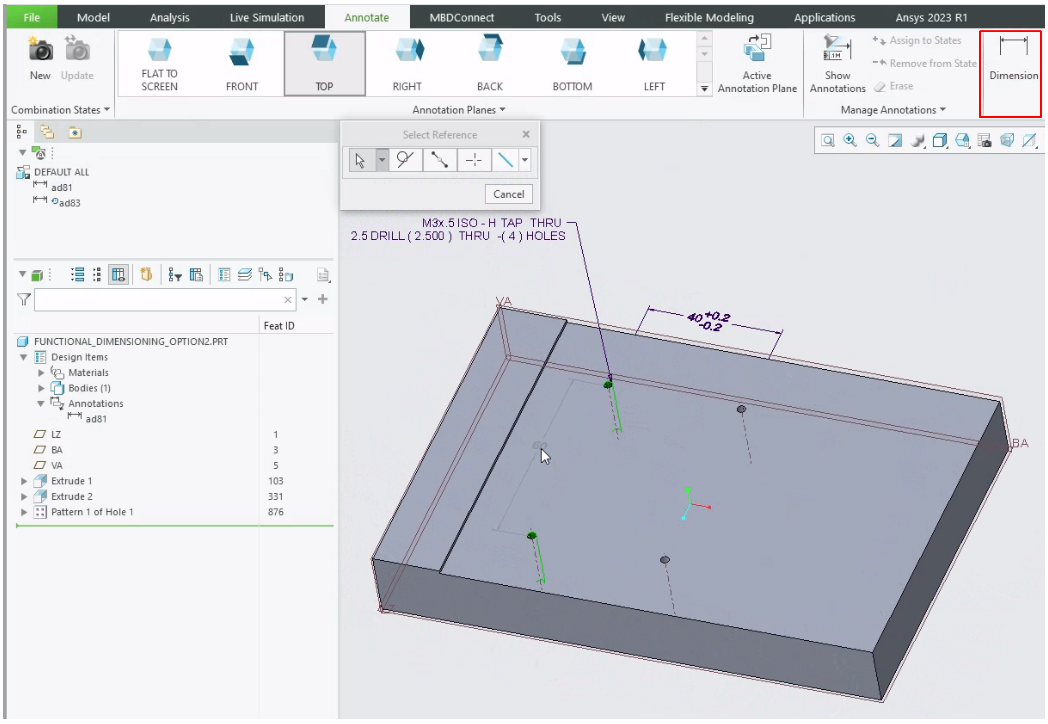
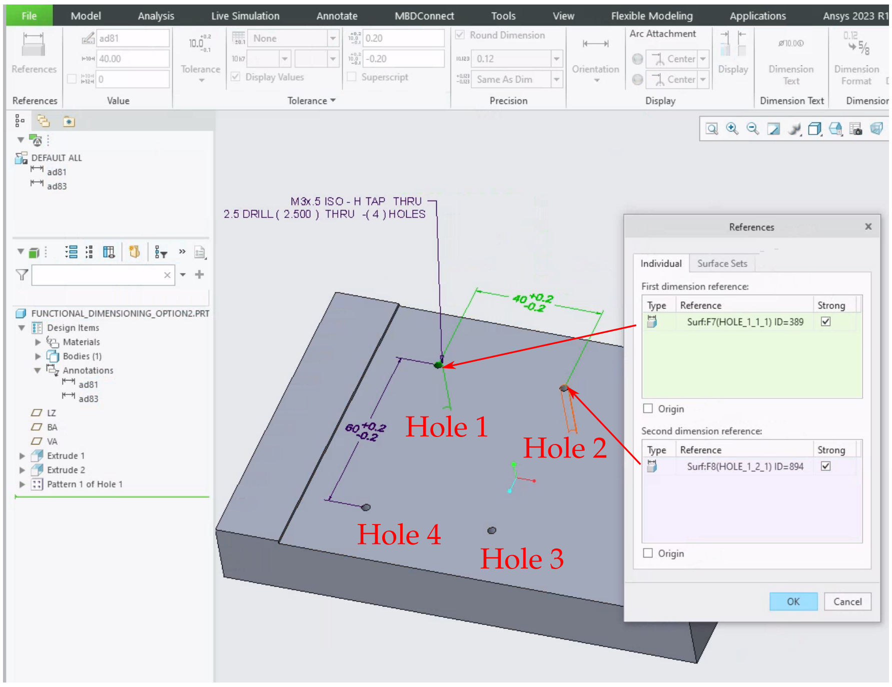


Figure 5.51: Dimensions 40 and 60 are created manually as driven dimensions

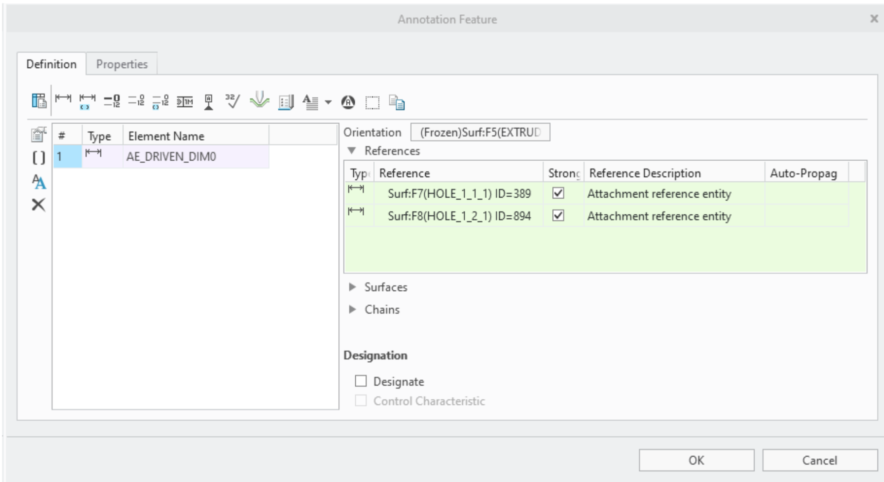
Unlike the driving dimensions, semantic references are automatically assigned to the driven dimension (Figure 5.52a). The entities (points, axes, edges, planes, surfaces), selected by the designer to create the linear dimension between them, are automatically assigned as semantic references to the driven dimension. To check whether these semantic references have multiple functions, the driven dimension is transformed into an annotation feature. The references of this annotation feature are then queried. This shows that they not only indicate what the dimension refers to, but also serve as so-called attachment references (Figure 5.52b). These are the entities to which the auxiliary dimension lines of the driven dimension are connected. It is the designer's responsibility to select the correct semantic references to create the driven dimensions. Correct here means usable for other applications, such as generating measurement programs for CMMs[1]. This means that the semantic references shown in Figure 5.52 are not correct. They only refer to holes 1 and 2, not 3 and 4. From a human point of view, the dimension does refer to all the four holes. From a software point of view it does not. Holes 1 and 2 are at a distance of $40^{+0.2}_{-0.2}$. If a general tolerance ISO 2768-m is applied to the model, then the distance between holes 3 and 4 will be seen as $40^{+0.3}_{-0.3}$. Figure 5.53 shows the designer has selected all the surfaces of all the holes as semantic surfaces. In the CAD kernel that is used by Creo Parametric, a cylinder

[1] CMM stands for Coordinate Measuring Machine

consists of 2 halves. As there are 4 holes, this makes 8 halves. However, there are 10 surfaces assigned to the dimension, as shown in Figure 5.53. When this dimension is converted to annotation feature, it becomes clear that the two original surfaces still act as attachment references and the other surfaces act as the “real” geometry references.



(a) Surfaces selected to create the driven dimension are now automatically used as semantic references



(b) The semantic references are used as attachment references

Figure 5.52: Semantic references are automatically assigned to the driven dimension

From a software perspective, two questions remain unanswered. The first question is what is the relationship between the surfaces. Which surfaces belong together? Specifically, in the case of a hole, which surfaces comprise a hole? The second question concerns the determination of the entities between which the distance is established by the dimension. Is it always possible to determine which surfaces are on which side of the dimension?

The first question can be answered by using "surface sets" or "chains" (in the case of edges). In Creo thinking, these are also called “collections”. An example of a surface set is the concept of "loop surfaces". Here, all surfaces are selected that are connected to

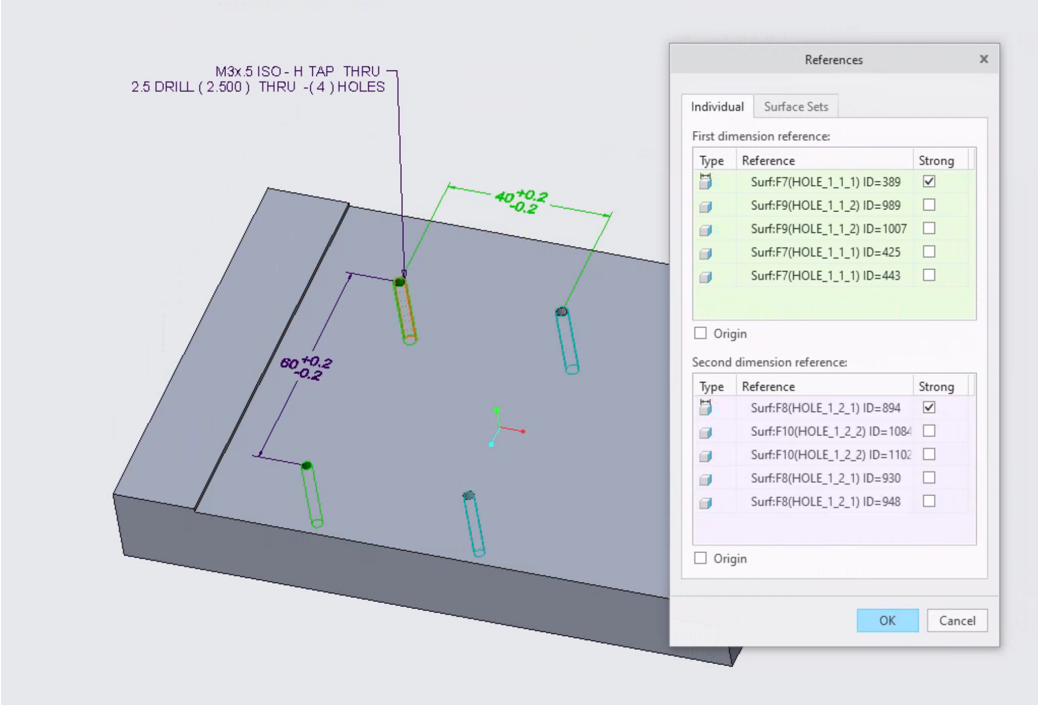


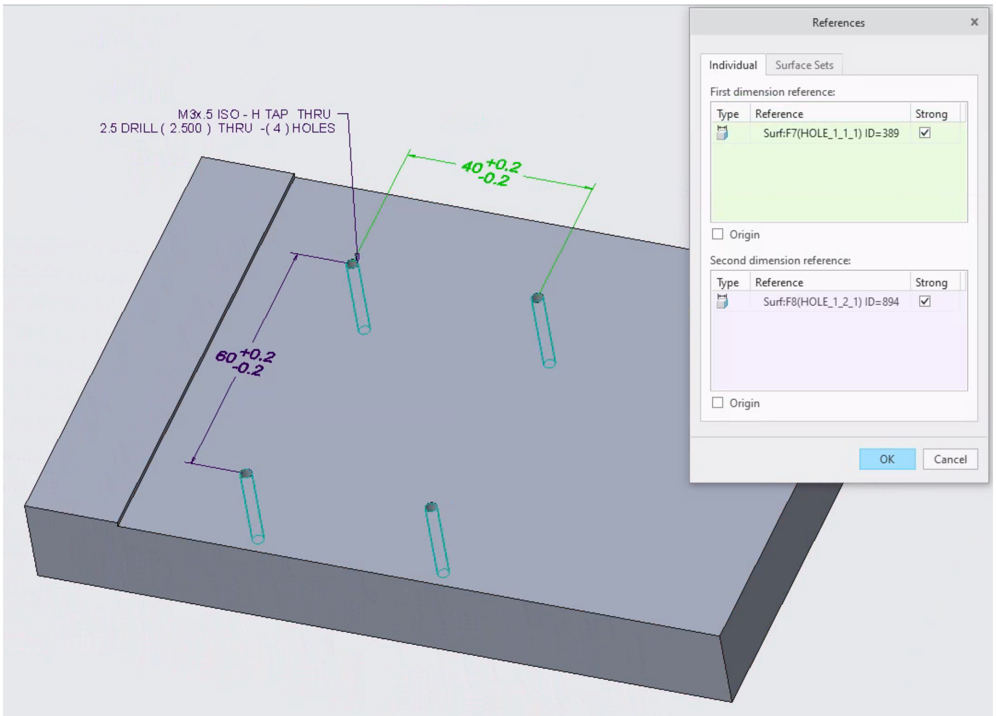
Figure 5.53: All the surfaces of the four holes are now specified as semantic surfaces

a contour selected by the user. If the contour changes by removing or adding an edge, the selected surfaces change accordingly. This type of surface set can be used to define the holes (Figure 5.54). When the driven annotation is converted to an annotation feature, the definition of the surface sets is retained (Figure 5.55).

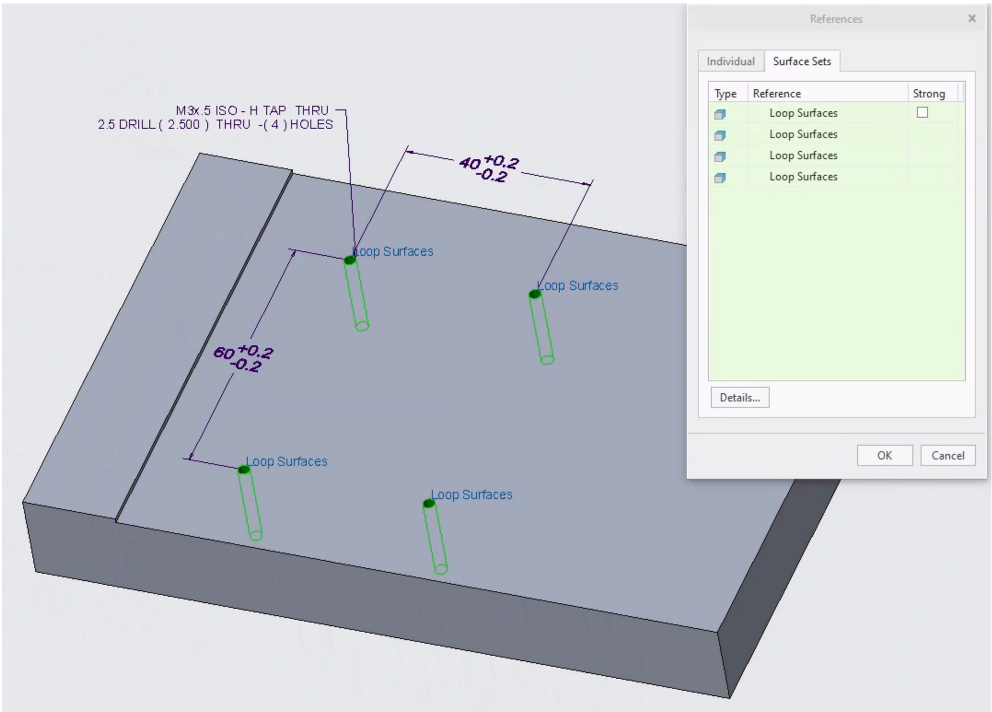
When using collections such as "surface sets", the second question seems superfluous. In the case of the distance between two holes, there are only two attachment references. The distance defined by the dimension can then be determined from these references. However, it is not always that simple. Consider the example in Figure 5.56. The dimension 95 has an asymmetric tolerance. To generate CNC toolpaths, not a width of 95, but of $95 - \frac{0.1}{2} = 95.05$ must be programmed. Depending on the tolerances assigned to the rest of the model, there are three possibilities:

1. surfaces 1, 2 and 3 have to be moved to the outside of the model
2. only surfaces 1 and 2 have to be moved to the outside of the model
3. only surface 3 should be moved to the outside.

So it is important to determine which surfaces belong together. If the driven surface is created as a so-called stand-alone dimension, this seems possible. Figure 5.54a shows two surfaces specified as "First dimension reference" and a third one as "Second dimension reference". The research during this PhD showed that this was not the case. PTC Creo has a software library called ProToolkit that can be used to develop applications that run in a layer on top of Creo. The APIs available in this library did not allow the distinction between "First Dimension Reference" and "Second Dimension Reference". This was reported to PTC. They were kind enough to provide an additional API for this purpose in the new version of Creo. However, this could not be tested in this PhD study because by the time the new API was implemented, the ProToolkit licence necessary to do this was no longer available for the PhD study. When the driven dimension is converted to an annotation feature, it becomes clear that there is no longer a distinction between the surfaces (Figure 5.58).



(a) The selected "individual surfaces" act as the reference attachments



(b) The selected "surfaces sets" are the surfaces comprising the holes the dimension is referring to

Figure 5.54: Using "surface sets" to specify exactly what the dimension is referring to

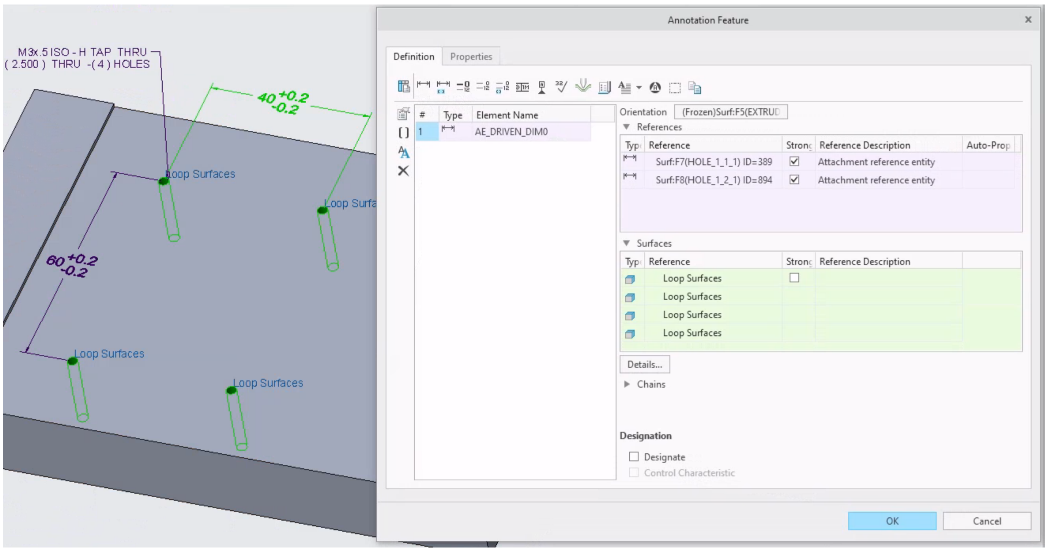


Figure 5.55: When the driven annotation is converted to an annotation feature, the definition of the surface sets is retained

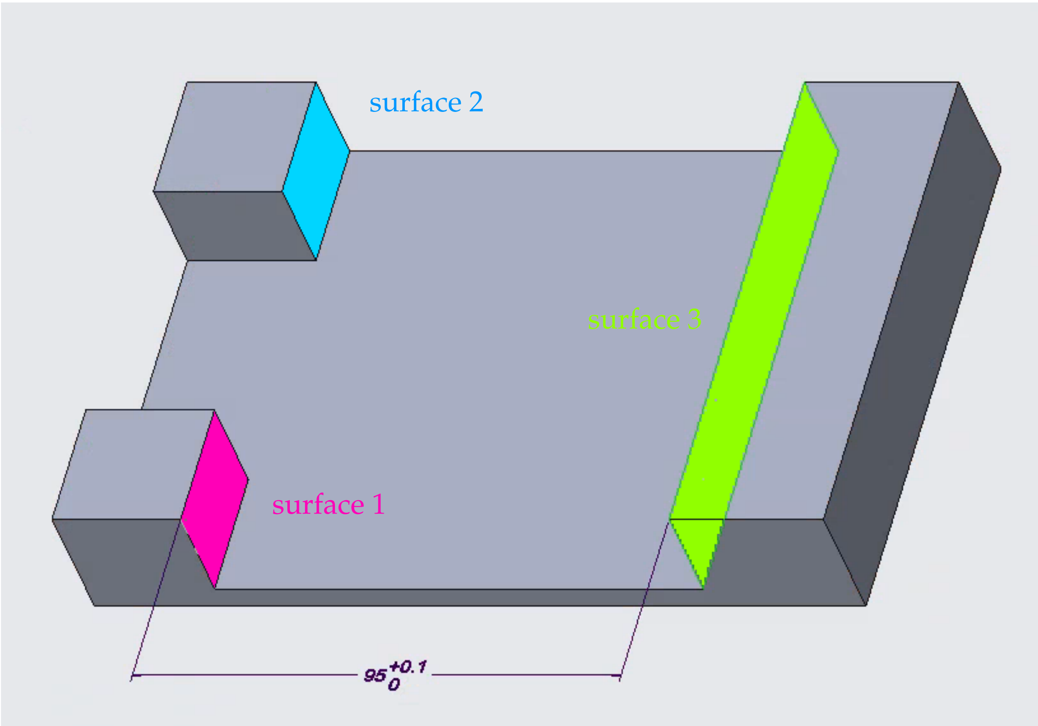


Figure 5.56: An example of a dimension where it is important to know which surfaces belong together

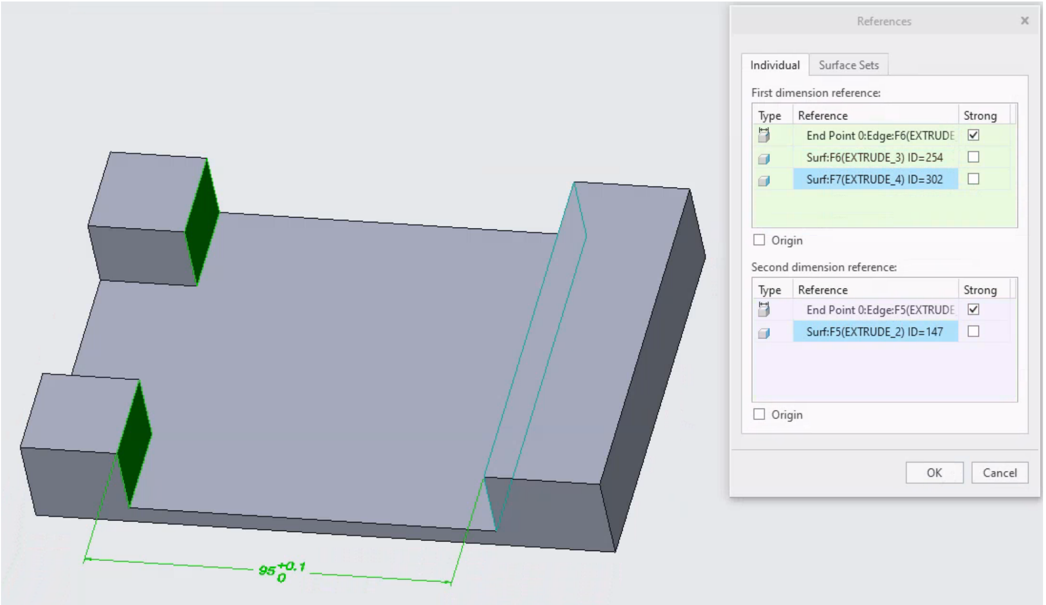


Figure 5.57: Ostensibly it is possible to distinguish between surfaces belonging to the “First Dimension Reference” and surfaces belonging to the “Second Dimension Reference”

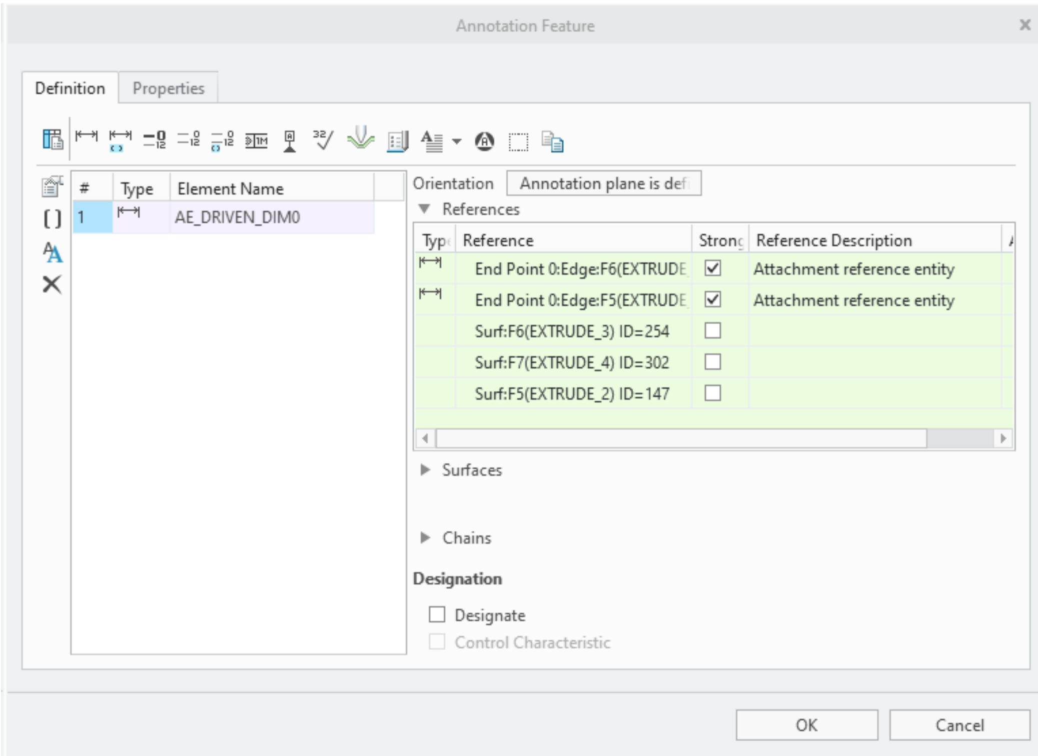


Figure 5.58: After converting the driven dimension to an annotation feature, it becomes clear that the surfaces can no longer be divided into “First Dimension Reference” and “Second Dimension Reference”

With respect to the note specifying the hole itself M3x.5 ISO . . . , this is exactly the same as is described under Option 1.

Since the driven dimension is derived from the model, a change in the model by adjusting the hole pattern will also be reflected in the dimensional numbers of the driven dimension. If the original hole pattern was created with functional dimensioning in mind, this will be retained.

Annotation features

As mentioned previously, the dimensions 40 and 60 in the MBD model in Creo Parametric can be created in different ways. They can be catalogued in different ways. The one that is used in this subsection is

1. Driving dimensions (discussed in a previous subsection)
2. Driven dimensions (discussed in the previous subsection)
3. Annotation features (will be discussed in this subsection)

Annotation features can be thought of as containers that can contain multiple driven dimensions. They can be used to create a group of annotations. Everything discussed at driven dimensions (p. 93) also applies here.

Option 3: Create four holes using the sketched hole feature

As mentioned, there are several ways to create the four tapped holes in PTC Creo Parametric, the CAD system used to demonstrate some of the problems that can occur. Some of the options available are

1. Create four separate tapped holes using the standard hole function

2. Create four tapped holes by creating a pattern from a threaded hole feature

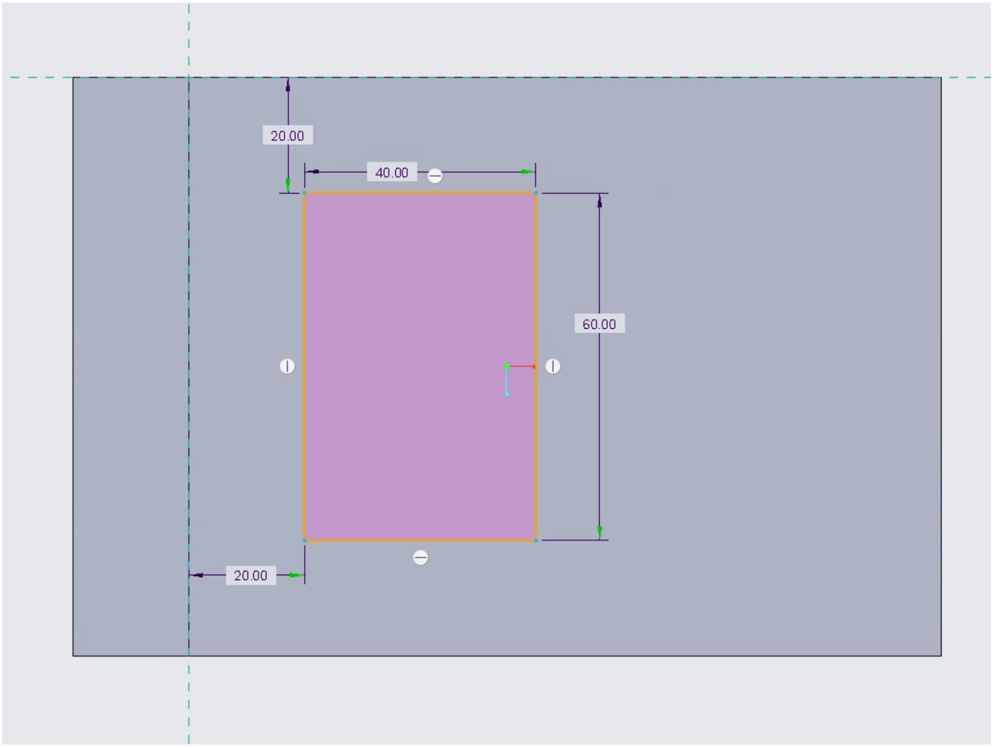
3. Create four threaded holes using the sketched hole feature.

The first two options have been covered in the previous subsections. This subsection covers the third option. A sketched hole feature is an option in PTC Creo Parametric that allows you to create holes while defining their position by endpoints and/or midpoints in a sketch (Figure 5.59). Comparing the dimensioning scheme of the sketch in Figure 5.59a with the one in the MBD model in Figure 5.60, it can be concluded that they are the same. The dimensioning scheme of the sketch conforms to the principles of "functional dimensioning", which is now the result of a proper implementation of "design intent". In the case of option 1, where the four holes are created as individual holes, changing a dimension value can result in the desired grouping of the holes being broken. Changing the dimension from 60 to 40 means that the four holes no longer lie on a rectangle (Figure 5.45a). In the case of option 2, where the four holes are defined as a pattern, the four holes will remain on a rectangle even if the dimension is changed from 60 to 40 (Figure 5.45b). This is also the case with option 3. Everything discussed under option 2 applies here too.

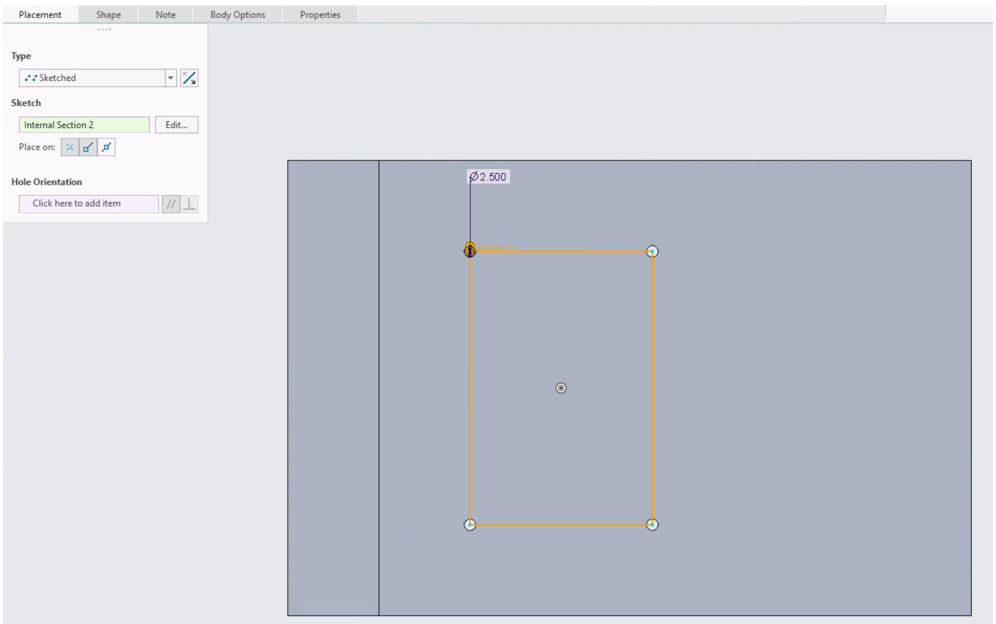
5.3.2 Symmetric and asymmetric tolerances

Symmetric tolerances

With regard to symmetrical tolerances as in the example in subsection 5.3.1, the impact of a discrepancy between the dimensioning scheme used to create the hole features and the dimensioning scheme relevant to production is minimal. If the dimensioning itself is correct, namely if it is such that the product can be assembled when its dimensions remain within the specified tolerance then the impact on the manufacturing of the product is minimal. As the tolerances are symmetrical the nominal values of the



(a) The sketch is dimensioned according to the principles of functional dimensioning, the centres of the holes are automatically assigned to the endpoints of this sketch



(b) The holes are placed at the endpoints of the sketch entities

Figure 5.59: The holes are created using the option "sketched holes"

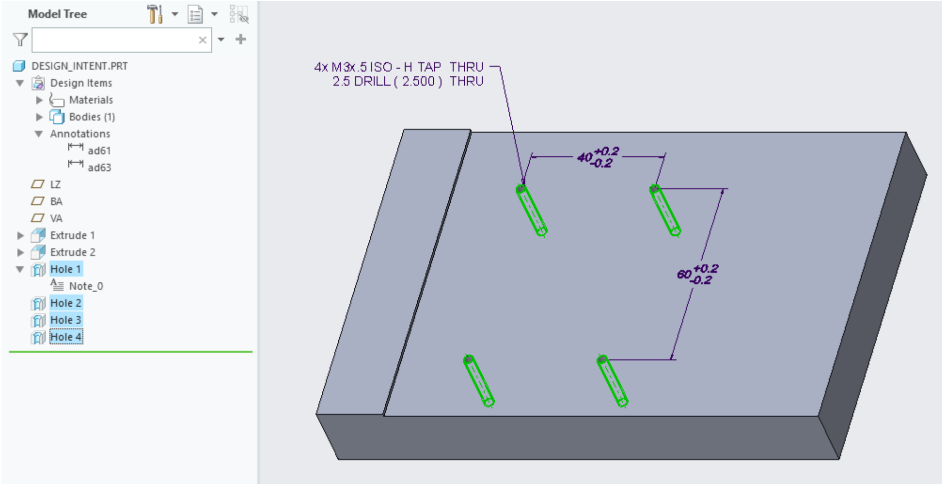


Figure 5.60: functional dimensioning scheme applied in an MBD view

dimensions don't need to be changed and can be used directly for the production of the model. The only effect of the discrepancy between the dimensioning schemes is that the designer and production people must be careful that the grouping of holes (design intent) is maintained when the value of a dimension is changed (Figure 5.15). It is therefore safe to conclude that it is sufficient for the quality and production departments just to be able to read the annotations.

5.3.3 Asymmetric tolerances

Consider the example in Figure 5.61 discussed in the introduction to this chapter (section 5.2).

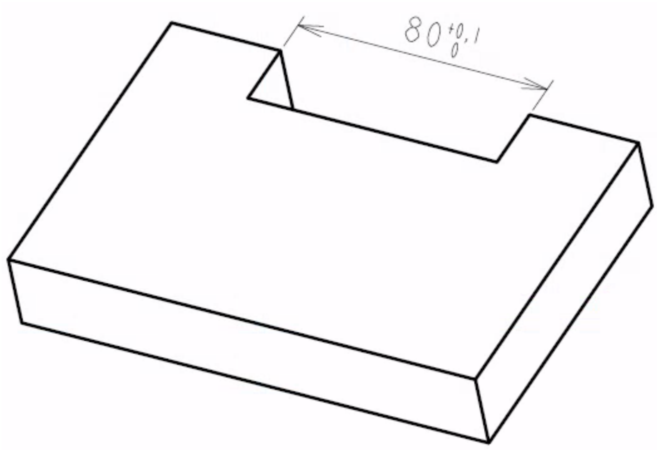


Figure 5.61: Cut with asymmetrical tolerance

In this case, the mere reading of the dimensions is enough for the quality department to be able to approve or reject the product, but not for the manufacturing department. The rectangular cutting operation may not be performed on the nominal size. An offset of 0.05 mm must be created in order to bring the nominal size into the centre of the specified tolerance field (see Figure 5.62) before the cutting operation can be performed on a CNC machine.

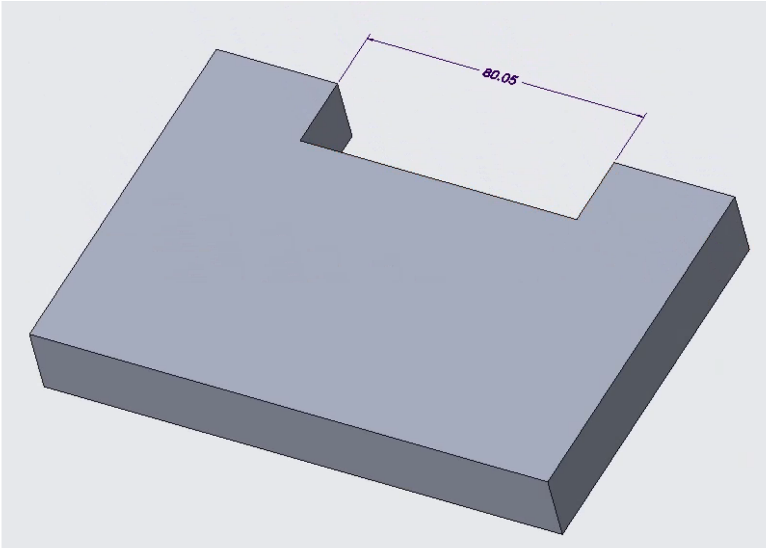


Figure 5.62: Nominal value is changed so it is now in the centre of the specified tolerance field

Two situations can arise:

1. The first is that the manufacturing department and the designer use two different CAD packages. As a result, the CAD model must be exchanged using a neutral exchange format such as STEP. The issue of these formats is discussed in chapter 6.

2. A second situation is where the manufacturing department and the designer use the same CAD package. If the dimensioning scheme used to create the cut feature does not match the dimensioning scheme applied to the MBD model, it is not always easy to modify the cut feature taking into account tolerances assigned to other related dimensions. This has been discussed in the introduction of this chapter (see section 5.2).

Some proponents of MBD advocate banning the use of asymmetric tolerances to solve this problem. However, this leads to other problems. This is discussed in chapter 7.

6

Neutral exchange formats

6.1 Introduction

Within MBD, the 3D model is the authority (Beckers et al. 2016). For MBD to be successful, it is imperative that the exchange of CAD models between the different stakeholders is as smooth as possible. Since not all stakeholders use the same CAD/CAM system, this means that either their system is capable of reading the supplied models in the native format they are created in or that neutral data exchange formats such as STEP, among others, can be used (Wardhani et al. 2016, p. 1). Neutral here refers to the fact that these formats are independent of the CAD system used (Jimmy Nguyen 2023). The previous chapters dealt with the native model. This chapter examines two of the most commonly used formats for exchanging 3D CAD data between CAD/-CAM systems, IGES[1] and STEP (Gielingh 2008; Qin et al. 2017). The literature review revealed that the accuracy of the transfer of the geometry from a CAD model is rarely questioned. It is assumed to be always correct. However, Gerbino 2003 and Gielingh 2008 indicate that the most critical problems encountered when exchanging data are caused by differences in the internal accuracy applied by CAD systems and differences in implementation. It is therefore investigated how the internal accuracy applied when creating a CAD model is transferred between different CAD systems. Here a distinction is made between the accuracy used for geometry (IGES, STEP AP214) and the accuracy used for representing assigned dimensional tolerances (STEP AP242). It then examines how this accuracy affects the correct transfer of CAD model geometry between different CAD systems and how different mathematical implementations affect the accuracy with which a model can be exchanged and affect the transfer of semantic references. Finally, two formats that are increasingly gaining ground in the field of MBD, namely QIF that is mainly focused on metrology (Hedberg Jr et al. 2019) and 3D PDF for general accessibility (Pfouga et al. 2018) are briefly discussed.

6.2 Model accuracy

Section 2.2 discussed the different types of accuracies, namely absolute and relative accuracy and curve tolerance, used in CAD/CAM systems. When neutral exchange formats are used, the accuracy applied in the exchange file is always absolute. Ideally, the absolute accuracy used in the CAD model should exactly match the absolute accuracy used in the exchange file. However, this is by no means always the case. Some CAD systems ensure that this matching happens automatically. Other systems allow the user to impose a specific accuracy for the exchange file or use a default

[1] The last version of IGES, version 5.3, dates from 1996. Nevertheless, IGES is still used and can also be used for MBD, albeit to a limited extent. This is because semantic references are not preserved in this format. Only presentation PMI are supported (Page 2006).

value. In the last two cases the accuracy used for the exchange file is independent of the accuracy of the original CAD model. As a result, exchanging CAD models between different CAD systems is not a simple matter of exporting from system one and importing into system two. The user must be very familiar with the operation of the CAD system used.

To check how the accuracy specified in a CAD system is transferred to an IGES or STEP file and vice versa, a cube was created in various CAD systems (Inventor 2022, CATIA V5-6R2022 SP1, Siemens NX Version 2019 Build 2501, PTC Creo 8.0.4.0) with its centre at the origin of the global coordinate system and with the dimensions 0.004 mm × 0.004 mm × 0.004 mm. This cube is exported as an IGES and a STEP file and imported into the various CAD systems. A cube was chosen for two reasons:

1. The first reason is to avoid the influence of the formulation and the approximation of splines.
2. The second reason is to have a geometry with the same required accuracy according to the three coordinates axes X, Y and Z.

The latter is done to keep it as simple as possible. The value 0.004 mm is chosen to have a value that is small enough to challenge the CAD system in terms of accuracy, but of an order of magnitude that could occur in real-world models. This order of magnitude is used in the application of the ISO System of Limits and Fits. The value 0.004 has been chosen to avoid the number 0.01 in case of rounding. The result of a cube of this size is that the absolute accuracy of this model does not exceed the value of 0.004 mm. If this were not the case, the eight vertices of the cube would be indistinguishable from each other and would be considered as eight coinciding points.

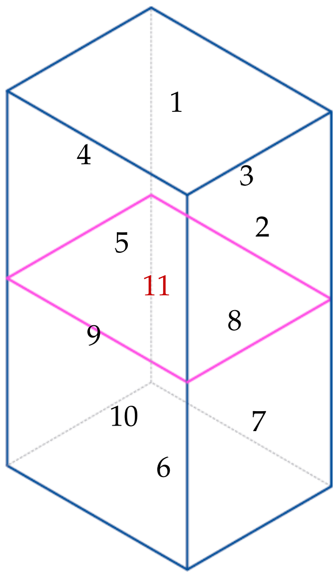
6.2.1 How is model accuracy handled in an IGES file

In an IGES file, the absolute accuracy can be found in the "Global Section" of the file header (see Figure 6.1). It is called the "minimum resolution" (Nasr et al. 2007). In the IGES 5.3 specification this is defined as 'Minimum User-Intended Resolution. This "required, no default" field specifies the smallest distance between coordinates, in model-space units, that the receiving system shall consider as discernible (e.g., if the value is .0001, postprocessors shall consider as "coincident" any coordinate locations in the file which are less than .0001 model-space units apart.).' (Page 2006).

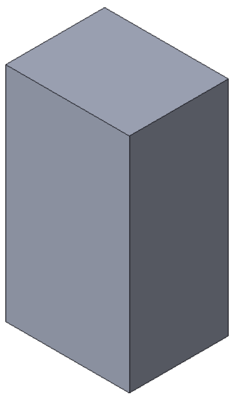
```
START RECORD GO HERE.                                                   S      1
1H,,1H;,5HPart1,9HPart1.igs,62HDASSAULT SYSTEMES CATIA Version 5-6 ReleaG      1
se 2022 - www.3ds.com,35HV5-6R2022_5.32.1.0.01-11-2022.18.00,32,75,6,75,G      2
15,5HPart1,1.0,2,2HMM,1000,1.0,15H20220906.135000,0.001,1.0E+04,8Hu00672G      3
59,4HUCLL,11,0,15H20220906.135000,;                                     G      4
     108       1       0       0       0       0       0       001010001D      1
     108       0       0       1       0                               0D      2
```

Figure 6.1: Absolute accuracy of 0.001 mm specified in the "Global Section" of an IGES file

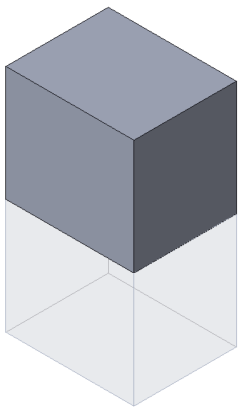
Two criteria are used to determine whether an import is successful or not. A first criterion is whether the geometry is imported correctly. This can be done unambiguously because this is a CAD model with an analytical shape, a cube. A second criterion is whether the exported model can be imported as a solid. A distinction must be made between a model containing only a solid and a model containing a solid and a surface. There are two methods that can be applied to export this model to IGES. With the first method the CAD system tries to retain the solid. With the second method the model is exported as a collection of surfaces. Originally, IGES only allowed surface models to be defined. If the surfaces clearly bound a volume, the receiving CAD system could use this to redefine a solid. However, if the original model is a mixture of a solid and surfaces, it is no longer so clear to redefine the original solid. The additional surfaces can lead to multiple combinations that result in a different solid (see Figure 6.2). For this reason, later versions of the IGES standard added the ability to define solid models in IGES. However, this option is not supported by all CAD systems. In the tests carried out here, in order to avoid any ambiguity in redefining the original solid, only the case where there is only one solid in the original model was tested. If the native CAD model contains only a solid and is exported as surfaces, it could be checked whether the collection of surfaces that form the boundary of a volume is automatically converted to a solid after import. This is not done in these tests. When both a solid and a surface model are present in the native model and only surfaces are defined in the IGES file, this may complicate the automatic recognition of the solid body when both solids and surfaces are present in the original CAD model.



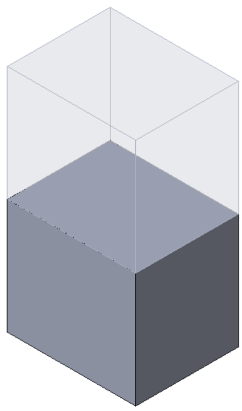
(a) The original CAD model contains surface 11 and a solid defined by surfaces 1, 6, 2, 3, 4, 5, 7, 8, 0, 10



(b) A first possible solid result after import



(c) A second possible solid result after import



(d) A third possible solid result after import

Figure 6.2: Possible result for solids after import of the IGES file

Export from Inventor 2022

A cube was created with the aforementioned dimensions and exported to IGES (see Figure 6.3).

As the dimensions of the cube are 0.004 mm × 0.004 mm × 0.004 mm, the smallest distance between two vertices is 0.004 mm. Therefore, in order to be able to distinguish the two vertices, the expected absolute accuracy is at most 0.004. However, as shown in Figure 6.4, the file has an accuracy of 0.01 mm.

To study the effect of this incorrect accuracy (0.01 mm instead of 0.004 mm), the IGES file was read into all the CAD systems for this test.

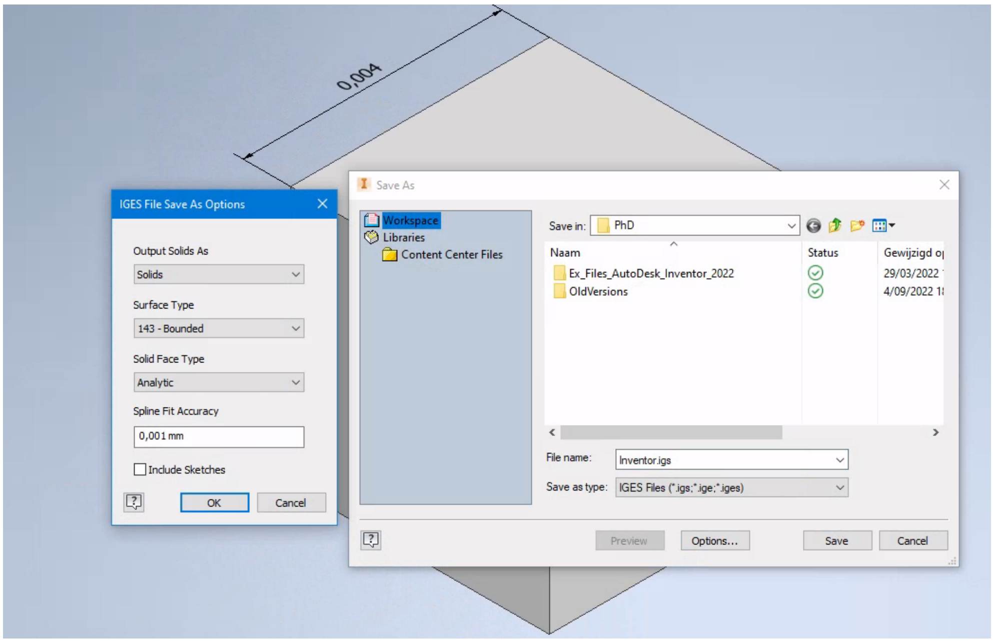
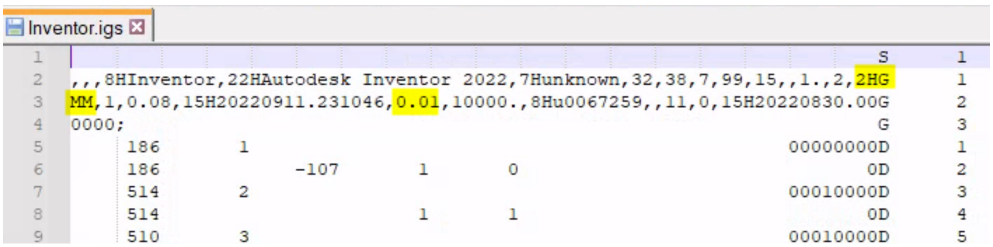


Figure 6.3: Settings that were used to export the Inventor 2022 model to IGES




```
                                                                        S      1
,,,8HInventor,22HAutodesk Inventor 2022,7Hunknown,32,38,7,99,15,,1.,2,2HG      1
MM,1,0.08,15H20220911.231046,0.01,10000.,8Hu0067259,,11,0,15H20220830.00G      2
0000;                                                                   G      3
     186       1                                                00000000D      1
     186              -107       1       0                            0D      2
     514       2                                                00010000D      3
     514                         1       1                            0D      4
     510       3                                                00010000D      5
```

Figure 6.4: IGES file created by Inventor 2022 has an absolute accuracy of 0.01 mm

Import in CATIA V5-6R2022 SP1

Many CAD users rarely change import settings, therefore the default options for IGES import have been used (see Figure 6.5).

Reading in the IGES file fails completely. All that remains of the original cube is a triangle (see Figure 6.6).

To check whether this is really caused by incorrect accuracy in the IGES file, it was manually changed from 0.01 to 0.004 by editing the IGES file directly in a text editor.

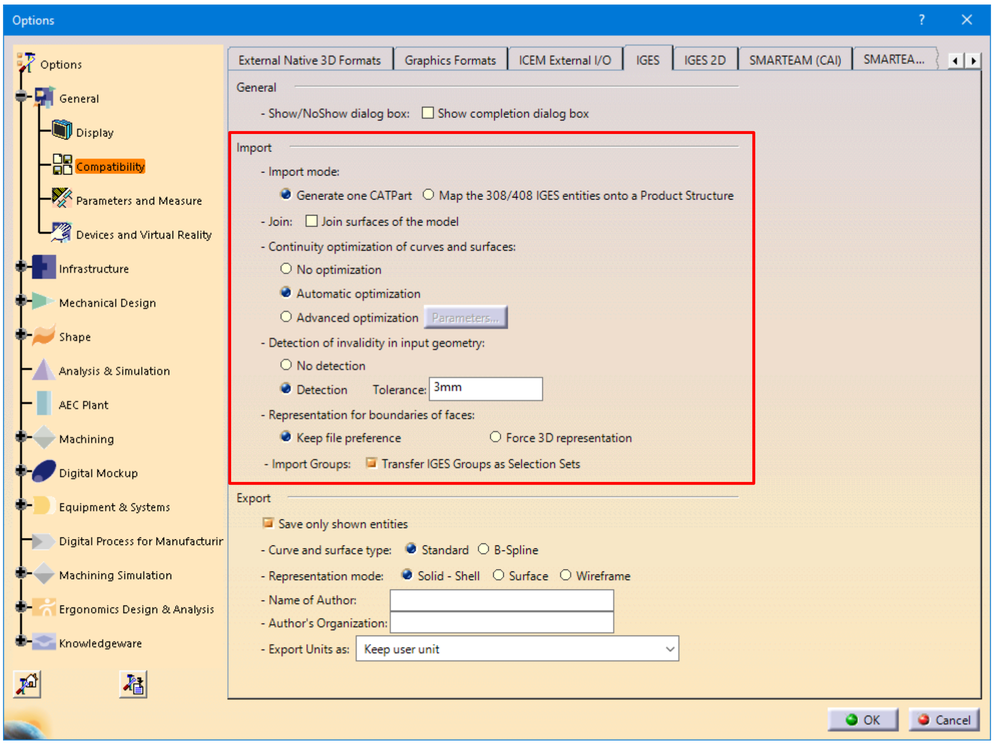


Figure 6.5: Default settings for importing IGES files in CATIA v5

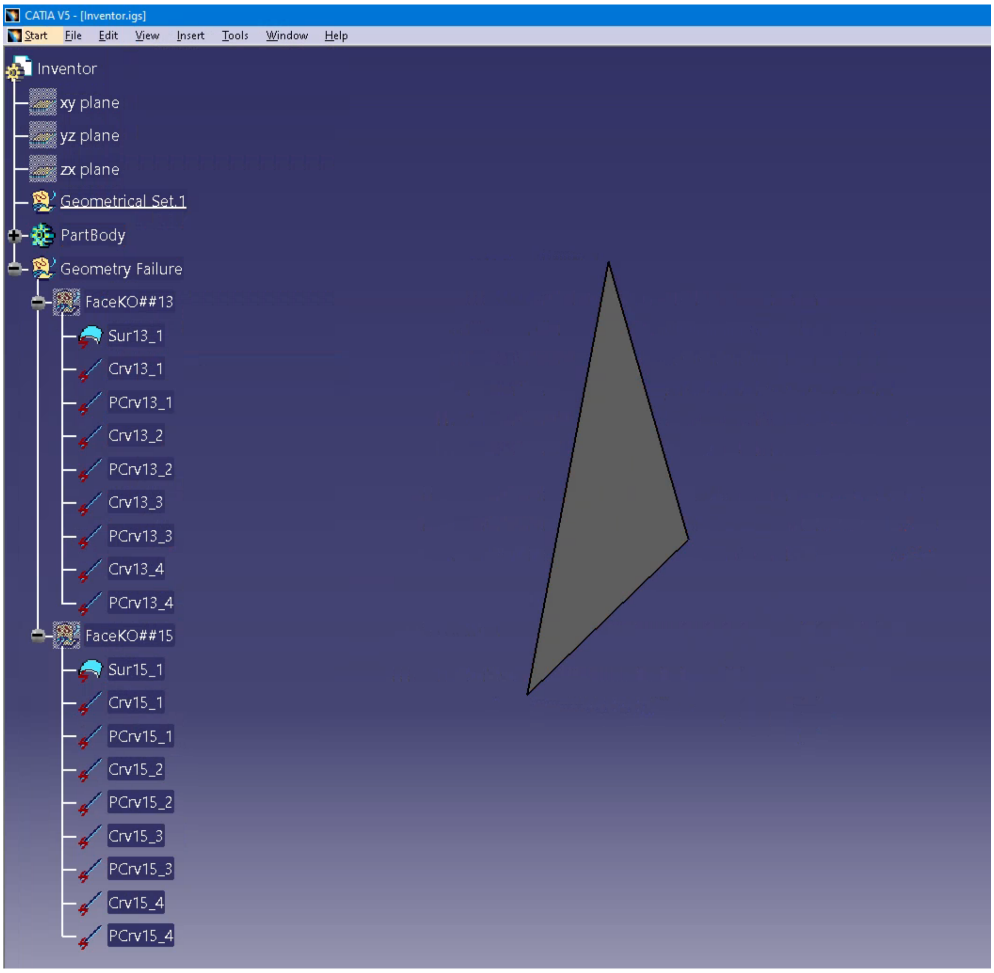


Figure 6.6: Importing the IGES file in CATIA V5 fails completely. All that remains of the original cube is a triangle.

The file is then re-read in CATIA V5. The reconstruction of the geometry still fails completely. So there is another reason for this failure. To find out the accuracy used internally, the model was exported back to IGES and the accuracy defined here was checked. This accuracy is 0.001 mm, which is the internal accuracy CATIA uses for ordinary (medium-sized) models. By default, Inventor exports a solid in IGES to "Solid" (see Figure 6.3). Although this is allowed by the IGES specification (Mattei 1993; Page 2006), this is an option not supported by all CAD systems. To check if the export setting to export a solid as a solid is causing the IGES file to be imported incorrectly, the cube is exported to IGES again in Inventor 2022, but this time the "Output Solids As" option is set to "Surfaces" (see Figure 6.7). The result can be seen in Figure 6.7

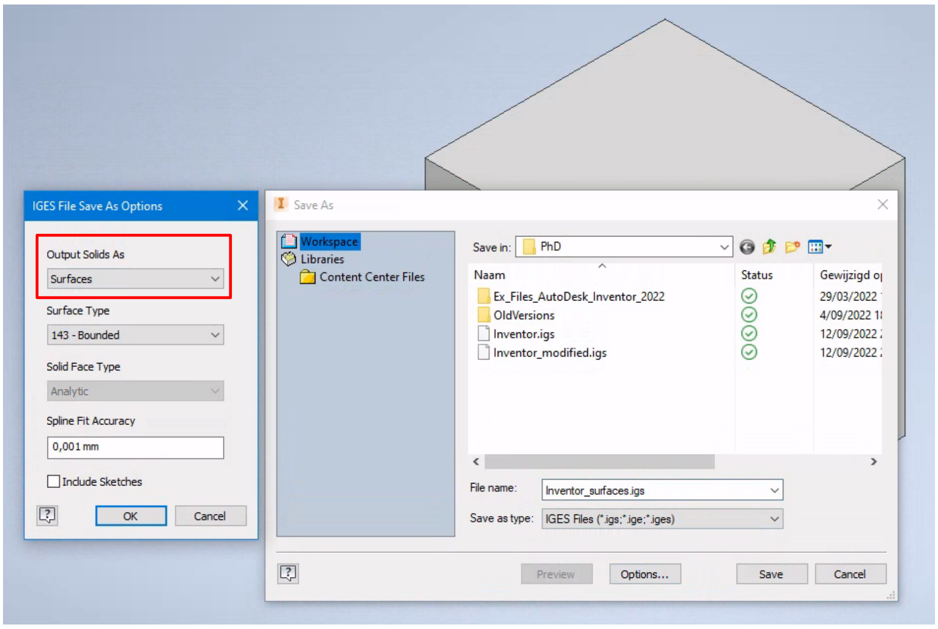


Figure 6.7: Exporting solids as surfaces by Inventor 2022

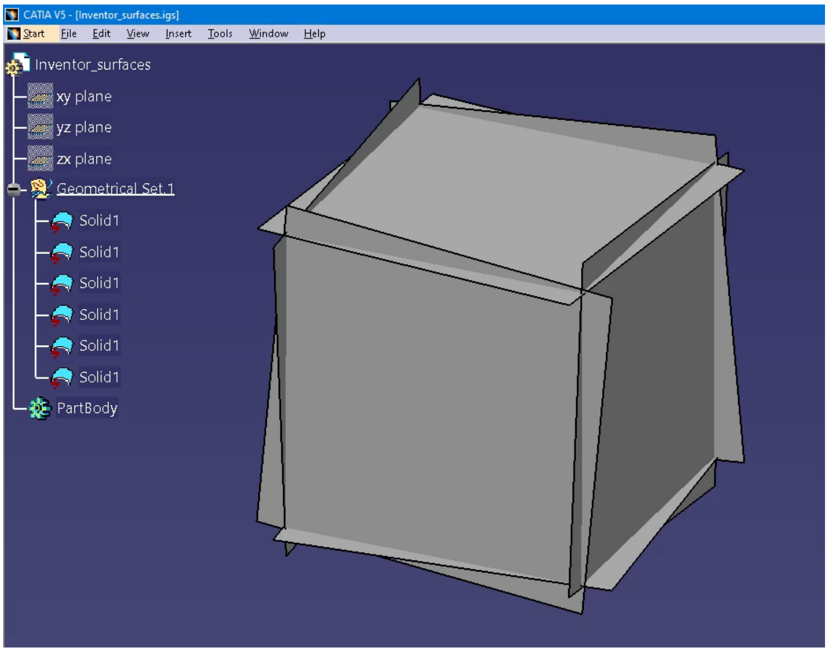


Figure 6.8: After exporting the solid as surfaces, there is an improvement, but still not a satisfactory result

The fact that the surfaces do not match (see Figure 6.8) may be caused by the accuracy being incorrectly specified in the IGES file or by the way the surfaces are defined in the IGES file. Manually changing the accuracy from 0.01 to 0.004 gives the same incorrect result. In contrast, when the surfaces are defined as "Trimmed" instead of "Bounded" (see Figure 6.9), the result is correct both in terms of the shape and the dimensions. Six square surfaces are now obtained (see Figure 6.10). These still need to be converted into a solid in CATIA V5. From the above it can be concluded that the accuracy specified in the IGES file is ignored.

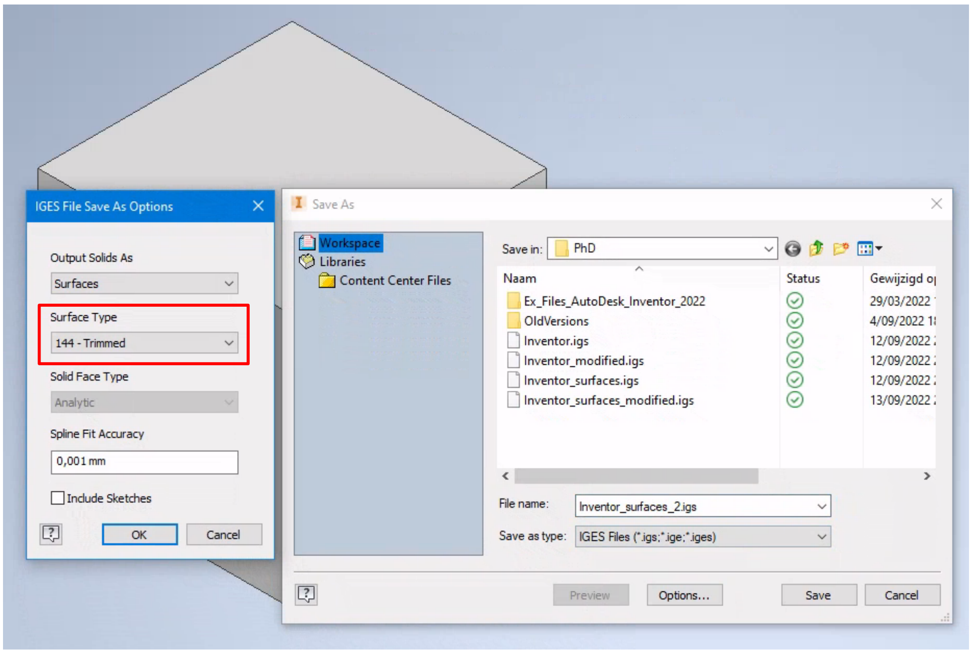


Figure 6.9: Defining surfaces as "trimmed surfaces" when exporting by Inventor 2022

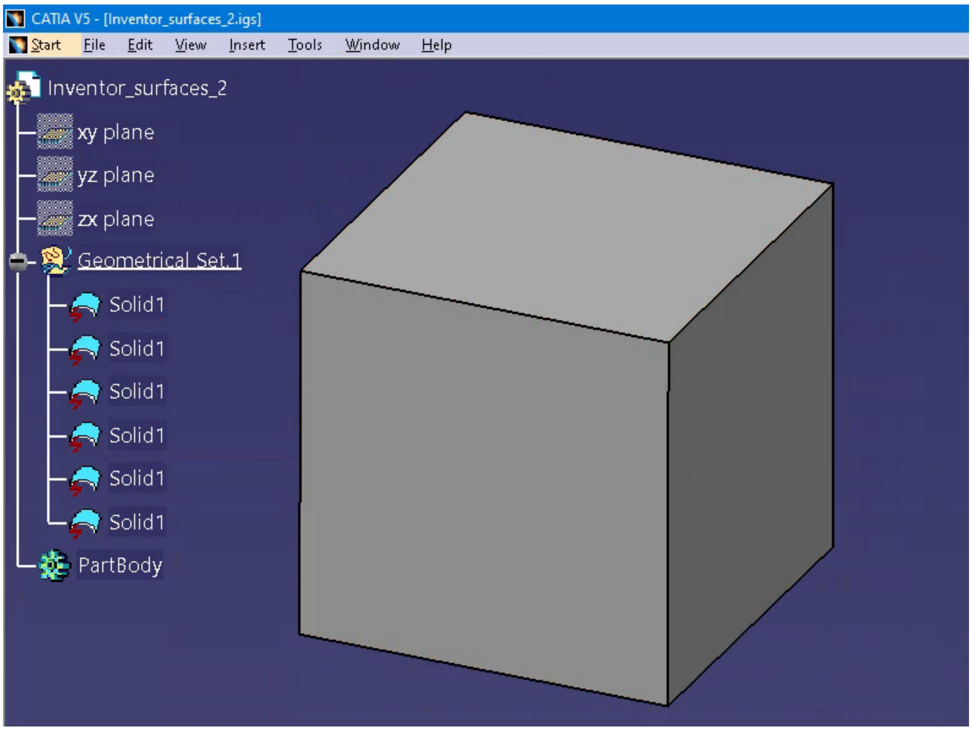


Figure 6.10: After exporting the solid as trimmed surfaces, the result is correct both in terms of the shape and the dimensions.

An overview of all test results for the IGES files generated by Inventor 2022 and imported into CATIA v5 is shown in Table 6.1.

Table 6.1: A summary of the results for CATIA V5 for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	No	No	No	No	Yes	Yes
Export[1] accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Solid recognised	No	No	-	-	-	-

A full report of the tests with exporting to IGES by the other CAD systems can be found in the Appendix on page A1. A summary of all these tests for the different CAD systems is given in Table 6.2, Table 6.3, Table 6.4 and Table 6.5 on the following pages. The unit used in all tables is mm. Two things stand out:

1. The first point to note is that the results suggest that the so-called "minimum resolution" defined in the IGES standard is completely ignored by the various CAD systems. Each CAD system handles this in its own way. Inventor always uses the same accuracy (0.01 mm). CATIA and Siemens NX use the accuracy set to create a new mechanical part. When using the "Model Accuracy:Automatic" or "Model Accuracy:Internal" option, Creo determines the accuracy based on the size of the model (PTC 2023a). The "Model Accuracy:Automatic" option is the one listed in the tables.

2. The second point to note is that some CAD systems cannot import the file they themselves have exported.

[1] The imported model is re-exported to IGES and the accuracy defined in this IGES file is examined.

Table 6.2: Summary of the results of exporting an IGES file generated by Inventor 2022 using different export settings and importing it into other CAD systems

IGES settings export from Inventor 2022																								
Output Solids	Solids				Solids				Surfaces				Surfaces				Surfaces				Surfaces			
Surface Type	Bounded				Bounded				Bounded				Bounded				Trimmed				Trimmed			
Solid Face Type	Analytic				Analytic				Analytic				Analytic				Analytic				Analytic			
Spline Fit Accuracy	0.001				0.001				0.001				0.001				0.001				0.001			
Accuracy IGES file	0.01				0.004				0.01				0.004				0.01				0.004			
	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo
Import successful	Yes	No	Yes	Yes	Yes	No	Yes	Yes	Yes	No	Yes	Yes	Yes	No	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Export accuracy	6.93e-07	0.001	1.0e-05	6.93e-07	6.93e-07	0.001	1.0e-05	6.93e-07	6.93e-07	0.001	1.0e-05	6.93e-07	6.93e-07	0.001	1.0e-05	6.93e-07	6.93e-07	0.001	1.0e-05	6.93e-07	6.93e-07	0.001	1.0e-05	6.93e-07
Solid recognised	Yes	No	Yes	Yes	Yes	No	Yes	Yes	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-	-

Table 6.3: Summary of the results of exporting an IGES file generated by CATIA V5 using different export settings and importing it into other CAD systems

IGES settings export from CATIA V5								
Output Solids	Solids				Surface			
Surface Type	Standard				Standard			
Accuracy IGES file	0.001				0.001			
	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo
Import successful	Yes	No	Yes	Yes	Yes	Yes	Yes	Yes
Export accuracy	0.01	0 001	1e-05	6.93e-07	0.01	0.001	1e-05	6.93e-07
Solid recognised	Yes	-	No	Yes	-	-	-	-

Table 6.4: Summary of the results of exporting an IGES file generated by Siemens NX using different export settings and importing it into other CAD systems

IGES settings export from Siemens NX								
Output Solids	Solids				Surface			
Surface Type	Standard				Standard			
Accuracy IGES file	1e-05				1e-05			
	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo
Import successful	Yes	Yes	Yes	Yes	No	No	No	no
Export accuracy	0.01	0.001	1e-05	6.96e-07	-	0.001	1e-05	0.087
Solid recognised	Yes	No	No	Yes	-	-	-	-

Table 6.5: Summary of the results of exporting an IGES file generated by PTC Creo using different export settings and importing it into other CAD systems

IGES settings export from PTC Creo								
Output Solids	Solids				Surface			
Surface Type	Standard				Standard			
Accuracy IGES file	0.001				0.001			
	Inventor	CATIA	NX	Creo	Inventor	CATIA	NX	Creo
Import successful	Yes	No	Yes	Yes	Yes	Yes	Yes	Yes
Export accuracy	0.01	0.001	1e-05	6.96e-07	0.01	0.001	1e-05	6.96e-07
Solid recognised	Yes	Yes	Yes	Yes	-	-	-	-

6.2.2 STEP

When used in the context of MBD, a distinction must be made between STEP AP203 and STEP AP214 on the one hand and STEP AP242 on the other. STEP AP203 and STEP AP214 focus on model geometry, while STEP AP242 adds support for 3D annotations. This will be discussed in the following sections.

STEP AP203 and STEP AP214

There are two versions of STEP AP203, namely ISO 10303-203:1994 also called "AP203 Edition 1" which is further subdivided into an "international standard" and an "extended international standard" (PTC 2022b) and ISO 10303-203:2011 also called "AP203 Edition 2" (*ISO 10303-203* 2022). There are three editions of STEP AP214 (ISO 10303-214:2001, ISO 10303-214:2003 and ISO 10303-214:2010) (*ISO 10303-214* 2022). In terms of model accuracy, all these versions use absolute accuracy. In the STEP standard, this is called 'global uncertainty' (Boy et al. 2014, p. 8). The value is determined by the parameter "`uncertainty_measure_with_unit`". (Pratt and Anderson 2001; PDES 1998). Figure 6.11 shows the parameter with the arguments "0.005" referring to the value of the accuracy and "#12" referring to the units used, namely mm.

```
#167=CARTESIAN_POINT('Axis2P3D Location',(40.,0.,0.)) ;
#8=PRODUCT_RELATED_PRODUCT_CATEGORY('part',$,(#5)) ;
#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,'distance_accuracy_value','CONFUSED CURVE UNCERTAINTY') ;
#29=STYLED_ITEM(' ',(#28),#21) ;
#28=PRESENTATION_STYLE_ASSIGNMENT((#27)) ;
#27=SURFACE_STYLE_USAGE(.BOTH.,#26) ;
#26=SURFACE_SIDE_STYLE(' ',(#25)) ;
#25=SURFACE_STYLE_FILL_AREA(#24) ;
```

```
#6=PRODUCT_DEFINITION_FORMATION_WITH_SPECIFIED_SOURCE('',' ',#5,.NOT_KNOWN.) ;
#20=SHAPE_DEFINITION_REPRESENTATION(#11,#19) ;
#12=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;
#13=(NAMED_UNIT(*)PLANE_ANGLE_UNIT()SI_UNIT($,.RADIAN.)) ;
#14=(NAMED_UNIT(*)SI_UNIT($,.STERADIAN.)SOLID_ANGLE_UNIT()) ;
#16=(GEOMETRIC_REPRESENTATION_CONTEXT(3)GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#15))GLOBAL_UNIT_ASSIGNED_CONTEXT((#12,#13,
_CONTEXT(' ',' ')) ;
ENDSEC;
```

Figure 6.11: Absolute accuracy of 0.005 mm specified in a STEP file

Some CAD systems ensure that the accuracy used in the STEP file automatically matches the accuracy used in the original model (see Figure 6.12).


```
#153=DIMENSIONAL_EXPONENTS(0.E0,0.E0,0.E0,0.E0,0.E0,0.E0,0.E0);
#155=PLANE_ANGLE_MEASURE_WITH_UNIT(PLANE_ANGLE_MEASURE(1.745329251994E-2),#154);
#156=(CONVERSION_BASED_UNIT('DEGREE',#155)NAMED_UNIT(*)PLANE_ANGLE_UNIT());
#158=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(1.23E-1),#152,'closure',
'Maximum model space distance between geometric entities at asserted connectivities');
#159=(GEOMETRIC_REPRESENTATION_CONTEXT(3)GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((
#158))GLOBAL_UNIT_ASSIGNED_CONTEXT((#152,#156,#157))REPRESENTATION_CONTEXT
('ID1','3'));
```

Figure 6.12: Absolute accuracy of the CAD model designed with PTC Creo Parametric corresponds to that in the exported STEP file

To verify the accuracy defined in an exported STEP AP203/AP214 file, a CAD model was created of a cube whose dimensions are 0.004 mm $\times$ 0.004 mm $\times$ 0.004 mm. A cube was chosen for the reasons described on page 103. MBD requires not only the exchange of geometry, but also the exchange of colours assigned to the 3D model and of annotations applied to the 3D model. Of the different versions of STEP AP203 and AP214, only the second edition of STEP AP203 and all versions of STEP AP214 support the use of annotations, albeit only presentation PMI (PTC 2022a). For some of the CAD systems in this test, notably Siemens NX and Inventor, it is not immediately clear to the designer which version of AP203 is being used, creating doubt as to whether the 3D annotations assigned in the CAD model are actually included in the STEP file. Only by looking at the value of the `FILE_SCHEMA` parameter in the STEP file (see Figure 6.13) can the designer find out which specific AP203 version is being used (see Table 6.6).

```
ISO-10303-21;
HEADER;
FILE_DESCRIPTION((''),'2;1');
FILE_NAME('PRT0001','2022-10-16T20:17:00',('u0067259'),('KULeuven-Diepenbeek'),
'CREO PARAMETRIC BY PTC INC, 2022014','CREO PARAMETRIC BY PTC INC, 2022014','');
FILE_SCHEMA(('CONFIG_CONTROL_DESIGN'));
ENDSEC;
DATA;
```

Figure 6.13: Specification of the application protocol used in the STEP file

Table 6.6: Specification which application protocol is used

	FILE_SCHEMA
AP203_ed1_is	'CONFIG_CONTROL_DESIGN'
AP203_ed1_is_ext	'CONFIG_CONTROL_DESIGN', 'GEOMETRIC_VALIDATION_PROPERTIES_MIM'
AP203_ed2	'AP203_CONFIGURATION_CONTROLLED_3D_DESIGN_OF_MECH ANICAL_PARTS_AND_ASSEMBLIES_MIM_LF { 1 0 10303 403 2 1 2 }'

Since all editions of STEP AP214 support annotations, it does not matter much which version is used. Therefore, the choice was made to export the cube to STEP AP214.

Export from Inventor 2022

A cube is created with the aforementioned dimensions and exported to STEP AP214 (see Figure 6.14). The option "Include Sketches" is not checked, so that only the solid with the associated surfaces is exported. The configuration parameter "Spline Fit Accuracy" has no influence because there are no splines in the model.

As the dimensions of the cube are 0.004 mm $\times$ 0.004 mm $\times$ 0.004 mm, the smallest distance between two vertices is 0.004 mm. Thus, the expected absolute accuracy is at most 0.004. However, as shown in Figure 6.15, the file has an accuracy of 0.01 mm. This value can not be the result of the rounding off of 0.004.

To study the effect of this incorrect accuracy (0.01 mm instead of 0.004 mm), the STEP file was read into all the CAD systems and re-exported to STEP AP214 to see the impact on accuracy in the newly generated STEP file.

Import in CATIA V5-6R2022 SP1

The default options for STEP import were used (see Figure 6.16). CATIA has an option to change the accuracy with which a CAD model is created to a limited extent. This is the "Scale" option (see Figure 6.17). Three settings are possible. The first is "Small range". The absolute accuracy of the CAD model is set to 1×10^{-5} mm. The second is "Normal range". The absolute accuracy of the CAD model is set to 1×10^{-3} mm. The third is the "Large range". The absolute accuracy of the CAD model is then set to 0.1 mm. These are fixed values in CATIA v5.

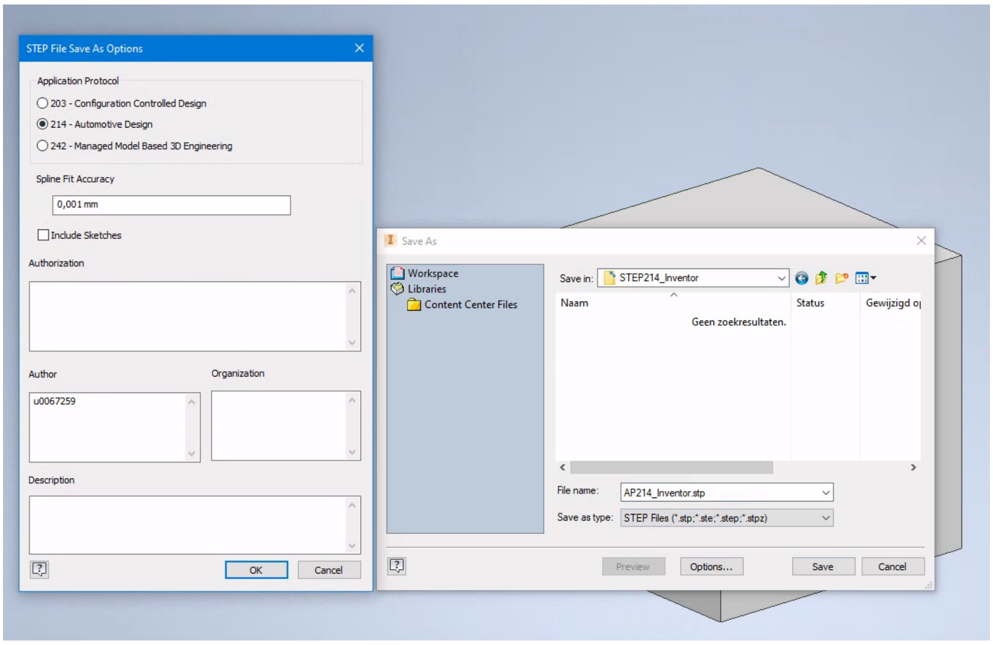


Figure 6.14: Settings used to export model to STEP AP214 in Inventor 2022

```
#188=CARTESIAN_POINT('Origin',(1.82267025552143,1.86728318492768,0.002));
#189=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#193,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
#190=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#193,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
```

Figure 6.15: STEP AP214 file created by Inventor 2022 has an absolute accuracy of 0.01 mm

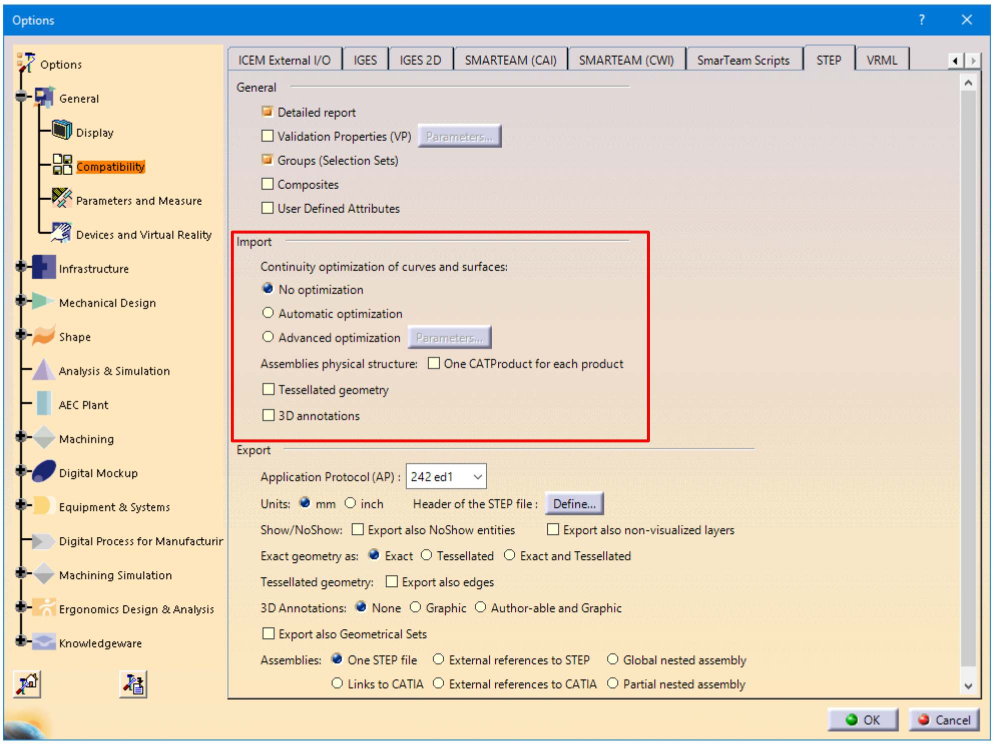


Figure 6.16: Default settings for importing STEP files in CATIA v5

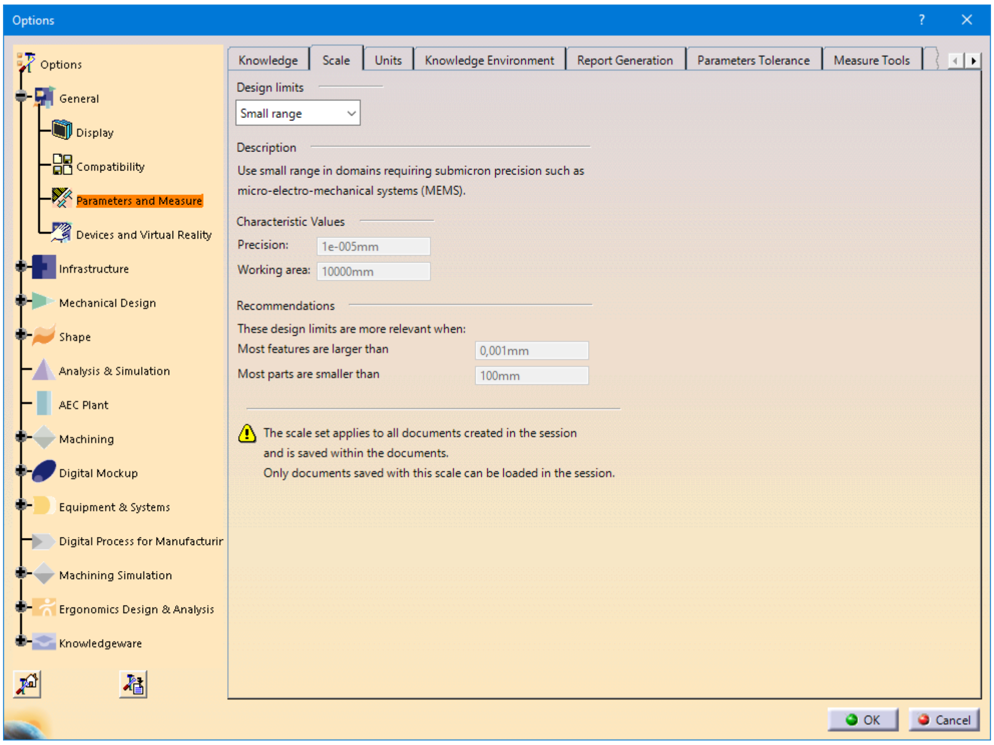


Figure 6.17: “Scale” configuration in CATIA to modify the absolute accuracy

Table 6.7: A summary of the results for CATIA V5 for the import of the STEP AP214 file generated by Inventor 2022

	Large range	Normal range	Small range
Accuracy STEP file	0.01 mm	0.01 mm	0.01 mm
Import successful	No	Yes	Yes
Internal accuracy used by CATIA	0.1 mm	0.001 mm	1×10^{-5} mm
Export accuracy	0.5 mm	0.005 mm	5×10^{-5} mm
Solid recognised	-	Yes	Yes

According IBM documentation (IBM 2003), 0.005 mm is a fixed value used for export, independent of the actual accuracy used, and cannot be changed by the user. This is based on the default setting for the design limits which is “Normal range”. Based on the results in Table 6.7, it is concluded that for CATIA, the accuracy of a STEP file exported by CATIA is equal to 5 × the set model accuracy for newly created CAD models.

A full report of the tests with exporting to STEP AP214 by the other CAD systems can be found in the Appendix on page A24. A summary of all these tests for the different CAD systems is given in Table 6.8, Table 6.9, Table 6.10, Table 6.11 on the following pages. The unit used is mm.

Table 6.8: Summary of the results of importing and exporting a STEP AP214 file generated by Inventor 2022

STEP AP214 generated by Inventor 2022									
Accuracy STEP file	0.01								
	Inventor	CATIA			NX	Creo			
		Large	Normal	Small		Automatic	External	Internal	Template (0.01)
Import successful	Yes	No	yes	Yes	Yes	Yes	Yes	Yes	No
Internal accuracy used	-	0.1	0.001	1e-5	-	6.928e-7	5.774e-6	6.928e-7	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.001	6.928e-7	5.774e-6	6.928e-7	0.01
Solid recognised	Yes	-	Yes	Yes	Yes	Yes	Yes	Yes	No

Table 6.9: Summary of the results of importing and exporting a STEP AP214 file generated by CATIA V5

STEP AP214 generated by CATIA V5									
Accuracy STEP file	0.005								
	Inventor	CATIA			NX	Creo			
		Large	Normal	Small		Automatic	External	Internal	Template (0.01)
Import successful	yes	No	Yes	Yes	Yes	Yes	Yes	Yes	No
Internal accuracy used	-	0.1	0.001	1e-5	-	6.928e-7	5.774e-6	6.928e-7	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.001	6.928e-7	5.774e-6	6.928e-7	0.01
Solid recognised	yes	-	Yes	Yes	Yes	Yes	Yes	Yes	No

Table 6.10: Summary of the results of importing and exporting a STEP AP214 file generated by Siemens NX Version 2019

STEP AP214 generated by Siemens NX Version 2019									
Accuracy STEP file	2e-5								
	Inventor	CATIA			NX	Creo			
		Large	Normal	Small		Automatic	External	Internal	Template (0.01)
Import successful	Yes	No	Yes	Yes	Yes	Yes	Yes	Yes	No
Internal accuracy used	-	0.1	0.001	1e-5	-	6.928e-7	5.774e-6	6.928e-7	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.001	6.928e-7	5.774e-6	6.928e-7	0.01
Solid recognised	Yes	-	Yes	Yes	Yes	Yes	Yes	Yes	No

Table 6.11: Summary of the results of importing and exporting a STEP AP214 file generated by PTC Creo 8.0.4.0

STEP AP214 generated by PTC Creo 8.0.4.0									
Accuracy STEP file	0.001								
	Inventor	**CATIA**			**NX**	**Creo**			
		Large	Normal	Small		Automatic	External	Internal	Template (0.01)
Import successful	Yes	No	Yes	Yes	Yes	Yes	Yes	Yes	No
Internal accuracy used	-	0.1	0.001	1e-5	-	6.928e-7	5.774e-6	6.928e-7	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.001	6.928e-7	5.774e-6	6.928e-7	0.01
Solid recognised	Yes	-	Yes	Yes	Yes	Yes	Yes	Yes	No

The overview in Table 6.11 clearly shows that the so-called "global uncertainty", as defined in the STEP standard, is completely ignored by the different CAD systems. The accuracy specified in the STEP file created by exporting the original CAD model is 0.001 mm. After import, the accuracy is different for each CAD system. The values are 0.5 mm, 0.01 mm, 0.005 mm, 0.001 mm, 5.774×10^{-6} mm, 6.928×10^{-7} mm. Each CAD system or CAD kernel handles the absolute accuracy applied to a model in its own way. If the accuracy after import does not match that of the original CAD model, this may affect the modification of a nominal dimension of the model to bring it to the centre of an assigned asymmetric tolerance. For example, if the accuracy is 0.01 mm and the dimension is $100^{0.005}_{0}$, it will not be possible to change the nominal value 100 to 100.0025 as 0.0025 is less than 0.01.

STEP AP242

STEP AP242 is used to exchange MBD models between different stakeholders using different CAD systems (Sarkar et al. 2019). As is the case with STEP AP203 and AP214, STEP AP242 also uses absolute accuracy to specify the model accuracy used. This model accuracy is specified with the parameter `UNCERTAINTY_MEASURE_WITH_UNIT`. The way the various CAD systems (Inventor, PTC Creo, Siemens NX and CATIA V5) deal with this parameter is the same as it is with STEP AP214. This means that everything that has been discussed in the previous sections about the handling of the model accuracy applies to all versions of STEP. When discussing the fact that CAD systems ignore the value of the `UNCERTAINTY_MEASURE_WITH_UNIT` parameter with people from the ProSTEP organisation, they explained that this parameter has been replaced by new parameters such as:

- `QUALIFIED_REPRESENATION_ITEM`
- `VALUE_FORMAT_TYPE_QUALIFIER`

These parameters were introduced in STEP AP242 to specify the accuracy assigned to dimensions created as annotations on the 3D model (Boy et al. 2014). Because of the contradiction, the parameters are used to specify the accuracy of the model geometry versus the parameters are used to specify the accuracy assigned to dimensions, this was further investigated through a series of tests. One such test is described in the next section.

Example

A model with different dimensions to those used in the previous tests was chosen. The reason for this is that the previous tests dealt with the accuracy intended for the construction of the model geometry. In this case it is about the accuracy of the assigned tolerances. How many digits are relevant and should be displayed. In Inventor 2022, a beam model is created with the dimensions 100 mm × 100 mm × 50 mm. In it, $100^{+0.035}_{-0.000}$, a dimension with a lower and an upper tolerance is created (see Figure 6.18). This model is exported to STEP AP242 (see Figure 6.19). Analysis of the STEP file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the dimension is stored as "presentation PMI " (Figure 6.20) as well as "representation PMI" (Figure 6.21). An excerpt from the STEP file can be seen in Figure 6.22.

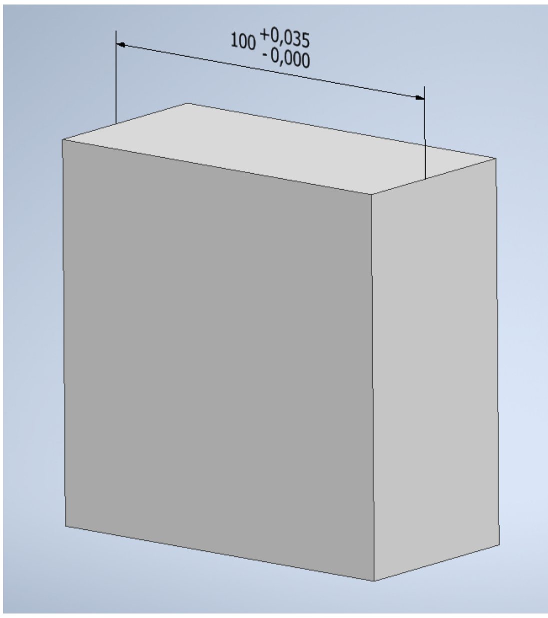


Figure 6.18: Beam model with a dimension with a lower and an upper tolerance (Inventor 2022)

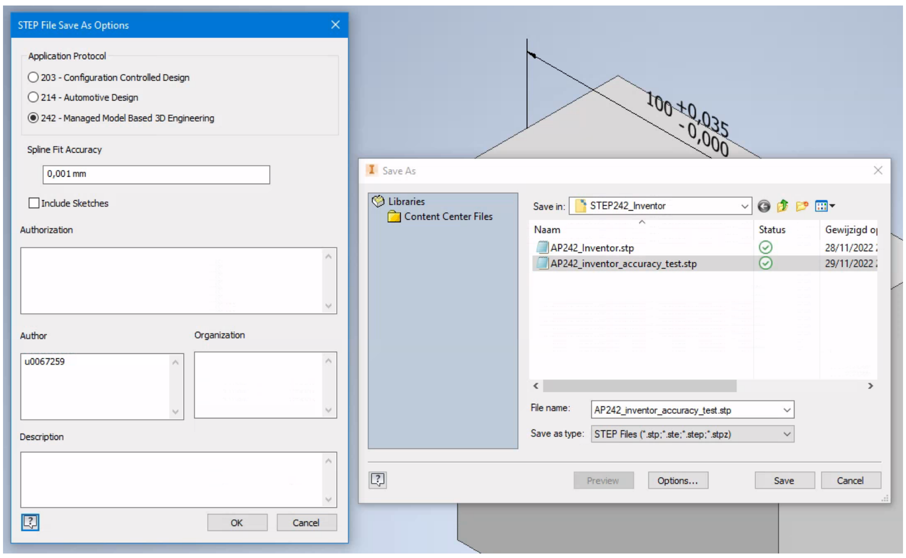


Figure 6.19: Settings used for export to STEP AP242 by Inventor 2022

draughting_callout (1)

ID	name	contents
78	Linear Dimension 1	(1) tessellated_annotation_occurrence 90

Figure 6.20: Excerpt from the results of the NIST STEP File Analyzer and Viewer showing part of the PMI presentation data structure of the dimension added to the 3D CAD model

dimensional_characteristic_representation (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PMI (A

			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
41	dimensional_location 52	shape_dimension_representation 42	100 +0.035 0	linear distance	100.0	nominal value	NR2 3.0	0.0 0.035	NR2 1.3	(2) plane 200 202 (2) advanced_face 207 209 (2) shape_aspect 58 59

Figure 6.21: Excerpt from the results of the NIST STEP File Analyzer and Viewer showing part of the PMI representation data structure of the dimension added to the 3D CAD model

```
#39=DRAUGHTING_MODEL_ITEM_ASSOCIATION(
'PMI representation to presentation link','',#52,#70,#75);
#40=DRAUGHTING_MODEL_ITEM_ASSOCIATION('','',#38,#70,#75);
#41=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#52,#42);
#42=SHAPE_DIMENSION_REPRESENTATION('',(#49),#356);
#43=MEASURE_QUALIFICATION('','upper bound',#45,(#51));
#44=MEASURE_QUALIFICATION('','lower bound',#46,(#51));
#45=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#358);
#46=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#358);
#47=TOLERANCE_VALUE(#46,#45);
#48=PLUS_MINUS_TOLERANCE(#47,#52);
#49=(
LENGTH_MEASURE_WITH_UNIT()
MEASURE_REPRESENTATION_ITEM()
MEASURE_WITH_UNIT(LENGTH_MEASURE(100.),#358)
QUALIFIED_REPRESENTATION_ITEM((#50))
REPRESENTATION_ITEM('nominal value')
);
#50=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.0');
#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');
#52=DIMENSIONAL_LOCATION('linear distance','',#59,#58);
#53=ID_ATTRIBUTE('ShapeAspect.1',#58);
```

```
#354=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
#355=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
#356=(
GEOMETRIC_REPRESENTATION_CONTEXT(3)
GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#354))
GLOBAL_UNIT_ASSIGNED_CONTEXT((#358,#360,#361))
REPRESENTATION_CONTEXT('','3D')
);
#357=(
GEOMETRIC_REPRESENTATION_CONTEXT(3)
GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#355))
GLOBAL_UNIT_ASSIGNED_CONTEXT((#358,#360,#361))
REPRESENTATION_CONTEXT('','3D')
);
#358=(
LENGTH_UNIT()
NAMED_UNIT(*)
SI_UNIT(.MILLI.,.METRE.)
);
```

Figure 6.22: Excerpt from the STEP AP242 file generated by Inventor 2022, with the parts relevant to the PMI representation data structure of the dimension added to the 3D CAD model marked

The dimension can be found in the following format in this STEP file.

Table 6.12: Summary of all the definitions shown in Figure 6.22 that are used to define the dimension as representation PMI in the STEP file

Dimension component	STEP

100	`#49=(LENGTH_MEASURE_WITH_UNIT()`
	`MEASURE_REPRESENTATION_ITEM()`
	`MEASURE_WITH_UNIT(LENGTH_MEASURE(100.),#358)`
	`QUALIFIED_REPRESENTATION_ITEM((#50))`
	`REPRESENTATION_ITEM('nominal value')`
	`);`
	`#50=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.0');`
	`#358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.) );`

+0.035	`#44=MEASURE_QUALIFICATION('','lower bound',#46,(#51));`
	`#46=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#358);`
	`#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');`
	`#358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.) );`

-0.000	`#43=MEASURE_QUALIFICATION('','upper bound',#45,(#51));`
	`#45=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#358);`
	`#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');`
	`#358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.));`

Table 6.12 specifies the accuracy of the display of the dimension as follows

```
#50=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.0');
```

This indicates that the dimension value should be displayed with 3 digits before the decimal point and no digits after it.

The accuracy of the display of the upper and lower tolerance is determined by

```
#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3')
```

This indicates that the value of the upper and lower tolerance should be displayed with 1 digit before the decimal point and 3 digits after it.

The absolute accuracy of the CAD model in this STEP file is 0.01 mm as can be seen in Table 6.13.

Table 6.13: Model accuracy of 0.01 mm defined within the STEP file

Model accuracy	STEP

0.01	`#354=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,`
	`'DISTANCE_ACCURACY_VALUE',`
	`'Maximum model space distance between geometric entities at asserted c`
	`onnectivities');`

units	`#358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.));`

The accuracy specified for the creation of the model geometry is 0.01 mm. This means that the smallest distance between two points that the CAD system can still identify as two separate points is 0.01 mm. The display accuracy of the assigned tolerances is three decimal places and is therefore equal to one µm. This means that the display accuracy of the dimension is higher than the absolute accuracy of the CAD model. As this is an asymmetric tolerance, the manufacturer should set the nominal size 100 to the centre of the tolerance field in order to generate the correct toolpaths. This means that the value 100 should be $100 + \frac{0.035-0}{2} = 100.0175$. With a model accuracy of 0.01 this will be rounded to 100.02 which in this case is a valid value. However, when the imposed tolerance field becomes smaller than the model accuracy, this is no longer the case. On this basis, it can be concluded that the new accuracy introduced with STEP AP242 is only "cosmetic" and only applies to the display of the dimension in the MBD model and does not take into account any changes to be made by the manufacturing personnel.

To check how other CAD systems handle the new STEP AP242 parameters `QUALIFIED_REPRESENTATION_ITEM` and `VALUE_FORMAT_TYPE_QUALIFIER` together with the STEP parameter `UNCERTAINTY_MEASURE_WITH_UNIT`, the STEP file generated by Inventor 2022 is read into the other CAD systems CATIA v5, Siemens NX and PTC Creo Parametric.

Import in CATIA V5-6R2022 SP1

The following options for STEP import were used to import the STEP file generated by Inventor 2022 into CATIA v5 (see Figure 6.23).

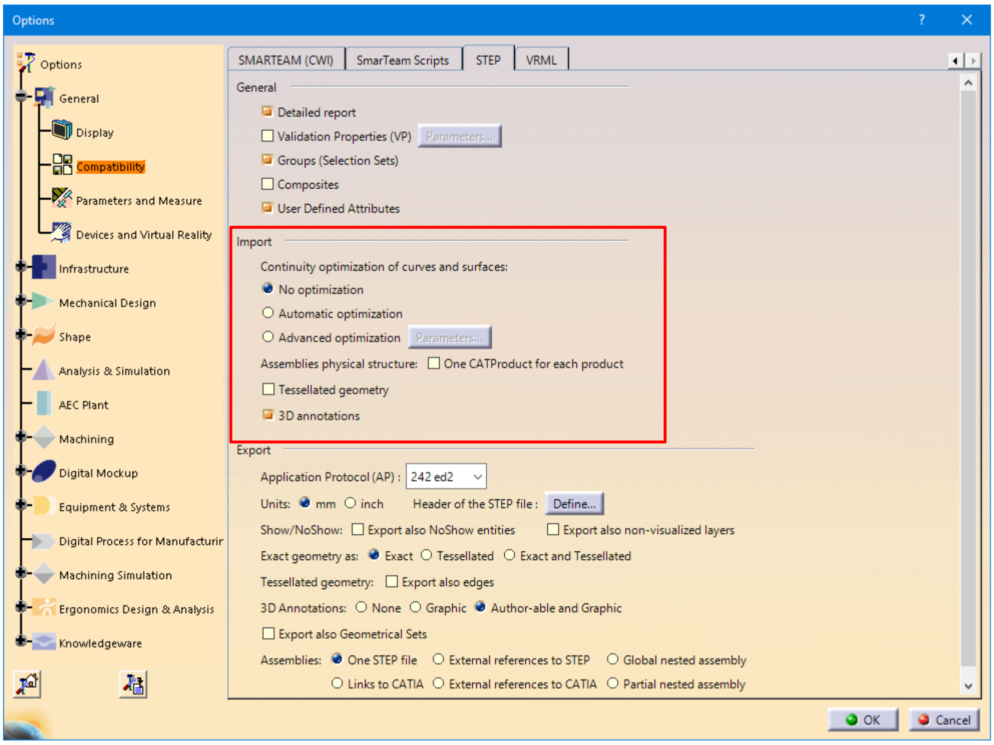


Figure 6.23: Settings for importing STEP files in CATIA v5

The "Scale" option used within this CATIA model is "Normal range"". The absolute accuracy of the CAD model is hereby set to 1×10^{-3} mm (see page A25). The feature tree in Figure 6.24 shows that the annotation is imported as "presentation PMI".

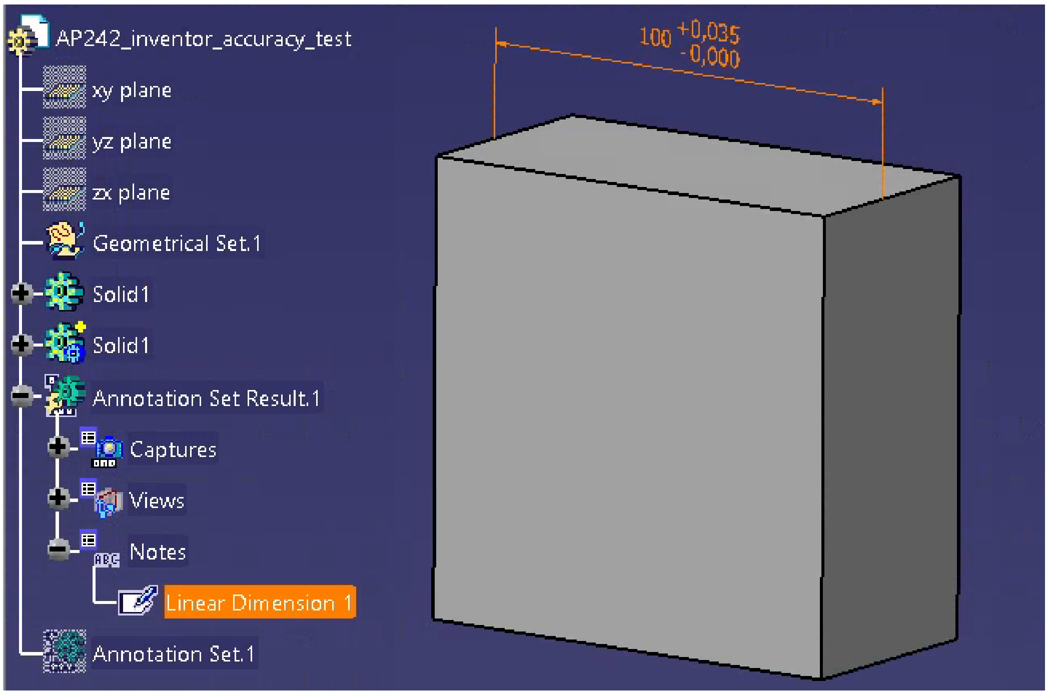


Figure 6.24: Result of importing the STEP AP242 file generated by Inventor 2022 into CATIA V5, showing that the dimension is converted to a note

To check whether the “representation PMI” present in the STEP file created by Inventor 2022 was actually lost when imported into CATIA v5, the imported model is re-exported to a STEP AP242 file. Analysis of the resulting STEP file with the “NIST STEP File Analyzer and Viewer” (Lipman 2017) shows that the annotation is saved as "presentation PMI" (see Figure 6.25). This can be determined by the text “tessellated_annotation_occurence”. This “presentation PMI” is specified as “general tolerance” (see Table 6.14). There is no “representation PMI” data structure in this STEP file.

The parameter `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8) in this re-exported STEP file defines the absolute accuracy used to construct the CAD geometry as 0.005 mm.

draughting_callout (1)

ID	name	contents
376	Linear Dimension 1	(1) tessellated_annotation_occurrence 375

Figure 6.25: Excerpt from the results of the NIST STEP File Analyzer and Viewer showing part of the PMI presentation data structure of the dimension present in the STEP AP242 file re-exported by CATIA V5

Table 6.14: Excerpt from the definition of the dimension in the STEP AP242 file created by Inventor and imported in and re-exported by CATIA V5

STEP

```
#374=PRESENTATION_STYLE_ASSIGNMENT((#373)) ;
#373=CURVE_STYLE(' ',#371,POSITIVE_LENGTH_MEASURE(0.129999995232),#372) ;
#376=DRAUGHTING_CALLOUT('Linear Dimension 1',(#375)) ;
#375=TESSELLATED_ANNOTATION_OCCURRENCE('Linear Dimension 1',(#374),#366) ;
#372=DRAUGHTING_PRE_DEFINED_COLOUR('white') ;
#371=DRAUGHTING_PRE_DEFINED_CURVE_FONT('continuous') ;
#366=(GEOMETRIC_REPRESENTATION_ITEM()REPOSITIONED_TESSELLATED_ITEM(#370)
REPRESENTATION_ITEM('general tolerance')TESSELLATED_GEOMETRIC_SET((#399,#409,#419,
#429,#439,#449,#459,#469,#479,#489,#499,#509,#519,#529,#539,#549,#559,#569))
TESSELLATED_ITEM()) ;
#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,
'distance_accuracy_value','CONFUSED CURVE UNCERTAINTY') ;
#12=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;
```

A full report of the tests with exporting to STEP AP242 by the all CAD systems can be found in the Appendix on page A35. Below is a summary of all these tests for the different CAD systems. The unit used is mm.

Table 6.15: Summary of the results of importing and exporting an AP242 file generated by Inventor 2022 with Inventor, CATIA, NX and Creo

STEP AP242 export from Inventor 2022									
Accuracy STEP file	0.01								
Display accuracy dimension	NR2 3.0								
Display accuracy tolerance	NR2 1.3								
Tolerance type	linear distance								
PMI Type: Presentation/Representation	Representation								
	Inventor	CATIA			NX	Creo			
		Large	Normal	Small		Automatic	External	Internal	Template(0.01)
Import successful (model geom)	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Import PMI Type (*Pres/Repres*)	P	P	P	P	R	R	R	R	R
Internal accuracy used	-	0.1	0.001	1e-05	-	0.015	0.01	0.015	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.005	0.015	0.01	0.015	0.01
Export display accuracy dimension	-	-	-	-	NR2 3.1	-	-	-	-
Export display accuracy tolerance	-	-	-	-	NR2 1.2	-	-	-	-
Tolerance type preserved	-	No	No	No	No	No	No	No	No
Export PMI Type: *Pres/Repres*	-	P	P	P	R	P	P	P	P

Table 6.16: Summary of the results of importing and exporting an AP242 file generated by CATIA V5 with Inventor, CATIA, NX and Creo

STEP AP242 export from CATIA V5	
Accuracy STEP file	0.005
Display accuracy dimension	NR2S 3.3
Display accuracy tolerance	NR2S 0.3
Tolerance type	linear distance
PMI Type: Presentation/Representation	Representation

	Inventor	CATIA			NX	Creo			
		Large	Normal	Small		Automatic	External	Internal	Template(0.01)
Import successful (model geom)	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Import PMI Type (Pres/Repres)	P	R	R	R	R	R	R	R	R
Internal accuracy used	-	0.1	0.001	1e-05	-	0.015	0.005	0.015	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.005	0.015	0.005	0.015	0.01
Export display accuracy dimension	-	NR2S 3.3	NR2S 3.3	NR2S 3.3	NR2 3.3	-	-	-	-
Export display accuracy tolerance	-	NR2S 0.3	NR2S 0.3	NR2S 0.3	NR2 1.2	-	-	-	-
Tolerance type preserved	-	Yes	Yes	Yes	No	No	No	No	No
Export PMI Type: Pres/Repres	-	R	R	R	R	P	P	P	P

Table 6.17: Summary of the results of importing and exporting an AP242 file generated by Siemens NX Version 2019 with Inventor, CATIA, NX and Creo

STEP AP242 export from Siemens NX Version 2019	
Accuracy STEP file	0.005
Display accuracy dimension	NR2 3.1
Display accuracy tolerance	NR2 1.3
Tolerance type	linear distance
PMI Type: Presentation/Representation	Representation

	Inventor	CATIA			NX	Creo			
		Large	Normal	Small		Automatic	External	Internal	Template 0.01
Import successful (model geom)	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Import PMI Type (Pres/Repres)	P	P	P	P	R	R[1]	R[1]	R[1]	R[1]
Internal accuracy used	-	0.1	0.001	1e-05	-	0.015	0.005	0.015	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.005	0.015	0.005	0.015	0.01
Export display accuracy dimension	-	-	-	-	NR2 3.1	-	-	-	-
Export display accuracy tolerance	-	-	-	-	NR2 1.2	-	-	-	-
Tolerance type preserved	-	Yes	Yes	Yes	No	No	No	No	No
Export PMI Type: Pres/Repres	-	P	P	P	R	P	P	P	P

[1] The display (dimension and tolerances) is not human readable in the 3D model.

Table 6.18: Summary of the results of importing and exporting an AP242 file generated by PTC Creo 8.0.4.0 with Inventor, CATIA, NX and Creo

STEP AP242 export from PTC Creo 8.0.4.0									
Accuracy STEP file	0.005								
Display accuracy dimension	NR2 3.1								
Display accuracy tolerance	NR2 1.3								
Tolerance type	linear distance								
PMI Type: Presentation/Representation	Representation								
	Inventor	CATIA			NX	Creo			
		Large	Normal	Small		Automatic	External	Internal	Template 0.01
Import successful (model geom)	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes	Yes
Import PMI Type (Pres/Repres)	P	P	R	R	R	R	R	R	R
Internal accuracy used	-	0.1	0.001	1e-05	-	0.015	0.01	0.015	0.01
Export accuracy	0.01	0.5	0.005	5e-5	0.005	0.015	0.01	0.015	0.01
Export display accuracy dimension	-	NR2S 3.3	NR2S 3.3	NR2S 3.3	NR2 3.3	-	-	-	-
Export display accuracy tolerance	-	NR2S 0.3	NR2S 0.3	NR2S 0.3	NR2 1.2	-	-	-	-
Tolerance type preserved	-	Yes	Yes	Yes	No	No	No	No	No
Export PMI Type: *Pres/Repres*	P	R[1]	R[1]	R[1]	R	P	P	P	P

6.2.3 Discussion

The previous sections examined how the Inventor, CATIA, Siemens NX and PTC Creo CAD systems deal with absolute accuracy as defined in a STEP AP242 file. Absolute accuracy is used to describe the topology of a 3D CAM model. As well as STEP AP242, there are other STEP formats such as STEP AP203 and STEP AP214. Only STEP AP242 was considered in the study for two reasons:

1. STEP AP242 is intended for MBD models
2. Absolute accuracy is defined in the same way in all STEP formats.

The latter makes sense because STEP can be considered an encapsulating format where one new variant builds on the previous one (Figure 6.26). A STEP file described in the EXPRESS format (ISO 10303-21) can be compared to a file in an XML-based format, where applications can extract from the file what they know and skip what they do not know. This makes it possible to extend a format without adversely affecting the operation of existing applications. The similarity in structure between the EXPRESS format originally used for STEP files and an XML-based format is also the reason why later versions of STEP can be described in both EXPRESS and XML formats.

The results show that the different CAD systems in this test treat absolute accuracy for describing geometry in IGES, STEP AP203, STEP AP214 and STEP AP242 in a similar manner. The accuracy, specified in these neutral exchange formats, is completely ignored. CAD systems internally apply their own accuracy determined by the CAD kernel used. For export to STEP, a fixed value is used that is independent of the internally used accuracy. The only exception is PTC Creo. This CAD system creates an IGES and a STEP file with the same accuracy used internally.

[1] An additional presentation PMI is created in another plane.

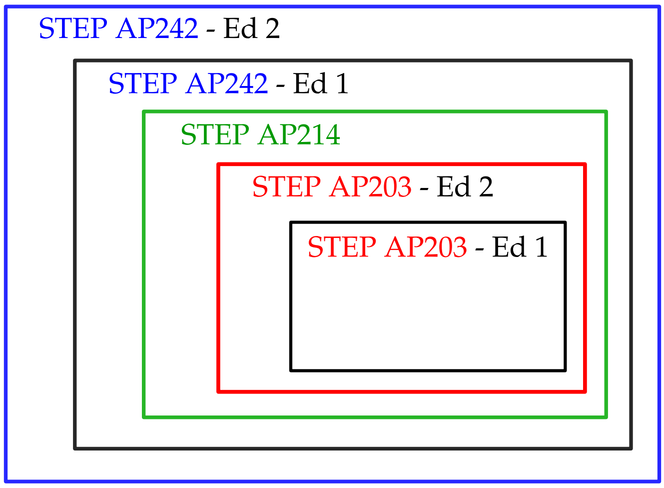


Figure 6.26: Every STEP variant builds on the previous one

The fact that the accuracy of the model as specified in a STEP file is completely ignored in both the export and, more importantly, the import, is likely to have a negative impact on the time taken to import a STEP file. Ignoring the STEP file accuracy, together with the fact that most CAD companies do not develop their own STEP interface but license it from third parties, is possibly the reason why importing a large STEP file can sometimes take hours. This complicates the use of STEP to exchange MBD models between stakeholders and encourages the use of the same CAD systems by all stakeholders, with the risk of potential vendor lock-in.

With the introduction of STEP AP242, a new type of accuracy was also introduced. This new accuracy is not used to describe the topology of the 3D CAD model, but to specify the accuracy with which dimensions and their associated tolerances should be displayed. The results show that, like the accuracy used to describe the topology of a 3D CAD model, this new type of accuracy is also completely ignored by the various CAD systems. This is especially evident when an imported STEP file is re-exported. In that case, sometimes even the dimension type changes. Representation PMI becomes presentation PMI and in some cases additional changes occur, such as a linear dimension becoming a diameter or a radial dimension (Siemens NX) or a weld symbol (PTC Creo).

As discussed with the model accuracy, a possible explanation for this behaviour can be found in the way the STEP interface is implemented. Implemented here refers to two things. The first is how the STEP interface itself is implemented. Is this the CAD system manufacturer's own implementation or does it use dedicated third-party libraries. This may affect the extent to which the STEP standard is fully implemented and how up-to-date this implementation is with the latest version of the STEP standard. The second is the mapping of the data structure used to describe a 3D annotation in the STEP file with the data structure used for this purpose in the CAD system itself. Regarding the use of third-party libraries, examination of the header of the STEP files generated by the various CAD systems indicates that at least Inventor and Siemens NX use a third-party library for their STEP-interface (Figure 6.27 and Figure 6.28). The header of the STEP files shows that they were generated using software from the company STEP Tools, Inc. Two components of this software are the ROSE and Rose Math libraries (Loffredo 2017). This is confirmed by the presence of two dlls in the installation folder of Inventor and Siemens NX, *rose_x64_vc15.dll* and *rosemath_x64_vc15.dll*.

The model accuracy used to describe the topology of the CAD model and the accuracy with which nominal dimensions and their associated tolerances must be displayed are completely independent of each other. This can result in the model accuracy being lower than the display accuracy of tolerances. If the tolerances are asymmetric, this can potentially cause problems when adjusting the model to change the nominal size to the centre of the specified tolerance range.

```
ISO-10303-21;
HEADER;
/* Generated by software containing ST-Developer
 * from STEP Tools, Inc. (www.steptools.com)
 */

FILE_DESCRIPTION(
/* description */ ('',
'CAx-IF Rec.Pracs.---Representation and Presentation of Product Manufa
cturing Information (PMI)---4.0---2014-10-13'),
/* implementation_level */ '2;1');
```

Figure 6.27: Part of header of STEP file generated by Inventor

```
ISO-10303-21;
HEADER;
/* Generated by software containing ST-Developer
 * from STEP Tools, Inc. (www.steptools.com)
 */
/* OPTION: using custom renumber hook */

FILE_DESCRIPTION(
/* description */ (
'CAx-IF Rec.Pracs.---Representation and Presentation of Product Manufa
cturing Information (PMI)---4.0---2014-10-13',
'CAx-IF Rec.Pracs.---3D Tessellated Geometry---0.4---2014-09-14','2;1',
'CAx-IF Rec.Pracs.---User Defined Attributes---1.5---2016-08-15'),
/* implementation_level */ '2;1');
```

Figure 6.28: Part of header of STEP file generated by Siemens NX

Regarding data structures, the reason for the change in dimension type is most probably a discrepancy between the data structure used in the CAD system itself and the data structure in the STEP file. This is usually caused by the development history of a CAD system. In the automotive and aerospace industry, it is a requirement that technical data be available and readable for a very long period of time. Chambolle 2013 mentions a required lifecycle of product definition data in the automotive industry of more than 30 years. To help achieve this goal, the LOTAR standard was created. LOTAR is an acronym and stands for **LO**ng-**T**erm **A**rchiving and **R**etrieval. LOTAR 2023 defines this as the specification of how digital product and engineering data, such as 3D CAD/CAM and PDM data, can be archived and retrieved while maintaining engineering intent throughout the product lifecycle. The requirements are laid down in Europe in the CSN EN 9300-003 standard and in the US in the NAS 9300-003 standard. For a CAD system, the implication is that the system must be able to read the original CAD data correctly over a period of more than 30 years. The company using the CAD system decides what the definition of "original CAD data" is. This could be the native CAD files or the designs in a neutral exchange format or a combination of both. When the manufacturer of a CAD system needs to add new functionality to the 3D annotations to be consistent with the current MBD philosophy, the manufacturer must choose between different ways in which this can be implemented taking into account the requirements imposed by the LOTAR standard. Consequently, new functions are often developed alongside or on top of existing 3D annotation functions. It can be assumed that for reasons of compatibility (LOTAR) and development cost, often no changes are made to the existing data structures for new MBD related functions. Instead the choice is made to keep the data structure of the functions already present unaltered and to create new data structures for new functions. This is reflected in the way annotations are exported to the STEP file. In fact, how this is done depends on how the annotations are created in the native CAD model. Specifically, over time, a CAD system will have multiple functions for creating 3D annotations, all of which have their own data structure and are also exported to STEP in a different way. For example, there are three different functions in PTC Creo to create a linear dimension as a 3D annotation. In turn, these three functions can be divided into annotation elements and stand-alone annotations (Nascimento 2017). The first function creates a “driving dimension”. This is an annotation element because it does not stand alone, but belongs to a feature used to create the CAD model. In the native model, this is created as representation PMI. The ability to specify specific semantic references was not available from the start and only added later on. To STEP, it is exported as presentation PMI. The second function creates a “driven dimension”. When created directly, this is a stand-alone annotation because it does not belong to any other feature. In the native model, this is created as representation PMI. To STEP, it is exported as representation PMI. The third function creates an “annotation feature”. This function can contain driven dimensions as annotation elements. It has extensive capabilities for assigning semantic references. Annotation features were added in Creo to facilitate the implementation of MBD. In the native model, this is created as representation PMI. To STEP, it is exported as representation PMI. To the end user, all these ways are presented as equivalent options. This is not the case and consequently creates a lot of difficulties and confusion. These kinds of issues complicate the use of a neutral exchange format like STEP for exchanging MBD models between different stakeholders. This is especially the case when the models have to be modified by stakeholders. To avoid such problems, it is increasingly recommended that all stakeholders use the same CAD system (Made-in Europe 2022). The native CAD model is then the exchange format of choice instead of a neutral exchange format such as STEP. This solution has two major drawbacks. The first is the danger of vendor lock-in. The second is the big cost for small suppliers working for multiple customers who each have their own CAD system. In that case, the supplier is required to have each customer’s CAD system. Regarding the transfer of 3D annotations, the conclusion of this study is that, in order to make it as clear as possible for the user and preserve as much data as possible, two requirements need to be met. The first requirement is to simplify the user interface so that only MBD-compliant methods are available where all data is retained when the model is exported to an MBD-compliant neutral exchange format. The second requirement is to ensure that the 3D annotations in a STEP file are created in the CAD system such that they are retained when re-exporting to STEP. All existing functions that don’t adhere to these two requirements should be depreciated.

6.3 Model geometry

6.3.1 Introduction

In the MBD methodology, the 3D model is the authority. Consequently, all dimensions that have to meet the general tolerance (e.g. ISO 2768-m) are not explicitly defined via 3D annotations. They are determined solely by the geometry of the CAD model. Since not all stakeholders involved in the design and manufacturing of a product use the same CAD system, the CAD models of this product are exchanged via neutral formats such as STEP. This means that these STEP files will also be used to generate CNC toolpaths. Manufacturers must therefore have complete confidence that the geometry is transferred with sufficient accuracy via STEP. This section examines whether this is the case. Two criteria are used to verify this. The first is the strictest tolerance for linear dimensions of the ISO 2768 standard. This is ± 0.05 mm (Table 6.19). The second is a tolerance of ± 1 µm. This value has been chosen because many CNC operations need to be executed within a tolerance range of a few µm to a few hundredths of a mm. The CAD model is created using nominal values to which a tolerance is assigned. When an asymmetric tolerance is applied, the nominal value needs to be changed to the middle of the tolerance field to be able to generate the optimal toolpath. This can only be done when the CAD model is transferred with an adequate accuracy. An adequate accuracy is an accuracy better than the smallest applied tolerance.

Table 6.19: ISO 2768-f (ISO 1989)

General tolerances for linear dimensions without individual tolerance indications						
Tolerance class		*Permissible deviations for basic size range (mm)*				
Designation	*Description*	0.5 up to 3	over 3 up to 6	over 6 up to 30	...	over 1000 up to 2000
f	*fine*	±0.05	±0.05	±0.1	...	±0.5

This PhD research found that three things can have an impact on the resulting accuracy when exchanging 3D models using STEP:

1. The nature of the exchanged geometry. This geometry can be analytical or spline-based or a combination of both.
2. The model accuracy of the original CAD model and the model accuracy applied within the receiving system.
3. The model accuracy used within the STEP file.

6.3.2 Effect of the nature of the exchanged geometry

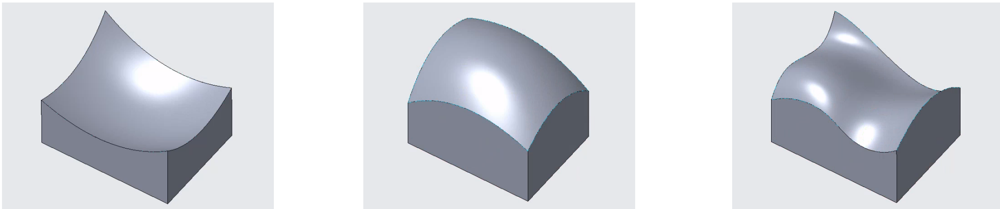
Bijnens et al. 2018 examined the effect of the nature of the exchanged geometry. Using the default settings of the CAD systems, they came to the following conclusions:

- There are large differences between CAD systems studied.
- Analytical shapes are transferred more accurately than spline-based forms
  - The largest deviation occurring in analytical model transfer is 0.155 µm.
  - The error values occurring in spline-based model transfer range from 3.183 µm to 21.424 µm.

6.3.3 Effect of the model accuracy of the original CAD model with a spline-based form

Test procedure

Three CAD models (Figure 6.29) with a spline-based form were created with a 200 mm × 150 mm rectangle as the ground plane and an average height of 100 mm. They were created with two different model accuracies, namely 0.01 mm and 0.001 mm. The first CAD model has a concave shape. The second has a convex shape. The third has a mixed concave and convex shape. The three models were created in PTC Creo, each with an absolute model accuracy of 0.01 mm and of 0.001 mm. PTC Creo was chosen because the module for determining the deviation of measurement points from a CAD model was available for it. In PTC Creo, a beam shape was created on which a cutting operation was performed using a spline-based surface as a patch surface. Two tests were carried out. In a first test, these CAD models were exported to STEP AP242 (Figure 6.30) with the default coordinate system as reference and imported into Inventor 2022, CATIA v5, Siemens NX Version 2019 and Creo 9.0.3.0. The original spline surface used as a patch surface was filtered out of the STEP file. To do so, the "Annotations" option was unchecked, so the "Quilts" an "Hidden entities" options could be unchecked. This was done to ensure that the same surface was always selected to measure the deviation. An STL model was then created using the coordinate system of the STEP file as reference. A chord height of 0.05 mm was set. The value of the chord height determines the distance between the vertices of the STL triangles. The vertices themselves lie on the surface approximated by the triangles. The unique vertices are filtered from the STL file and stored in a separate file. The resulting point cloud is merged with the original CAD modal by aligning the coordinate system of the point cloud with the coordinate system of the original CAD model.



(a) Concave shape (b) Convex shape (c) Mixed concave/convex

Figure 6.29: Three CAD models created in PTC Creo Parametric

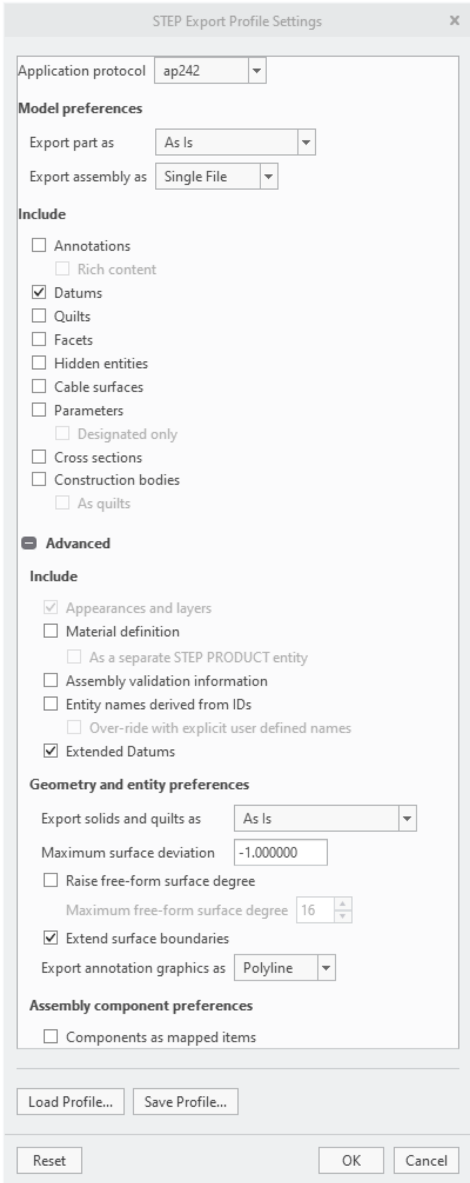


Figure 6.30: Creo export settings used for export to STEP AP242

The deviation of the point cloud from the original model is determined using the Assembly Verify Module. The deviation can be regarded as a measure of the accuracy with which the CAD model has been exchanged from Creo to another CAD package. In a second test, the imported STEP file was exported back to STEP AP242. This was done to simulate the situation where the receiver needs to make changes to the STEP file and send it back to the customer. This file was then imported into PTC Creo using “Model Accuracy: Automatic”. After this, an STL was generated and the deviation of the vertices with the original file was measured.

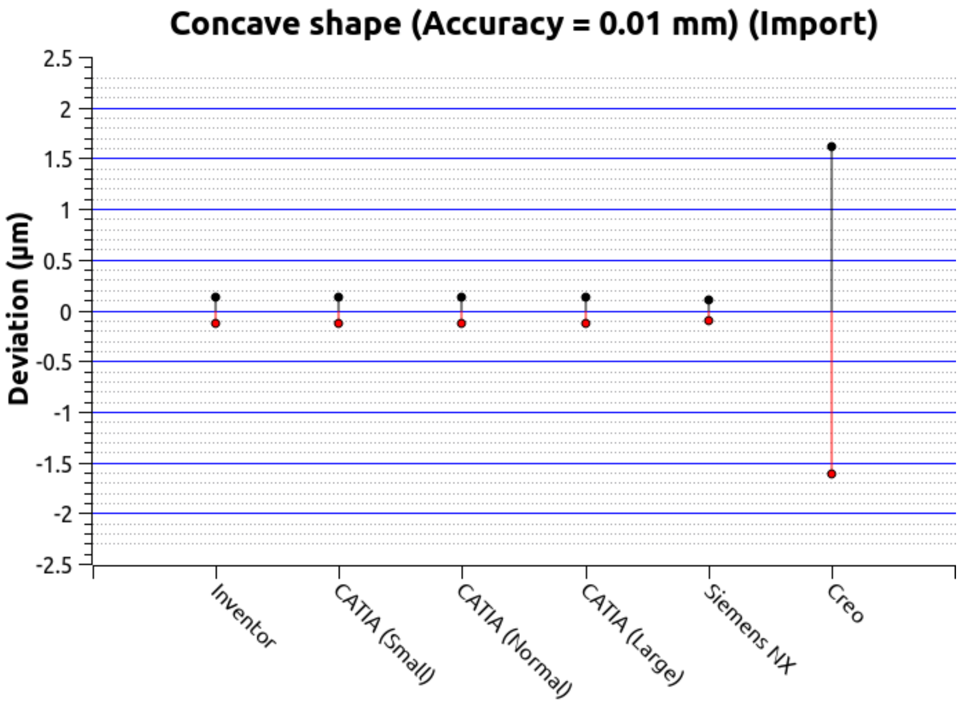


Figure 6.31: Deviation of the imported STEP model from the original model

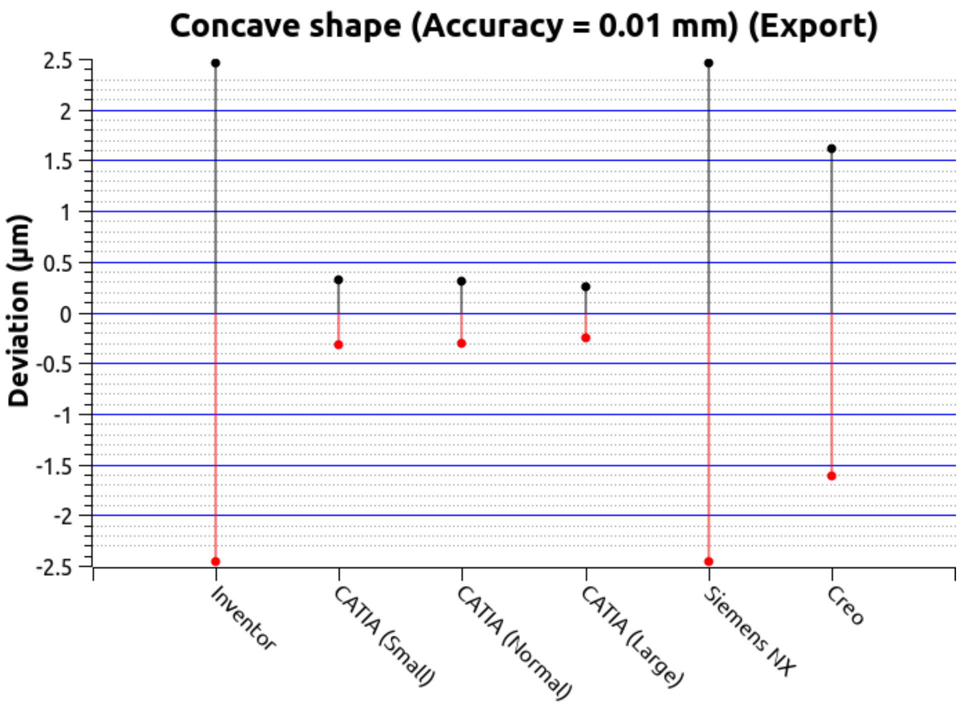


Figure 6.32: Deviation of the re-exported STEP model from the original model

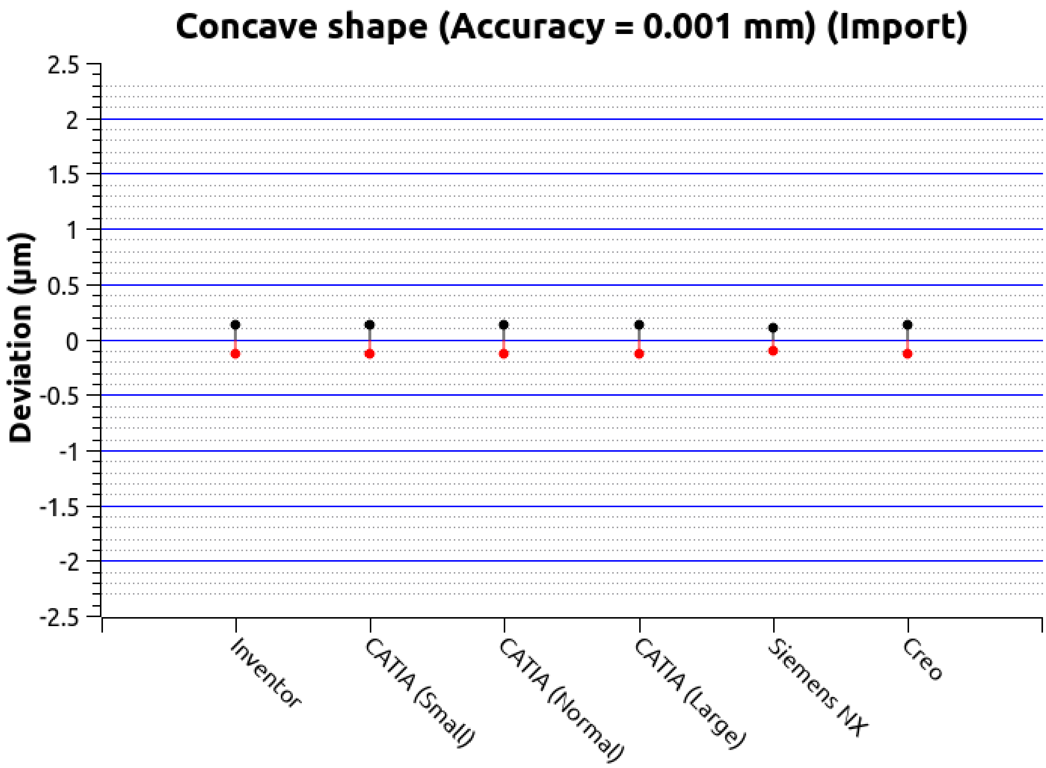


Figure 6.33: Deviation of the imported STEP model from the original model

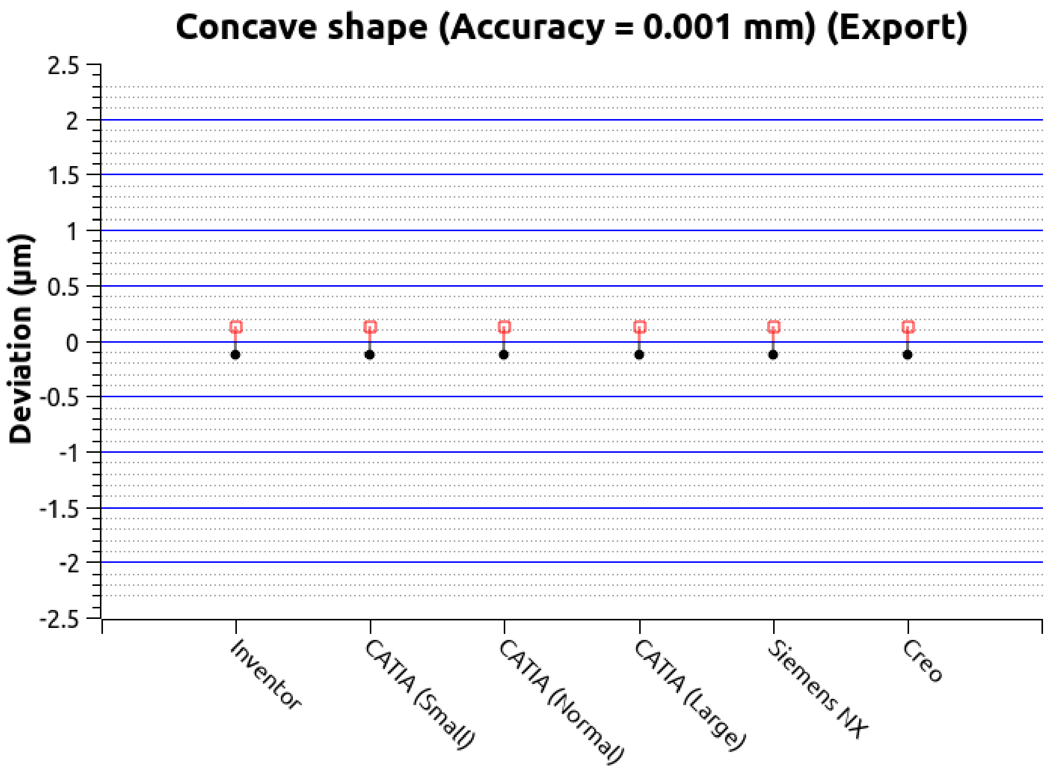


Figure 6.34: Deviation of the re-exported STEP model from the original model

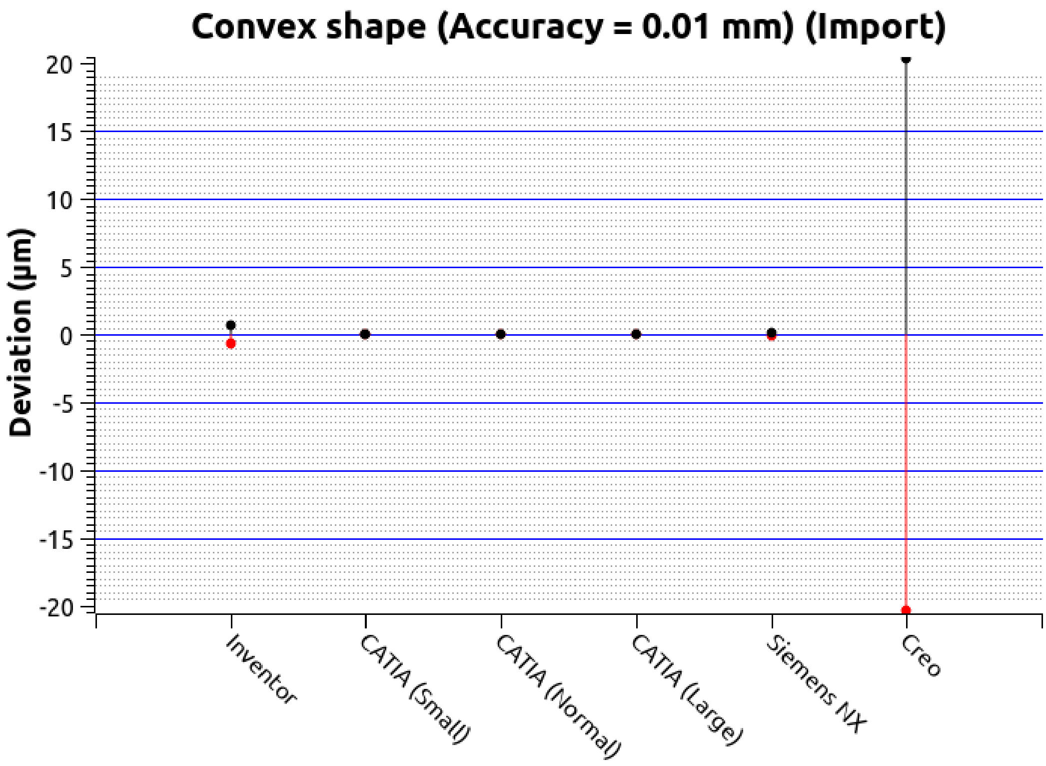


Figure 6.35: Deviation of the imported STEP model from the original model

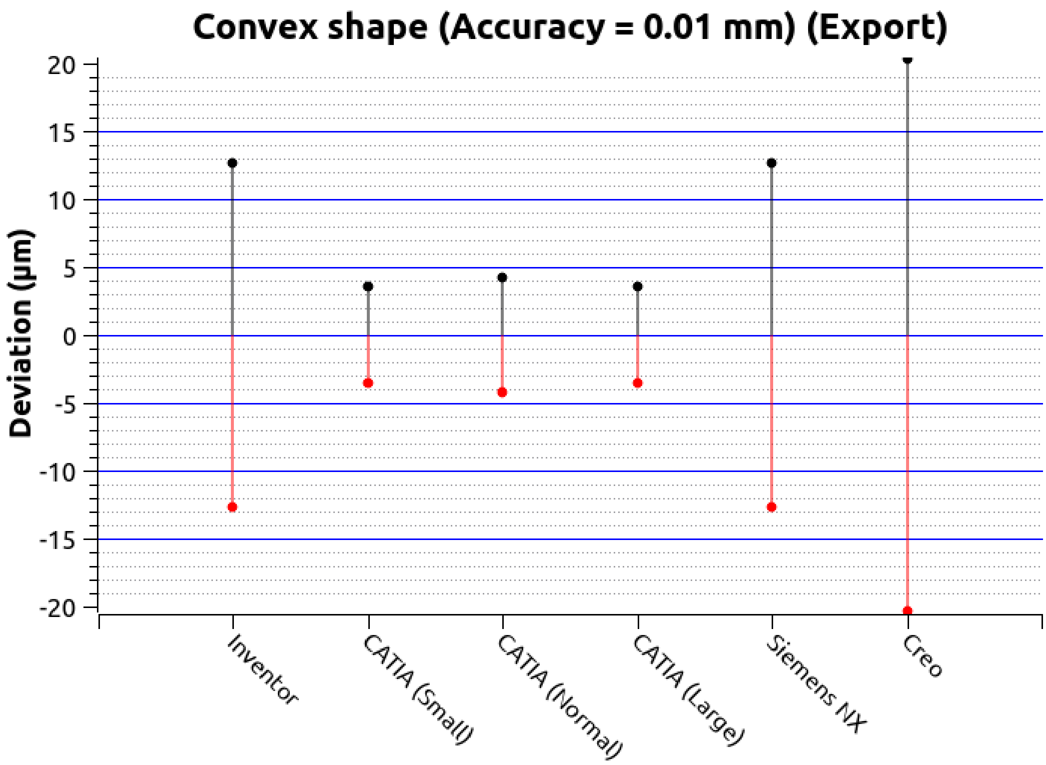


Figure 6.36: Deviation of the re-exported STEP model from the original model

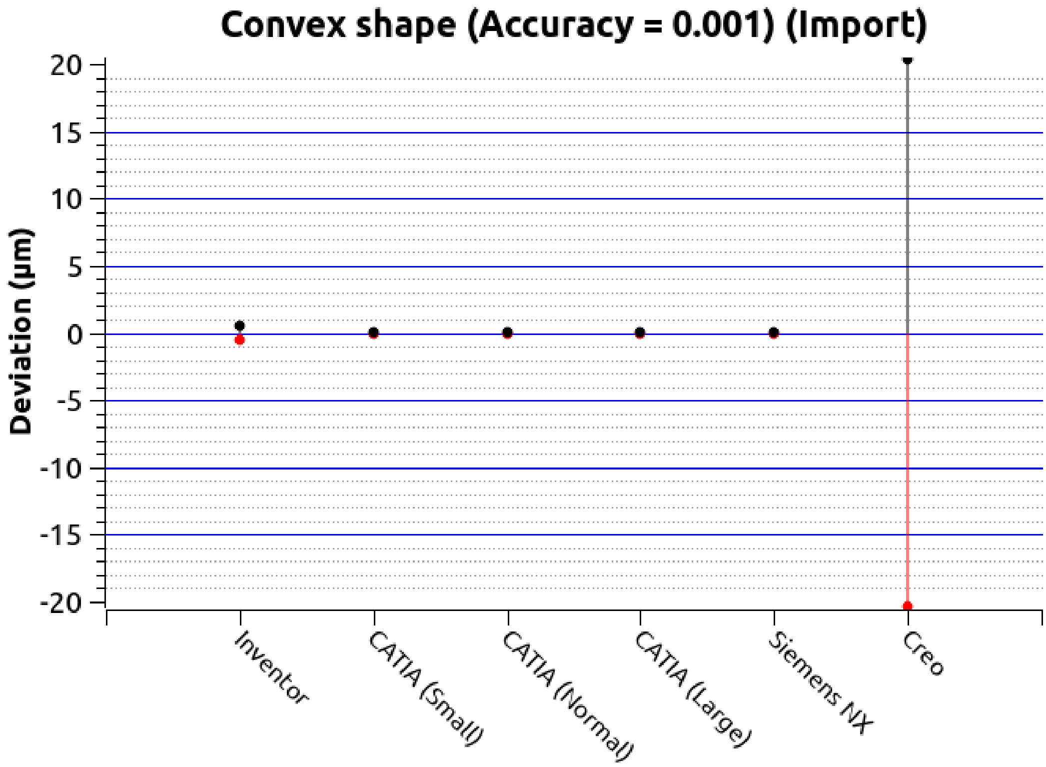


Figure 6.37: Deviation of the imported STEP model from the original model

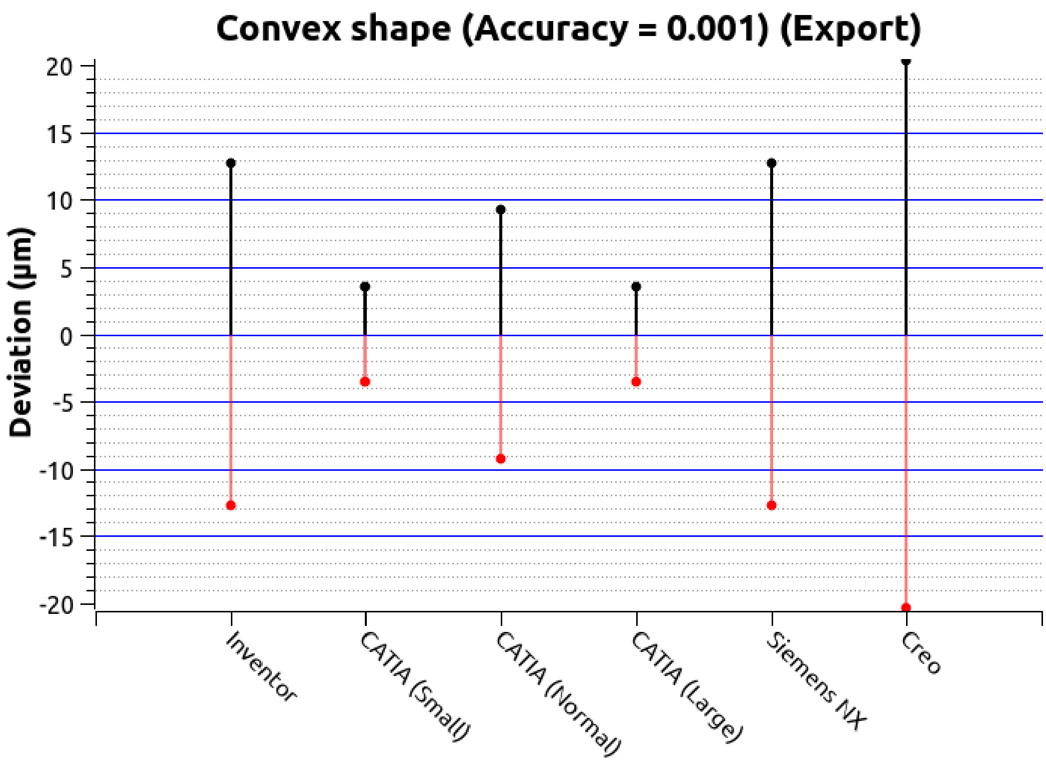


Figure 6.38: Deviation of the re-exported STEP model from the original model

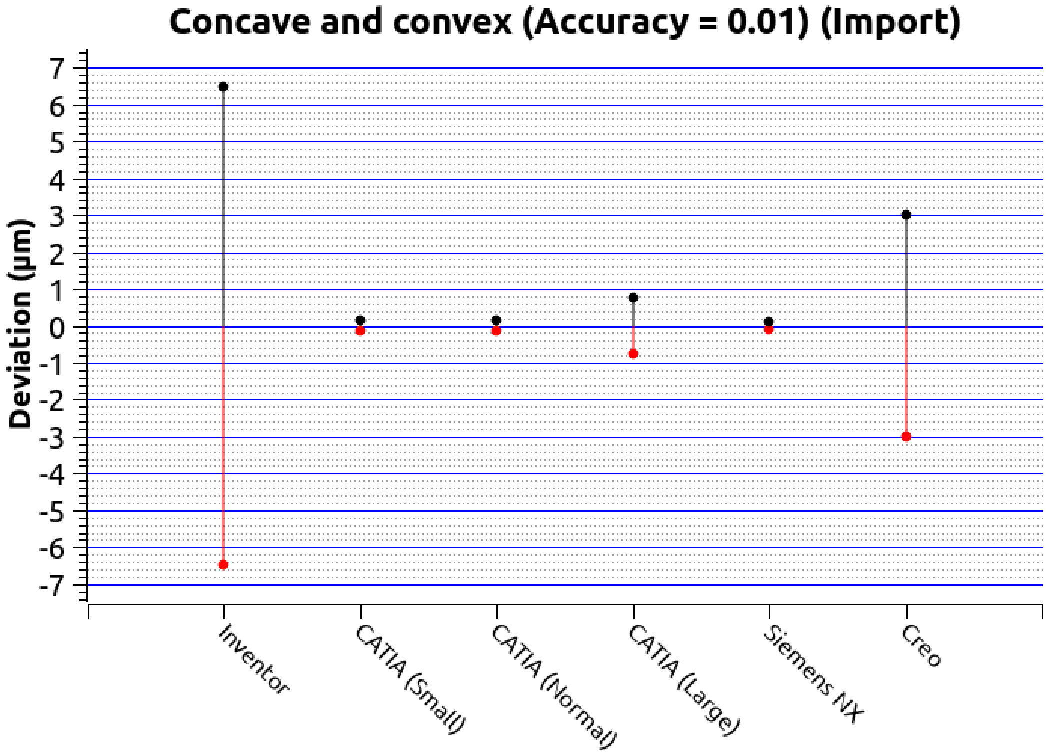


Figure 6.39: Deviation of the imported STEP model from the original model

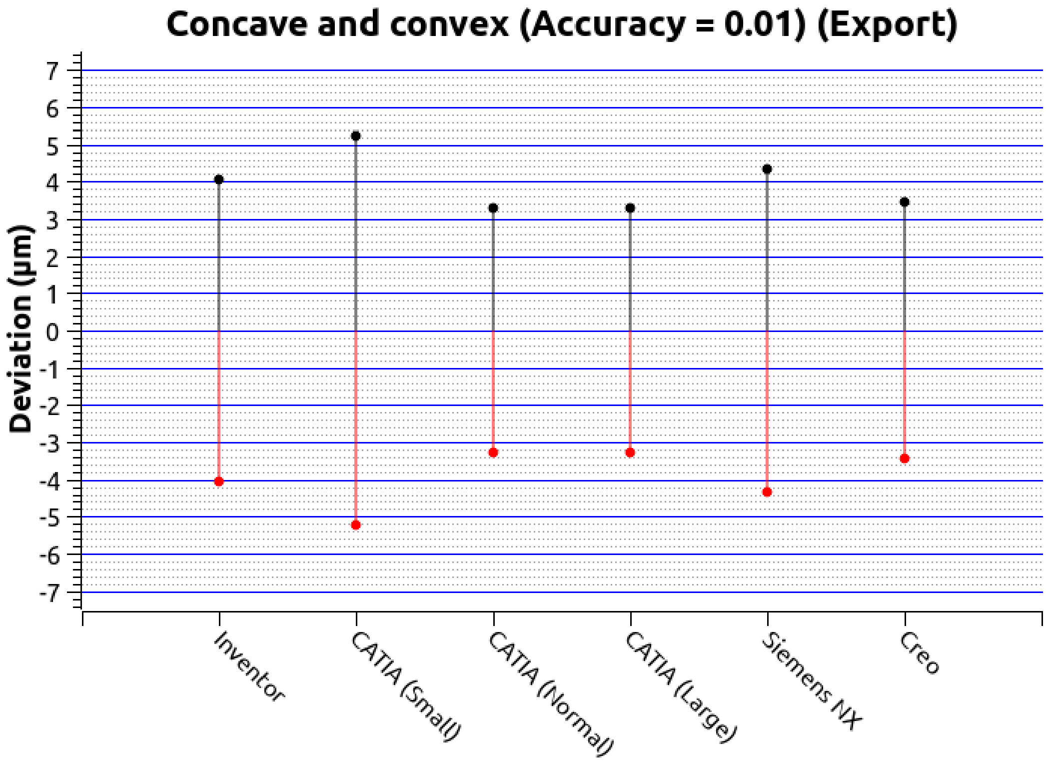


Figure 6.40: Deviation of the re-exported STEP model from the original model

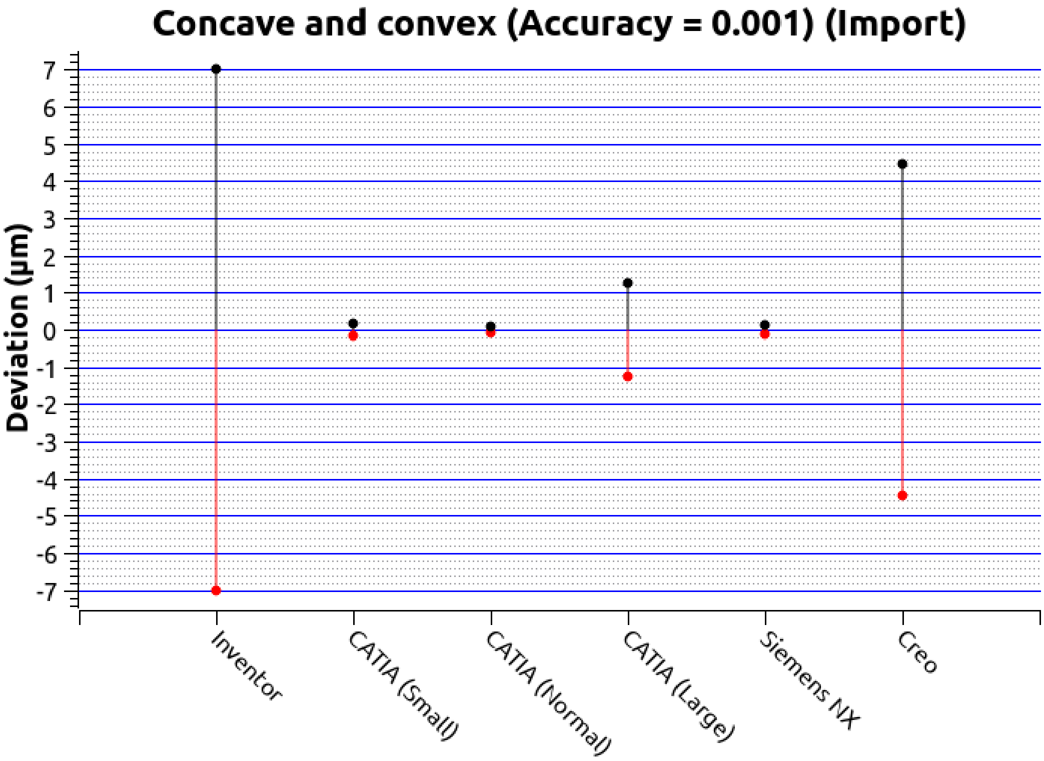


Figure 6.41: Deviation of the imported STEP model from the original model

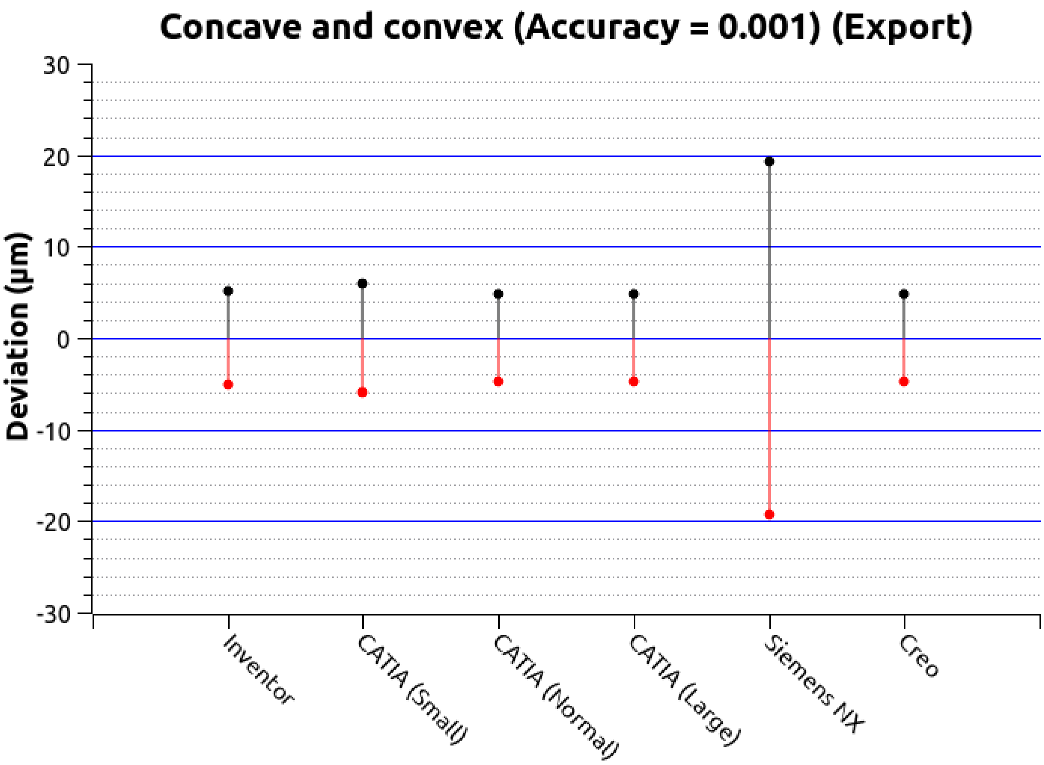


Figure 6.42: Deviation of the re-exported STEP model from the original model

When the various stakeholders are working together on the realisation of a project, it is often not enough to be able to just transfer CAD models in one direction. Changes made by one stakeholder to the imported STEP model often need to be communicated to another stakeholder via STEP. In the case of curved surfaces, these changes can affect the shape of the part of the CAD model that remains unchanged. This is illustrated in the following example.

To check how an effective modification of a STEP file affects the geometry of the imported model, the following test was set up. The model with the largest deviations in the previous tests, is the model with the convex surface. This model was created in PTC Creo and exported to a STEP AP242 file. This STEP file, which was also used in the above test, was modified in the receiving CAD system. This modification consists of making a hole with a diameter of 70 mm in the centre of the model (Figure 6.43). This hole runs through the entire model. For three CAD systems (Inventor, Siemens NX and PTC Creo, all with their own CAD kernel), the largest deviation that occurred in the original test with a convex surface is within the circle of this hole (Figure 6.44). The size 70 mm was chosen to have a change that would be large enough to have a significant impact. The procedure was then followed as described in the previous tests. The comparison model was the original model without a hole. The results are shown in Figure 6.45, Figure 6.46, Figure 6.47 and Figure 6.48.

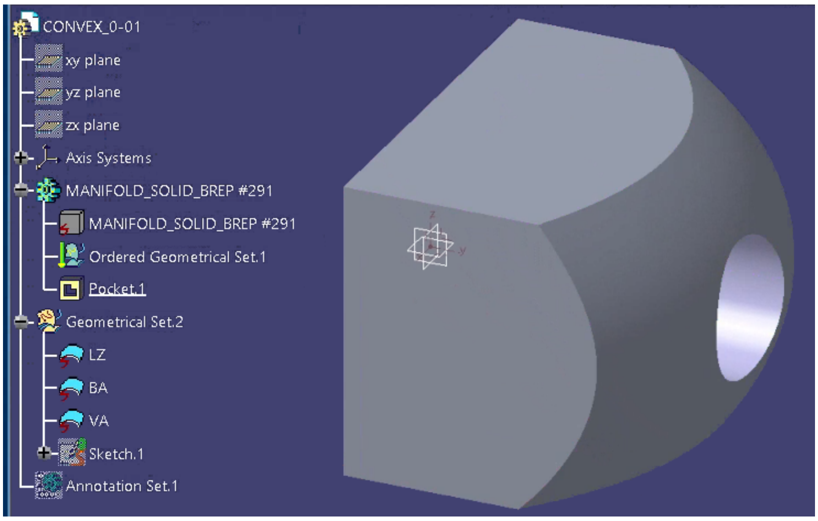


Figure 6.43: Hole created in the centre of the model imported in CATIA V5

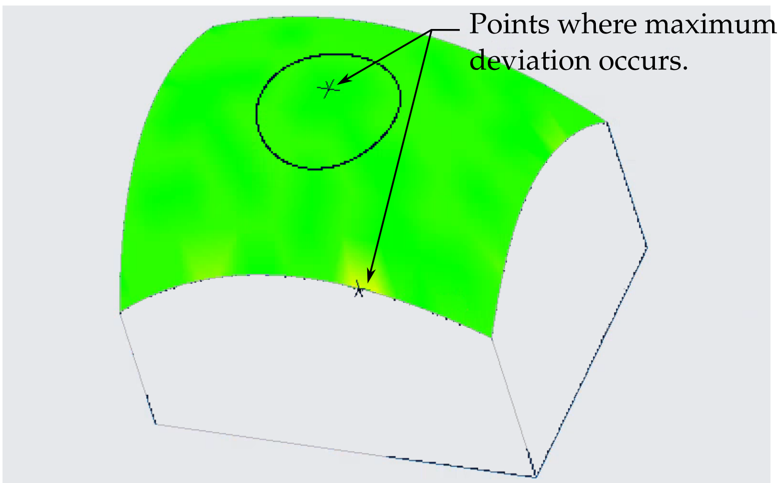


Figure 6.44: Maximum deviation occurs within circle of hole in a re-exported STEP AP242 file by Siemens NX

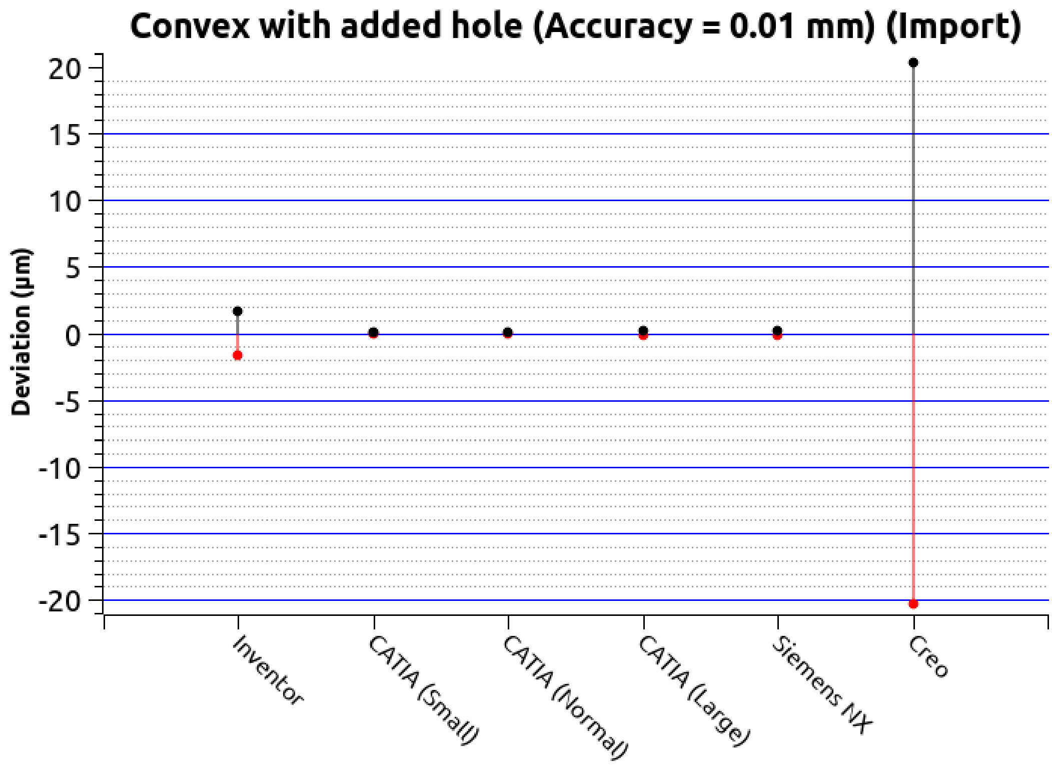


Figure 6.45: Deviation of the imported STEP model from the original model

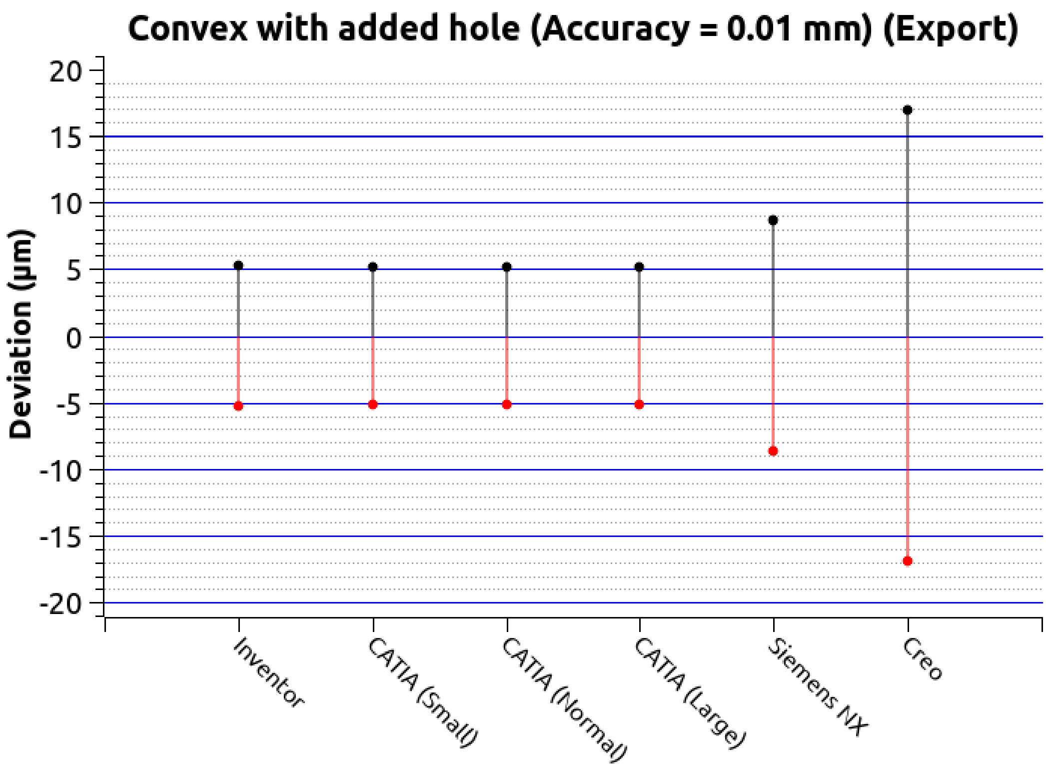


Figure 6.46: Deviation of the re-exported STEP model from the original model

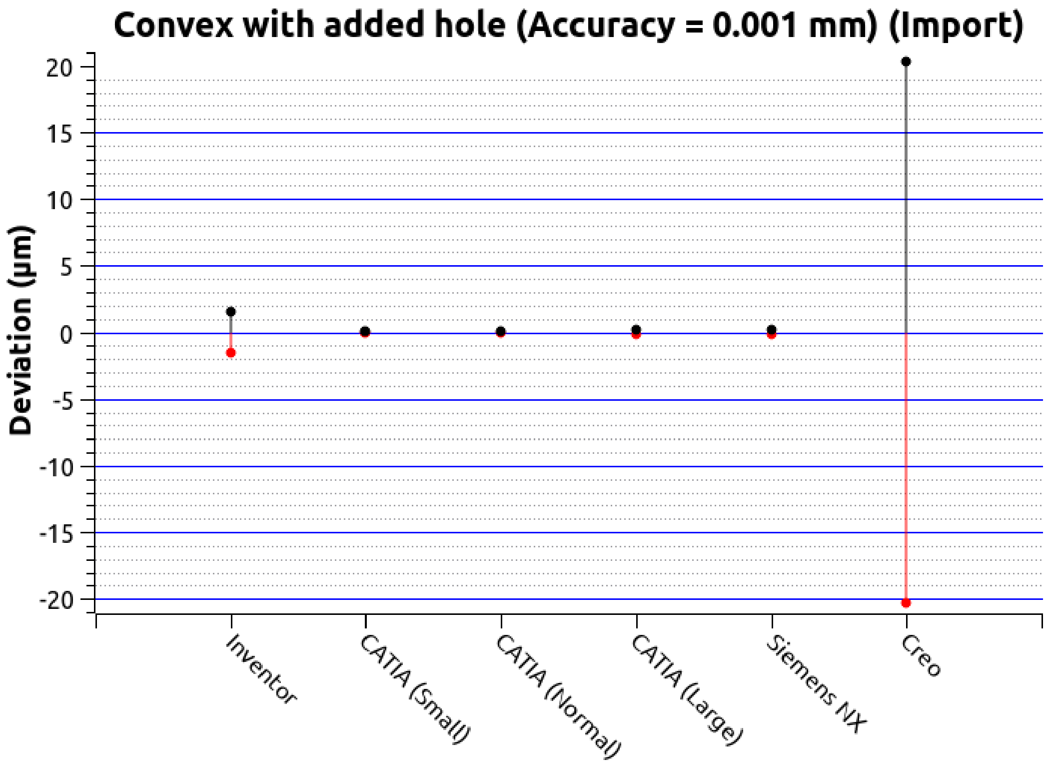


Figure 6.47: Deviation of the imported STEP model from the original model

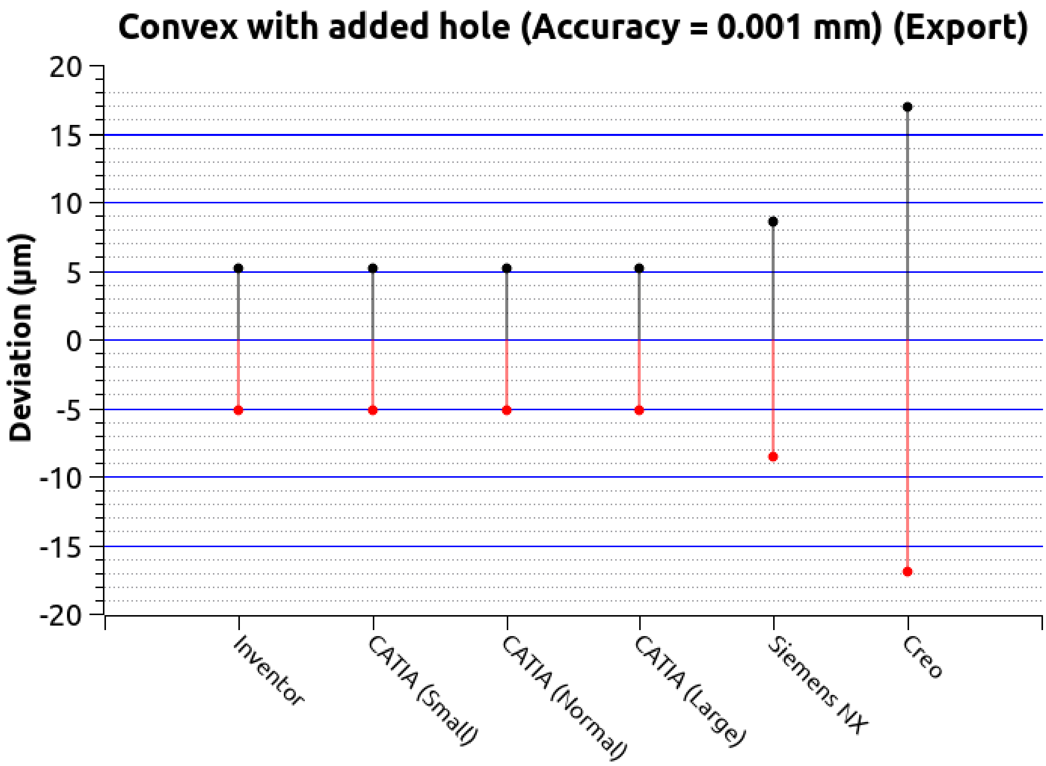


Figure 6.48: Deviation of the re-exported STEP model from the original model

6.3.4 Discussion

In MBD, the 3D model is "the authority". There are several factors that greatly affect the accuracy with which a CAD model can be transferred from one CAD system to another. These are

- The nature of the geometry. Analytical shapes are transferred much more accurately than spline-based shapes.
- The shape of the geometry. The tests examined three different spline-based shapes: concave, convex and a mix of both. When convex shapes appear in the model, they are transferred least accurately.
- The accuracy with which the STEP file is generated.
- The configuration (tweaking) of the receiving CAD system. An absolute model accuracy of 0.001 mm seems to lead to the best result in most cases.
- Re-exporting to STEP. When this is done, there is a large deterioration in accuracy.

The smallest tolerance field in DIN ISO 2768 is ± 0.05 mm. The largest deviation found in the tests is ± 0.02 mm. The deviations resulting from conversion to the STEP format are therefore well below the tolerances specified in DIN ISO 2768. This applies both to a simple export/import operation and to operations where the imported STEP file is re-exported to STEP again. Making changes to the imported geometry in the receiving CAD system does not seem to make this worse. However, this needs to be assessed on a situation-by-situation basis. DIN ISO 2768 is the standard specified as the generally applicable tolerance for dimensions for which no explicit tolerance values are specified. In that case, the 3D model can be used directly to generate CNC toolpaths. It becomes different when explicit tolerances with widths of up to two hundredths of a millimetre are assigned. In that case, direct use of the 3D model to generate CNC toolpaths when spline-based geometry is involved may lead to products that do not meet the allowed tolerances.

6.4 Model parameters

One of the requirements for MBD is the availability of annotations as "representation PMI" in the CAD model. In theory, this makes it easier to use these annotations in other applications, e.g. automatic generation of First Article Inspection Documents, (semi-)automatic generation of measurement programmes and CNC programmes. "Representation PMI" is a method of linking the information of an annotation as metadata to the "presentation PMI" of the same annotation (see also subsection Annotations). However, there may also be other metadata stored in the CAD model that is not linked to an annotation. An example of such metadata are variables used by PDM/PLM systems. This section examines the extent to which these variables can be passed on to other CAD systems when the exchange is based on STEP AP242. To this end, a brick-like model with dimensions of 200 mm $\times$ 150 mm $\times$ 50 mm was created in Inventor 2022, CATIA v5-6R2022, Siemens NX Version 2019 and PTC Creo 9.0.3.0. Shape and size are irrelevant. It is just a model to which parameters can be assigned. Three parameters were created in these CAD models, namely *Make_Part*, *VersionID* and *Material_name*. *Make_Part* is a boolean parameter which can hold the values Yes/No. *VersionID* and *Material_name* are text strings. The parameter name *Material* has been avoided because it is often used as an internal parameter by various CAD systems. The CAD model is exported to STEP AP242, with the options for correct export of parameters and annotations activated. This STEP file is then read into each of the four CAD systems. It is then checked that the three parameters have been transferred correctly. Finally, the imported STEP file is exported back to STEP AP242 by the receiving CAD system. Then, this STEP file is imported back into the original CAD system and it is checked whether the parameters are still present.

6.4.1 Inventor 2022

The three parameters *Make_Part*, *VersionID* and *Material_name* were created as custom iProperties (see Figure 6.49). Analysis of the STEP file exported by Inventor 2022 by the NIST STEP File Analyzer and Viewer shows that no parameters are present within the file. As a result, the other CAD systems cannot read parameters and no parameters are retained when re-exporting to STEP AP242 (Table 6.20 and Table 6.21).

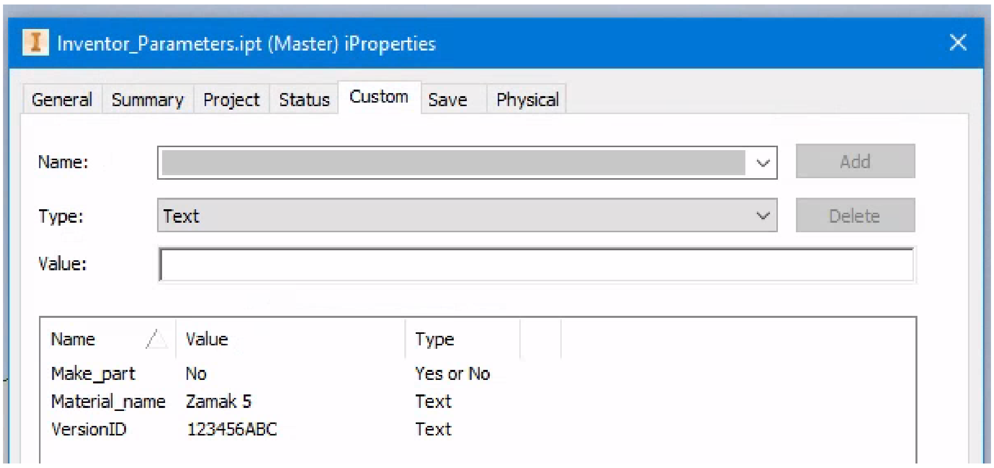


Figure 6.49: Parameters added as custom iProperties in Inventor 2022

Table 6.20: Success rate of transferring parameters by exporting to STEP AP242 by Inventor 2022

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	-	-	-	-
VersionID	-	-	-	-
Material_name	-	-	-	-

Table 6.21: Success rate of transferring parameters by re-exporting to STEP AP242 and re-importing in Inventor 2022

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	-	-	-	-
VersionID	-	-	-	-
Material_name	-	-	-	-

One possible solution to this problem is to write a script using the Inventor VB.NET API that adds the custom iProperties parameters as STEP parameters to the exported STEP file and vice versa. This can be done relatively easily because there is no connection between the parameters and the model topology.

6.4.2 CATIA V5-6R2022

The three parameters *Make_Part*, *VersionID* and *Material_name* were created as "Formulas" (see Figure 6.50). Analysis of the STEP file exported by CATIA V5 by the NIST STEP File Analyzer and Viewer shows that all the parameters and their contents are present within the file. Table 6.24 shows how these parameters are defined in the STEP AP242 file. The parameters are retained when importing and re-exporting by CATIA v5 and PTC Creo Parametric 9.0.3.0. They are lost completely when imported and re-exported by Inventor 2022 and Siemens NX Version 2019 (Table 6.22 and Table 6.23).

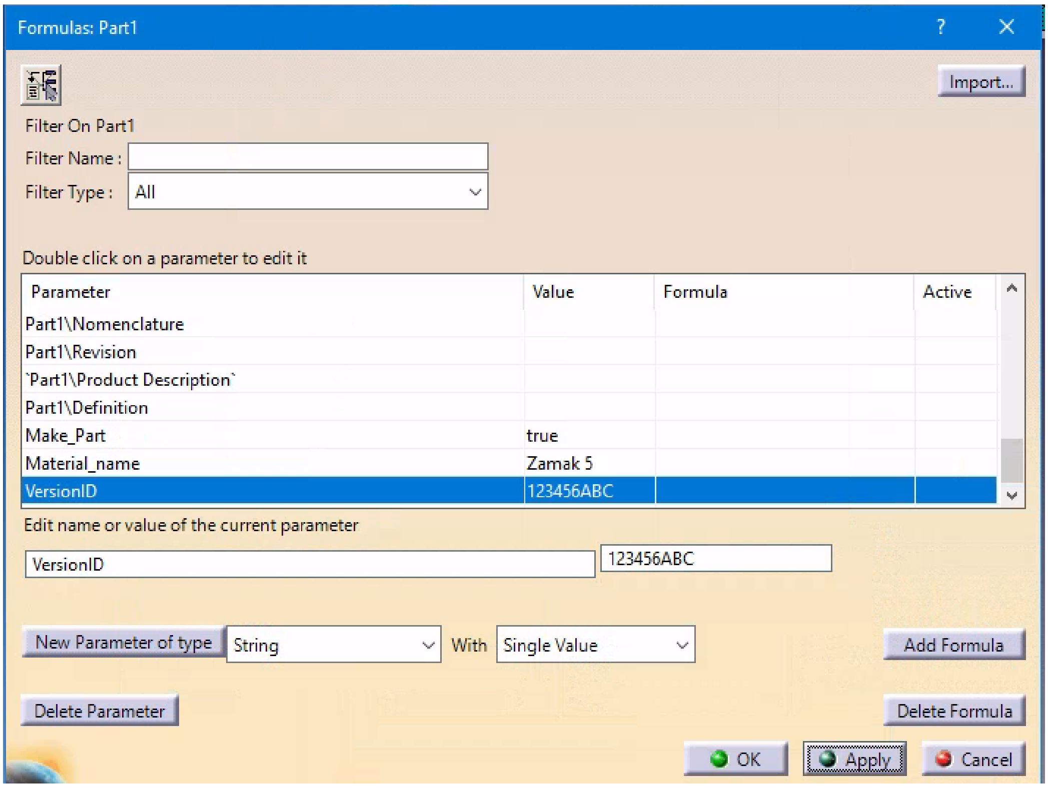


Figure 6.50: Parameters added as "Formulas" in CATIA V5

Table 6.22: Success rate of transferring parameters by exporting to STEP AP242 by CATIA v5

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	-	+	-	+
VersionID	-	+	-	+
Material_name	-	+	-	+

Table 6.23: Success rate of transferring parameters by re-exporting to STEP AP242 and re-importing in CATIA v5

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	- -	+	- -	+
VersionID	- -	+	- -	+
Material_name	- -	+	- -	+

Table 6.24: Excerpt from the definition of the parameters in the STEP AP242 file created by CATIA V5

STEP

```
#10=PRODUCT_DEFINITION(”,’ ’,#6,#3) ;
#189=PROPERTY_DEFINITION(’Make_Part’,’user defined attribute’,#10) ;
#190=GENERAL_PROPERTY(”,’Make_Part’,”) ;
#191=GENERAL_PROPERTY_ASSOCIATION(”,”,#190,#189) ;
#192=REPRESENTATION(”,(#194),#16) ;
#193=PROPERTY_DEFINITION_REPRESENTATION(#189,#192) ;
#194=BOOLEAN_REPRESENTATION_ITEM(”,.T.) ;

#195=PROPERTY_DEFINITION(’Material_name’,’user defined attribute’,#10) ;
#196=GENERAL_PROPERTY(”,’Material_name’,”) ;
#197=GENERAL_PROPERTY_ASSOCIATION(”,”,#196,#195) ;
#198=REPRESENTATION(”,(#200),#16) ;
#199=PROPERTY_DEFINITION_REPRESENTATION(#195,#198) ;
#200=DESCRIPTIVE_REPRESENTATION_ITEM(”,’Zamak 5’) ;

#201=PROPERTY_DEFINITION(’VersionID’,’user defined attribute’,#10) ;
#202=GENERAL_PROPERTY(”,’VersionID’,”) ;
#203=GENERAL_PROPERTY_ASSOCIATION(”,”,#202,#201) ;
#204=REPRESENTATION(”,(#206),#16) ;
#205=PROPERTY_DEFINITION_REPRESENTATION(#201,#204) ;
#206=DESCRIPTIVE_REPRESENTATION_ITEM(”,’123456ABC’) ;

#207=PROPERTY_DEFINITION(’attribute validation property’,”,#10) ;

#12=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;
#13=(NAMED_UNIT(*)PLANE_ANGLE_UNIT()SI_UNIT($,.RADIAN.)) ;
#14=(NAMED_UNIT(*)SI_UNIT($,.STERADIAN.)SOLID_ANGLE_UNIT()) ;
#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,’distance_accuracy_value’,
’CONFUSED CURVE UNCERTAINTY’) ;
#16=(GEOMETRIC_REPRESENTATION_CONTEXT(3)GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#15))
GLOBAL_UNIT_ASSIGNED_CONTEXT((#12,#13,#14))REPRESENTATION_CONTEXT(’ ’,’ ’)) ;
```

6.4.3 Siemens NX Version 2019

The three parameters *Make_Part*, *VersionID* and *Material_name* were created as “Expressions” (see Figure 6.51). Analysis of the STEP file exported by Siemens NX Version 2019 by the NIST STEP File Analyzer and Viewer shows that that no parameters are present within the file. As a result, the other CAD systems cannot read parameters and no parameters are retained when re-exporting to STEP AP242 (Table 6.25 and Table 6.26).

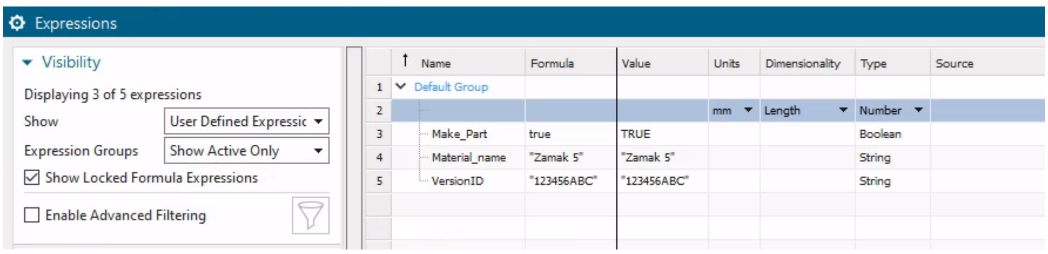


Figure 6.51: Parameters added as “Expressions” in Siemens NX

Table 6.25: Success rate of transferring parameters by exporting to STEP AP242 by Siemens NX

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	-	-	-	-
VersionID	-	-	-	-
Material_name	-	-	-	-

Table 6.26: Success rate of transferring parameters by re-exporting to STEP AP242 and re-importing in Siemens NX

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	-	-	-	-
VersionID	-	-	-	-
Material_name	-	-	-	-

One possible solution to this problem is to export the "Expressions" to an Exp-file. This is a plain text file (Figure 6.52). A computer programme can then be written that extracts the parameters from the STEP file and converts them into an Exp-file and vice versa. This can be done relatively easily because there is no connection between the parameters and the model topology.

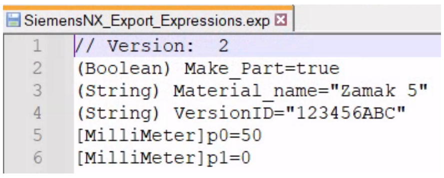


Figure 6.52: Siemens NX Expressions exported to an Exp file

6.4.4 PTC Creo Parametric 9.0.3.0

The three parameters *Make_Part*, *VersionID* and *Material_name* were created as "Part parameters" (see Figure 6.53). Analysis of the STEP file exported by PTC Creo Parametric 9.0.3.0 by the NIST STEP File Analyzer and Viewer shows that all the parameters and their contents are present within the file. Table 6.29 shows how these parameters are defined in the STEP AP242 file. The parameters are retained when importing and re-exporting by CATIA v5 and PTC Creo Parametric 9.0.3.0. They are lost completely when imported and re-exported by Inventor 2022. Although the parameters were not recognised and read in Siemens NX, they were retained in the re-exported STEP file (Table 6.27 and Table 6.28).

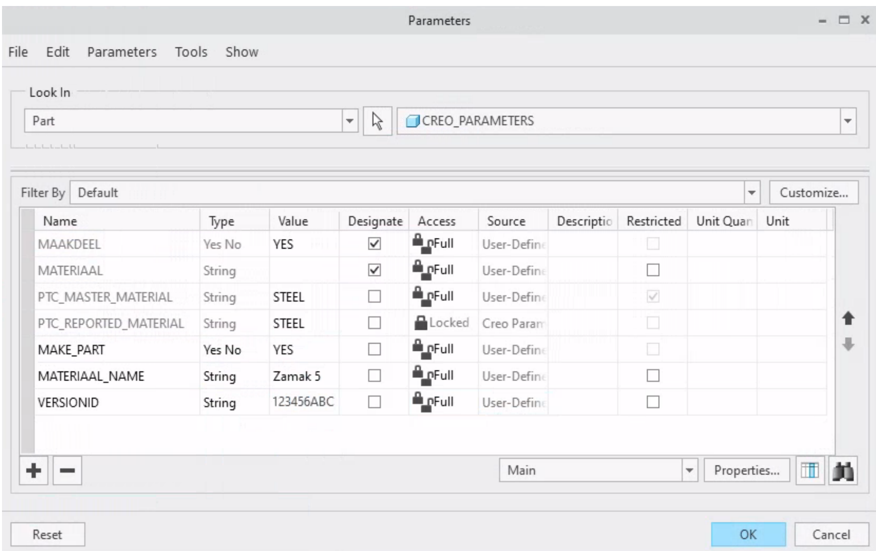


Figure 6.53: Parameters added as "part parameters" in Creo Parametric

Table 6.27: Success rate of transferring parameters by exporting to STEP AP242 by Creo Parametric

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	-	+	-	+
VersionID	-	+	-	+
Material_name	-	+	-	+

Table 6.28: Success rate of transferring parameters by re-exporting to STEP AP242 and re-importing in Creo Parametric

	Inventor	CATIA v5	Siemens NX	PTC Creo Parametric
Make_Part	- -	+	-/+	+
VersionID	- -	+	-/+	+
Material_name	- -	+	-/+	+

Table 6.29: Excerpt from the definition of the parameters in the STEP AP242 file created by Creo Parametric 9.0.3.0

STEP

```
#229=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(1.E-2),#224,’closure’,
’Maximum model space distance between geometric entities at asserted connectivities’);

#340=GENERAL_PROPERTY(”,’MAKE_PART’,’user defined attribute’);
#341=GENERAL_PROPERTY_ASSOCIATION(’user defined attribute’,”,#340,#339);
#342=BOOLEAN_REPRESENTATION_ITEM(’MAKE_PART’,.T.);
#346=GENERAL_PROPERTY(”,’MATERIAAL_NAME’,’user defined attribute’);
#347=GENERAL_PROPERTY_ASSOCIATION(’user defined attribute’,”,#346,#345);
#348=DESCRIPTIVE_REPRESENTATION_ITEM(’MATERIAAL_NAME’,’Zamak 5’);
#352=GENERAL_PROPERTY(”,’VERSIONID’,’user defined attribute’);
#353=GENERAL_PROPERTY_ASSOCIATION(’user defined attribute’,”,#352,#351);
#354=DESCRIPTIVE_REPRESENTATION_ITEM(’VERSIONID’,’123456ABC’);

#224=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.));

#230=(GEOMETRIC_REPRESENTATION_CONTEXT(3)GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((
#229))GLOBAL_UNIT_ASSIGNED_CONTEXT((#224,#227,#228))REPRESENTATION_CONTEXT
(’ID1’,’3’));
#241=APPLICATION_CONTEXT(’automotive design’);
#242=APPLICATION_PROTOCOL_DEFINITION(’international standard’,
’config_control_design’,1994,#241);
#243=PRODUCT_DEFINITION_CONTEXT(”,#241,’design’);
#244=PRODUCT_CONTEXT(”,#241,’mechanical’);
#245=PRODUCT(’CREO_PARAMETERS’,’CREO_PARAMETERS’,’NOT SPECIFIED’,(#244));
#246=PRODUCT_DEFINITION_FORMATION_WITH_SPECIFIED_SOURCE(’1’,’LAST_VERSION’,#245,
.MADE.);
#247=PRODUCT_DEFINITION(’design’,”,#246,#243);

#339=PROPERTY_DEFINITION(’MAKE_PART’,’user defined attribute’,#247);
#343=REPRESENTATION(”,(#342),#230);
#344=PROPERTY_DEFINITION_REPRESENTATION(#339,#343);
#345=PROPERTY_DEFINITION(’MATERIAAL_NAME’,’user defined attribute’,#247);
#349=REPRESENTATION(”,(#348),#230);
#350=PROPERTY_DEFINITION_REPRESENTATION(#345,#349);
#351=PROPERTY_DEFINITION(’VERSIONID’,’user defined attribute’,#247);
#355=REPRESENTATION(”,(#354),#230);
#356=PROPERTY_DEFINITION_REPRESENTATION(#351,#355);
```

6.5 Feature characteristics

6.5.1 Introduction

In addition to the topology of the CAD model and the parameters associated with the model, there is another way in which additional information about the CAD model can be communicated. These are the feature characteristics, such as “feature hierarchy” and “feature properties”, which are discussed in the following subsections.

6.5.2 Feature hierarchy

Consider the example of a hole passing through two mounted parts (see Figure 6.54).

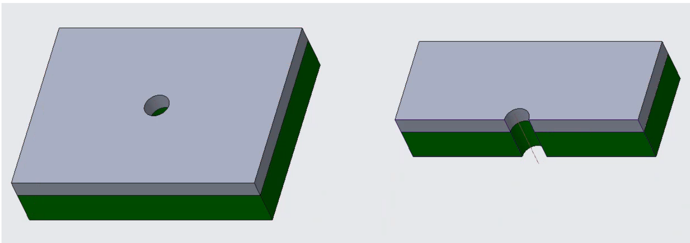


Figure 6.54: Feature hierarchy

There are several ways in which the hole in this assembly could have been created. A first possibility is that the hole was present in these two parts before they were assembled on top of each other. A second possibility is that the parts were assembled on top of each other without a hole, after which the hole was created in the assembled parts. In several CAD systems, this distinction can be made in the way the hole is created. In this way, the manufacturing department can be informed at what time this hole should be made. This subsection examines the extent to which this information specified in the CAD system is retained after export to STEP AP242.

In PTC Creo Parametric, for example, this can be done via the so-called intersection mode. When the “Top Level” option is selected, the hole is made only at assembly level and not in the individual parts (see Figure 6.55). When the “Part Level” option is selected, the hole is created at the level of the individual parts (see Figure 6.56).

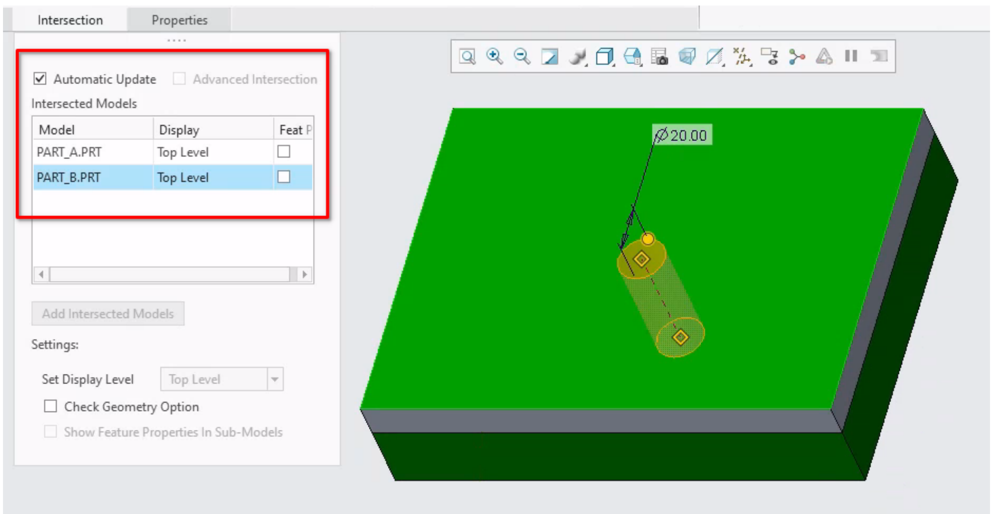


Figure 6.55: Hole created with the option “Top Level” in PTC Creo. The hole now only exists in the assembly

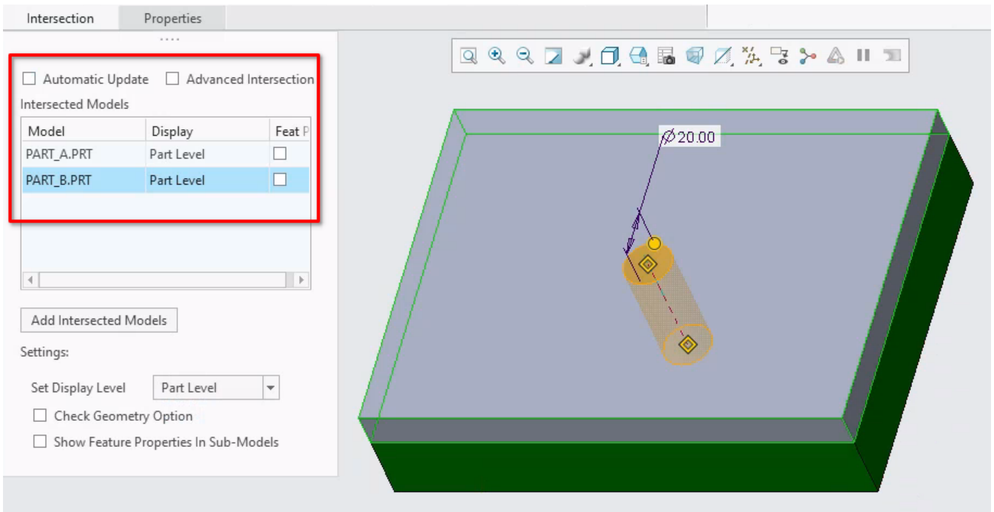


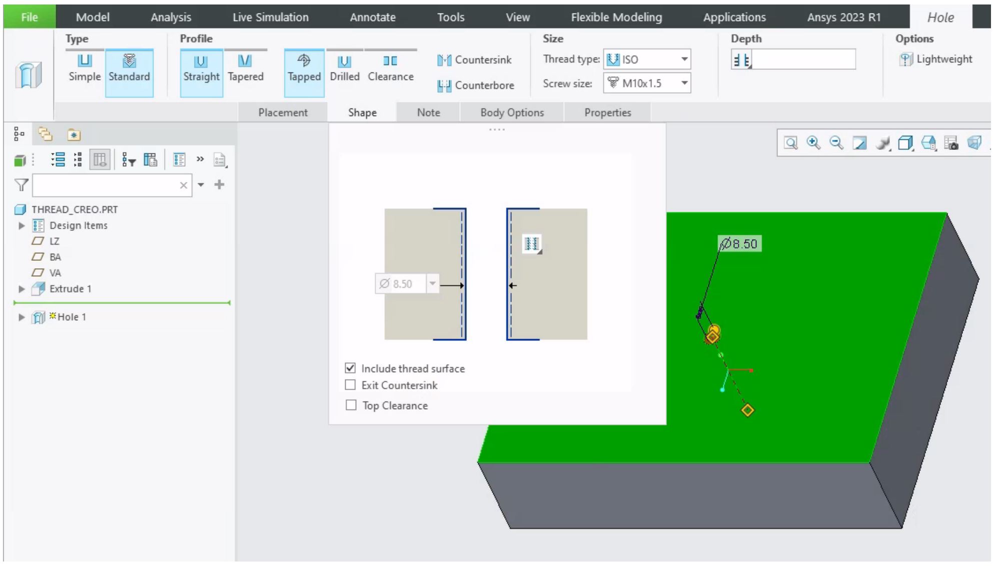
Figure 6.56: Hole created with the option “Part Level” in PTC Creo
There is now a hole in each part

The use of functionality such as “Top Level” and “Part Level” is a way of putting some intelligence into the CAD model that can indicate when the hole should be made during the production process. This intelligence is only available in the native CAD format. The tests showed that when the assembly model is exported to STEP, this information is lost. Even though the holes are defined as “Top Level”, they are present in the individual parts.

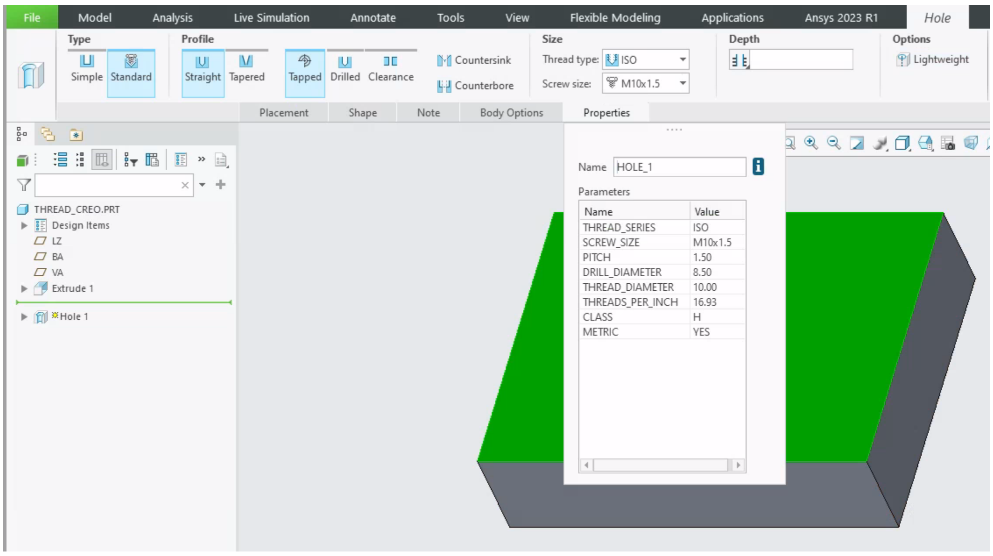
6.5.3 Feature properties

The holes discussed so far are so-called “simple holes”. This means that the shape of the hole is entirely determined by the geometry created by the designer. Tapped holes are another matter entirely. The geometry of the thread is not created in the CAD system, but by the tapping operation on the machine.

Just about every CAD system has the ability to create tapped holes (Figure 6.57a, Figure 6.58a, Figure 6.59, Figure 6.60a). The hole is not modelled with a geometric model of the thread. Only the core diameter of the hole is created after which the specifications of the thread are assigned to the hole as parameters in a manner unique to each CAD system. (Figure 6.57b, Figure 6.58b, Figure 6.60b). These parameters are added as metadata to the hole feature. When the model is exported to STEP AP242, these metadata are lost. This occurs with all the CAD systems tested. Only the hole with the core thread diameter is retained. All thread information such as which thread, pitch, depth is lost. So any information that was present in the CAD model about the tapped holes disappeared after export. The information can only be derived from annotations assigned to the tapped holes.

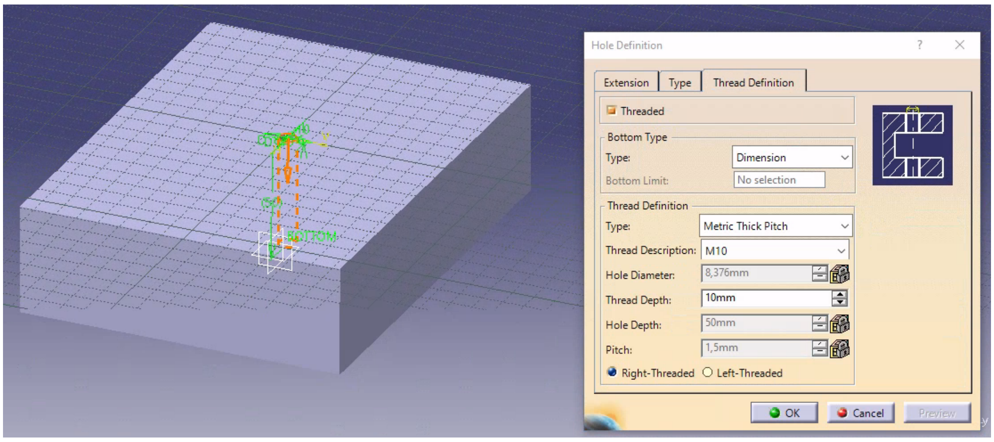


(a) Thread hole creation

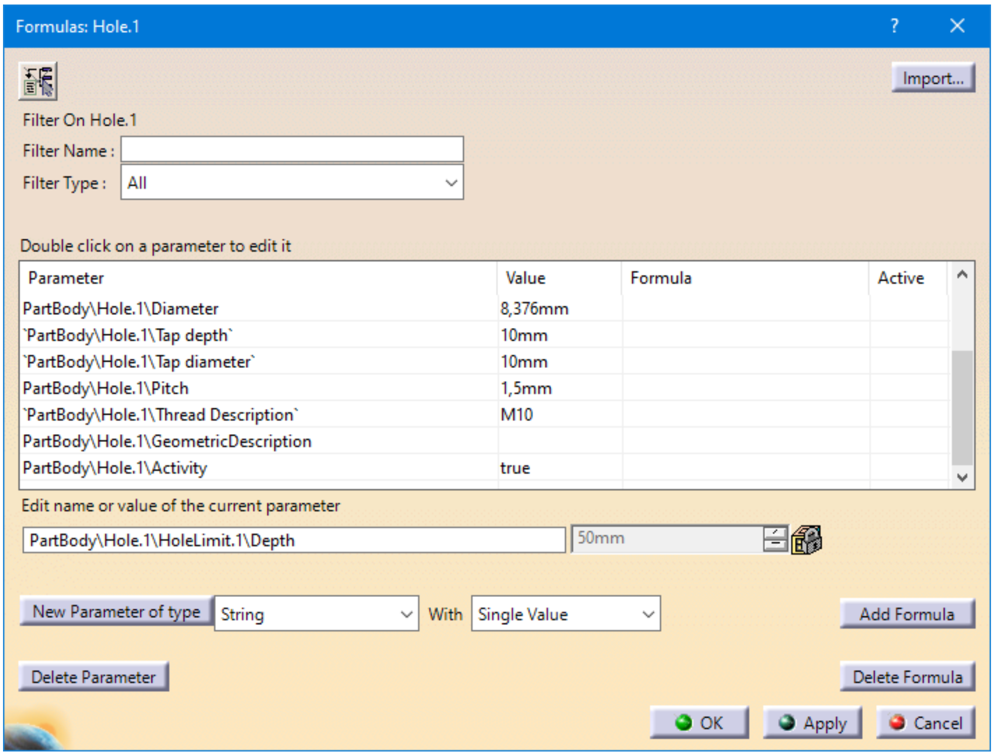


(b) Threaded hole parameters

Figure 6.57: Creation of a threaded hole in PTC Creo Parametric



(a) Thread hole creation



(b) Threaded hole parameters

Figure 6.58: Creation of a threaded hole in CATIA V5

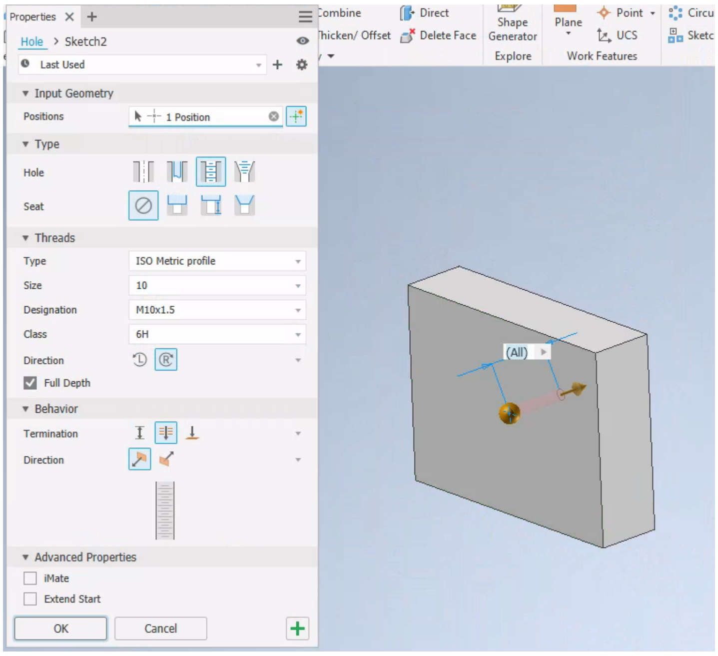
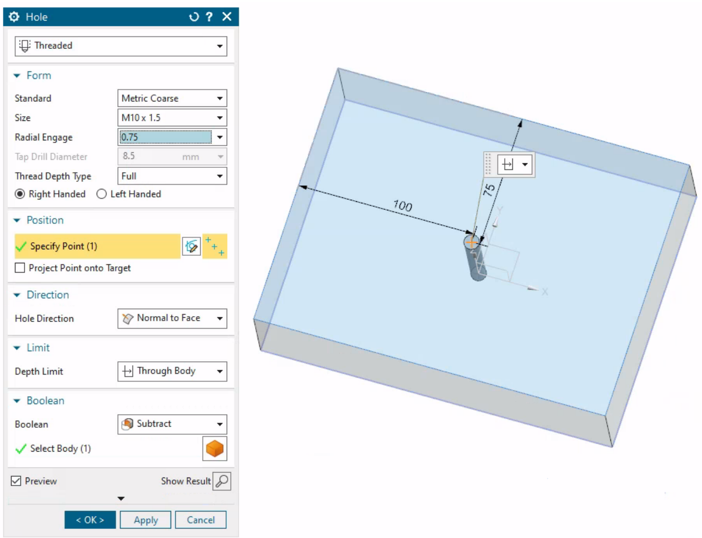
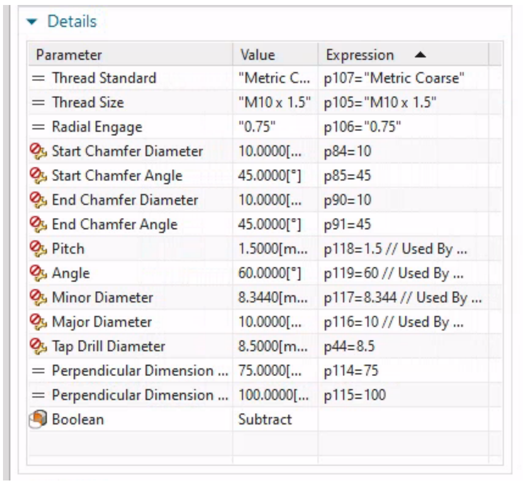


Figure 6.59: Creation of a threaded hole in Inventor 2022



(a) Thread hole creation



Details

Parameter	Value	Expression
Thread Standard	"Metric C...	p107="Metric Coarse"
Thread Size	"M10 x 1.5"	p105="M10 x 1.5"
Radial Engage	"0.75"	p106="0.75"
Start Chamfer Diameter	10.0000[...	p84=10
Start Chamfer Angle	45.0000[°]	p85=45
End Chamfer Diameter	10.0000[...	p90=10
End Chamfer Angle	45.0000[°]	p91=45
Pitch	1.5000[m...	p118=1.5 // Used By ...
Angle	60.0000[°]	p119=60 // Used By ...
Minor Diameter	8.3440[m...	p117=8.344 // Used By ...
Major Diameter	10.0000[...	p116=10 // Used By ...
Tap Drill Diameter	8.5000[m...	p44=8.5
Perpendicular Dimension ...	75.0000[...	p114=75
Perpendicular Dimension ...	100.0000[...	p115=100
Boolean	Subtract	

(b) Threaded hole parameters

Figure 6.60: Creation of a threaded hole in Siemens NX

In PTC Creo Parametric, a tapped hole is built up by a normal hole with the core diameter of the thread and a surface around this hole with the outside diameter of the thread. This surface is used to represent the screw thread in a 2D drawing by means of a three-quarter circle (Figure 6.61). The previously mentioned parameters of the thread are also assigned to this surface. This surface becomes a separate surface in the model after export to STEP AP242. The CAD system importing the STEP file cannot do anything with is. However, it can be used to manually determine the depth of the thread.

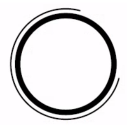


Figure 6.61: Three-quarter circle representing the screw thread in a 2D drawing

The reason for all this is that STEP only focuses on transferring the topological structure of a 3D model. The thread is not represented in 3D because the geometry of the product does not determine the thread geometry, but rather the production tool being a thread cutting die or tapping tool.

From all this it can be concluded that STEP AP242 is not ideal from a production point of view. Information that can be used to automate operations such as tapping holes is no longer available. Information that can be used for part assembly, such as hole definition on part level or on assembly level, is also lost. These shortcomings should be addressed in future versions of the STEP standard.

6.6 QIF

MBD is associated with the term "smart manufacturing". It requires a seamless flow of information between different stakeholders such as design, production and quality control (Ram et al. 2021). An example is the transfer of measurement results by storing them in the MBD model of the product to be controlled. This is something STEP cannot do. Partly because of this, a group of companies decided to develop a new format. This format is called QIF.

QIF stands for Quality Information Framework and began to gain prominence about 10 years ago when the DMSC (an acronym of the Dimensional Metrology Standards Consortium) submitted QIF v1.0 for ANSI standardisation in 2013 (Quality Magazine 2013). The latest standard QIF v3.0 was released in December 2018. In 2020 it became a new ISO Standard ISO 23952:2020 (Metrology News 2020). Zhao et al. 2012 define QIF as "an integrated and holistic set of information models which, if widely adopted, can enable the effective exchange of metrology data throughout the entire manufacturing quality measurement process – from design to planning to execution to analysis". It is a new neutral exchange format for CAD models that - as the name already suggests - focuses on quality control. It makes use of XML to store the data and can be considered as the successor of the STEP AP219 standard that never got fully developed as it was originally intended (Zhao et al. 2012). QIF not only enables the exchange of CAD geometry, along with all MBD-related information such as 3D annotations with semantic references, but also allows measurement protocols to be stored in the same QIF file. The results of these measurement protocols can again be stored in the same file. According to the specifications that are freely available at `https://qifstandards.org/download/` there is full support for the parameters of threaded holes. QIF is often considered the format that overcomes the shortcomings of STEP in terms of MBD. However, the QIF format is not yet supported by many

CAD/CAM packages. The company CAPVidia has developed plug-ins for PTC Creo (Figure 6.62), SolidWorks and Siemens NX for importing and exporting QIF files.

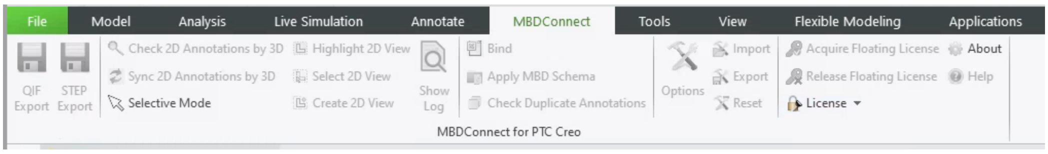


Figure 6.62: MBDConnect, QIF plug-in for PTC Creo developed by CAPVidia

For PTC Creo, this plug-in is developed using ProToolkit. ProToolkit is a library made available by PTC to develop applications on top of PTC Creo using the C programming language. What the QIF plug-in can and cannot do is entirely determined by the functionality of the APIs available within this library. Due to the limited number of CAD systems for which the QIF plug-in is available, claims for QIF could not be investigated in this PhD study.

6.7 3D PDF

To view a STEP file, the user needs specialist software such as a CAD system or a CAD viewer. This makes it less suitable for some stakeholders, such as sales. 3D PDF is proposed as a possible alternative that does not require special software if no manipulation of the model, such as measurements, is required.

For this reason, 3D PDF is a format that is often put forward as a medium for communicating an MBD model to the various stakeholders (Miralbes-Buil et al. 2020). If a company wants to use 3D PDF for this purpose, it should be aware that 3D PDF is a so-called encapsulating format that can contain multiple types of formats under the same name. Two of these formats that can be embedded in a 3D PDF, namely U3D and PRC, are supported by default by the free Adobe Acrobat Reader software (Adobe 2023). U3D stands for the Universal 3D format. Originally, development was started in 2004 by a consortium called 3D Industry Forum with more than 30 members including Intel, Adobe, Microsoft (Smith 2004). In 2005 it became an ECMA[1] standard known as ECMA-363. Shortly afterwards, the consortium was dissolved. At first it only supported tessellated formats. The latest edition of the standard of 2007 (the fourth edition) added support for curves and surfaces based on B-splines and NURBS (International 2007). PRC stands for Product Representation Compact. Its development began in 2002 by the TTF Group, which was later acquired by Adobe. In 2014, it finally became an ISO standard, ISO 14739-1:2014. From the beginning, the goal was to be able to do much more than just define a tessellated CAD model. The goal was to preserve the history tree, topology, assemblies, geometry, layers and colours combined with very high data compression (TTF Group 2003; Yoders 2007). In addition to U3D and PRC, more than 50 other formats, including proprietary CAD formats, are supported via plug-ins for Adobe Acrobat Reader. So a 3D PDF generated using one CAD system does not necessarily have the same capabilities as a 3D PDF generated using another CAD system. Most CAD systems require a special license to activate the module to create 3D PDF files. That module may have been developed by the CAD system manufacturer itself or be a plug-in from another company. Four very well-known companies that develop translator plug-ins are Anark, Datakit, Tetra4D and Theorem Solutions.

The following test examined the 3D PDF export module of Inventor, Siemens NX and PTC Creo. The reason that CATIA v5 was not included is that no 3D PDF export plug-in could be obtained for running the test. This was overcome by using sample

[1] EMCA stands for European Computer Manufacturers Association
https://www.ecma-international.org/

3D PDF files from the company Theorem Solutions, which develops 3D PDF modules for CATIA v5, among others. The mixed concave and convex model from subsection 6.3.3 was used for this test. The use of a double-curved surface in the CAD model makes it easier to check whether or not faceting has been used in the 3D PDF model. A 3D annotation was added to the original CAD model to test the transfer of PMI data. Figure 6.63, Figure 6.64, Figure 6.65 and Figure 6.66 show the resulting 3D PDF files. On first inspection, all models look the same as in the CAD systems. When the render mode in Acrobat Reader is changed from solid to shaded wireframe, the true nature of the model is revealed. Figure 6.67, Figure 6.68, Figure 6.69 and Figure 6.70 show that the topology of the model is no longer the same as in the CAD systems, but has changed from a B-rep to a triangulated model. To check whether an annotation was presentation PMI or representation PMI, the annotation was selected in Acrobat Reader's "Model Tree". The "Copy Data" option was then selected and pasted into a text editor.

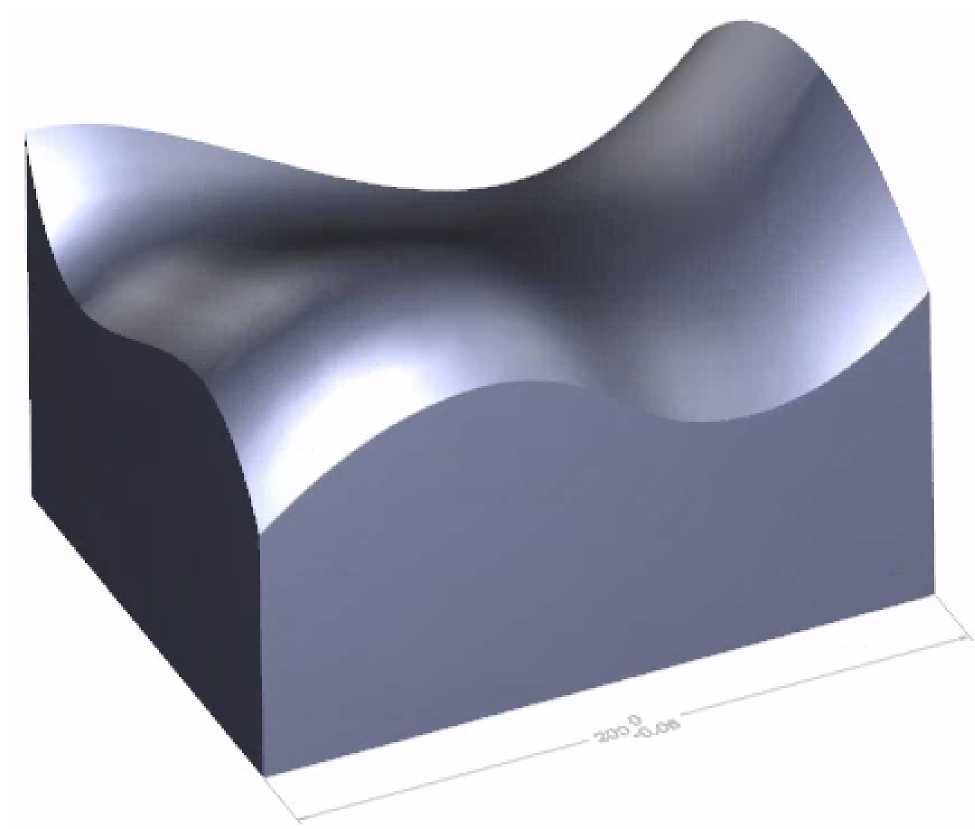


Figure 6.63: Display of the 3D PDF file created by PTC Creo in Adobe Acrobat Reader

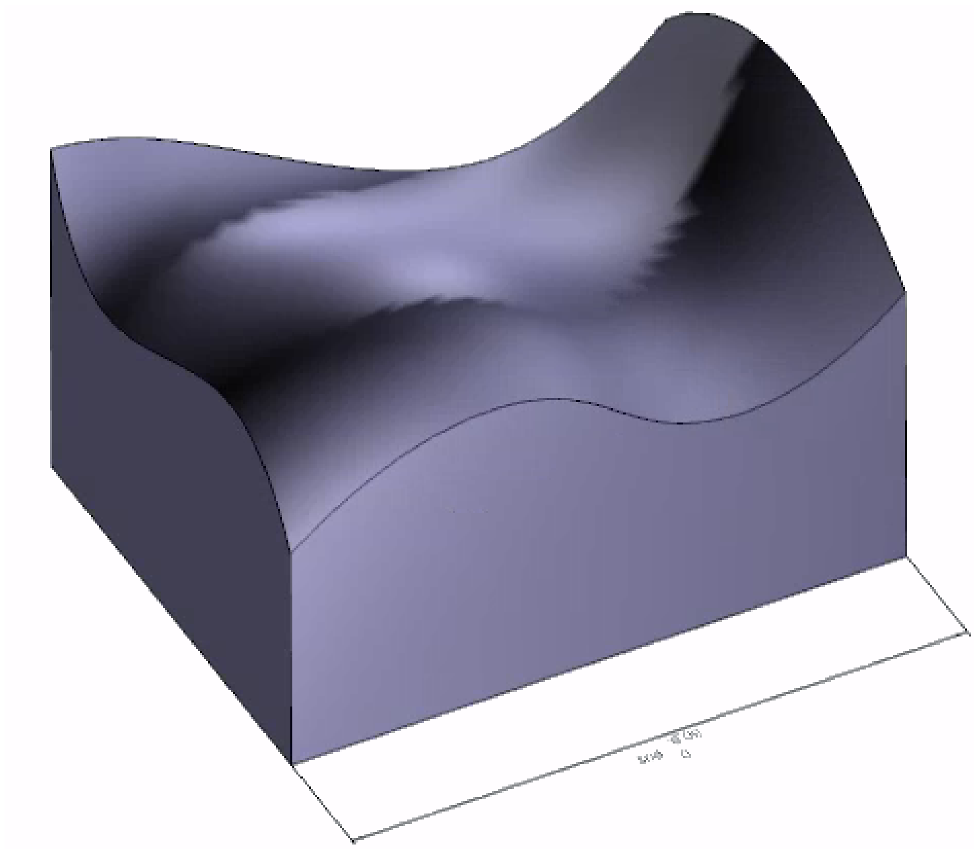


Figure 6.64: Display of the 3D PDF file created by Siemens NX in Adobe Acrobat Reader

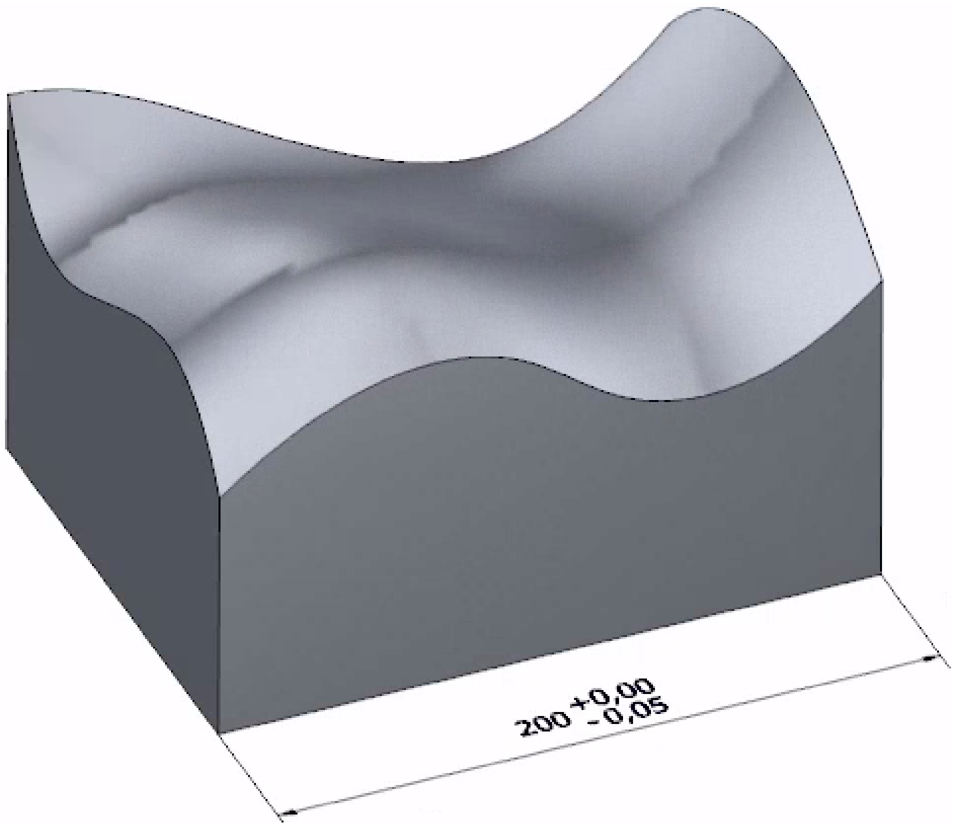


Figure 6.65: Display of the 3D PDF file created by Autodesk Inventor in Adobe Acrobat Reader

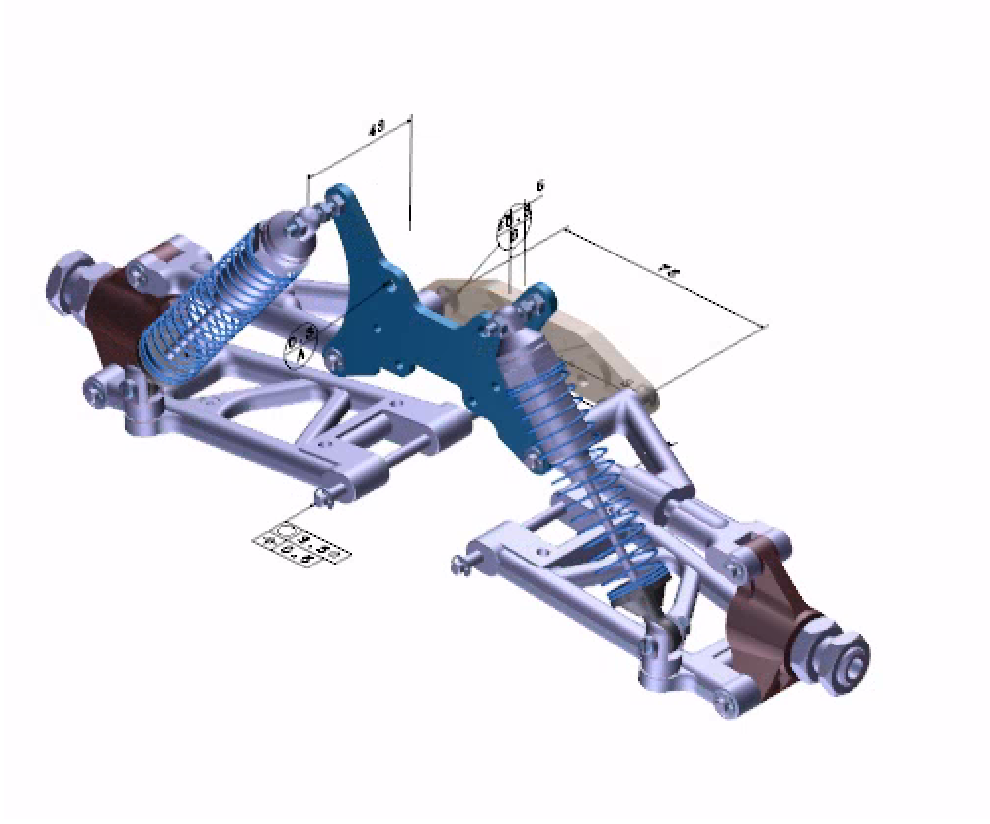


Figure 6.66: Display of the sample 3D PDF file created by Theorem Solutions' software package

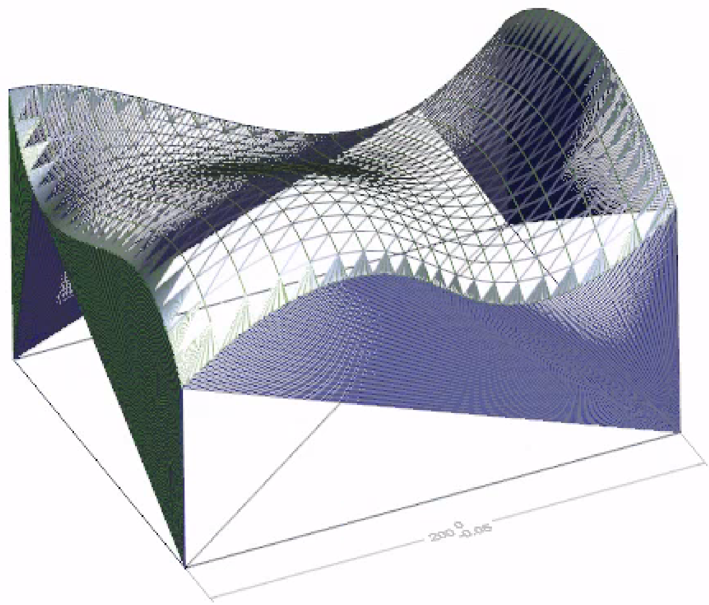


Figure 6.67: Display as shaded wireframe of the 3D PDF file created by PTC Creo in Adobe Acrobat Reader

The PDF file created by PTC Creo contains the following line

```
«/DV/DEFAULT/Filter/FlateDecode/Length 73233/Subtype/U3D/Type/3D/VA
[29 0 R 30 0 R 31 0 R 32 0 R 33 0 R 34 0 R]»stream
```

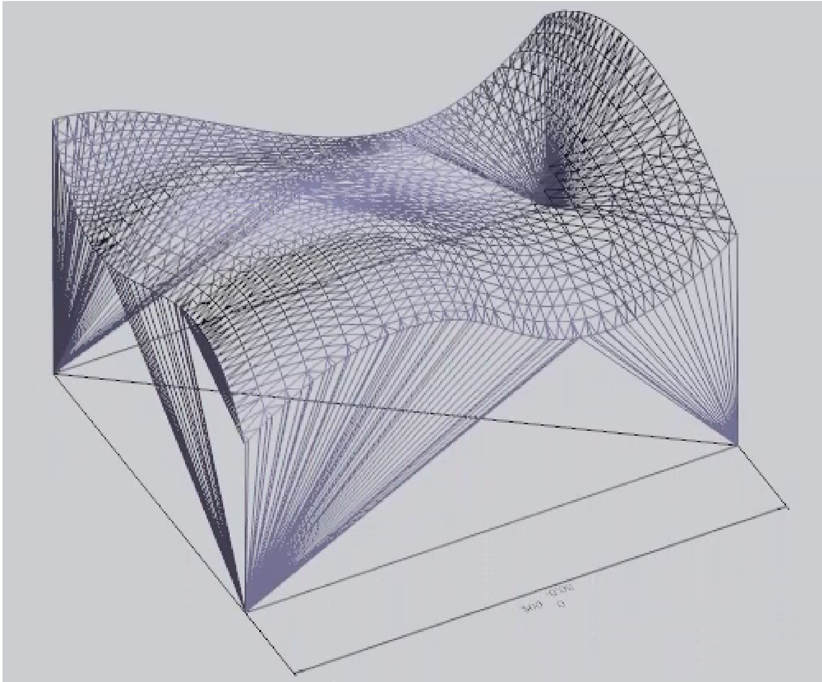


Figure 6.68: Display as shaded wireframe of the 3D PDF file created by Siemens NX in Adobe Acrobat Reader

The PDF file created by Siemens NX contains the following line

```
«/3DOwners[144 0 R]/AN 143 0 R/Length 109664/OnInstantiate 48 0 R
/Subtype/PRC/Type/3D/VA[136 0 R]»stream
```

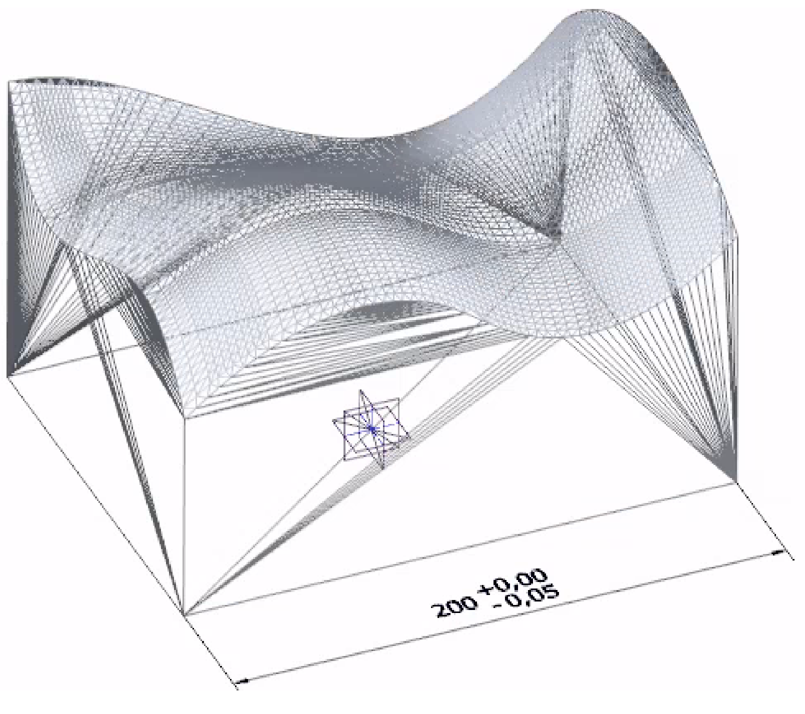


Figure 6.69: Display as shaded wireframe of the 3D PDF file created by Autodesk Inventor in Adobe Acrobat Reader

The PDF file created by AutoDesk Inventor contains the following line

```
«/AN«/PC -1/Subtype/Linear/TM 1/Type/3DAnimationStyle»/Filter/FlateDecode
/Length 253048/OnInstantiate 60 0 R/Subtype/PRC/Type/3D/VA 101 0 R»stream
```

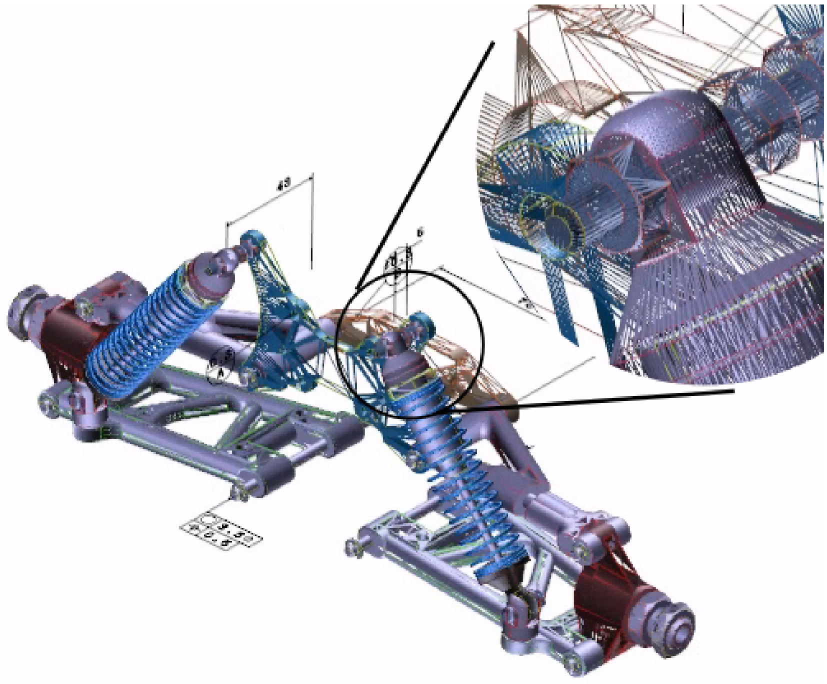


Figure 6.70: Display as shaded wireframe of the 3D PDF file created by Theorem Solutions' software package

The PDF file created by Theorem Solutions' software contains the following line

```
«/AN 321 0 R/Length 567708/OnInstantiate 322 0 R/Subtype/PRC/Type/3D/VA
[323 0 R 324 0 R 325 0 R 326 0 R]»stream
```

Table 6.30: Internal format used in the 3D PDF file

	Creo	**Siemens NX**	**Inventor**	**Theorem Solutions**
Internal format	U3D	PRC	PRC	PRC
Model type	Triangulated	Triangulated	Triangulated	Triangulated
Type of PMI	Presentation	Presentation	Presentation	Presentation

In the case of Siemens NX, the lack of suitable test software meant that it could not be determined whether the annotation had semantic references. However, it could be determined that the nominal value could be read, but not the tolerances. For this reason, the Type of PMI was assigned the value Presentation. The same was the case for Inventor.

In the case of Theorem Solution's demo model, it could be established that the annotation had semantic references associated with it. The associated faces coloured red when the annotation was selected (see Figure 6.71). However, the value of the annotation could not be extracted. For this reason, the Type of PMI was assigned the value Presentation. The highlighting did not occur in the 3D PDF files generated by PTC Creo, Siemens NX and Inventor 2022.

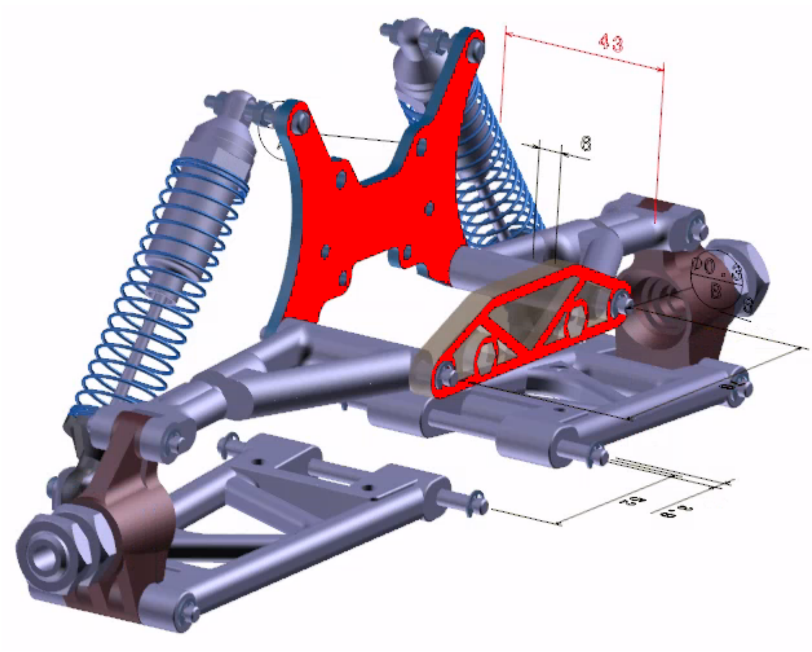


Figure 6.71: Semantic references of the annotation are highlighted in red in the display of the 3D PDF file created by Theorem Solutions' software package

The manual of the 3D PDF software from the company Tetra4D describes that the software has the ability to save the geometry of the CAD model simultaneously as a B-rep model and as a faceted model in the 3D PDF file (Tetra4D 2023, p. 14). This allows the 3D PDF file to be used by stakeholders with different needs. Examples of such stakeholders are sales people and manufacturing personnel. Sales people have enough with the visualisation of the model. This can be done perfectly with the faceted model. Manufacturing personnel need the B-rep model to generate CNC toolpaths. If the B-rep model is saved in the 3D PDF file, this can be done using special software that must be purchased additionally such as, for example, the Tetra4D Converter. This software can convert a B-rep model that is saved in the PRC format in the 3D PDF file to IGES, ParaSolid, STEP, VRML, and STL (Tetra4D 2023, p. 63).

Based on these initial preliminary tests, it can be concluded that 3D PDF can be a solution to pass the MBD model along the chain of stakeholders with all their different needs. From the tests, the preliminary conclusion can be drawn that all the options offered by the different CAD systems as standard modules are only aimed at visualising the model. At best, a list of all measurements can be extracted from the 3D PDF file. The info that can be extracted depends on whether the annotations are saved as presentation PMI or representation PMI. To what extent this can be implemented in the plug-ins depends on how level the API's of the development libraries of the various CAD systems allow the third-party developers to go. In any case, being able to do more with the MBD model stored in the 3D PDF file will require additional investment from the stakeholders involved.

7 Current practices in MBD

7.1 Introduction

One of the research questions of this PhD study is "How are PMI annotations implemented and used in major CAD/CAM/CAE packages in relation to the MBD philosophy". The question "How are PMI annotations implemented" has already been discussed in the previous chapters. This chapter takes a closer look at the question "How are PMI annotations used". It distinguishes between

- Avoiding asymmetric tolerances
- Working with colour coded tolerances
- Design for eXcellence (DfX) / Design for Manufacturing (DfM) / Design for Assembly (DfA)

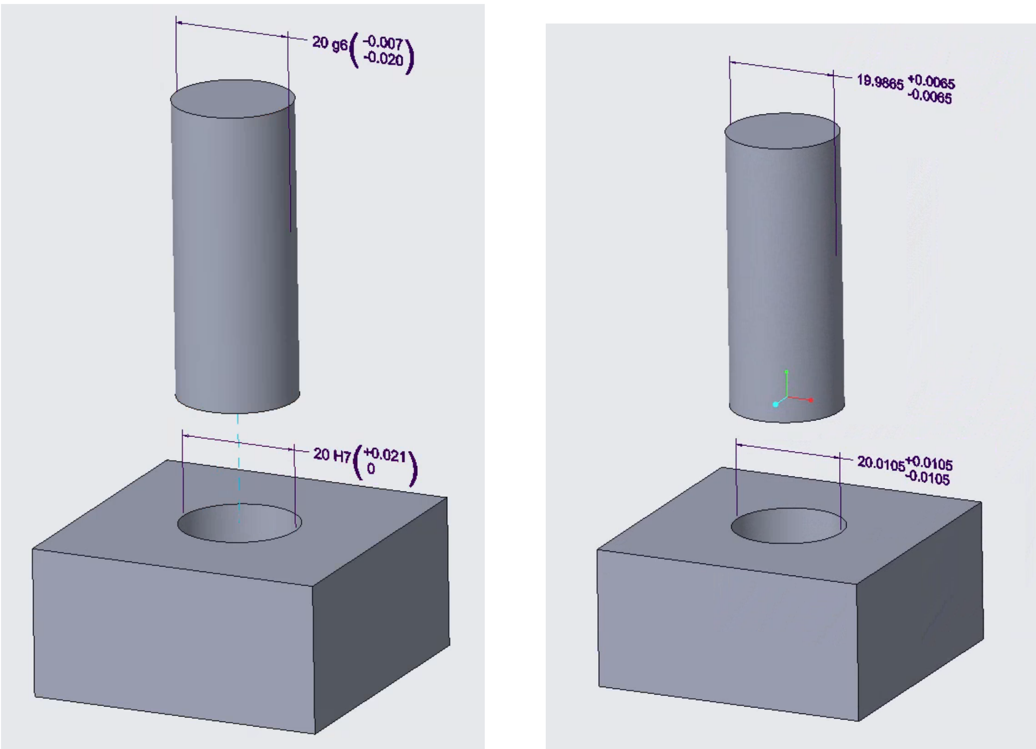
7.2 Avoidance of asymmetric tolerances

Chapter 5 (see page 68 and 101) discussed in more detail the impact of using asymmetric tolerances on the effective manufacturing of the CAD model, including toolpath generation. This showed that, depending on how the CAD model was constructed by the designer, it could be easy or very difficult to adapt the model to the needs of the manufacturing department. Discussions with representatives from PTC and companies assisting others in implementing the MBD philosophy revealed that the use of asymmetric tolerances (even those of the ISO System of Limits and Fits) was strongly discouraged for this very reason. Indeed, this avoids the need for the manufacturing department to change the nominal dimensions to bring them in the centre of the tolerance field to make it easier to generate the CNC toolpaths. However, a number of other issues are overlooked and this creates new problems.

A first issue that is often overlooked is that one of the raisons d'être of the ISO System of Limits and Fits is to standardise the tools needed. These may be quality control tools such as go/no-go gauges and calipers. An example of such a tool is the Insize 4124 range of go/no-go plug gages (Insize 2023). They can also be tools such as reamers used to produce a hole with a specific tolerance. An example of such a tool is the CoroReamer®835 manufactured by Sandvik Coromant (Sandvik Coromant 2022), which is used to produce holes with an H7 fit. When the specific notation of the ISO System of Limits and Fits, with its associated asymmetric tolerances, can no longer be used, the manufacturing department's job becomes more difficult, not easier. It becomes much harder to determine how a hole should actually be made to achieve what the designer intended. This can work if very strict agreements are made between the designer and the manufacturing department (Nyffenegger Felix et al. 2020). This is why it is often limited to those companies where design and manufacturing are in the same company. Taking into account the issues discussed in chapter 6 that can arise when exchanging CAD models between different systems, it can be concluded that the best communication between stakeholders is often only achieved when all stakeholders are using the same CAD/CAM system.

A second issue that is often forgotten is that asymmetric tolerances say something about how parts are assembled. A negative asymmetric tolerance indicates that this part is assembled into another part. A positive asymmetric tolerance indicates that another part is assembled into this part. Asymmetric tolerances ensure that, if sized correctly, parts will always fit. The use of asymmetric tolerances makes it clear to the manufacturing department what the designer's intention is and ensures that the parts will fit, even if each part is manufactured by a different company. Two examples can be seen in Figure 7.1 and Figure 7.2.

A third point is that this may affect the price quoted in a tender for the production of the part. Proponents of avoiding asymmetric tolerances in an MBD model claimed in meetings attended during this PhD study that this was not the case. Written requests for studies to prove this were never answered. One Dutch company said that there were three issues that were included in the software for automatically generating an initial quotation. The first issue was the amount of material to be milled. This was determined by determining the volume difference between the smallest standard rough block and the part to be produced. A second was the number of GD&Ts in the model. A third was the number of dimensions with x number of decimal places. The more decimal places the higher the price estimate.



(a) The assigned tolerances of the ISO System of Limits and Fits indicate the shaft is assembled into the hole and ensure it fits and specify how it must fit

(b) The assigned symmetrical tolerances make it more difficult to see if the shaft will always fit and how it must fit

Figure 7.1: Asymmetric versus symmetric tolerances

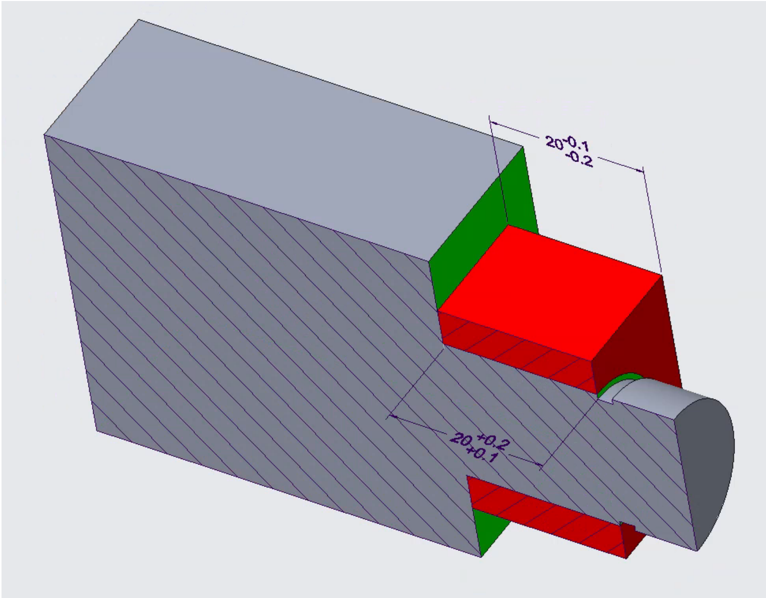


Figure 7.2: The assigned asymmetric tolerances ensure that the red part can always be secured with a circlip

7.3 Working with colour codes for tolerances

To make the MBD model clearer, colours are being used in the MBD model. They can represent surface treatments (Bergen 2002). They can also be used to avoid overloading the MBD model. A particular colour represents a particular tolerance. Example of this are shown in Figure 7.3 and Figure 7.4.

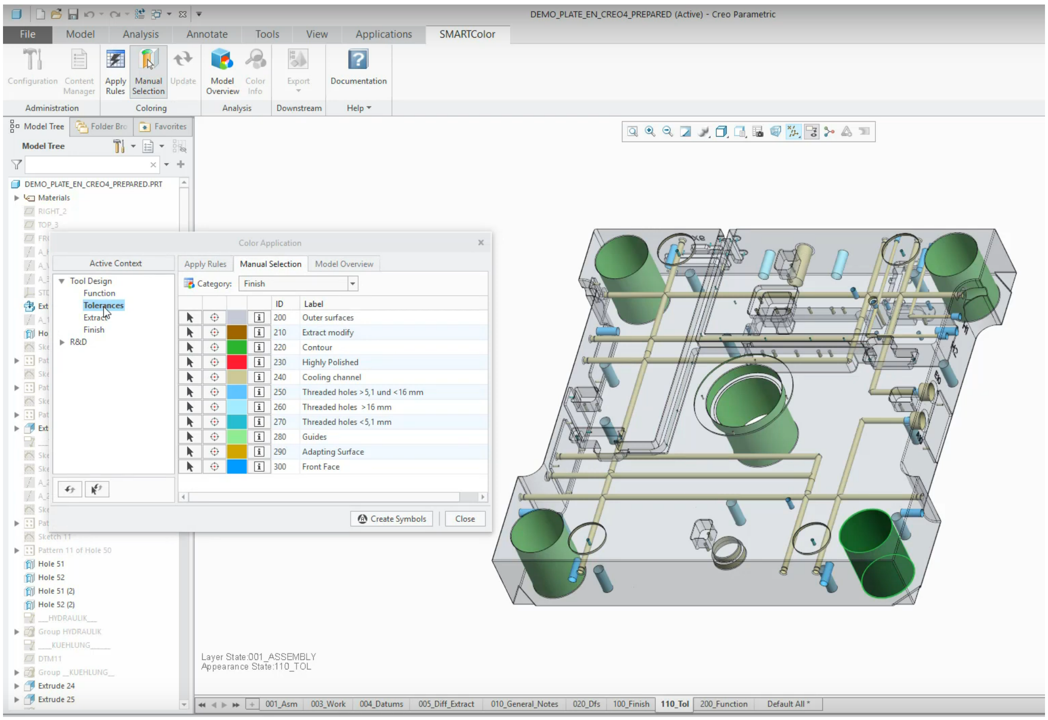


Figure 7.3: Example of the use of colours to represent a particular tolerance with SmartColor® in PTC Creo (©b&w software)

When the number of tolerances applied to a model is limited (Jack 2013 states a good design contains as few tolerances as possible) the use of colour in the MBD model makes it easier to see where each tolerance has been applied. The German

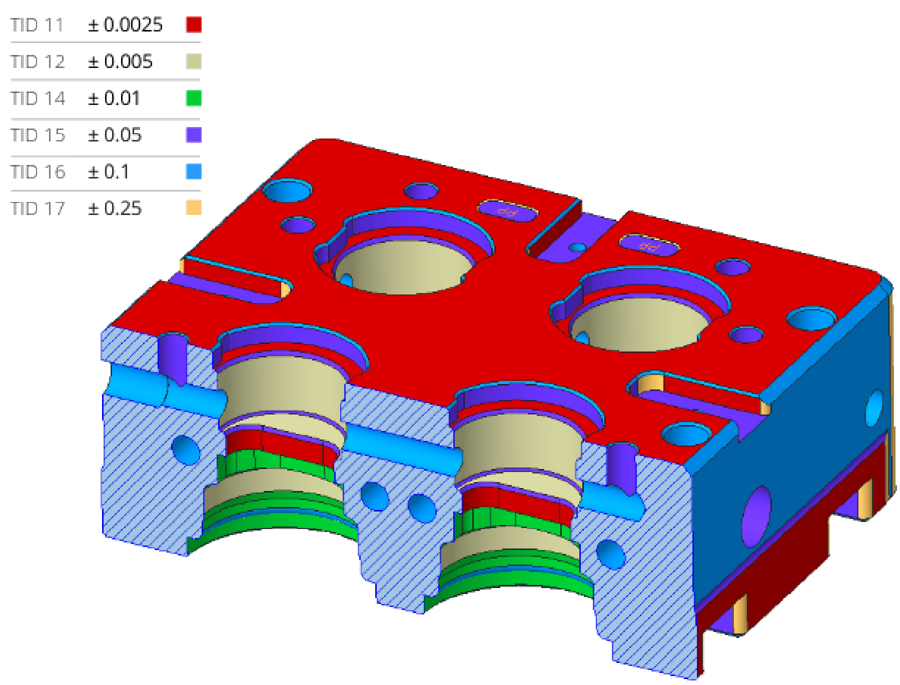


Figure 7.4: Example of the use of colours to represent a particular tolerance with ColorCoding® in PTC Creo (©Software Factory)

company Software Factory claims that the use of colours in an MBD model can reduce NC programming up to more than 70% (see screenshot from Twitter in Figure 7.5).

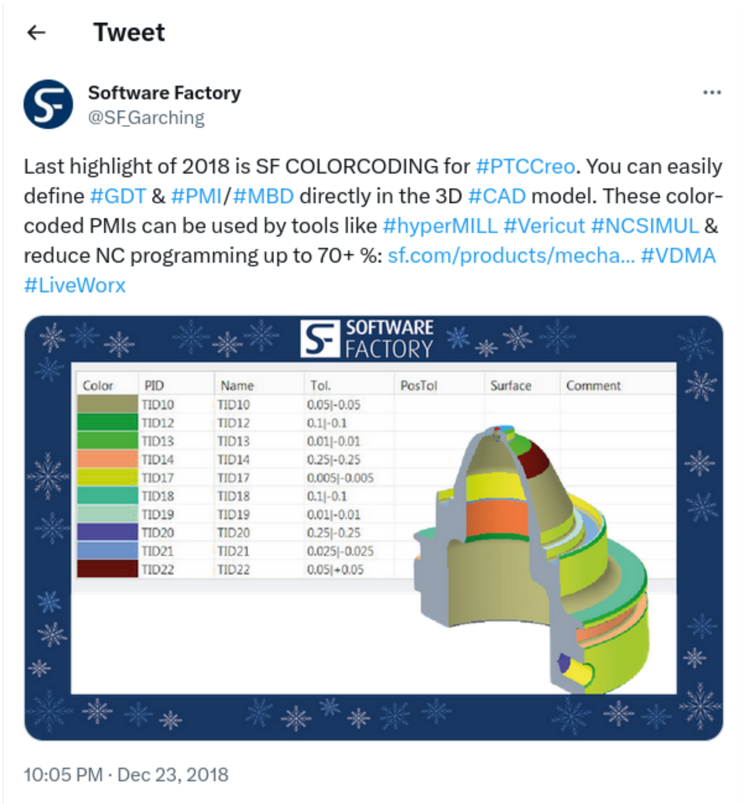
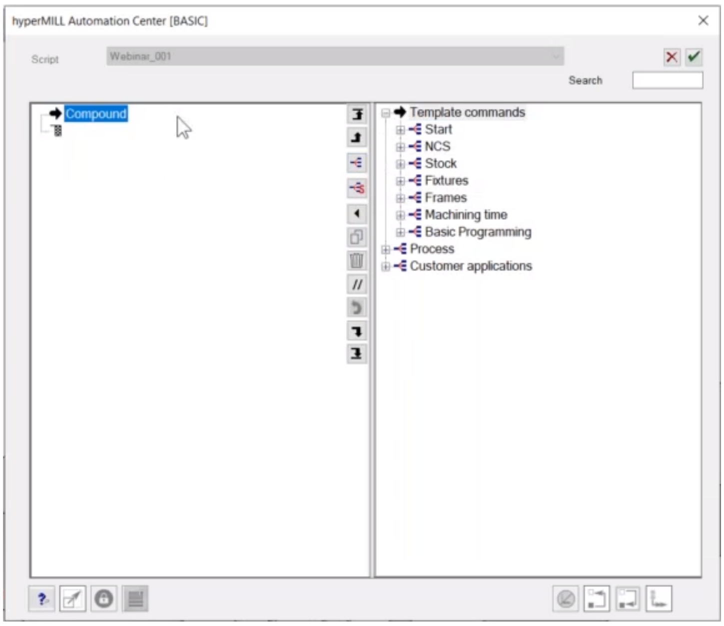
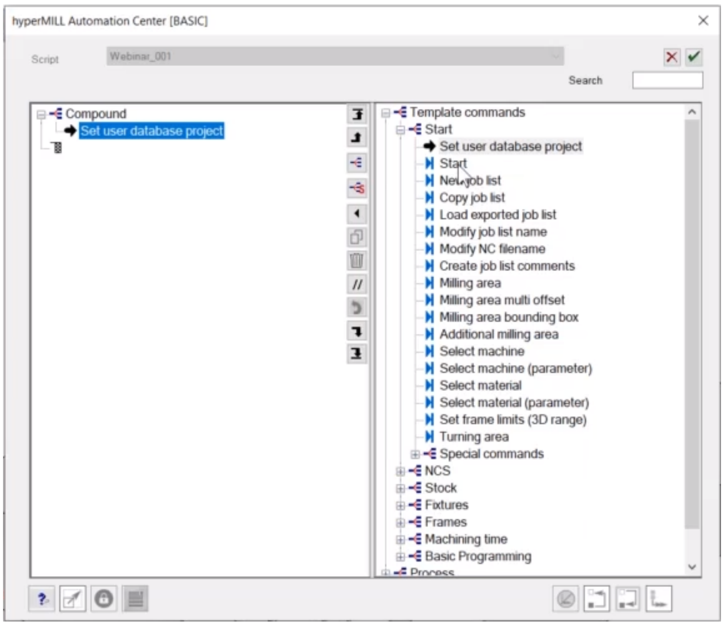


Figure 7.5: A publicity tweet by the company Software Factory claiming the use of colours in an MBD model can reduce NC programming up to more than 70%

In one of their customer's white paper "IGS GeboJagema *Model Based Definition: Nice-to-have or must-have for an automated, flawless industry 4.0 production environment*" it is stated that the objective is to have 75% of the CNC programming automated (Doorakkers 2019). Software Factory's tweet and the website of their customer IGS GeboJagema mention the HyperMill CAM package. It is therefore plausible to assume that this is the CAM package used to achieve the automations described in the white paper. HyperMill is a CAM package developed by Open Mind Technology. The package has extensive automation capabilities through the *Automation Center* module. This makes it possible to (semi-)automatically create toolpaths based on feature recognition (such as holes, stepped holes, open and closed pockets), select surfaces based on colour, select macros based on selected colour and much more (OPEN MIND Technologies 2019).



(a)



(b)

Figure 7.6: Screenshots of some of the dialogues in HyperMill's Automation Center module to give an idea of what can be included in the automation of toolpath generation

In the white paper it says 'The FEM should be 100% nominal, which means no unsymmetrical tolerances, but only +/- tolerances can be used.'. FEM is an acronym for Functional Engineering Model. This is the 3D CAD file. The white paper also explains that the LTE team prepares the workflow for the different manufacturing departments to produce the FEM, in this case the different parts of a mould assembly. LTE stands for Logistic Technical Engineering. The different steps of the workflow are captured in MBD models derived from the FEM. This way of working produces good results. There are several reasons for this. The first reason is that it is always the same type of product, namely injection moulds, where a lot of things are common between the different moulds and therefore recurring. This can be taken into account when automating toolpath generation. A second and perhaps the most important reason is that both the design and the manufacture of the designed parts are done in the same company. This means that design and production are very well aligned and it is clear to the manufacturing department what the design department means, which can eliminate the need to communicate the purpose of a product through asymmetric tolerances. It is a very different story when the design and manufacture of the parts are done by different companies that do not have a close relationship with each other. At that point, everything is determined by the annotations made in the MBD model. As already described in section 7.2, the lack of asymmetric tolerances, such as those described in the ISO System of Limits and Fits, makes it more difficult for the manufacturing department to find out how the part will be used. The latter refers to whether the part will fit into or receive another part. The lack of annotations as defined in the ISO system of limits and fits also makes it more difficult to find out whether standardised tools can be used.

As mentioned above, only symmetric tolerances are allowed in the MBD model in this white paper. What if asymmetric tolerances are still used in the MBD model? Consider the example in Figure 7.7 which shows a part with a rectangular opening with a width of $100^{+0.2}_{0}$. The blue colour indicates that an asymmetric tolerance ${}^{+0.2}_{0}$ should be applied to the coloured surfaces.

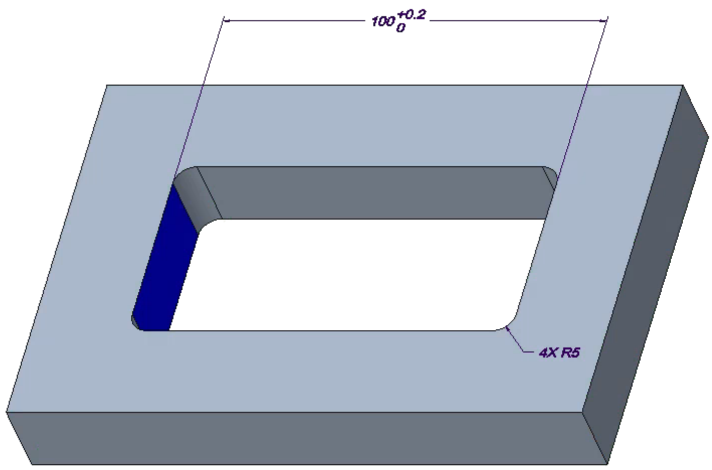
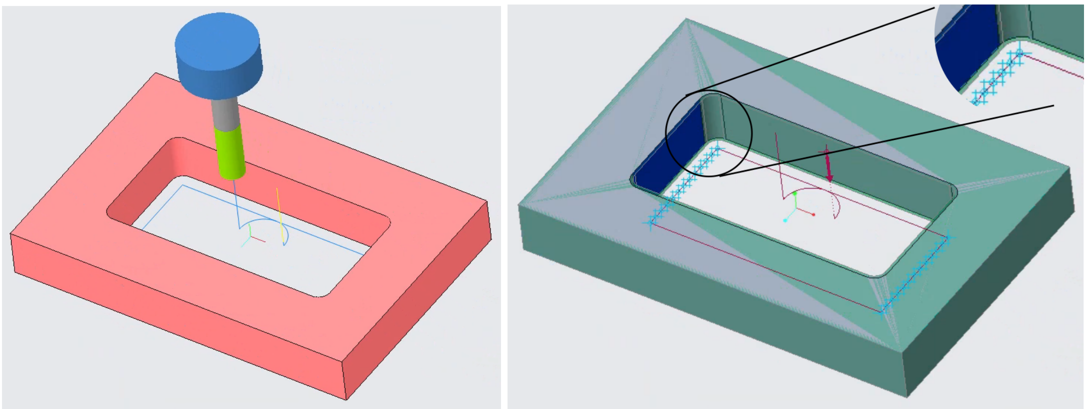


Figure 7.7: The colour blue is assigned to an asymmetric tolerance of ${}^{+0.2}_{0}$

The blue colour can be used by a macro in a CAM package such as HyperMill to automatically generate a +0.1 offset for the CNC toolpath running along the coloured surfaces. On first inspection, this seems to work as well as the method described in the white paper. The difference is in the simulation of the generated CNC toolpaths. In the use case described in the white paper, the CAD model can be used directly for simulation. This is because only symmetrical tolerances have been used. The nominal dimensions of the model correspond to the centre of the different tolerance fields. This is not the case in the example shown in Figure 7.7. In this case, the nominal size 100 is not the centre of the tolerance field. It should be changed to $100 + \frac{0.2-0}{2} = 100.1$. The CAM package cannot do this automatically. As a result, the path offset by 0.1 is simulated on the original nominal model. This will result in the simulation showing an undercut or damage to the part that is not there in reality.



(a) Simulation of the finishing toolpath in PTC Creo

(b) Gouge check of the finishing toolpath. The blue crosses indicate the place where gouging occurs.

Figure 7.8: Screenshots of the simulation in PTC Creo of the finishing toolpath

None of the elements of the philosophy described in the white paper

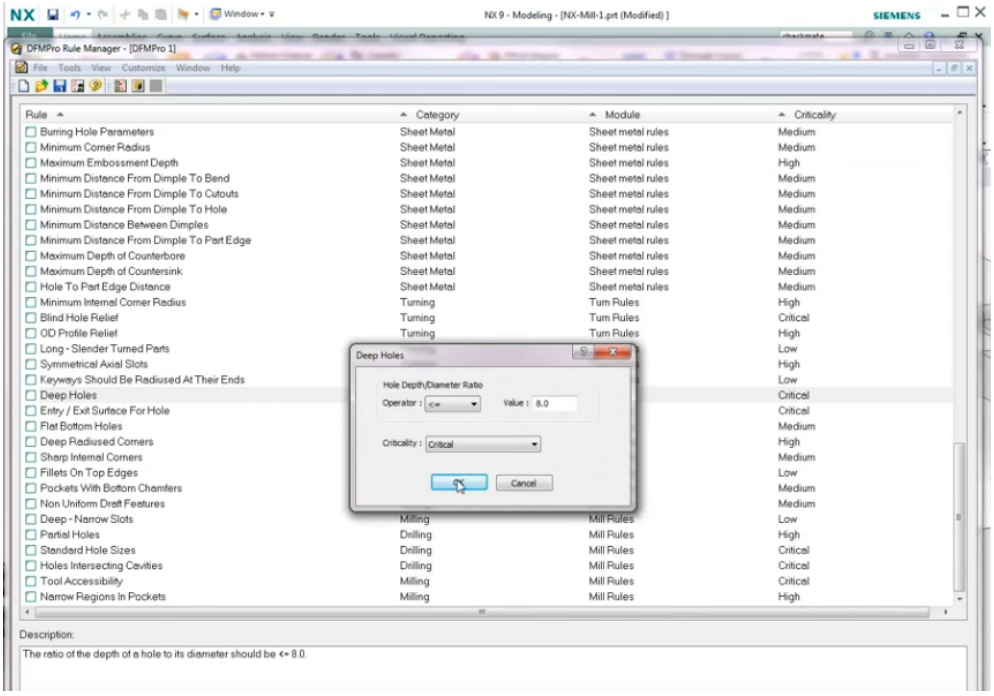
- Prohibition of the use of asymmetric tolerances
- The use of colours to indicate the symmetric tolerance used and the semantic references, namely the surfaces to which this symmetric tolerances applies.

are mandatory in the description of MBD. It can therefore be concluded that this has more to do with optimising internal communication between departments than with the application of a general MBD philosophy.

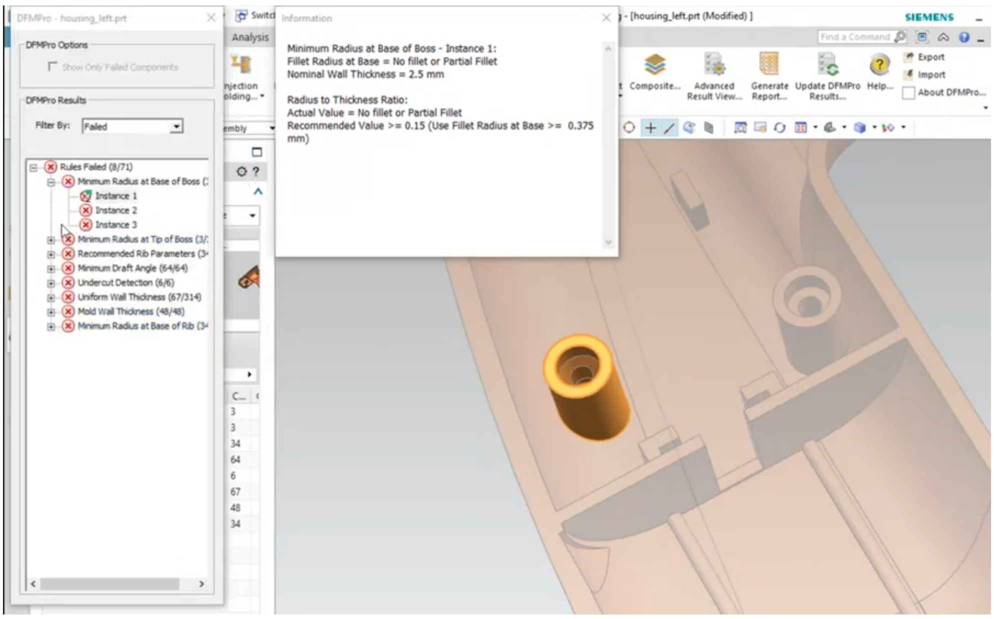
7.4 Design for Manufacturing

In Jack 2013, Design for Manufacturing, abbreviated DfM, is defined as 'a collection of rules and approaches for making parts easier to manufacture at lower costs'. Giachetti 1999 states about DfM 'the objective is that by considering manufacturing early in the design process, the design can be favorably influenced to improve quality, reduce cost, and decrease time-to-market'. In Ullmann 2003, professor David Ullman defines DfM as 'establishing the shape of components to allow for efficient, high-quality manufacture'. DfM can therefore be seen as a philosophy, a way of working, in which the manufacturability of the product and the constraints of the manufacturing process are taken into account during the design process. There are software packages available to help the designer incorporate this into the design, such as DFMPro. DFMPro is developed by HCL Technologies as a plug-in for CATIA v5, Creo Parametric, Siemens NX and SolidWorks. It acts as a kind of expert system, checking the design against a long list of rules (see Figure 7.9a). The designer can accommodate this to insure the design is more manufacturable (see Figure 7.9b).

In some webinars and seminars on MBD that were attended during this PhD study, it is sometimes said that MBD enables automatic toolpath generation. This is then referred to as Design for Manufacturing. The example used to prove this is the automatic detection of holes to be drilled and/or tapped. This has nothing to do with MBD as such. It is a property of the hole feature (Figure 7.10) within a particular CAD/CAM system.



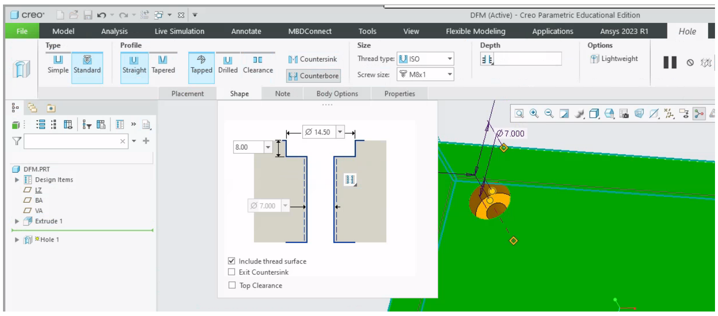
(a) Overview of some of the rules implemented in DFMPro



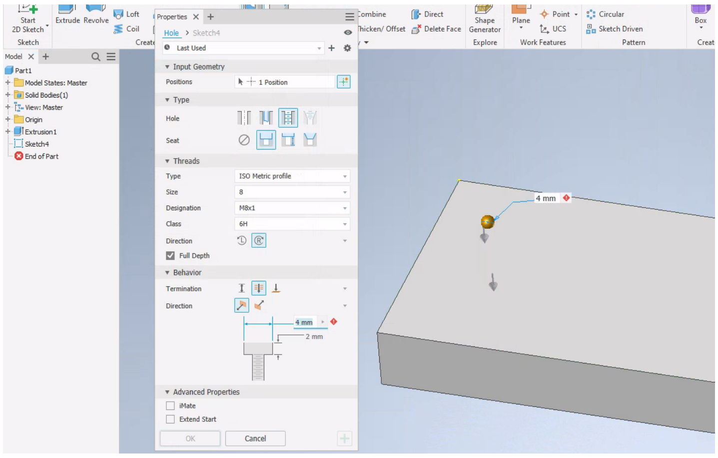
(b) An example of the type of advice DFMPro can provide. In this case , the figure shows the recommendations for a boss in an injection moulded product.

Figure 7.9: DFMPro plug-in module for Siemens NX

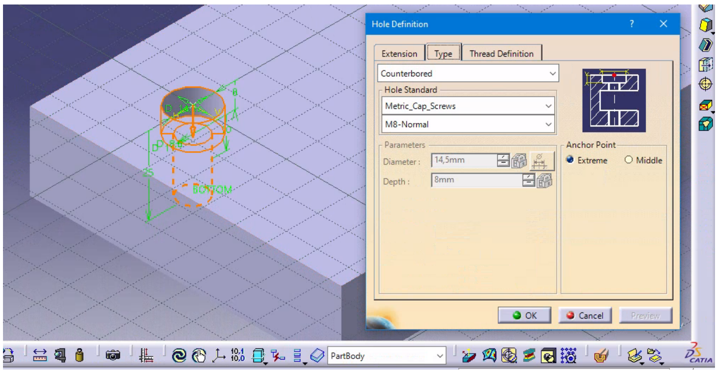
These features are recognised by the native CAM system, which can interrogate the native CAD model. This is possible because the CAD model behaves like a database, where the features used to build the model are records in that database and the feature parameters are the fields in those records. By querying these fields, the hole parameters (Figure 7.11) can be used by the CAM system to identify and generate the drilling and tapping operations.



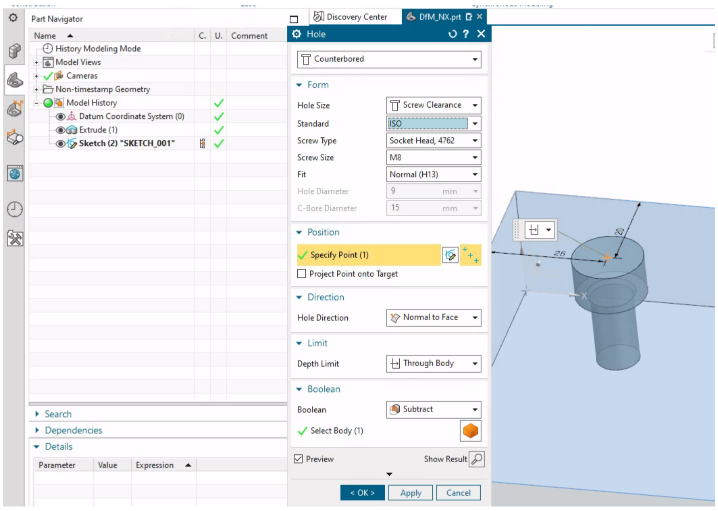
(a) The hole feature in PTC Creo



(b) The hole feature in Autodesk Inventor

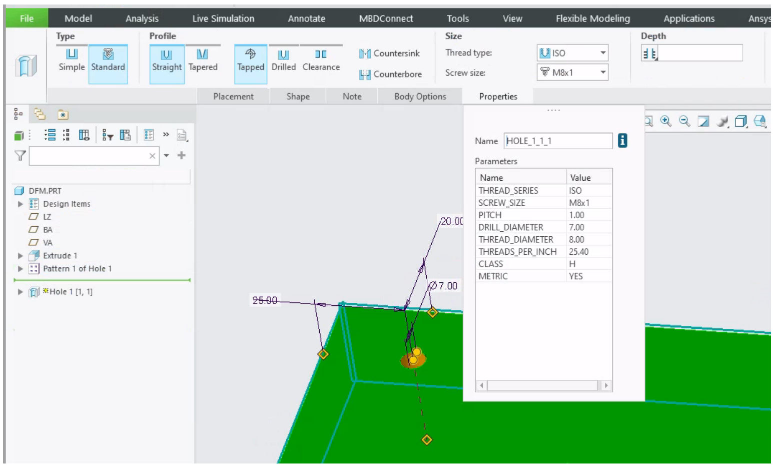


(c) The hole feature in CATIA v5



(d) The hole feature in Siemens NX

Figure 7.10: Examples of the use of the hole feature in Creo, Inventor, CATIA and NX



Name	Value
THREAD_SERIES	ISO
SCREW_SIZE	M8x1
PITCH	1.00
DRILL_DIAMETER	7.00
THREAD_DIAMETER	8.00
THREADS_PER_INCH	25.40
CLASS	H
METRIC	YES

Figure 7.11: The parameters defined within the hole feature of PTC Creo

Some CAD systems allow the user to link a macro or template to a feature. One such CAD system is PTC Creo, where a manufacturing template can be linked to a feature. One possible application of this is to link a spot drilling operation, a hole drilling operation, . . . to a hole feature. This allows a company to create its own library of standardised holes with corresponding CNC operations. When these are used by the designer to create holes in the design, the required CNC programme is generated at the same time.

Because everything related to drilling and tapping in the CAM module is linked one-to-one with the hole feature, the ability to automatically generate CNC toolpaths is lost when the CAD model is exported to a neutral exchange format such as STEP AP242. This is because only the topology, namely the surfaces used to describe the geometry of the model, is stored in the STEP AP242 file. The relationship between the surfaces, such as the surfaces used to build the hole, is gone. All the CAM system can do is recognise the holes and infer their diameter based on the cylindrical surfaces present in the STEP model. Even the term "cylindrical surface" is confusing here. This is because in most cases it refers not only to the shape but also to the mathematical description of the surface. In fact, a cylindrical surface can be described in two ways: as an analytical cylindrical surface or as a NURBS-based surface. A method to describe a circle using four splines is described in Jankauskas 2010. Most CAD systems can only recognise analytical cylindrical surfaces as holes (see Figure 7.12). On first inspection, this may seem like a far-fetched problem, as it would seem that the designer had to deliberately define a cylinder with splines in order to have this problem. However, the splines do not have to be created by the designer, they could be the result of a previous operation.

If a CAD model contains threaded holes, the problem is exacerbated when this model is exported to a format like STEP AP242. There are several reasons for this. The first reason is that threads are not included in the various STEP standards currently used in CAD/CAM systems: STEP AP203, STEP AP214 and STEP AP242. The reason for this is that with standard threads, the shape of the thread is not determined by the CAD geometry, but by the tool used to make the thread, the tapping tool, or the standard cycle of a CNC lathe. Since each CAD system has its own way of indicating that a hole feature contains threads (Cheney et al. 2015), this also affects how these threads are exported to STEP.

Some CAD systems export the thread as a separate cylindrical surface around the hole with the core diameter. This is the case for Creo Parametric (see Figure 7.13).

When the exported STEP file is imported by other CAD systems or by other CAD systems or by PTC Creo itself, this separate cylindrical surface is useless. It cannot be

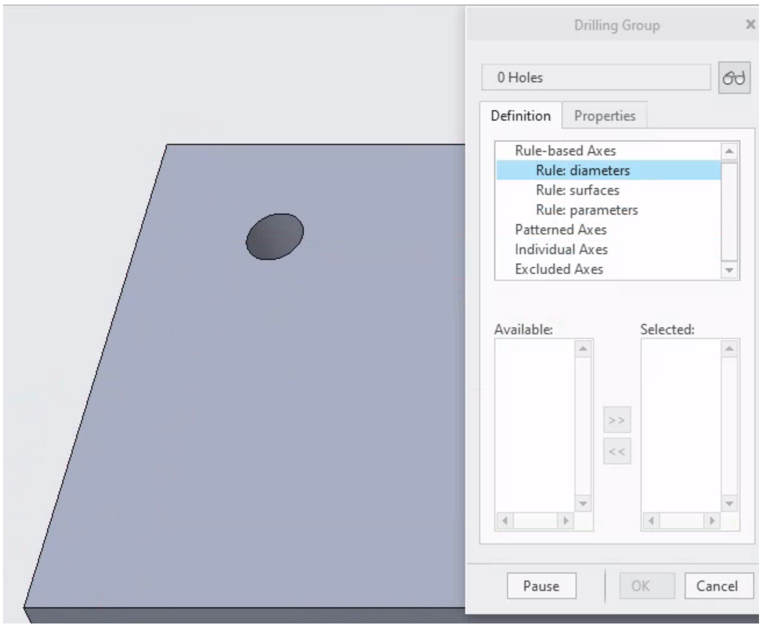


Figure 7.12: Creo doesn't recognise the hole defined by splines, no diameters listed under Available

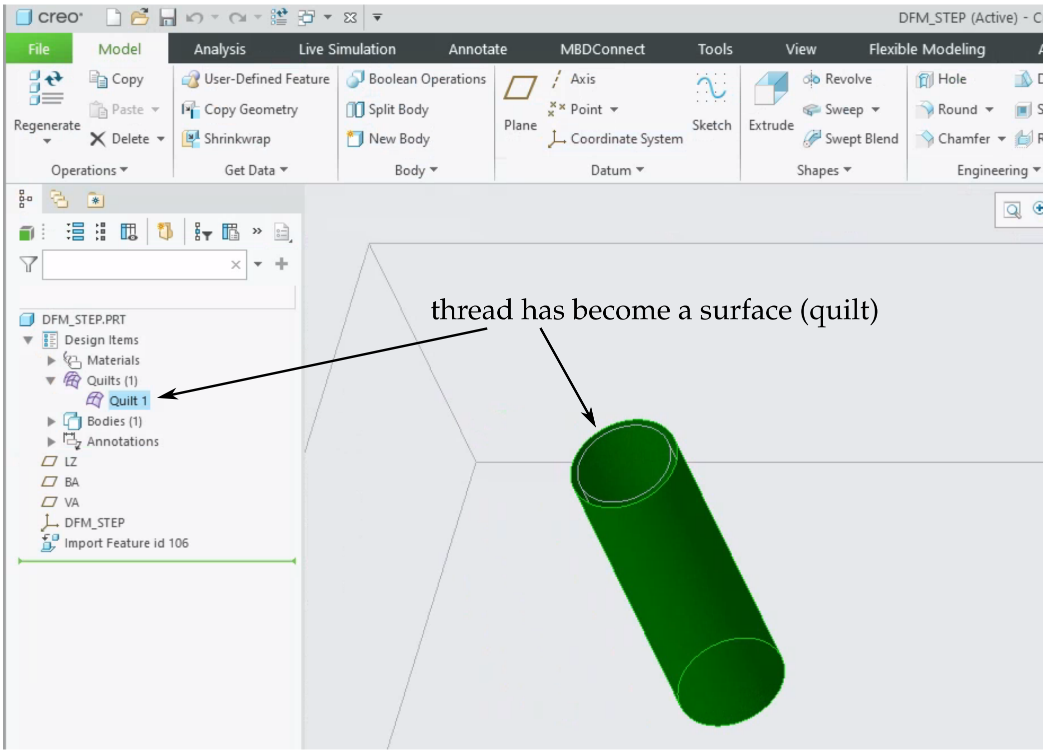


Figure 7.13: The thread of a hole is converted to a surface by Creo when exported to STEP. The green surface is the thread surface selected in the imported STEP file. The feature tree identifies this as a quilt which is a surface.

recognised as a thread and will be interpreted by a number of CAD systems, including SolidWorks, as an error in the imported model.

Other CAD systems only export the hole with the core diameter. An example of such a CAD system is Autodesk Inventor. Th value of the core diameter is determined differently by each CAD system. For some CAD systems, it is the diameter of the hole to be pre-drilled before tapping. For other CAD systems, it is the core diameter of the thread.

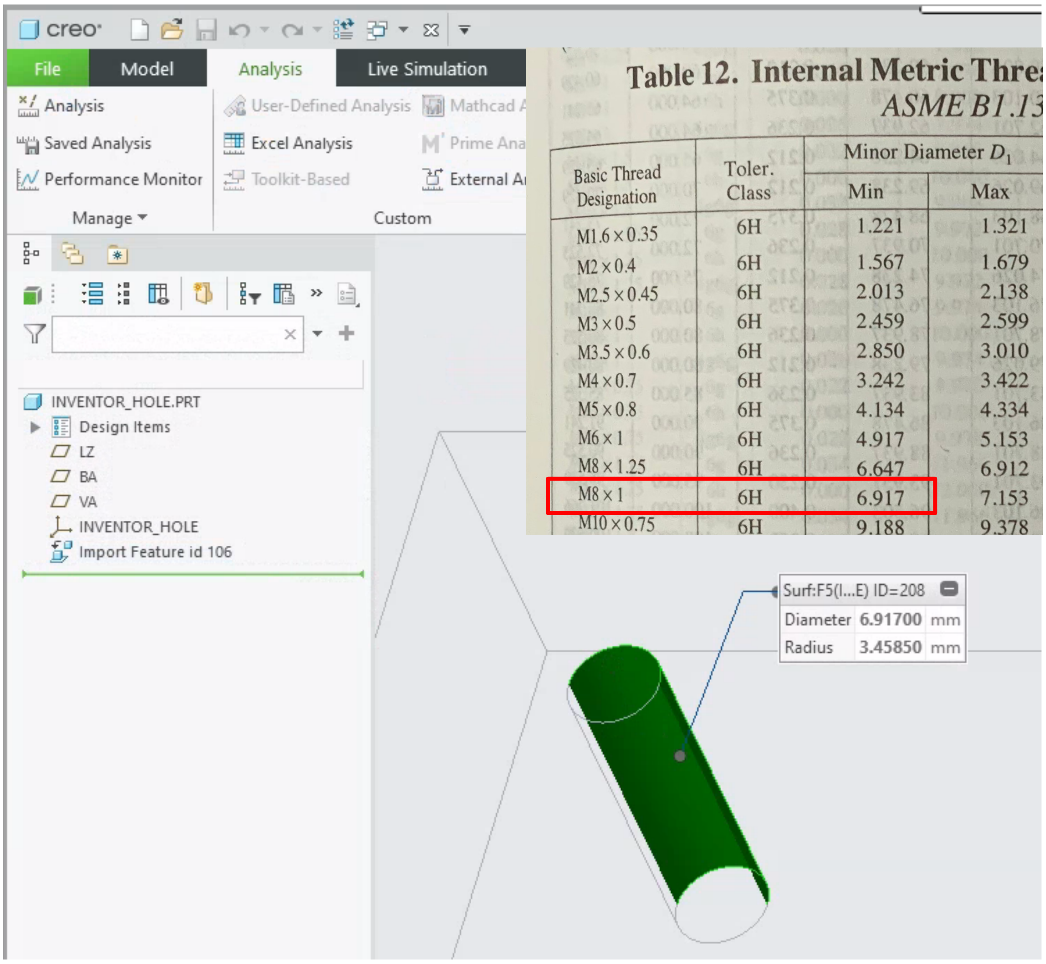


Figure 7.14: The thread of a hole is converted by Inventor to a hole with the minor thread diameter when exported to STEP.

In most cases, thread depth information is lost when the model is exported to STEP. The advantage of "Design for Manufacturing" in automatically recognising holes with or without threads is therefore lost if features cannot be used while retaining them. The mention of retaining features is important here because many CAD systems advertise the ability to read native formats from other CAD systems, but do not mention that in most cases this is only about transferring the topology, not the features. The ability to read a native format while retaining only the topology is therefore not enough to take advantage of this so-called "Design for Manufacturing" functionality.

One way of being independent of how the tapped hole was created, and thus overcoming the loss of feature-related information when exporting to STEP, is to annotate the tapped hole. This annotation or PMI should be machine-readable according to the MBD philosophy. There are several reasons why annotating a tapped hole may not be an adequate solution at present.

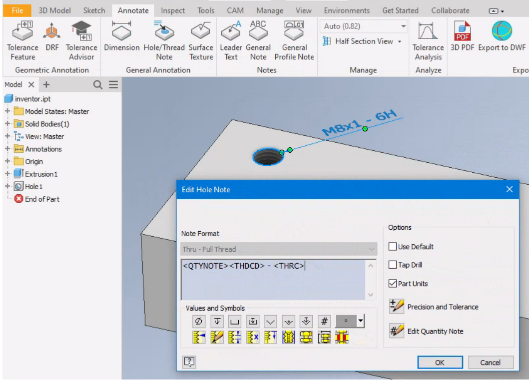
A first reason is that simply being machine-readable is not enough. Machine-readable means that the annotation has been assigned parameters that can be read by a CAD/CAM system. However, each CAD/CAM system gives different values to the same parameters. This makes it difficult to fully utilise the information contained in an MBD model by a CAD/CAM system other than the one in which the model was created. The holes in Figure 7.15 are M8 × 1 holes drilled and tapped throughout the model. The annotations were made in each CAD system using standard auto-generation of the annotation with default settings. They were then exported to STEP AP242 and imported into PTC Creo Parametric where the contents of the parameters associated with the hole annotation was examined.

A second reason is that the way the annotation is created affects the export to STEP. As a result, the annotation is machine-readable in the original CAD system and is not machine-readable after export to STEP. This has been discussed in chapter 3.

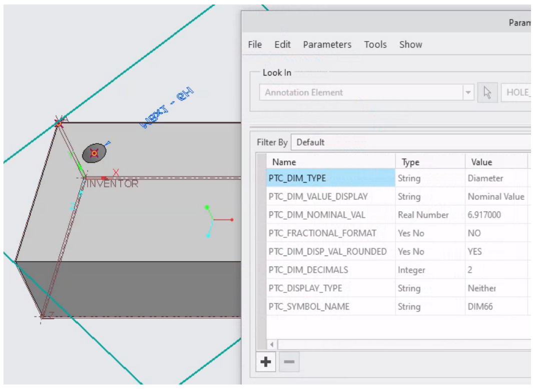
A third reason is closely related to the second and concerns semantic references. Semantic references are the geometric entities to which an annotation refers. It is possible that the designer has overlooked to specify them. This is particularly common when the annotation refers to multiple holes. Some CAD systems such as Siemens NX have special modules[1] that can help to prevent this. Another possibility is that they were lost during the export to STEP.

A fourth reason is that although there is an ISO and an ASME standard that defines how annotations should be formatted, not everyone follows it. Some examples are shown in Figure 7.16 and Figure 7.17. This shows several screenshots of YouTube videos from companies showing different ways of indicating how many holes the annotation refers to. Companies often have their own standards. This makes it difficult to work with other stakeholders. This is especially the case if the standards used are not explicitly shared and they do not use the same CAD package.

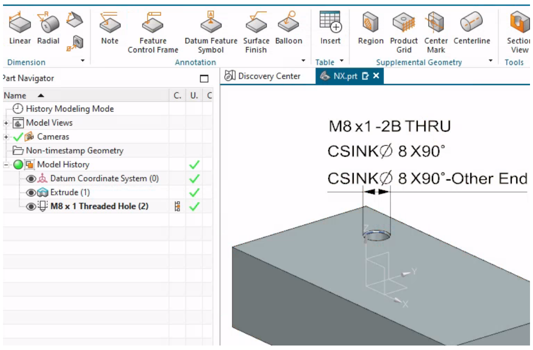
[1] The Siemens module can be seen in action in this video https://www.youtube.com/watch?v=JD_CcBKYiKQ



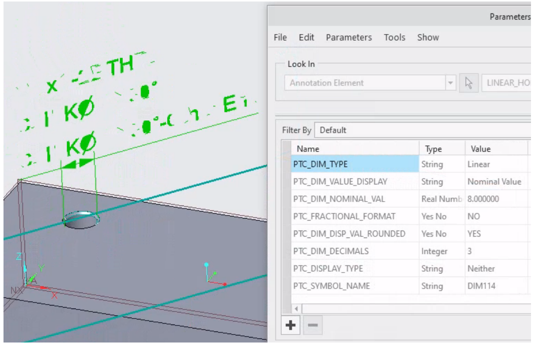
(a) Hole M8 × 1 created in Inventor and annotated with default settings



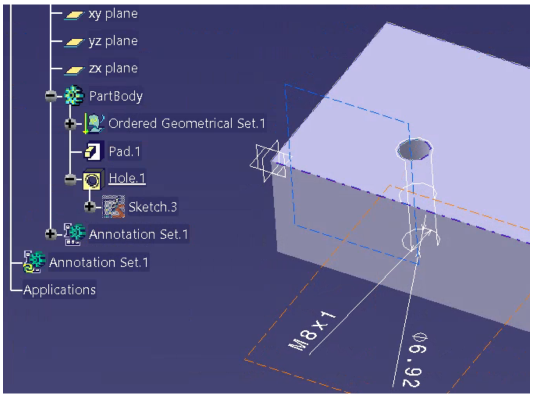
(b) Inventor STEP file imported into Creo and the parameters associated with hole annotation examined



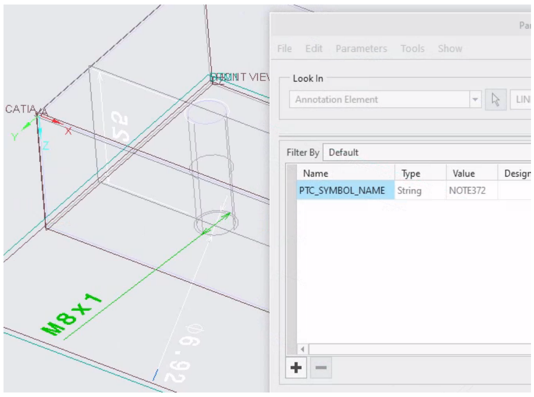
(c) Hole M8 × 1 created in NX and annotated with default settings



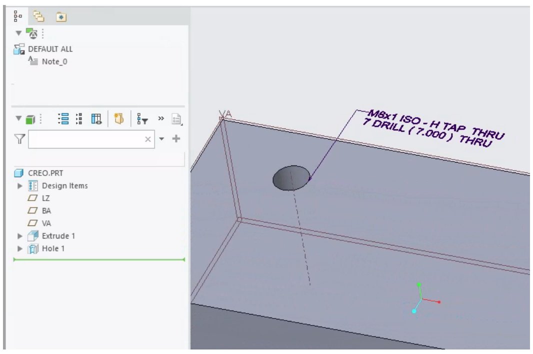
(d) NX STEP file imported into Creo and parameters associated with hole annotation examined



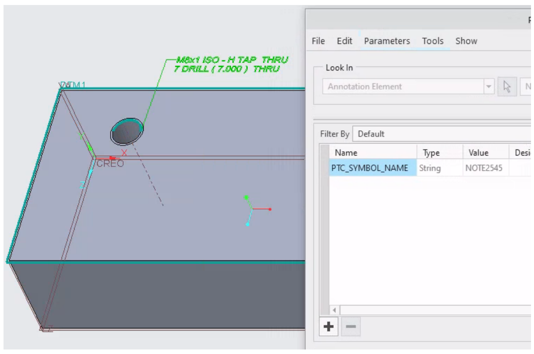
(e) Hole M8 × 1 created in CATIA v5 and annotated with default settings



(f) CATIA STEP file imported into Creo and parameters associated with hole annotation examined

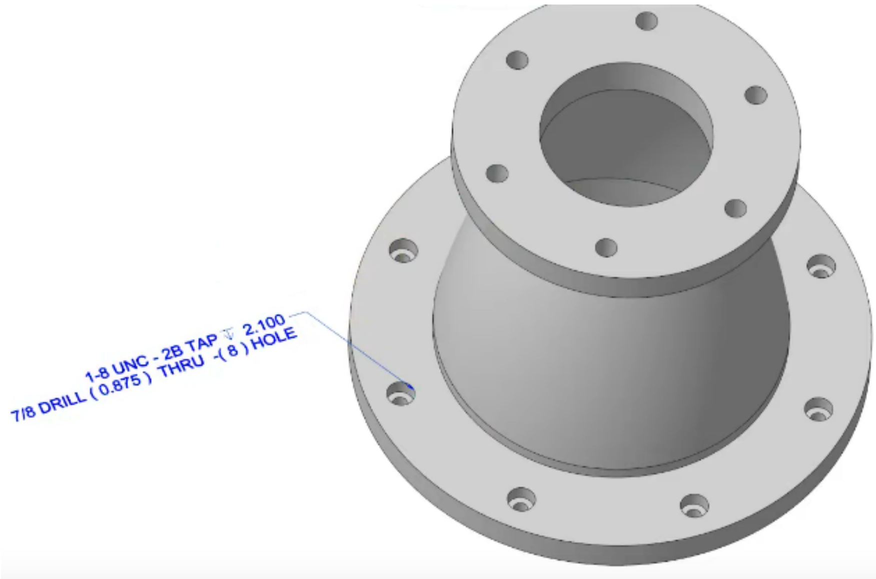


(g) Hole M8 × 1 created in Creo and annotated with default settings

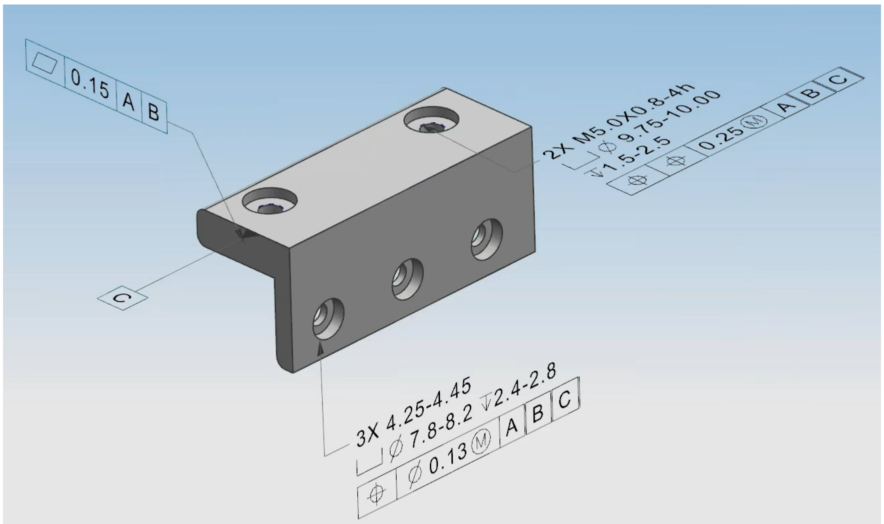


(h) Creo STEP file imported into Creo and parameters associated with hole annotation examined

Figure 7.15: Holes M8 × 1 created in 4 CAD systems and exported to STEP AP242

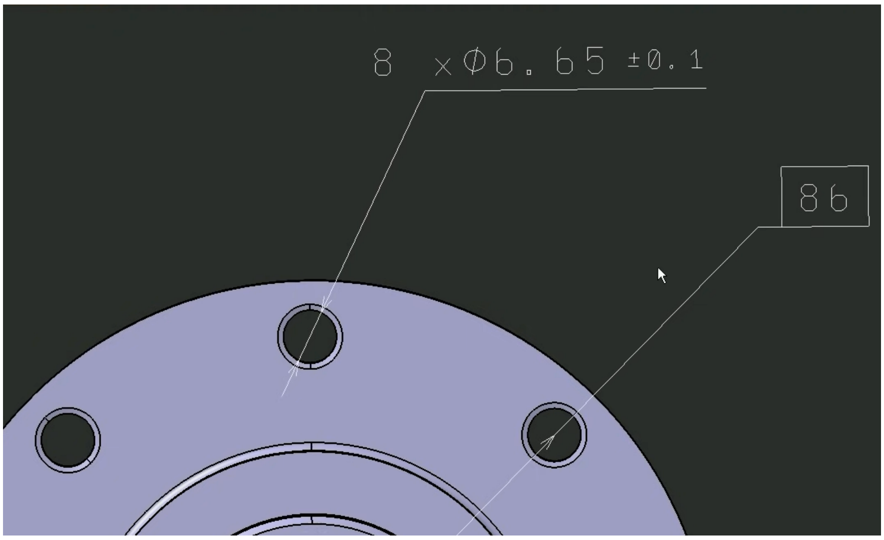


(a) Number of holes indicated as: (8) HOLE
Screenshot from https://www.youtube.com/watch?v=Hpz03Hqrb04

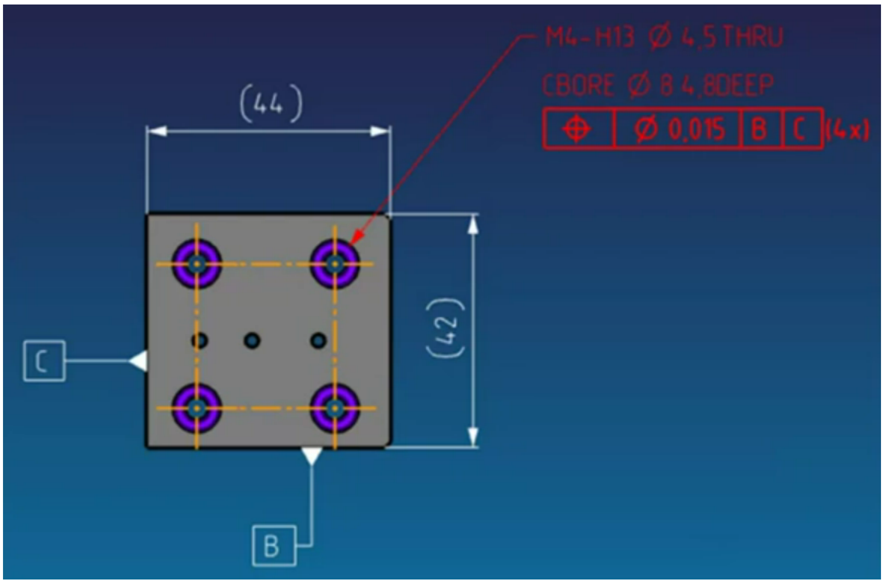


(b) Number of holes indicated as: 2X and 3X
This is in accordance with ASME Y14.5
Screenshot from https://www.youtube.com/watch?v=AWyilGFq0GI

Figure 7.16: Examples of how the number of holes is displayed in the MBD model



(a) Number of holes indicated as: 8 x
Screenshot from https://www.youtube.com/watch?v=WQUjodi7Izs&t=388s



(b) Number of holes indicated as: (4x)
Screenshot from https://www.youtube.com/watch?v=UFScx4QmnZA

Figure 7.17: Additional examples of how the number of holes is displayed in the MBD model

7.5 Conclusion

The three approaches discussed in this chapter are used to avoid some problems. However, they also create new ones.

The first approach, which avoids the use of asymmetric tolerances, solves the difficulty of having to modify the geometry of the model. This is a difficulty because the geometry has to be modified taking into account all the existing tolerances. There are two possible situations that can arise. The first possibility is that the geometry has to be modified on the native model. If the designer has applied functional dimensioning in the features used to create the model, then it is relatively easy to modify the model. Many CAD systems have special modules for this purpose. An example of this is the "Dimension Boundaries" function in PTC Creo (see Figure 7.18). If the designer has not applied functional dimensioning in the features used to create the model, then modifying the model is a time-consuming task. A second possibility is that the model is supplied in a neutral exchange format such as STEP. In this case, the difficulties are of the same nature as with a native model that has not been built with functional dimensioning in mind.

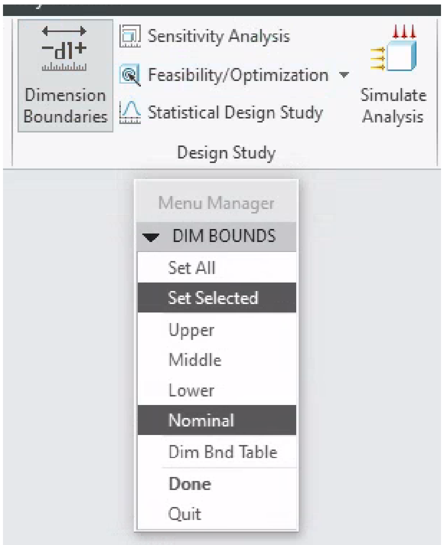


Figure 7.18: The "Dimension Boundaries" module in PTC Creo makes it possible to modify the nominal values of the model to the centre of the assigned tolerance fields

The consequence of avoiding asymmetric tolerances is that there must be very good agreements with the people who have to manufacture the design. They must be familiar with the designer's way of working. This is possible when design and manufacturing are in the same company or always involve the same partners. Whereas with asymmetric tolerances it is clear how the design will be used or assembled, this is no longer the case. Avoiding a standard such as the ISO System of Limits and Fits also ignores the fact that many tools such as reamers and inspection tools such as gauges are standardised for use with the ISO System of Limits and Fits. This can lead to additional difficulties if production has to be outsourced.

The second approach, working with colour-coded tolerances, makes it easier to identify areas to which the same tolerance fields have been assigned. As noted in this chapter, this often follows the first approach, which avoids the use of asymmetric tolerances. This avoids two difficulties. The first difficulty that is avoided is the need to match the geometry to the centre of the tolerance field. This makes manufacturing both easier and more difficult. Easier because the geometry does not need to be adjusted. More difficult because, as already indicated as a consequence of the first approach, it is no longer clear how the part will be used. This has an impact on the milling tools and the inspection tools. A second difficulty that is avoided is the occurrence of false undercuts. If asymmetric tolerances are used and the geometry is not adjusted and the colour codes are only used to automatically create offset toolpaths, the simulation of these toolpaths will be done on the original nominal model and will show undercuts. The question then arises as to what is a real undercut and what is a false undercut. Using colour-coded tolerances has the same consequences as the first approach. There must be robust agreements between the designer and those who have to manufacture the design. This is possible if the design and manufacturing departments are in the same company or always work with the same partners.

As for the third approach, the concept of "Design for Manufacturing", abbreviated as DfM, is a universally applicable philosophy when it comes to considering how a product should or can be made when designing a product. The situation is very different when it comes to feature recognition for the automatic generation of CNC toolpaths. In this case, two conditions must be met if a company wants to apply "Design for Manufacturing" in this sense. The first is that the CAD/CAM system used must be able to reuse CAD features such as holes and tapped holes in the CAM module. A second condition is that the designer must also effectively apply the CAD features that can be reused in the CAM module, such as hole features, with or without macros and/or templates to optimise their use, in the design. When neutral exchange formats such as STEP are used, these properties are largely lost. As a result, if DfM is seen as reusing CAD features to automatically generate CNC toolpaths, it can only be done if all stakeholders are using the same CAD/CAM package. This is necessary both to avoid errors in complex geometries such as double curved surfaces (see chapter 6) and to enable adequate feature recognition. To enable adequate feature recognition when using neutral exchange formats such as STEP, support for threaded holes should be added to the standard. Currently, the transfer of threaded holes in STEP is not facilitated. This means that if a threaded hole is exported without an associated PMI annotation, the information about the thread (type, specification, depth) is completely lost.

8 Software development

8.1 Introduction

Two research questions of this PhD study are

- How are PMI annotations defined in current exchange standards and implemented and used in major CAD/CAM/CAE packages with regard to the MBD philosophy?
- How can the current PMI definitions be enhanced to facilitate automated use in later stages of the manufacturing process?

With regard to the first research question, the previous chapters have covered how PMI annotations are defined in current exchange standards and how they are used in major CAD/CAM/CAE packages. This chapter looks in more detail at how they are implemented in a CAD/CAM/CAE package.

With regard to the second research question, it was investigated whether a software package could be developed that could

- read the data of a representation PMI annotation (linear dimension with an asymmetric tolerance) in a STEP AP242 file
- determine the semantic references
- adjust the nominal value of the CAD geometry to the centre of the asymmetric tolerance field.

The ultimate goal was to adapt the CAD geometry so that it could be used directly to generate CNC toolpaths using a CAM package. The aim was to develop this software package as independent of CAD systems as possible. It was investigated whether it was possible to develop the software using kernels as Open Cascade and ACIS. When this proved unsuccessful, it was decided to develop this software package as a plug-in for PTC Creo Parametric using the ProToolkit software library and read the data of a representation PMI annotation within a native PTC Creo file and within a STEP AP242 file. This chapter takes a closer look at how PMI annotations are implemented in the APIs of the programming libraries of a CAD system such as PTC Creo.

8.2 Open Cascade

Open Cascade is an open source 3D CAD kernel. Originally called CAS.CADE, it was developed by the French company Matra Datavision, who used it in the CAD/CAM package they developed, Euclid. After Euclid was sold to the French company Dassault, the developer of the CAD/CAM package CATIA, it became open source in 1994. The name was then changed to Open Cascade.

The idea of using Open Cascade was motivated by the fact that it is free. The development of the software started in 2018. According to the manual of the then Open Cascade version 7.3.0, only STEP AP203 and STEP AP214 are supported. Basic support for STEP AP242 has been introduced in recent releases[1]. This would mean that only the geometric data in a STEP file could be read and visualised, but not the PMI annotations. To test this, a simple STEP viewer was developed using the Open Cascade libraries to handle the STEP format and wxWidgets toolkit to create a GUI. The open source 3D CAD package FreeCAD was used for additional testing. The choice of FreeCAD was determined by the fact that FreeCAD itself uses Open Cascade as the internal geometry processing engine.

A test model with a linear dimension created as a representation PMI annotation was modelled in PTC Creo Parametric 7.0.4.0 and exported to a STEP AP242 file (Figure 8.1). The resulting STEP AP242 file is imported and visualised in the STEP viewer that was developed using Open Cascade and wxWidgets (Figure 8.2).

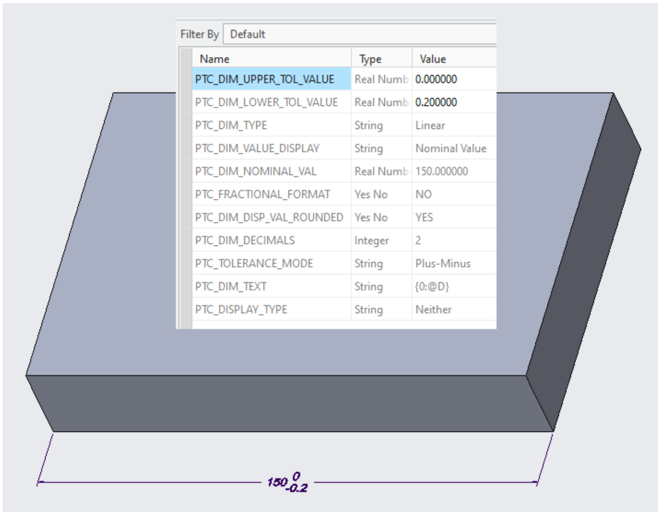


Figure 8.1: Test model with a linear dimension created as a representation PMI annotation modelled in PTC Creo 7.0.4.0 and exported to STEP AP242

The image in the viewer (Figure 8.2) shows that the geometry has been imported. However, the 3D annotation has disappeared.

STEP is an encapsulated format, which means that different versions build on each other (Figure 8.3).

[1] On 19 August 2023, the online user guide for Open Cascade 7.7.0 states that some parts of AP242 are supported (Open CASCADE Technology 2023).

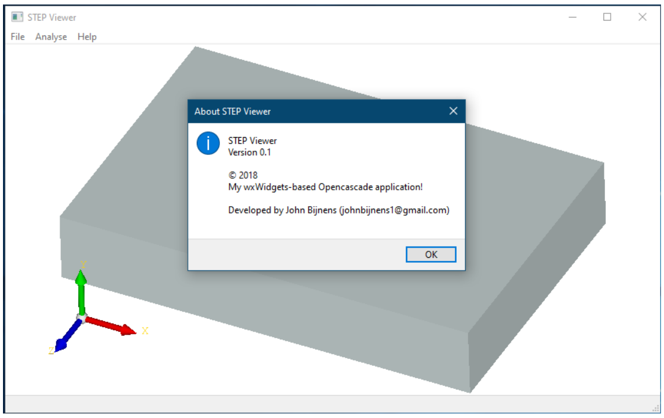


Figure 8.2: STEP AP242 file created by PTC Creo imported into the STEP viewer developed using Open Cascade and wxWidgets

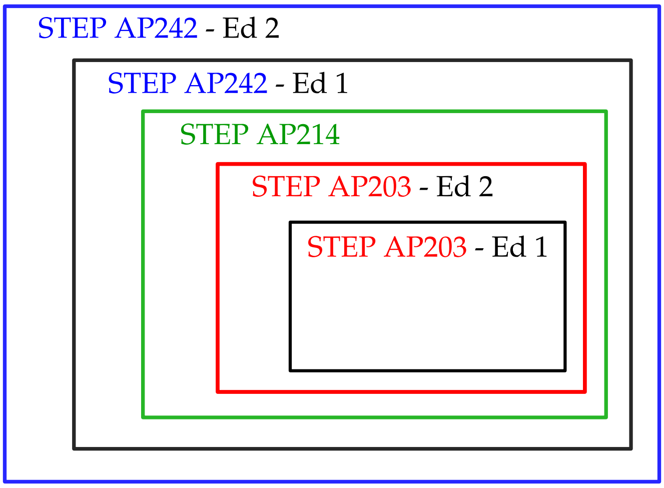


Figure 8.3: Every STEP variant builds on the previous one

A STEP file is described using a data modelling language, EXPRESS, which is defined in the ISO 10303-21 standard. Although EXPRESS is originally more of a full programming language and XML is more of a markup language, there is a reasonable similarity between the two (Peak et al. 2004). As a result, the processing of a STEP file by a CAD/CAM package can be compared to the processing of an XML file. This means that known elements can be read and unknown elements can be skipped. In this way, a package that does not support AP242 specific elements, such as representation PMI annotations, can still read the geometric data described in AP203 and AP214, while skipping the AP242 specific elements. In theory, it should therefore be possible, on the one hand, to read the geometry of an MBD model exported in STEP AP242 into a CAD system that does not support STEP AP242, or does not support it fully, and, on the other hand, to read out the PMI representation contained in the STEP AP242 file and link it to the topological elements of the geometry, using software developed specifically for this purpose. The model in Figure 8.1 is used to test this assumption. For this test, this model was exported to STEP AP242. Figure 8.4 shows a mapping of the relevant parts of the exported STEP AP242 file in relation to the 3D annotation:

Parameter #293 contains the definition of the linear dimension with a value of 150 mm.

Parameter #301 specifies how the associated tolerance is defined. In this case it is a PLUS_MINUS_TOLERANCE.

Parameter #300	indicates which parameters define the lower and upper limits of the tolerance range. In this case they are parameters #298, which has a value of -0.2, and #299, which has a value of 0.
Parameter #297	indicates which parameters define the semantic references to which the linear mapping refers. In this case they are parameters #295 and #296. These two parameters in turn refer to two other parameters that contain the id of the geometric entities between which the linear dimension lies, namely #1085 and #1087.
Parameter #1085	refers to plane #176.
Parameter #1087	refers to plane #148

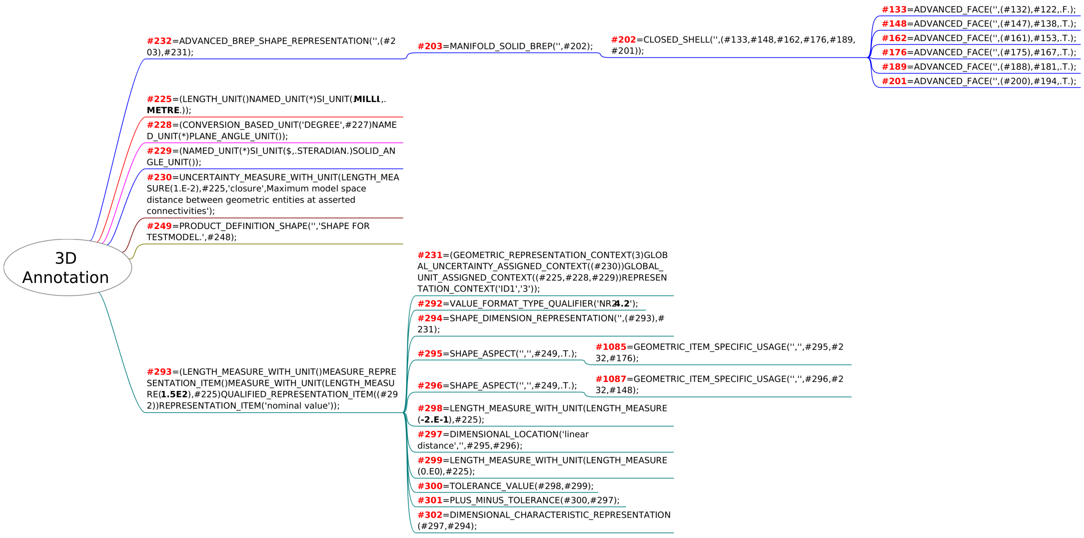


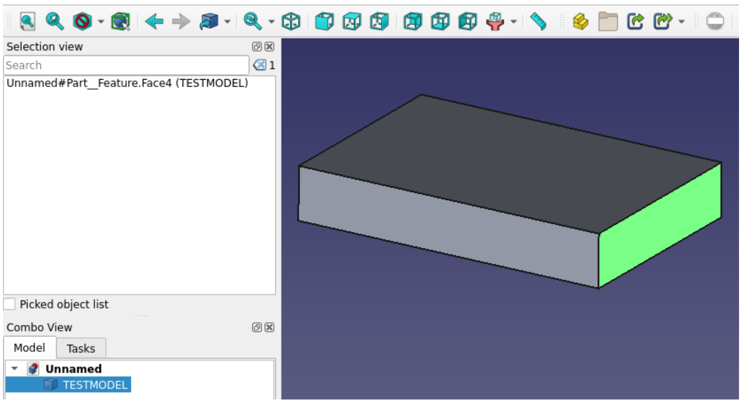
Figure 8.4: Mapping of the relevant parts in the STEP file related to the 3D annotation

The following procedure was used for the test:

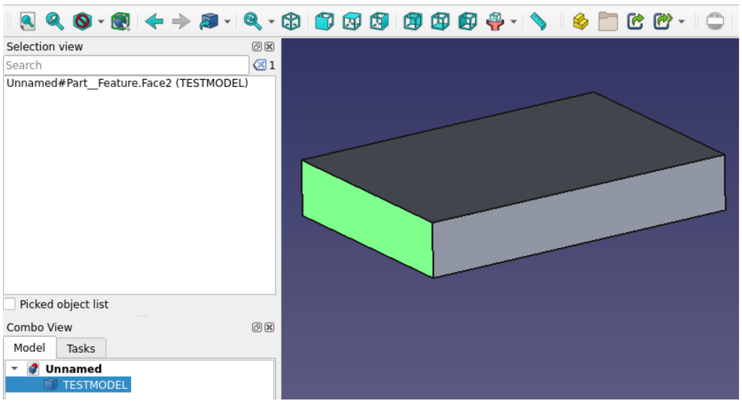
1. Import the STEP FILE into FreeCAD
2. Parse the STEP file using separate software:
   - Detect the PMI in the CAD model
   - Determine the semantic references of that PMI
3. Link the semantic references to topological elements (surfaces, edges, . . .) in the FreeCAD model

Step 3 of this procedure could not be performed successfully. This is because the topological elements of the CAD model do not have the same IDs as in the STEP model after import. As described above, the linear dimension in the test model refers to two surfaces identified in the STEP file with IDs 176 and 148. After importing a STEP file into FreeCAD, all topological elements (points, curves, surfaces) of the CAD model are renumbered and given new IDs (Figure 8.5). As a result, all original IDs are lost. So the original IDs 176 and 148 of the two semantic references of the linear dimension no longer exist.

It is therefore no longer possible to identify the semantic references to which the linear dimension refers. This in turn makes it impossible to adjust the geometry, such as changing a nominal size to the centre of the assigned asymmetric tolerance field. It was investigated whether it was possible to add a parameter to each topological element that would take the original id as a value. In this way, the semantic references could be found again in the model imported by OpenCascade. It turned out that this could only be done by reading the STEP file element by element and recreating it



(a) The semantic reference that originally had ID #176 now has a new ID Face4



(b) The semantic reference that originally had ID #148 now has a new ID Face2

Figure 8.5: After importing the STEP file, the semantic references have new IDs

in OpenCascade itself. This amounts to the development of a completely new STEP interface. For this reason, this route was no longer pursued.

8.3 ACIS

As the original aim was to develop a software package independent of existing CAD systems, other CAD kernels were sought. ACIS is a commercial 3D CAD kernel developed by Spatial Corporation, part of Dassault Systemes. Unlike OpenCascade, ACIS has full support for STEP AP242. Discussions about obtaining the necessary software libraries dragged on too long. It was also unclear what the final cost would be. As this PhD is entirely self-funded, this route was not pursued further.

8.4 PTC Creo Parametric

As it was not possible to develop a software package independent of an existing CAD package using OpenCascade and ACIS, the decision was made to develop a package on top of an existing CAD package. The CAD package had to meet the following conditions:

- Full support for STEP AP242
- API available for programming an application on top of the CAD package
- Be affordable as this PhD is entirely self-funded.

On this basis, the PTC Creo Parametric CAD package was selected. There are 6 different levels at which a user can develop a script or software package. These are:

- Mapkeys
- Relations, Family tables, User Defined Features (UDF)
- ProProgram
- Web.Link
- J-Link
- Creo Toolkit

Each of these levels has its own capabilities and limitations that define its scope.

8.4.1 Mapkeys

"Mapkeys" is a function that allows the designer to store keystrokes and mouse clicks in a macro. This macro can be assigned to a specific key combination (see Figure 8.6). This allows repetitive operations to be performed very quickly. Mapkeys are strongly linked to a specific version of Creo Parametric. If changes have been made to the GUI of the package with a new version of Creo Parametric, this may result in a mapkey no longer working.

8.4.2 Relations, Family tables, User-Defined Features (UDF)

Relations

"Relations" allow the designer to define relationships between dimensions using formulas. These can be dimensions that

- belong to the sketch of a particular feature
- are used to create one particular feature

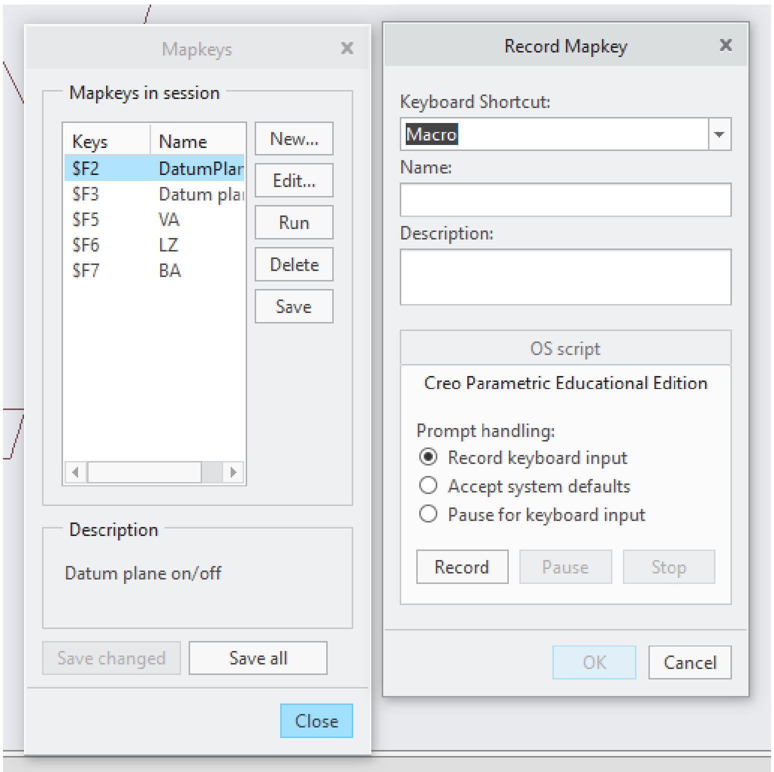


Figure 8.6: Options available when creating a new mapkey

- belong to several different features
- are used when assembling parts and, for example, determine the distance between parts
- ...

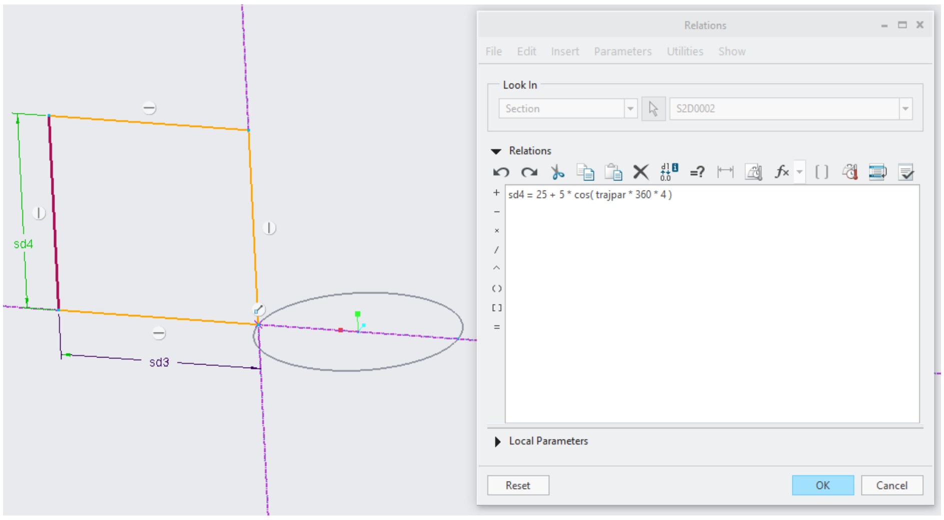
There is a very limited ability to use conditional statements. This is limited to an IF - (ELSE) - ENDIF construction (PTC 2023e). Figure 8.7 shows an example of applying a relation using the dimensions of a particular feature.

Family tables

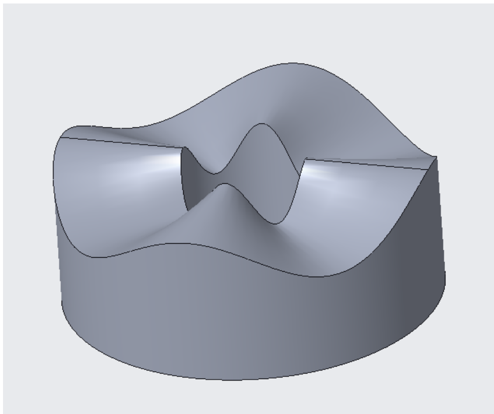
As the name suggests, the "Family Tables" module allows designers to quickly generate a series of related components. These parts, known as a "family", are generated by creating a table of the component's dimensions and features, which may or may not be included (PTC 2023b). Relations can also be utilised within this process. An example of creating a family table is shown in Figure 8.8.

User-Defined Features

The "User-Defined Features" module allows the designer to create new features by combining existing features into a new feature. This new feature can be called like any other feature (extrude, revolve, ...) where the designer can enter the desired number values and options (PTC 2023d). Relations and UDFs can also be used here. An example of an application is the creation of a new feature to automate the application of a boss design (see Figure 8.9) to a plastic injection moulded product.



(a) A relation created in a sweep feature using dimensions from different levels of the feature



(b) The end result of the relation applied to the sweep feature

Figure 8.7: A relation using the dimensions of a particular feature

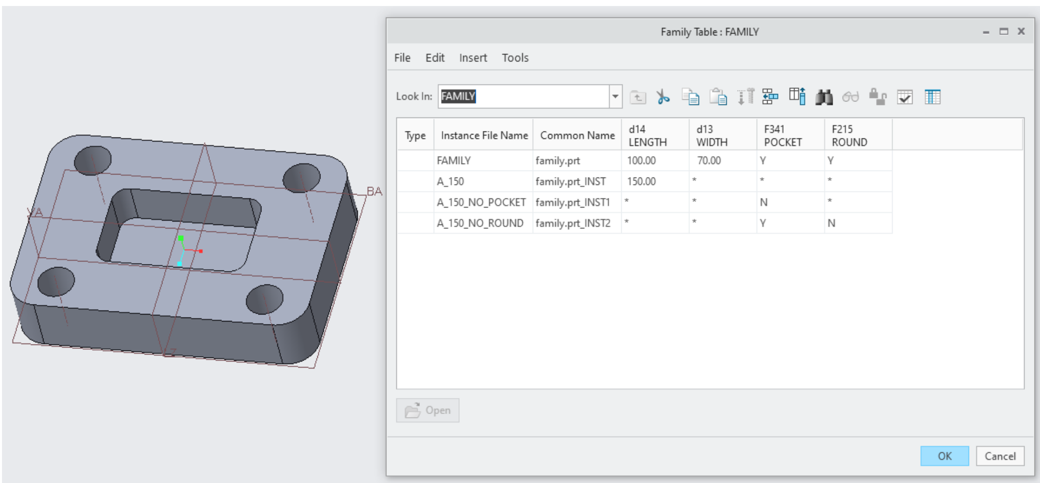


Figure 8.8: Dialogue box displayed when creating a family table

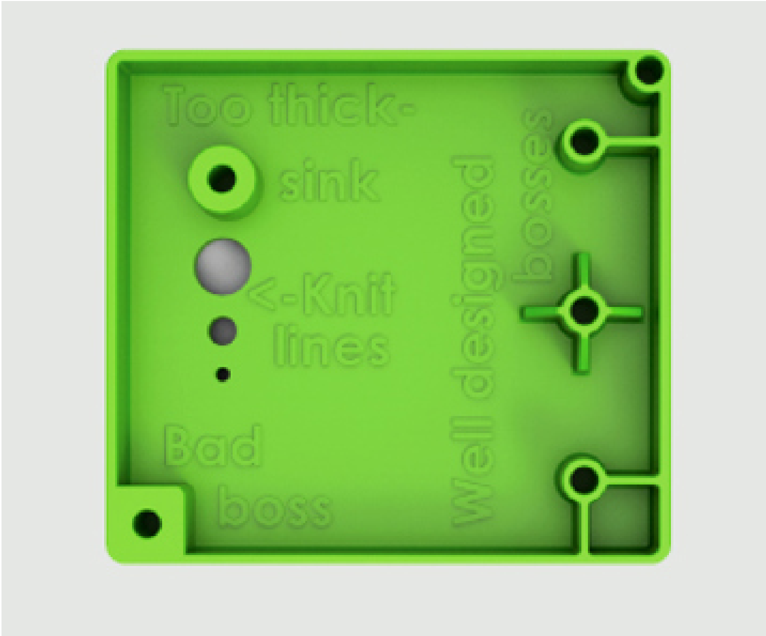


Figure 8.9: Top view of a plastic product with multiple boss designs (round, square, with and without support)

8.4.3 Web.Link

This is a library that allows web applications to interact with Creo Parametric. This means that a developer can create a GUI via a web page, where the communication can take place via JavaScript. The prerequisite is that the web page is accessed via Creo Parametric's internal web browser (EACPDS 2017). Web.Link allows objects in the CAD model to be manipulated from a JavaScript program. This includes modifying or deleting objects. However, it is not possible to create objects directly (PTC Inc. 2021).

Based on the information in the user manual, it was concluded that not only was it not possible to create objects directly in the CAD model, but also not possible to retrieve all the desired information from 3D annotations, such as semantic references, among others. Therefore, Web.Link was not considered suitable for developing the software to read the PMI annotations present in the CAD model and to modify the geometry based on the information from these PMI annotations.

8.4.4 J-Link

This is a library that allows Java applications to interact with Creo Parametric. There are two major differences with Web.Link. The first difference is that a developer can choose how to develop the GUI. It is no longer necessary to do this via a web page in Creo Parametric's internal web browser. A second difference is that the Java application can run synchronously or asynchronously. Synchronous means that when the application is called from Creo, it must first be closed before work can continue in Creo. Asynchronous means that when the application is called from Creo, work can be done both in the Java application and in Creo. Since PTC Creo Parametric 4.0, J-link has been called Object Toolkit Java (PTC 2023f). Besides J-Link or Object Toolkit Java, there is also VB API. This is a library that allows you to use Visual Basic instead of Java.

Based on the information in the user manual, it was concluded that there were severe limitations to creating objects directly in the CAD model. Therefore, J-Link or Object Toolkit Java and VB API were not considered suitable for developing the software to read the PMI annotations present in the CAD model and to modify the geometry based on the information from these PMI annotations.

8.4.5 Creo Toolkit

This is a library whose APIs provide relatively low-level access to the CAD model in PTC Creo Parametric using the C programming language. By relatively low-level, it is meant that these APIs provide deeper access to the CAD system compared to Web.Link and J-Link, but they are not the underlying layer of the CAD system itself, rather an additional layer on top of it. As with J-Link, Toolkit applications can run synchronously or asynchronously.

Based on the information in the user manual, it was concluded that there should be no restrictions on creating objects directly in the CAD model and that it should be possible to retrieve any desired information from 3D annotations, such as values and semantic references. It was therefore decided to use the Creo Toolkit and to develop the software for this PhD study in C.

PTC uses a license server to control access to the different software modules. The license server holds a license file containing codes that unlock access to a particular module. The list of available modules includes the Toolkit module (see Figure 8.10). Early in the development of the software, it became clear that the toolkit license alone was not sufficient to write the software. In fact, the Toolkit license does not allow the use of the APIs needed to query the 3D annotations in a CAD model. This requires a separate license, Toolkit-for-3D_Drawings.

```
==================================================
Configured Option Modules
==================================================

The following Creo Parametric options are installed:
(Note: license extensions are marked with a '*')

3AXIS_MOLD_TOOLING             Expert_Machinist               NC-CHECK
4DNavigator                    Expert_Moldbase                NC-MILL
ADDITIVE_MANUFACTURING         FEATURE                        NC-SHEETMETAL
ADVANCED_RENDER_2              FEM-POST                       NC-TURN
AE                             Flexible_Modeling              NC-WEDM
ANIMATE                        Freestyle_Design_Extension     NOTEBOOK
ASSEMBLY                       GDT_ADVISOR                    ObjectToolkitCpp
Advanced_Framework             GDT_ENTERPRISE                 Options_Modeler
Advanced_Rendering_III         Generative_Design              Options_Modeler_Basic
Behavioral_Modeler             HARNESS-MFG                    PHOTORENDER
Build_Process_Interface        INTERFACE                      PIPING
CABLING                        INTF-CADDS5                    PLOT
CASTING                        INTF-CADDS5                    PROCESS_ASM
CDT                            INTF_for_STEP                  PROCESS_MFG
CFD_BASIC                      Import_Data_Doctor             PRODUCT_INSIGHT
CMM                            Interactive_Surface_Design     PTC_Creo_Unite
COMPOSITE                      Intralink_8_0                  PlasticAdvisor
CREO_ANSYS                     LEGACY                         Progressive_Die
CREO_SIMULATION_LIVE           LIBRARYACCESS                  REPORT
CREO_SIMULATION_LIVE_FLUID     Layout_3D_Integration          Reverse_Engineering
Clearance_Creepage             Layout_Authoring               SCAN_TOOLS
DATA_for_PDGS                  Legacy_Migration               SHEETMTL-DES
DETAIL                         MOLDDESIGN-2                   SURFACE
DIAGRAM                        MOLD_ANALYSIS_Lite             TOOLKIT
DIEFACE                        Manikin                        TOPOLOGY_OPT_ADVANCE
Design_Animation               Manikin_Analysis               Update_Control
Design_Exploration             Mechanism_Design               VERIFY
ECAD                           Mechanism_Dynamics             WELDING
ECAD_COLLABORATION             ModelCHECK                     Web.Link
EZ_TOLERANCE_ANALYSIS          Mold_Analysis                  Web.Publish
Education                      NC-ADVANCED
```

Figure 8.10: Overview of modules (including toolkit) available via the licence server

Launch a Toolkit application

One of the ways to call a toolkit application within Creo Parametric is through a DLL that is loaded when Creo Parametric is started. A special file with a .dat extension contains the settings that determine how the application should behave. This file must be specified in one of the config.pro configuration files loaded by Creo when

Creo is started. Figure 8.11 shows the contents of the creotk.dat file used to start the ModifyByPMI_2 application developed during this PhD study.

```
name ModifyByPMI_2
startup dll
exec_file ./ModifyByPMI_2.dll
text_dir ./text
allow_stop true
delay_start false
end
```

Figure 8.11: Contents of the creotk.dat file of the application ModifyByPMI_2

name	The name of the application that will be visible in Creo when it is registered. ModifyByPMI_2 is the name of the application developed during this PhD study.
startup	This specifies the type of file to be loaded to start the application. The type of file defines how Creo communicates with the application. In the case of a DLL, this is done by direct function calls.
exec_file	Here the full path and name of the application file is specified.
text_dir	The Toolkit User's Guide states 'This specifies the full path to the text directory that contains the language-specific directories. These directories contain the message files, menu files, resource files and UI bitmaps in the language supported by the Creo Parametric Toolkit application'.
allow_stop	If the value of this setting is set to true, this means that the application can be stopped and restarted from within Creo.
delay_start	If the value of *allow_stop* is set to true, setting the value of *delay_start* to true means that Creo will not start the Toolkit application until explicitly requested.
end	This indicates the end of the .dat file

8.5 ModifyByPMI_2 application

8.5.1 Conventions used in source code

Hungarian notation

Hungarian notation is a method of naming variables used in a program in a particular way.
The actual name of the variable starts with a capital letter and is preceded by a lower case indication of the data type stored in the variable. Some examples of naming variables in a C program:

szText[10]	A variable *szText* containing a C string of up to 10 characters, including the binary 0 that terminates the string. sz stands for string zero terminated.
pszText	A variable *pszText* containing a pointer to a C string. psz stands for pointer to string zero terminated.
dNumber	A variable *dNumber* containing a double precision floating point number.
pdDimension	A variable *pdDimension* containing an object of the Creo data type ProDimension.

The name of a global variable is preceded by g_. An example of such a variable is

g_bActivateDelayedProcessing	A global variable *g_ActivateDelayedProcessing* containing a boolean variable.

Naming of functions and procedures

In order to clearly distinguish between functions from the Creo ProToolkit libraries on the one hand and user-developed functions and procedures on the other, the names of the latter are preceded by the initials of the developer. In this case these are the letters jb. Examples of user-developed functions and procedures are:

jbCheckAsymmPMI	This is the main function which is used to launch the application. This function scans the model for 3D annotations and converts queued annotations into annotation features.
jbReadDimensionData	This function reads the data of a detected annotation and adds the annotation to the linked list.
jbReadAngularDimData	This function reads the data of a detected angular dimension and adds the annotation to the linked list.
jbFormatAngularData	This function writes the angular data into a tab formatted string

8.5.2 Structure of the ModifyByPMI_2 application

The software is divided into six main components:

1. The **launcher module**. This is the module that starts the application and builds the dialogue box.
2. The **scan module**. This module scans the CAD model for 3D annotations and stores them in a linked list.
3. The **visualisation module**. This module presents the detected annotation in a table in a clear and straightforward format. This includes highlighting a selected annotation and its semantic references in the 3D model, and indicating which annotations may cause problems when transferring the model to another stakeholder. These problems can be missing semantic references or the choice of an annotation type that causes data loss when exporting to STEP AP242.
4. The **conversion module**. This module allows the user to convert annotation types that cause data loss when exported to a STEP AP242 file to another type that avoids this data loss.
5. The **analysis module**. This module checks which dimensions have asymmetric tolerances in order to change the nominal size to the centre of the specified tolerance field if necessary.
6. The **export module**. This module allows the data structure of the detected annotations to be exported to a CSV file[1]. This can be imported into a spreadsheet for use in other applications, such as the preparation of First Article Inspection documents.

[1] CSV stands for Comma Separated Values. A CSV file is a file used to store tabular data, such as in a spreadsheet or database. The data values are stored in a line where the different values are separated by a specific character (comma, semicolon, tab, . . .).

The data structure

Before discussing the six different modules, the data structure used in all modules of the software is explained. The purpose of this data structure is:

- to store the PMI annotations that occur in an MBD model in a list
- to allow subdivision based on the type of annotation

The number of PMI annotations in an MBD model is unknown and may vary from model to model. The type of annotation is also unknown. Examples of types are linear dimension, radius, diameter, angular dimension, GD&T, note. To deal with the aforementioned unknowns, each type of annotation is stored in a data structure unique to that type. These data structures are considered objects and are stored in a linked list built with dynamically allocated memory blocks. The data structure is divided into a part common to all data structures and a part specific to a particular type of annotation. The common part is defined as a separate data structure. This data structure is included in the data structure of each annotation type and forms the beginning of the data structure of the annotation type (see Figure 8.12).

All data structures are defined as members of a union (see Figure 8.13). This means that all these data structures can be mapped to the same block of memory.

When the common data type *CommonData* (Figure 8.14) is mapped to the different memory blocks, it allows to iterate through these different memory blocks regardless of the type of data (linear dimension annotation, diameter annotation, . . .) they contain and the size they have (Figure 8.15).

AnnTypeT	iDimType
void	*prevItem
void	*nextItem
AnnInfoT	AnnInfo
char	szCombinedState[MAX_STRING_LEN]
char	szDimName[PRONAME_WIDTH]
char	szDimType[20]
char	szValueDisplay[20]
char	szDisplayType[20]
char	szTolTable[20]
int	iTolTableColumn
char	szTolType[25]
char	szUnits[15]
double	dDimNominal
int	iDimDecimals
double	dLowerTolerance
double	dUpperTolerance
ProBoolean	bFractionalFormat
ProBoolean	bDispValRounded
ProBoolean	bDimInspection
ProBoolean	bGeomModified
ProBoolean	bDimConverted2Feat
int	iNrParamsDetected

Linear dimension

AnnTypeT	iDimType
void	*prevItem
void	*nextItem
AnnInfoT	AnnInfo
char	szCombinedState[MAX_STRING_LEN]
char	szDimName[PRONAME_WIDTH]
char	szDimType[20]
char	szDisplayType[20]
char	szGtolType[20]
char	szGtolValue[50]
char	szGtolLeftText[50]
char	szGtolRightText[50]
char	szGtolTopNext[50]
char	szGtolBottomText[50]
char	szGtolPrimaryRef[50]
char	szGtolSecondaryRef[50]
char	szGtolTertiaryRef[50]
char	szUnits[15]
char	szRefComplete[4]
double	dPrimaryTol
char	szMaterialCondition[10]
char	szProjTolZone[10]
ProBoolean	pbStaticalTol
ProBoolean	pbDiameterSymbol
ProBoolean	pbFreeState
ProBoolean	pbSetBoundary
ProBoolean	pbUnordered
ProBoolean	bGeomModified
ProBoolean	bDimConverted2Feat
char	szGtol1Tol20]

geometrical tolerance

AnnTypeT	iDimType
void	*prevItem
void	*nextItem
AnnInfoT	AnnInfo
char	szCombinedState[MAX_STRING_LEN]
char	szDimName[PRONAME_WIDTH]
char	szDimType[20]
char	szRefComplete[4]
char	szNotes[MAX_NOTE_LEN]
ProBoolean	bWithLeaders
ProBoolean	bAnnotElem

Note

Figure 8.12: Some examples of data structures defined within the software.
The figure shows the definition for the linear dimension type, the gtol type and the note type. The part common to all types is indicated by the red rectangle.

```c
typedef union AnnItem
{
    AnnTypeT     AnnotationType;
    CommonDataT CommonData;
    LinDimT      LinearItem;
    ChamfDimT    ChamferItem;
    OrdDimT      OrdinateItem;
    AngDimT      AngularItem;
    DiaDimT      DiameterItem;
    RadDimT      RadiusItem;
    GTolT        GTolItem;
    SurfFinT     SurfFinItem;
    DatumsT      DatumsItem;
    NotesT       NotesItem;
    MiscT        MiscItem;
    SymbT        SymbItem;
} AnnItemT;
```

Figure 8.13: The definition of the union AnnItem which shows the different members. All the members can be mapped to the same memory block.

```c
typedef struct CommonData
{
    AnnTypeT  AnnotationType;
    void      *prevItem;
    void      *nextItem;
    AnnInfoT  AnnInfo;
    char      szCombinedState[MAX_STRING_LEN];
    char      szDimName[PRONAME_WIDTH];
} CommonDataT;
```

Figure 8.14: The definition of the CommonData structure

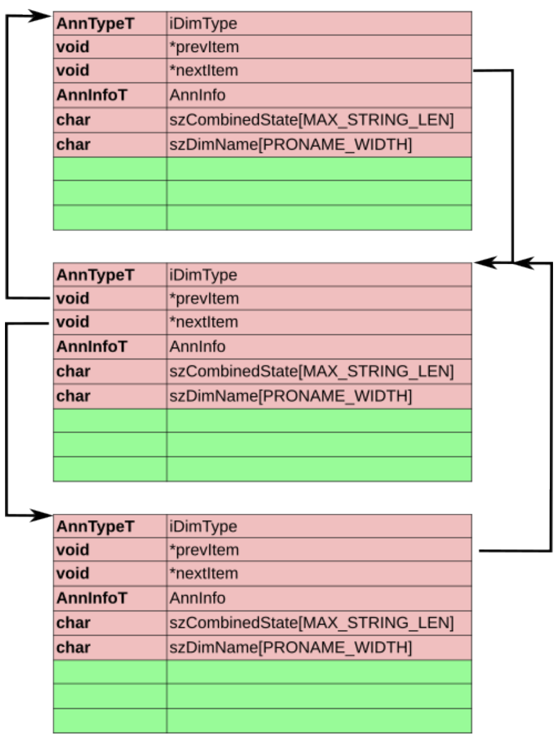


Figure 8.15: Mapping of the common data type *CommonData* allows cycling through the different memory blocks

The first item of the linked list is identified by the value NULL in the pointer *prevItem*. The last item of the linked list is identified by the value NULL in the pointer *nextItem*. The value of the integer *AnnotationType* (Figure 8.16) makes it possible to identify the correct annotation type stored in the memory block and to map the correct data structure to it in order to read its contents.

```
typedef enum AnnType
{
    TYPE_LINDIM   = 1,
    TYPE_CHAMFDIM = 2,
    TYPE_ORDDIM   = 3,
    TYPE_ANGDIM   = 4,
    TYPE_DIADIM   = 5,
    TYPE_RADDIM   = 6,
    TYPE_GTOL     = 7,
    TYPE_SURFFIN  = 8,
    TYPE_DATUMS   = 9,
    TYPE_NOTES    = 10,
    TYPE_MISC     = 11,
    TYPE_SYMB     = 12
} AnnTypeT;
```

Figure 8.16: The possible values of the variable *AnnotationType*

Separate data structures, *DlgMemoryToFreeT* and *DlgAsymmMemoryToFreeT*, have been created to track dynamically allocated memory, free it back and prevent memory leaks.

Module 1: The launcher module

The launcher module *jbCheckAsymmPMI* has two functions:

1. Launch the main part of the application. This parts creates the linked list of detected annotations and displays a dialogue box listing the detected annotations. These annotations are grouped according to their characteristics. Examples of characteristics are missing semantic references, possible data loss when exporting to STEP AP242. From this dialogue box the user can access the analysis module and the export module.
2. Conversion of annotations placed in the 'to be processed' queue by the user from annotation element to annotation feature to avoid data loss when exporting to STEP AP242. The dialogue box is then rebuilt.

Since no parameters can be passed within the call to the launcher function to specify its behaviour, this is solved by using a global variable *g_bActivateDelayedProcessing*.

If this global variable is false, the launcher module will execute the first function. The launcher module can call itself back with the global variable set to true. This will execute the second function. When the user-marked annotations are transformed, the linked list is rebuilt. Before the linked list is rebuilt, the memory allocated when the linked list was previously built is released.

Module 2: The scan module

Prior to an examination of the structure of the scan module, it is first necessary to identify the key considerations to be borne in mind when scanning annotations.

In PTC Creo, annotations can be classified in two ways, this is independent of the type of annotation (linear dimension, diameter dimension, . . .).

One way is to distinguish between:

1. driving dimensions — Dimensions that are derived directly from dimensions used to create a feature using the *Show Annotations* function.
2. driven dimensions — Dimensions that are created manually and have nothing to do with the dimensions used to create features.
3. annotation features — These can be thought of as containers that can contain multiple driven dimensions, GD&T, notes, ...

Another way is to distinguish between:

1. annotation elements — Annotations that belong to another feature like a driving dimension or a driven annotation, GD&T, note, . . . that is embedded in an annotation feature
2. stand-alone annotation — An annotation that stands on its own, a driven dimension that is not embedded in an annotation feature

As driven dimensions are manually created in the 3D model, semantic references are always assigned to the annotation. However, the correctness of these semantic references is not guaranteed. There are three reasons for this.

For the first, consider the example in Figure 8.17. When the annotation is queried to see if it has semantic references, the answer is yes. However, these semantic references are incorrect. The correct semantic references are the two faces highlighted in Figure 8.18. In some cases, companies try to deal with this by using special software. In PTC's case, this is ModelCheck. One of the modules in ModelCheck is RuleCheck. This module allows a company to create a list of rules to check that the CAD model conforms to the company's design standards (PTC 2023c). One of these rules could be that a 3D annotation must have faces as semantic references. In some cases, however, the annotation needs to have semantic references to the edges, which creates another problem. An example of this is shown in Figure 8.19. Checking with software such as ModelCheck in PTC Creo can therefore catch some of the problems, but not all. Some form of user intervention will be required.

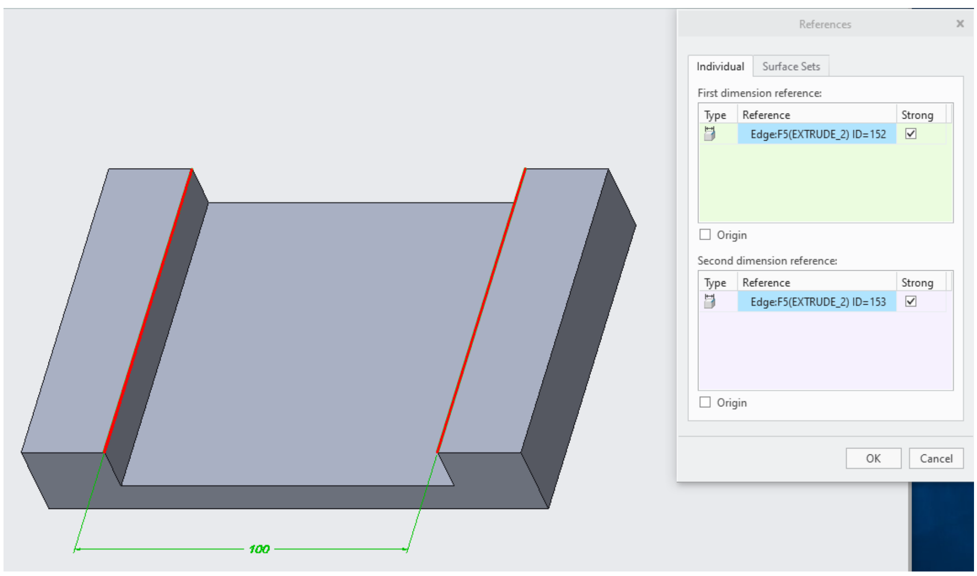


Figure 8.17: Driven dimension created manually in the model using the two red highlighted top edges of the rectangular cut. This is not what the designer intended.

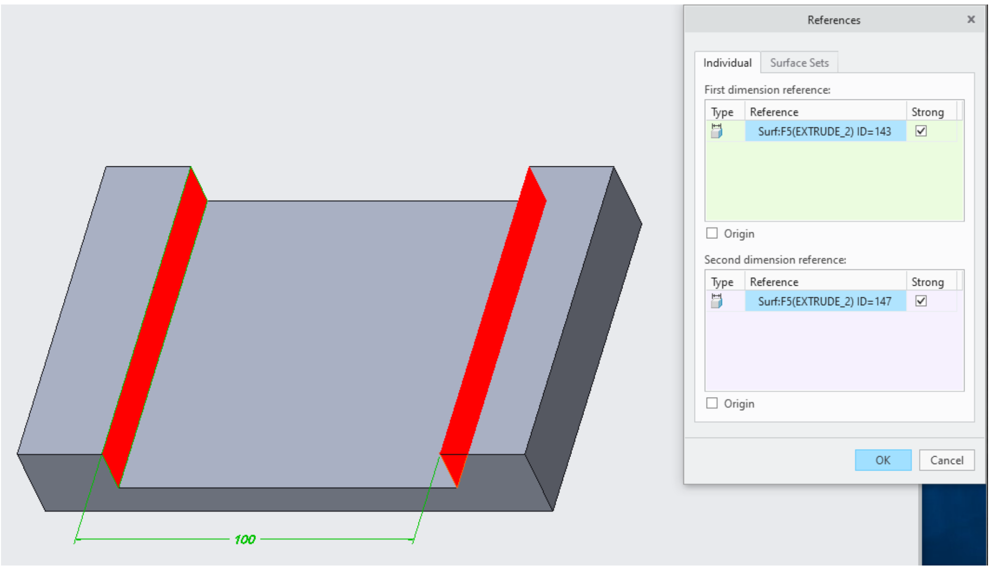


Figure 8.18: Driven dimension created manually in the model using the two red highlighted faces of the rectangular cut. This is what the designer intended.

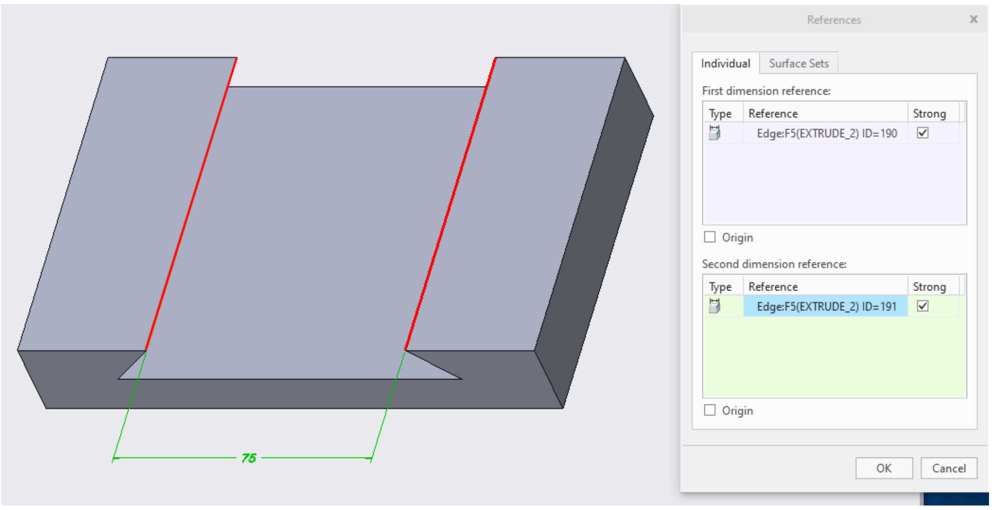


Figure 8.19: Driven dimension created manually in the model using the two red highlighted edges of the rectangular cut. This is what the designer intended.

The second reason why the correctness of semantic references is not always guaranteed is the fact that two types of semantic references can be assigned to an annotation. On the one hand, there are semantic references that are used to specify what the annotation is referring to. On the other hand, there are semantic references that are used as attachment references and to facilitate the visualisation of the annotation. The latter type can simultaneously serve the same function as the first type. An example of this is shown in Figure 8.20. Nevertheless, it is often challenging to ascertain this information through software. An illustrative example can be found in Figure 8.21. Some form of user intervention will be required.

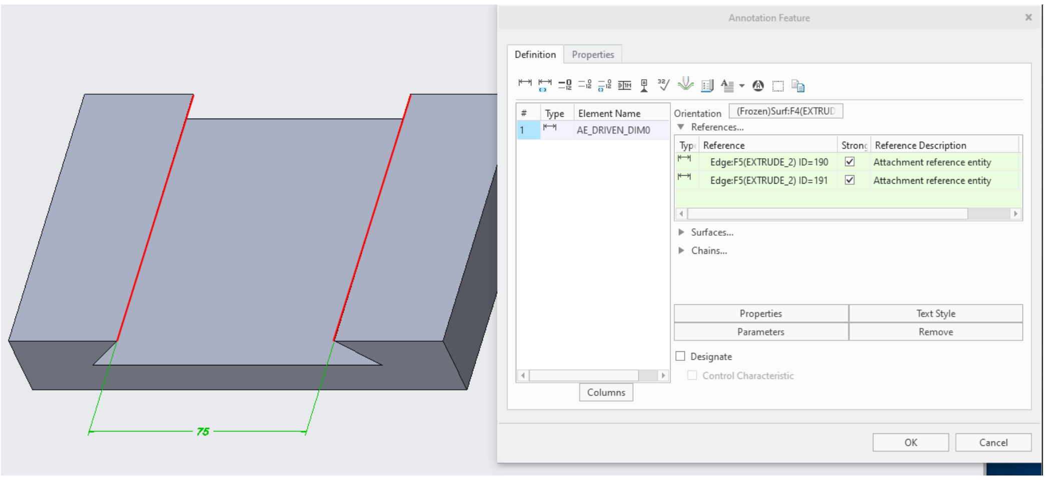


Figure 8.20: Driven dimension created manually as an annotation feature in the model using the two red highlighted edges of the rectangular cut. The edges are also the attachment references for the annotation.

There are two potential problems with the example in Figure 8.21.

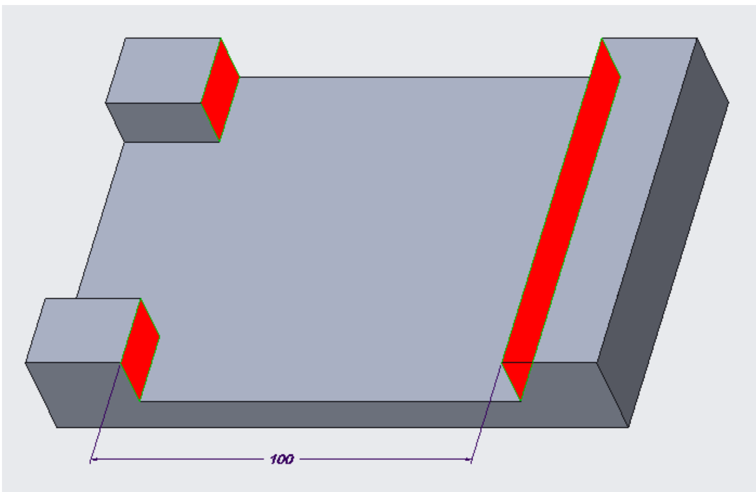


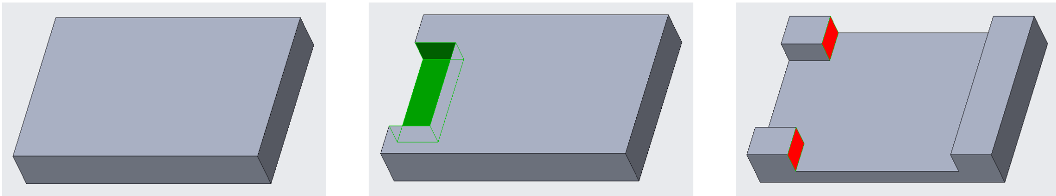
Figure 8.21: Driven dimension created manually as an annotation feature using the three red highlighted faces of the rectangular cut.

The first issue arises from the absence of a differentiation between the *First* and *Second* dimension references. One of the original objectives of the software was to modify the geometry of the model to the centre of the specified tolerance field. This cannot be achieved without knowing the specific surfaces associated with the *First* and *Second* dimension references.

A second issue arises following the export to a neutral exchange format such as STEP AP242. This is caused by the way the model is created, which affects the numbering of the topological elements used to build the model. A comparison of two distinct approaches to the construction of the model depicted in Figure 8.21 will serve to illustrate this.

Approach 1

Figure 8.22 shows the different steps used in the first approach.



(a) Step 1: Creation of a block

(b) Step 2: First cutting operation

(c) Step 3: Second cutting operation

Figure 8.22: The different steps to create the CAD model in a first approach

After the last step, the two faces in red in Figure 8.22c belong to the same surface. The logical consequence of this is that when an annotation is added to the model using the red and green faces shown in Figure 8.23 as semantic references, the two red surfaces appear as a single surface in the semantic reference dialog box, as does the green face.

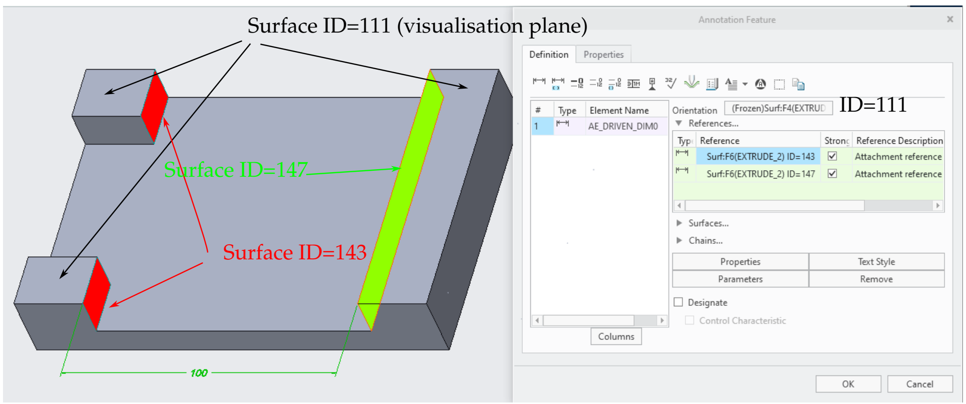


Figure 8.23: When creating the driven dimension as an annotation feature, the red and green-coloured faces are identified as two surfaces.

With regard to the native CAD model in question, it can be stated that it does not give rise to any additional issues within Creo Parametric with respect to the interpretation and meaning of the annotation. Nevertheless, the model's export to a neutral exchange format, such as STEP AP242, may result in additional complications. The two red-coloured faces that were previously identified as one surface have now been split into two separate surfaces. One of the surfaces has been assigned the original surface's ID indicating that the other surface is no longer considered a semantic reference (see Figure 8.24).

In addition to the fact that face 1 (see Figure 8.24) no longer corresponds to the original face 1 (the red-coloured faces in Figure 8.23), a third semantic reference has been added (face 3 in Figure 8.24). This third reference indicates the plane in which the annotation should be displayed. A method could not be identified within the Creo 6 Toolkit libraries that would enable the distinction between that define what the annotation refers to, and the reference that defines the plane in which the annotation should be displayed.

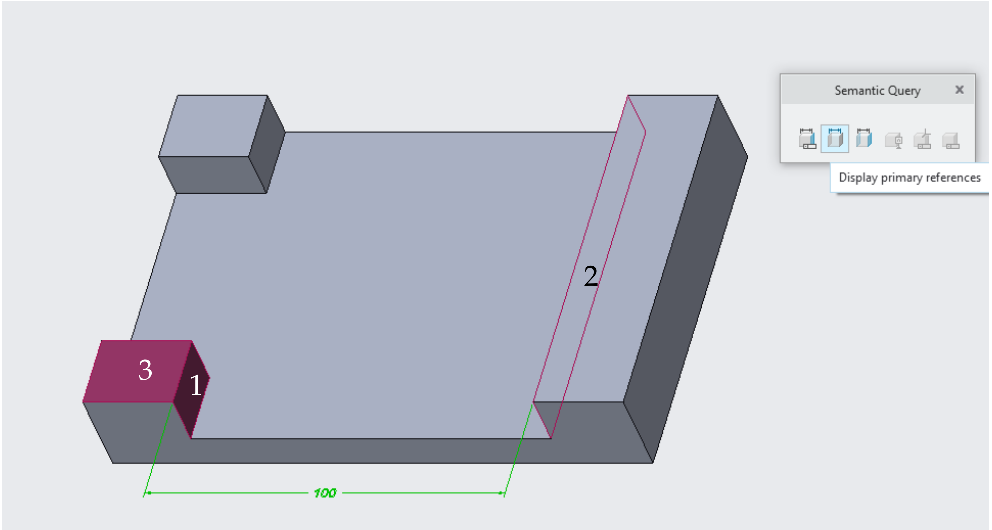
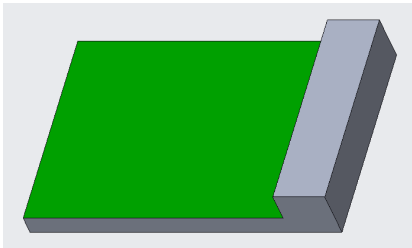


Figure 8.24: The semantic references 1, 2 and 3 after importing the STEP AP242 file created by exporting the model shown in Figure 8.23

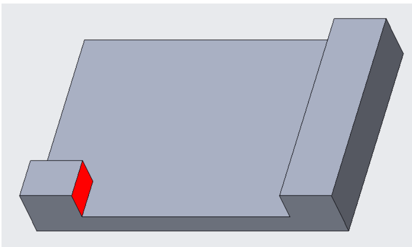
Approach 2

Figure 8.22 shows the different steps used in the second approach.

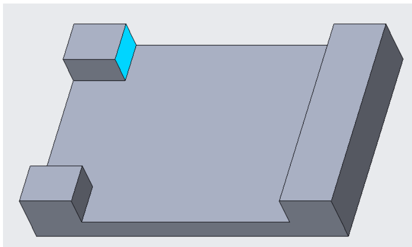
(a) Step 1:creation of a block



(b) Step 2:first cutting operation



(c) Step 3:adding first block



(d) Step 4:adding second block

Figure 8.25: The different steps to create the CAD model in a second approach

The two blocks on the left-hand side of the model are no longer the result of a single cut as was the case in the first approach (see Figure 8.22c). They have been added to the model as two separate blocks. This results in two distinct faces, one red (see Figure 8.25c) and one cyan (see Figure 8.25d). When an annotation is added to the model using the red, cyan and green-coloured faces shown in Figure 8.26 as semantic references, three surfaces appear in the semantic reference dialog box.

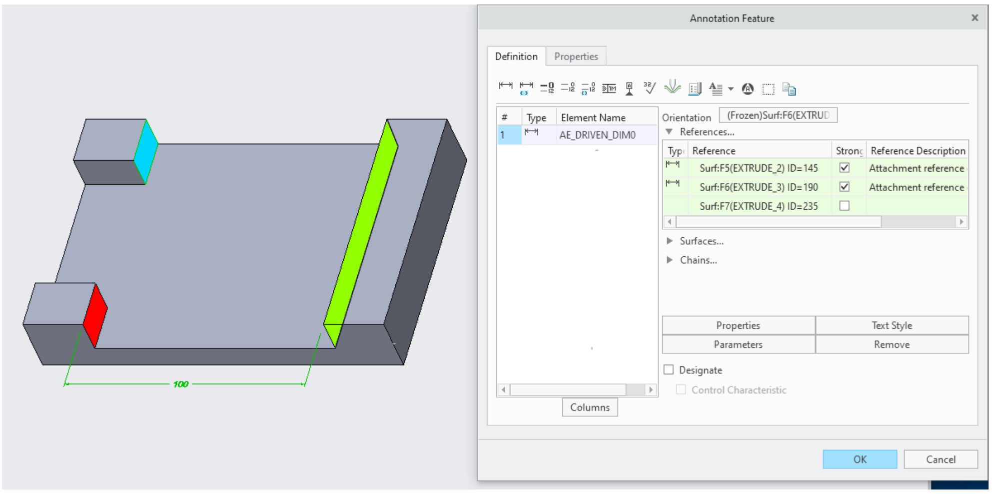


Figure 8.26: When creating the driven dimension as an annotation feature, the red, cyan and green-coloured faces are identified as three surfaces

With regard to the native CAD model in question, it does not present any additional problems within Creo. However, when the model is exported to a neutral exchange format such as STEP AP242, additional problems arise. Upon import of the resulting STEP file, the plane used to display the annotation becomes a semantic reference, resulting in the loss of the cyan surface shown in Figure 8.26 and its identification as a semantic reference.

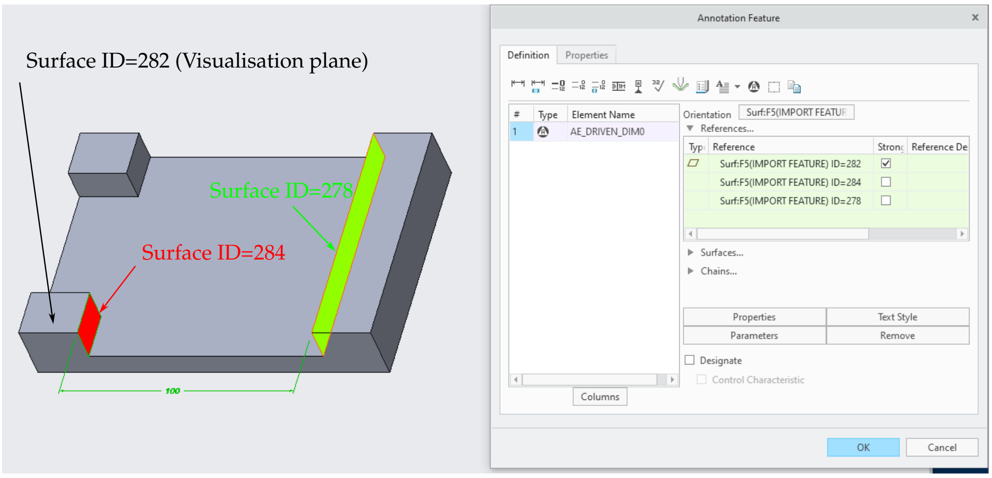


Figure 8.27: When creating the driven dimension as an annotation feature, the cyan surface (see Figure 8.26) is lost and is no longer identified as a semantic reference

The third reason why the correctness of the determination of the semantic references is not guaranteed, is that the Toolkit APIs are unable to correctly distinguish between *First dimension references* and *Second dimension references* (see Figure 8.28). When an annotation feature is used, the distinction between *First dimension references* and *Second dimension references* no longer exists (see Figure 8.29). If there are only two references, this distinction does not matter. It is different when there are more than two references (see Figure 8.30).

This issue was discussed with Michael Fridman, PTC's product manager for drawings and MBD. He acknowledged the problem and promised to provide an additional API in the Toolkit library of the next Creo versions that would allow the distinction to be made.

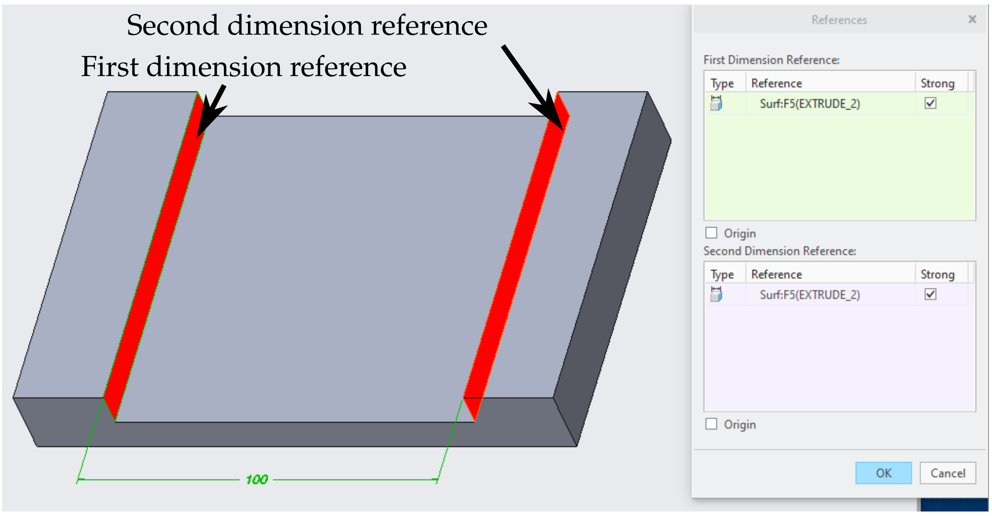


Figure 8.28: First and second dimension references

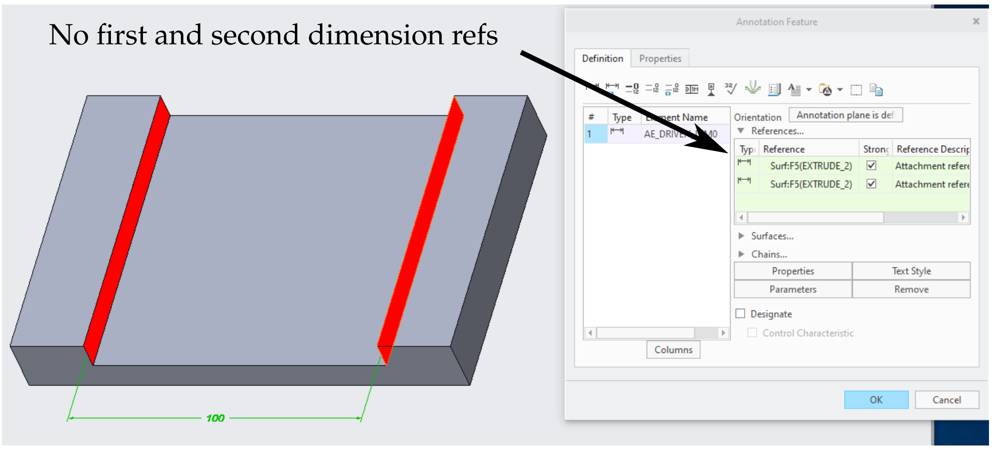


Figure 8.29: When an annotation feature is used to create the dimension, there is no longer a distinction between first and second dimension references

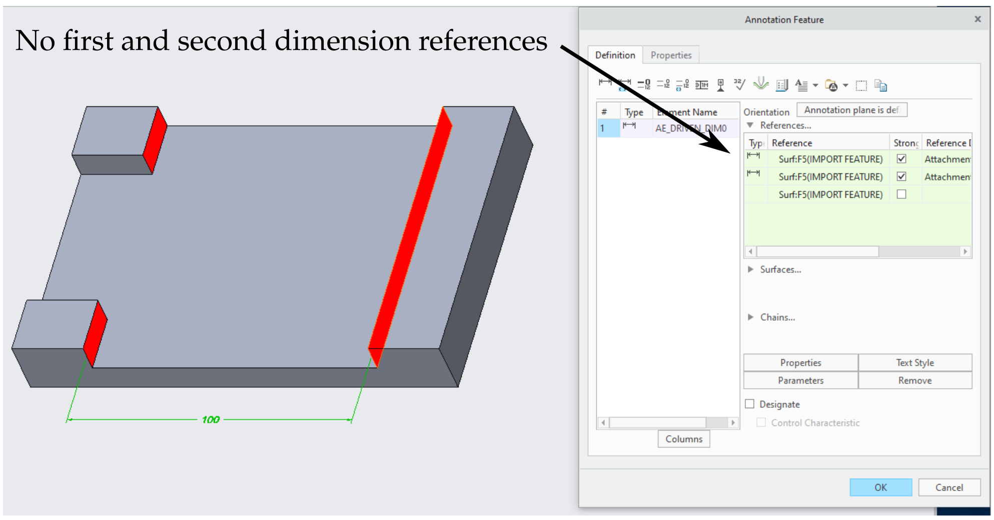


Figure 8.30: If there are more than 2 surfaces in the selection list, it is not possible to determine which surfaces are the first and which are the second dimension references

These observations lead to the following conclusion. When querying the semantic references within Creo Parametric, the aforementioned issues are primarily observed when utilising annotation features, with less pronounced effects observed when employing the “Show Annotation” function to create a driving dimension and the “Dimension” function to create a driven dimension. When querying the semantic references within a STEP file that was exported by Creo and imported back into Creo, regardless of the feature used in Creo to create the annotation in the original native Creo model, problems are encountered. Testing has demonstrated that analogous issues arise with models generated in alternative CAD systems, including Inventor, Siemens NX and CATIA, which have been exported to STEP AP242. These issues were pivotal in the decision to prioritise CAD models created in Creo. Nevertheless, it is insufficient to limit one’s focus to native Creo CAD models in order to resolve all issues. For example, even in a native Creo model, it is in some cases very difficult to impossible to always determine the exact semantic references of an annotation. This is the case with “First dimension references” and “Second dimension references” mentioned above. The original purpose of developing the software solution was to change the nominal values of the CAD model to the centre of the imposed tolerance field. As the dimensioning scheme used to create the model does not need to be the same as the annotations applied to the MBD model, the plan was to achieve this using Creo’s direct modelling functionality. This allows the model to be modified without having to consider how the model was created. However, it is essential to be able to retrieve the semantic references. This turned out not to be the case. For this reason, the purpose of the software development was changed to provide the designer with a tool to analyse and convert annotations in order to minimise data loss when exporting the Creo model to STEP AP242.

Having identified the potential challenges and limitations that the scan module must address, the subsequent stage is to implement the scan module in practice. The Creo Toolkit has so-called *visit* functions, such as *ProSolidAnnotationselemsVisit()*, among others, which should make it possible to go through all the 3D annotations present in the model and, based on this, to list the 3D annotations present in the model. These *visit* functions were originally used in the scanning module. During testing it was found that these *visit* functions did not produce the desired results. For example, not all annotations present in the model were always detected, or some annotations were detected twice, or annotations that had been deleted were still detected as valid annotations. It was not clear whether this was due to errors in the implementation of the Toolkit APIs used or to other reasons. A new detection method was therefore developed.
This new method of detection consists of a number of steps. These are implemented in the *jbBuildLinkedListAnnotations* function.

The first step is to determine how many combined views[1] there are and what names are assigned to them.

The second step

- checks whether there are annotations assigned to the detected combined views
- checks whether the detected annotations are valid or not
- builds the linked list of detected annotations

All of this is done in a number of sub-steps.
Combined views are scanned for annotations associated with those combined views.

[1] Combined views allow the designer to combine, apply and save multiple display states to a specific view. Possible display state elements are: model orientation, simplified representation, model style, cross section, exploded view (including cosmetic offset lines), layer state, appearance state.

These annotations can be annotation elements or annotation features. This is used in the first check to see if the annotation is valid or not. Valid here means that the annotation is in an active, non-deleted state. This check attempts to retrieve the data from the annotation using the *ProAnnotationElementGet* and *ProAnnotationelemFeatureGet* toolkit functions. If the value *PRO_TK_NO_ERROR* or *PRO_TK_NOT_EXIST* is returned by these functions, the annotation is considered valid for the time being. At the same time, the base annotation type is determined. The following base types can occur:

- *PRO_DIMENSION*
- *PRO_SURF_FIN*
- *PRO_GTOL*
- *PRO_SET_DATE_TAG*
- *PRO_NOTE*
- *PRO_SYMBOL_INSTANCE*.

Each base type is handled by a function designed for it:

PRO_DIMENSION	:	*jbReadDimensionData*
PRO_SURF_FIN	:	*jbReadSurfFinishData*
PRO_GTOL	:	*jbReadGTolData*
PRO_SET_DATE	:	*jbReadDatumData*
PRO_NOTE	:	*jbReadNoteData*
PRO_SYMBOL_INSTANCE	:	*jbReadSymbolData*

As the detection of the annotation is not done by a so-called visit function, but by querying combined states, and an annotation can occur on several combined states, care must be taken:

- to avoid duplicates
- to add the name of the combined state to the list of combined states on which the annotation is displayed.

Within each dimension type handling function, an additional check is made to see if the detected annotation is valid or not. This is done here by looking at the visualisation of the annotation using the *ProDimensionValuedisplayGet* toolkit function. In this way, an attempt is made to filter out annotations that have been deleted. The tests carried out showed that this method seems to work, at least for the test models.
More unique than the other handling functions is the *jbReadDimensionData* function. This is because the base type *PRO_DIMENSION* can contain different subtypes of dimensions:

- linear dimensions:
  - regular linear dimensions
  - ordinate dimensions
  - chamfer dimensions
- angle dimensions
- circle dimensions:
  - diameter dimensions
  - radius dimensions
  - arc length dimensions

Each handling function:

- Allocates temporary memory to store the data structure of the detected dimension type
- stores the annotation structure, which is used to check for possible duplicates, as annotations can be displayed on multiple combined states at the same time.

- Determines the annotation properties:
  - Determine whether the annotation is a driving dimension, a member of an annotation element, has semantic references, . . .
  - Determine the combined state the annotation was found on and store this in the data structure
  - In the event that the annotation is a dimension, it is necessary to ascertain whether the dimension is overridden or hidden. The term “overridden” is used to describe a situation in which the actual nominal value of a dimension is overwritten by the designer with an arbitrary value. This results in a loss of the relationship between the nominal value and the geometric dimensions, which in turn precludes the use of the dimension to adjust geometry.
  - Determine how the annotation is displayed. In case of a dimension, this implies the numerical or fractional form of tolerances, whether or not they have been assigned, and if they have been assigned, the manner of their representation. This includes the ISO system of limits and fits, whether or not they have been assigned.
  - In the event that the annotation is a dimension, it is necessary to ascertain which units have been employed.
  - Determine the name of the annotation
- In the event the annotation is a dimension, a flag (boolean variable *bGeomModified*) is set to false. If this flag is set it means the corresponding geometry was modified to the mid value of the specified tolerance.
- A flag (boolean variable *bDimConverted2Feat*) is set to false. If this flag is set it means this dimension was converted to an annotation feature in order to be able to access the semantic references.
- The temporary data structure containing all the information of the detected annotation is added to the linked list.
- The temporary memory that is allocated to store the data structure of the detected annotation is subsequently released.

Module 3: The visualisation module

The visualisation in the visualisation module refers to two things:

1. display the annotations found in a dialog window identified by DLG_PMI_NAME, grouped by category and probability of occurence of data loss when exporting to STEP AP242.
2. highlight an annotation selected in the dialog window with its associated semantic references, if any, in the CAD model.

Since the functionality of the visualisation module is made available to the user through the GUI, all names of C functions and C procedures developed for this module start with *jbUI*. This is also the case for all other custom C functions and C procedures related to the GUI.
The dialog window is created using the *jbUICreateDialogWindow* function. This function uses the data stored in the linked list created by module 2, the scan module, to list all valid annotations detected in the CAD model. These annotations are grouped in different categories.

The highlighting of a specific annotation in the CAD model is implemented in the following handling functions:

jbUIOptMnuSelectAction	:	Switch to the combined view selected by the user via the *OPTMNU_CMB_VIEWS* option menu in the *DLG_PMI_NAME* dialog window and update the annotations display accordingly.
jbUIOnListSelect	:	This is the handler for the select action in a list component. List components are used to display the various lists of annotations in the dialog window. When the boolean *bDirectHighlightEnabled* is set, selection of an annotation in the list component must result in automatic highlighting of that annotation in the CAD model.
jbUIHighlightItem	:	This is the function used to highlight the annotation selected in a list component in the dialog window, together with its associated semantic references in the CAD model. To highlight the semantic references, the function *jbUIHighlightReferences* is called. A distinction must be made between "driving dimensions" and "driven dimensions". In the case of "driven dimensions", the semantic references are attached to the dimension when the dimension is created. This is probably the reason why the references have to be collected using the ProToolkit function *ProAnnotationelemReferencesCollect()*. In the case of "driving dimensions", the semantic references are attached to the dimension after the dimension is created. This is probably the reason why the references need to be retrieved using the ProToolkit function *ProDimensionAdditionalRefsGet()*.
jbUICheckedHighlight	:	This is the handler for the *ChkBtn_Highlight* check button. When checked, the annotation selected in the list component must be highlighted immediately, without having to press the right mouse button and select the highlight option.
jbUISetStatusHighlightBtn	:	This function enables or disables the pop-up menu button to highlight the annotation that is selected in a list component of the dialog *DLG_PMI_Data* or *DLG_ASYMM_NAME*.

Module 4: The conversion module

This module allows the user to convert annotation types that cause data loss when exported to a STEP AP242 file, or whose semantic references cannot be retrieved, to another type where these problems do not occur. The annotations that are eligible for conversion are listed by annotation type in the "Semantic references not defined or not accessible" list box. The user is then able to select the annotations in the list box and add them to a queue for subsequent conversion in bulk. The module is initiated when the user clicks the "Process Queue" button in the dialog window. At that point, all annotation added to the queue are transformed into annotation features. In the event that semantic references are assigned to an annotation but cannot be queried, conversion of the annotation to an annotation function can enable this possibility.

There is no function within the ProToolkit APIs to convert an annotation to an annotation feature. However, in PTC Creo, the user can do this by selecting the annotation and then selecting "Create Annotation Feature" from the pop-up menu. The following functions have been developed to implement the functionality of the conversion module:

jbUIPushedQueueConv2AF: This is the handler function for the pop-up menu button "Btn_QueueConv2AF". This function adds the selected annotation to the queue of annotations to be converted to an annotation feature. To indicate that there are items in the queue to be processed, the boolean *bToProcess* is set and the list control display is updated to indicate the annotation has been added to the queue. As it is not possible to change one item in the list control, the contents of the entire list control must be rebuilt.

jbUIPushedProcess : Starts the processing (conversion to annotation feature) of the queued annotations. As there is no ProToolkit function to convert an annotation to an annotation feature, this is achieved by a workaround. Within Creo Parametric, the user can do this by selecting an annotation and then selecting "Create Annotation Feature" from the pop-up menu. Calling up the pop-up menu and selecting "Create Annotation Feature" is recorded in a macro. In Creo terminology, such a macro is called a *mapkey*. When the conversion process is started by *jbUIPushedProcess*, the global boolean variable *g_bActivateDelayedProcessing* is set to true and the main dialog window is closed. By setting the global boolean variable to true, the function *jbCheckAsymmPMI* that creates the main dialog window is called again. The annotations in the queue to be processed are now checked off one by one. This involves placing an annotation in the selection buffer and then calling the mapkey. This converts the selected annotation into an annotation feature. The main dialog window is then recreated.

Module 5: The analysis module

Originally this module was intended to adjust the geometry of the CAD model, based on the dimensions with asymmetric tolerance, so that these dimensions would have a symmetric tolerance. This facilitates the generation of CNC toolpaths for the manufacture of the model. As it was not always possible to retrieve the semantic references of these dimensions, this was abandoned. The new objective was to compile all dimensions with asymmetric tolerances into a convenient list, with the ability to highlight these dimensions in the CAD model to assist the CAM programmer.

The following functions have been developed to implement the functionality of the conversion module:

jbUIPushedAnalyse	:	This is the handler function for the button "Analyse". This function displays a new dialog window listing all dimensions with asymmetric tolerances assigned to them.

Module 6: The export module

This module enables the data structure of the detected annotations to be exported to a CSV file. This can then be imported into a spreadsheet for use in other applications, such as the preparation of First Article Inspection documents.

The *jbUIPushedExport* handler is responsible for this functionality. This handler is called when the user clicks the "Export CSV" button in the dialog window. Since each type of annotation (linear dimension, radial dimension, surface finish, note, . . .) is characterised by its own specific data, a separate CSV file is generated for each type of annotation.

GUI and functions of the application

The application is launched by clicking the application icon on the ribbon (see Figure 8.31).

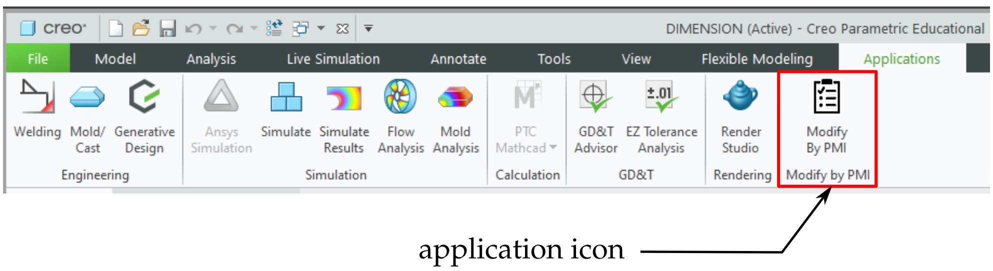
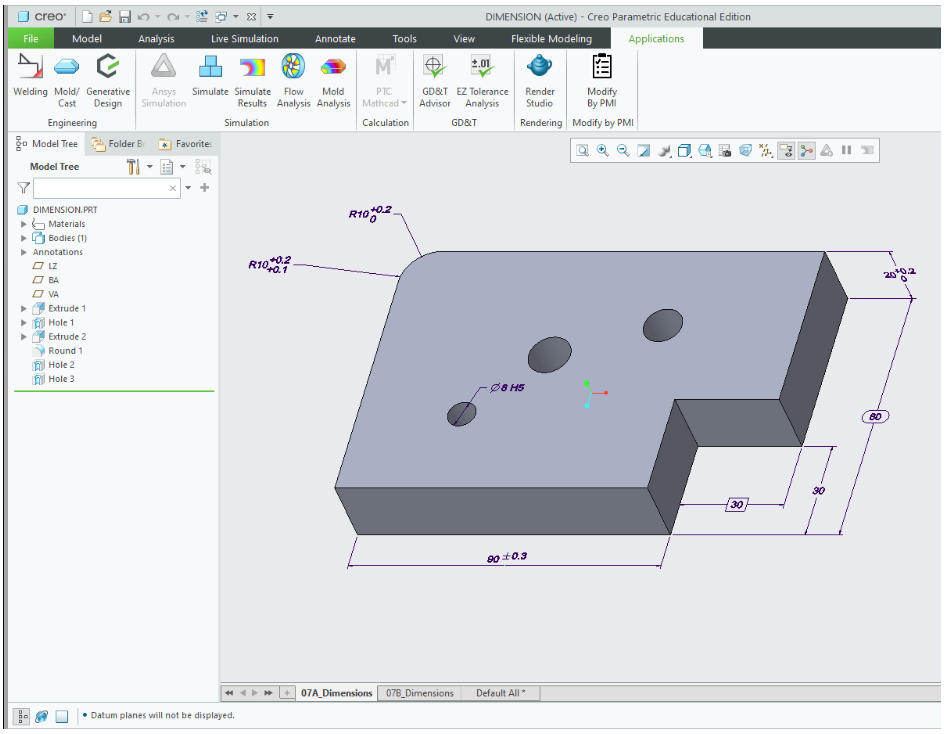


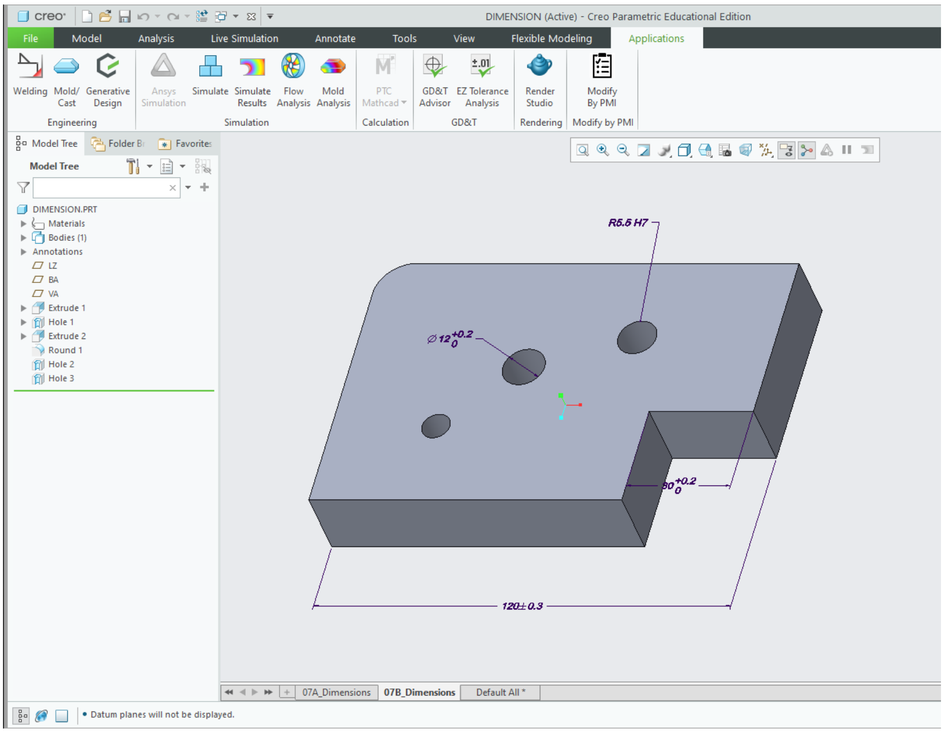
Figure 8.31: The application is launched by clicking the application icon on the ribbon

To illustrate how the application works, the example in Figure 8.32 is used. This is an MBD model with two sheets 07A_Dimensions and 07B_Dimensions containing model views with annotations. A screenshot of the software application window is shown in Figure 8.33. The software that has been developed has five functions:

1. Lists the annotations present in the MBD model, sorted by type (Linear, Chamfer, Ordinate, . . .)and by sheet
2. Indicates which annotations may cause problems
3. Lists dimensions with asymmetric tolerance fields
4. Redefine an annotation to preserve semantic references where possible
5. Export the detected annotations into a CSV format that can be imported into an Excel spreadsheet.



(a) First sheet with an annotated model view



(b) Second sheet with an annotated model view

Figure 8.32: The sheets containing the annotated views of the MBD model

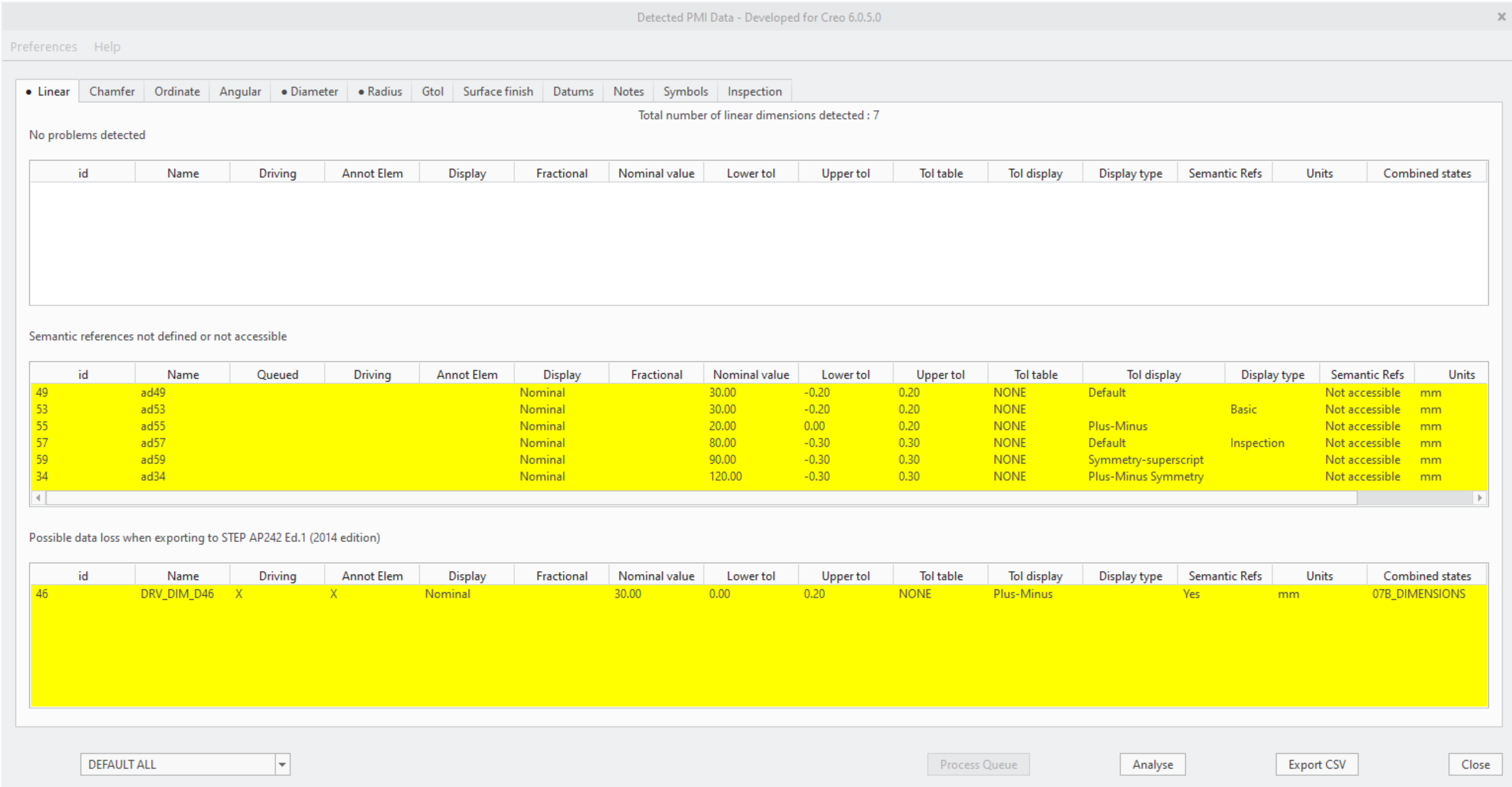


Figure 8.33: The software application window

The first function of the application is to list the annotations detected in the model. These are listed by type (Linear, Chamfer, . . .). A large dot in front of the annotation type tab name (Figure 8.34) indicates that this annotation type has been detected. This gives the designer an immediate indication of what type of annotations have been detected. The designer can also select an annotation and then use the popup menu that appears when the right mouse button is pressed to highlight it in the model (Figure 8.35). If the annotation is on a sheet other than the currently active sheet, that sheet will be displayed with the annotation and semantic references highlighted. The designer is made aware of these functions by a tooltip that appears when the mouse pointer is over an annotation in one of the lists and is not moved for a while (Figure 8.36).

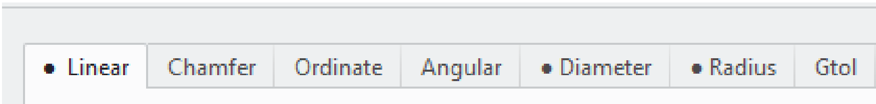


Figure 8.34: A large dot in front of the annotation type tab name indicates that this annotation type has been detected

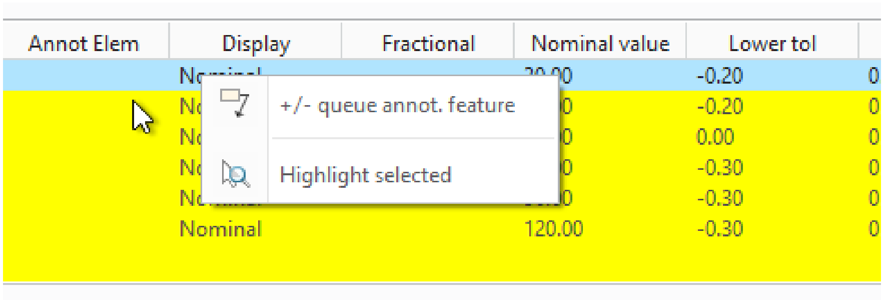


Figure 8.35: When an annotation is selected and the right mouse button is pressed, additional options appear, such as highlighting the selected annotation in the CAD model

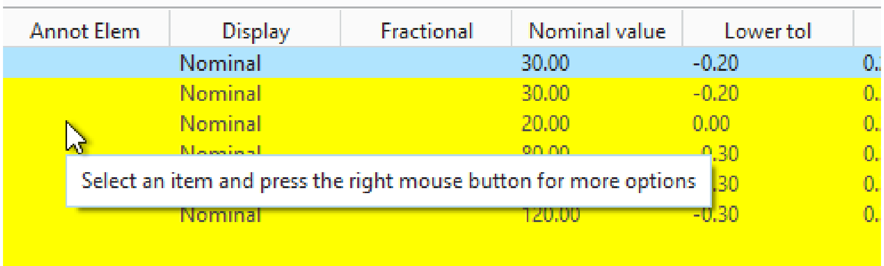
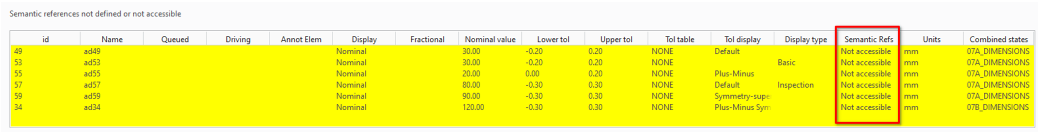


Figure 8.36: When the mouse pointer is over an annotation in one of the lists and is not moved for a while a tooptip appears to alert the designer to additional functions

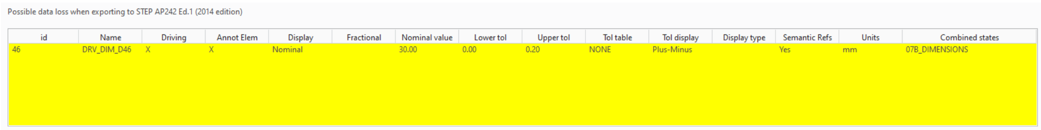
The second function of the application is to indicate which annotations may cause problems. Two yellow-coloured list boxes indicate annotations where problems may occur. The first list box lists annotations that have no semantic references or whose semantic references cannot be retrieved (Figure 8.37). When an annotation is selected in this list box and the right mouse button is pressed, another option is available in addition to highlighting the annotation in the CAD model (see Figure 8.35). This other option is to add or to remove the selected annotation from the conversion queue. This allows the designer to add all annotations that have semantic references, but whose semantic references cannot be queried, to a queue for later processing in one go. This processing involves converting these annotations into an annotation feature that allows the semantic references to be retrieved. The second box lists the annotations that will lose data when the MBD model is exported to STEP AP242 (Figure 8.38). In most cases these are driven dimensions. The only solution here is to delete these annotations and re-create them as driven dimensions or as an annotation feature containing a driven dimension.



Semantic references not defined or not accessible

id	Name	Queued	Driving	Annot Elem	Display	Fractional	Nominal value	Lower tol	Upper tol	Tol table	Tol display	Display type	Semantic Refs	Units	Combined states
49	ad49				Nominal		30.00	-0.20	0.20	NONE	Default		Not accessible	mm	07A_DIMENSIONS
53	ad53				Nominal		30.00	-0.20	0.20	NONE		Basic	Not accessible	mm	07A_DIMENSIONS
55	ad55				Nominal		20.00	0.00	0.20	NONE	Plus-Minus		Not accessible	mm	07A_DIMENSIONS
57	ad57				Nominal		80.00	-0.30	0.30	NONE	Default	Inspection	Not accessible	mm	07A_DIMENSIONS
59	ad59				Nominal		90.00	-0.30	0.30	NONE	Symmetry-supe		Not accessible	mm	07A_DIMENSIONS
34	ad34				Nominal		120.00	-0.30	0.30	NONE	Plus-Minus Sym		Not accessible	mm	07B_DIMENSIONS

Figure 8.37: The software application lists the annotations without semantic references or whose semantic references cannot be queried

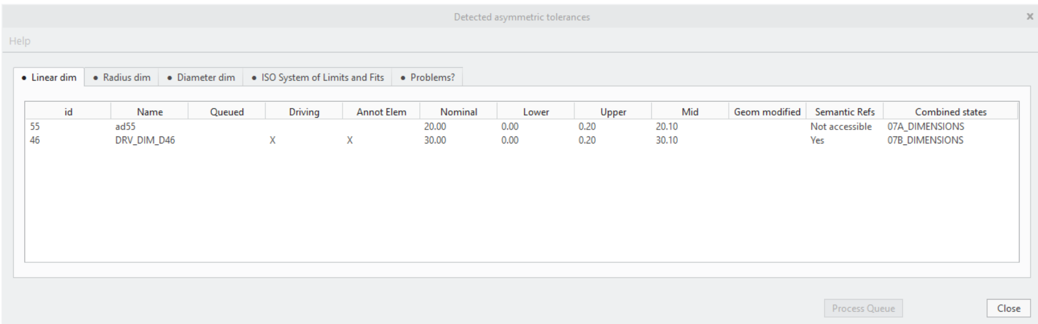


Possible data loss when exporting to STEP AP242 Ed.1 (2014 edition)

id	Name	Driving	Annot Elem	Display	Fractional	Nominal value	Lower tol	Upper tol	Tol table	Tol display	Display type	Semantic Refs	Units	Combined states
46	DRV_DIM_D46	X	X	Nominal		30.00	0.00	0.20	NONE	Plus-Minus		Yes	mm	07B_DIMENSIONS

Figure 8.38: The software application lists the annotations that may lose data when exported to STEP

The third function of the application is to lists dimensions with asymmetric tolerance fields. These dimensions are of particular interest to those responsible for making the model. In fact, it may be necessary to bring the nominal values of these dimensions to the centre of the tolerance field in order to mill or turn them correctly. This function can be accessed by clicking on the Analysis button at the bottom right of the application window. When this button is clicked, a new dialogue window appears with a list of all dimensions with asymmetric tolerances (Figure 8.39).



id	Name	Queued	Driving	Annot Elem	Nominal	Lower	Upper	Mid	Geom modified	Semantic Refs	Combined states
55	ad55				20.00	0.00	0.20	20.10		Not accessible	07A_DIMENSIONS
46	DRV_DIM_D46		X	X	30.00	0.00	0.20	30.10		Yes	07B_DIMENSIONS

Figure 8.39: When the Analyse button is clicked, a new dialogue window appears with a list of all dimensions with asymmetric tolerances

The fourth function of the application is to transform an annotation to preserve semantic references where possible. The aim is to transform annotations that can be determined to have semantic references, but whose semantic references themselves cannot be retrieved, into annotation features. Their semantic references can be retrieved. Transforming an annotation into an annotation feature is only possible if the annotation being transformed is a stand-alone annotation or a driven dimension. There is no API available within the Toolkit libraries to turn these annotations into annotation features, although this function is available within Creo Parametric. This has been solved by calling the function from Creo Parametric by means of a macro, which in turn is controlled from the Toolkit application. In order to make the conversion as smooth as possible for the designer, and to make it easier to call the macro from within the application, it is possible to add the annotations to be converted to a queue. This queue can be processed in one go by clicking on the Process Queue button.

The fifth function of the application is to export the detected annotations to a CSV file. Each type of annotation is exported to a separate CSV file (see Figure 8.40). These CSV file can be imported into another package, such as Excel, for further processing. For example, it is possible to generate First Article Inspection Documents.

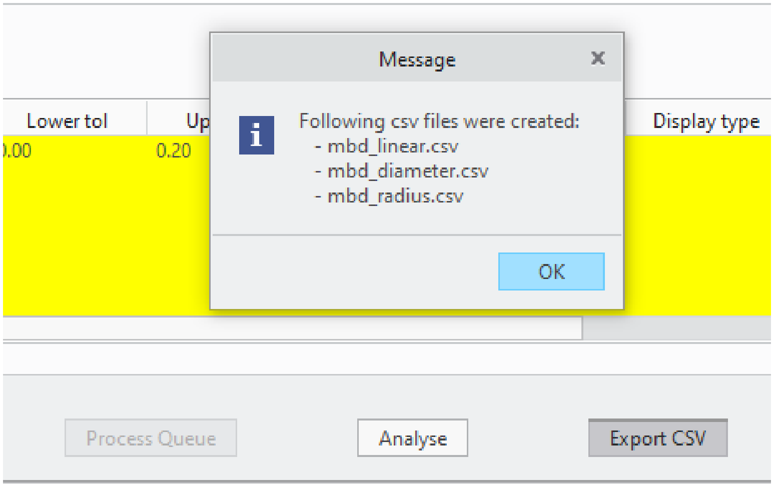


Figure 8.40: By clicking the Export CSV button each type of annotation is exported to a separate CSV file

The software has been tested by several people in the industry. Many thanks to Thomas Sarre of the company Innoptus for his valuable insights into the usability of the software.

8.6 Conclusion

The original aim was to develop a stand-alone software package that would be able to analyse the annotations in an MBD model and, on that basis, make modifications to the model to ensure that the nominal dimensions of the model were in the centre of the imposed tolerance field. This would make it easier for production staff to generate CNC programmes to produce the model within the imposed product specifications. Several solutions were considered. The use of an open source kernel such as Open-Cascade was rejected on the grounds that it did not have sufficient support for STEP AP242 with PMI representation. This made it impossible to read the value and semantic references of an annotation. The use of a commercial CAD kernel such as ACIS was rejected due to uncertainty regarding the cost of licenses. Finally, based on availability and cost, the choice was made to develop the software as a Toolkit application for PTC Creo Parametric. During its development, a number of problems arose which made it impossible to maintain its original purpose. For example, it was not always possible to detect semantic references, or the semantic references that were detected could not be correctly assigned. It was also discovered that the way a dimension was created as an annotation affected how it was transferred when exported to STEP AP242. This was discussed with representatives from PTC who said that they were aware of these problems and that the STEP interface would be adjusted in newer versions. For these reasons, the original purpose was abandoned and the software was developed with the aim of analysing the annotations present in the MBD model. It identifies which annotations may cause problems and provides the designer with a tool to automatically convert these annotations, where possible, so that the problems are either solved or minimised. A flowchart of the software can be seen in Figure 8.41.

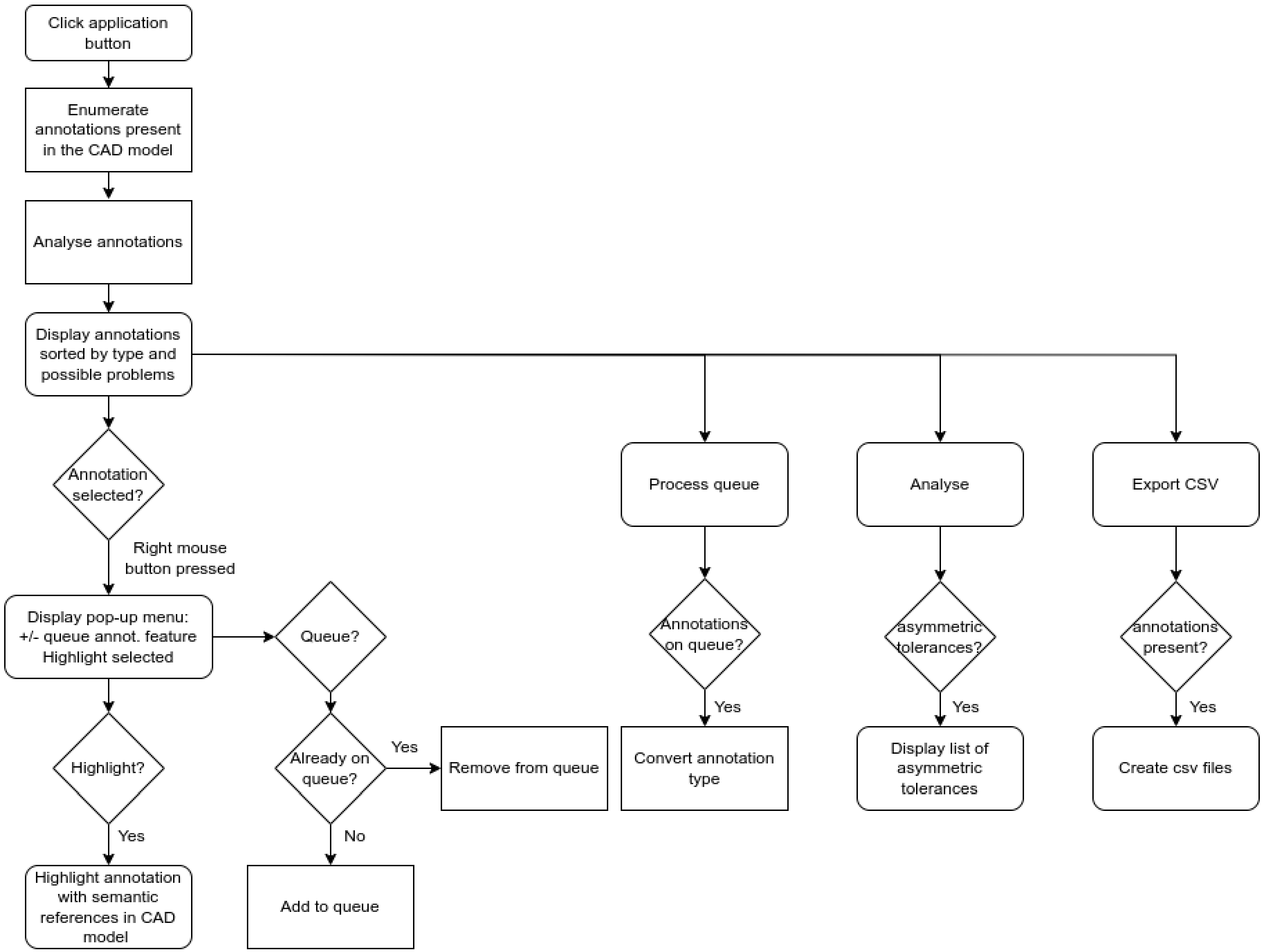


Figure 8.41: Flowchart of the developed software

9 Conclusions

9.1 Geometry

With MBD, the 3D model is considered the authority. This entails that all dimensions, complying with the generally applicable tolerance of the design, no longer need to be explicitly defined in the model as 3D annotations. They are exclusively determined by the geometry of the CAD model. This is only possible if the CAD model can maintain the same accuracy when transferred from one stakeholder to another. The tests carried out have shown that, when a model is exported to a neutral exchange format such as IGES or STEP, a distinction has to be made between so-called “analytical shapes” and “double-curved shapes”. “Analytical shapes” encompass shapes like beams, cylinders, cones, and the like. “Double-curved shapes” refer to shapes that are to be described with splines. If the model only contains analytical shapes, there are no accuracy issues. However, when the model contains double-curved shapes, deviations in accuracy may occur. These deviations are usually smaller than the smallest tolerance field included in the general tolerance specified by the ISO 2768 standard, which means they can be ignored. However, it is possible for the deviation to be larger in some cases, depending on the CAD system utilised, the CAD kernel utilised, the maximum spline degree, the configuration of the CAD system, and the nature of the model’s shape. Hence, it cannot be assumed that a model can be easily transferred from one CAD system to another without some measures being taken. It is therefore advisable to first align the configuration of the CAD systems between which the transfer takes place, based on thorough testing prior to starting the design process. This is because there are various options, such as the model accuracy and the maximum degree of the splines applied in a design, which cannot always be altered once they have been applied.

9.2 Annotations

All dimensions with tolerances that differ from the general design tolerance, along with all additional GD&T and other annotations that are crucial for manufacturing, must be explicitly created in the MBD model. The focus here is on machine readability. It should be possible to read out in software the type of annotation (linear dimension, diameter/radius dimension, angle dimension, GD&T, . . .), the corresponding values and the assigned semantic references (these are the geometric entities to which the annotation refers, such as the two faces between which a linear dimension is placed). Annotations that make this possible are called “PMI representation”. The abbreviation “PMI” stands for “Product Manufacturing Information”. The counterpart to “representation PMI” is “presentation PMI”. In this case, the annotation is depicted in the MBD model. Consequently, humans can read the annotation, but software cannot.

This study examines the following topics:

1. The different options provided by diverse CAD systems, including Inventor, CATIA, Siemens NX, and Creo Parametric, for creating 3D annotations.

2. The impact of CAD model construction on annotations in manufacturing, particularly in terms of modifying nominal dimensions to align them with the centre of an asymmetric tolerance field.

3. The impact of annotations on the manufacturing of features like tapped holes.

4. The effect of exporting CAD models to STEP AP242 on the annotations. Further discussion of this topic is presented in the next section.

With regards to the first topic, it can be observed that CAD systems offer several options for creating a 3D annotation that are seemingly equivalent and lead to the same end result. For instance, Siemens NX has two options for creating an annotation where the first option results in creating presentation PMI that is not machine-readable and the second option results in creating representation that is machine-readable. Creo Parametric has three options for creating an annotation, namely "Show Annotations", "Dimension" and "Annotation Feature". These three options are all seemingly equivalent. However, if the first option "Show Annotations" is used then machine-readability will be lost when exporting to STEP AP242. As far as Creo Parametric is concerned, training manuals, videos and conversations with trainers show that new users are often not made aware of the consequences of using a particular option to create a 3D annotation. To make the use of MBD models smoother by improving machine-readability, it would be desirable for this to always happen.

The second topic focuses on the ease with which manufacturing can adjust the nominal value of dimensions of the model so that they are in the middle of imposed (asymmetric) tolerance fields. A distinction needs to be made between changing nominal value of dimensions in the native CAD model and in an imported STEP model. The focus here is on the native CAD model. In the native CAD model, the way in which the model is constructed, and in particular the dimensioning scheme used, has a significant influence on the ease with which the nominal value of dimensions can be modified to the centre of tolerance field. This is something that CAD users should be made aware of. Too often, designing in a CAD system is seen as working with a lump of clay, where the dimensioning scheme used does not matter. The desired dimensioning scheme is then added to the MBD model afterwards. This approach has two disadvantages. Not only does it make it more difficult for the manufacturer to adjust the nominal dimensions, but it also costs the designer more time, and because the dimensioning scheme has to be created entirely manually rather than generated directly from the features, it also increases the likelihood of errors. This can have a significant impact on the cost price, for example due to incorrect First Article Inspection Documents. MBD proponents recommend solving this problem by eliminating all asymmetric tolerances. This solves one problem but creates another. There are two reasons for this. The first reason is the fact that some asymmetric tolerances, as defined in the ISO system of limits and fits, are produced and measured by purpose-built tooling. So, while eliminating asymmetric tolerances may make the designer's job easier, it makes the manufacturer's job harder. A second reason is that asymmetric tolerances also make it easier to determine how an assembly should be built. Removing them therefore makes this more difficult.

The third topic focuses on special features such as tapped holes. What is distinctive about threaded holes is that the geometry is determined not only by what the designer has drawn, but rather by the tools used in production. The hole is pre-drilled with a specific drill and then tapped with a tap designed for the specific thread. In order to automate the creation of tapped holes, it is not only necessary that the annotation used to indicate them is machine-readable, but also that the way in which they are annotated in the MBD model is fully compliant with the standard (ISO / ASME). If the latter is not the case, the content of the annotation can be read, but it becomes very difficult for the software to interpret the content. In practice, each company often uses it own standards, which makes automation difficult. Proponents of MBD say that this problem can be solved by using the DFM methodology. DFM stands for Design For Manufacturing. This means that by using the special hole features of a CAD system, the CAM system can recognise these holes and automatically generate the corresponding drilling and tapping operations with the appropriate tools. However, this overlooks the fact that this has nothing to do with MBD as such, but rather with the feature properties of the hole function of a CAD system. Since each CAD system uses its own system for defining threads, the implication is that DFM only works where native CAD models can be used. This almost always means using a CAD and CAM system from the same vendor. This makes communicating between different stakeholders using different CAD/CAM systems difficult.

9.3 Neutral exchange formats

Several neutral exchange formats are used in the industry. The word neutral implies that these are formats that are independent of a specific CAD/CAM system. In other words, they are formats whose specifications are not proprietary but are defined by standards (ISO / ANSI). This study examines IGES, STEP, QIF and 3D PDF, focusing on STEP because it is currently one of the most widely used formats for exchanging 3D models between different CAD/CAM systems.
This study examines the following questions:

1. How is the accuracy used to construct geometry in IGES and STEP defined and how do these formats handle it?
2. As the 3D model is the authority in MBD, the question arises as to whether complex CAD geometry (spline-based shapes) can be transferred from one CAD system to another via STEP with sufficient accuracy.
3. How well are parameters defined in the CAD model that contain additional information such as the material used, the revision number of the design, among others, preserved when exported to STEP?
4. How well are feature specific properties retained when a model is exported to STEP, such as the specification of the thread in a threaded hole and the retention of the hierarchical level (part or assembly) at which a hole was made?
5. What about the use of QIF and 3D PDF for the exchange of MBD CAD models between stakeholders?

With regards to the first question, the accuracy used to build the CAD model in the native CAD/CAM system, two issues can be identified based on the tests carried out in this PhD study. A first observation is that the accuracy used to build the CAD model in the native CAD/CAM system rarely (only PTC Creo Parametric does this) matches the corresponding accuracy defined in the STEP file. A second observation is that the same accuracy, defined in the STEP file, is almost always completely ignore by the CAD/CAM system importing the STEP file. The only exception seems to be PTC Creo Parametric, which gives the user the option of applying this accuracy to the imported geometry.

As for the second question, the results of the research of this PhD study cited in section 9.1 show that it should not be assumed that the exchange of complex geometry (spline-based shapes) is simply a matter of selecting the export and import functions in the respective CAD/CAM systems. The configurations of the CAD systems involved must be carefully matched to ensure that the exchange of CAD models via STEP is efficient as possible.

The results of this doctoral research aimed at answering the third question are disappointing. Only two of the four CAD systems tested are able to export and import parameters correctly. These are CATIA v5 and PTC Creo Parametric. Siemens NX is unable to convert the parameters in the STEP file to model parameters in the imported model, but the parameters are retained in some cases. When the STEP file is generated by CATIA v5, Siemens NX is unable to read the model parameters defined in the STEP file. When the imported model is exported back to STEP and read into CATIA v5 and PTC Creo Parametric, the parameters are lost. If the STEP file is generated by PTC Creo Parametric, Siemens NX will not be able to read the model parameters. When the imported model is exported back to STEP and read into CATIA v5 and PTC Creo Parametric, the parameters are retained. Examination of the STEP files generated by CATIA v5 and PTC Creo Parametric shows that both CAD packages use a different way of defining the model parameters in the STEP file. Both methods comply with the STEP standard. Within Inventor, parameters can be defined in the CAD model, but they can only be exported to a separate file. This means that in order to read parameters defined in a STEP file, a software package must first be written to extract these parameters from the STEP file and create them in the Inventor CAD model and vice versa. As far as could be verified, the STEP interface of CATIA v5 and PTC Creo Parametric is developed in-house by Dassault Systèmes and PTC respectively. For the STEP interfaces of Autodesk Inventor and Siemens NX, both companies use the software libraries of the US company STEP Tools.

The fourth question concerns two cases:

1. How are tapped holes transferred in STEP AP242?

2. How is the hierarchical level at which a cutting operation has been performed transferred in STEP AP242?

One of the problems mentioned in section 9.2 is that each CAD system uses its own system to indicate that a hole is threaded and what its properties are. This, together with the fact that the shape of standard threads is not determined by the geometry of the CAD model but by the tool (drill, tap), is one of the reasons why the transfer of threaded holes from one CAD system to another is sub-optimal and it is very difficult for CAM systems to recognise them in order to generate the necessary CNC programmes to manufacture these holes. Two other problems are non-compliance with ISO/ASME standards for the annotation of threaded holes, where companies use their own standards, and the fact that STEP has virtually no support for threaded holes. This is something that should definitely be addressed in future versions of the standard. Proponents of MBD suggest solving this problem with DFM, which essentially amounts to feature recognition of holes. In principle, this only works if all parties are using the same CAD/CAM package. Some CAD/CAM systems offer modules that allow reading other systems' native CAD formats while retaining all features. Some CAD/CAM systems offer modules that allow other systems' native CAD formats to be read while retaining all the features. Apart from the cost, there is also the risk that these modules will not support the latest version of the file formats of the other CAD systems, or that they will become unavailable. The reason is that these modules use libraries provided by the other CAD companies. So the company developing this module is completely dependent on the other CAD companies.

In researching an answer to the fifth question, it has often been suggested that QIF could be a good alternative to STEP and a good medium for communicating information in the flow from designer to finished product. QIF stands for Quality Information

Framework and, as the name suggests, is primarily focused on quality control. The main advantage of QIF over STEP is that QIF can be used not only to pass information such as the MBD CAD model, but also to return information such as the results of a measurement report. This makes it suitable as an information carrier in the flow from designer to finished product. What makes the QIF format less suitable is the fact that very few CAD/CAM packages support it. Another potential problem is that it is often developed as a plug-in to a CAD/CAM package. Capvidia, which co-founded the QIF format, develops such plug-ins for PTC Creo Parametric, Siemens NX and SolidWorks. The problem with plug-ins is that they depend on the APIs available in the target CAD/CAM system. This means that if the APIs do not allow to retrieve all the information of a 3D annotation, such as values and semantic references, then it cannot be made available in the QIF file. The ability to create 3D PDF files is available in many CAD systems. However, this does not automatically mean that these 3D PDF files can be used for MBD. The functionality available in most CAD systems allows the CAD model and applied 3D annotations to be exported to 3D PDF as a tessellated model with PMI representation. This 3D PDF file can be viewed using the freely available Adobe Acrobat Reader package. It is human readable but not machine readable. There are several commercial plug-ins on the market to enable the latter. However, this does not mean that these 3D PDF files are immediately usable. The plug-ins, together with special software, make it possible to use the 3D PDF file as an information carrier, where the required information, the CAD model with annotations, must be extracted from the 3D PDF file and converted into a format that can be read by another software package, such as a CAM package. The limitations are the same as those discussed previously with STEP.

9.4 Software development

The original aim of the software to be developed in this PhD study was to modify the CAD model to match the specified tolerances so that it could be used directly to generate CNC toolpaths. The nominal dimension values would be brought to the centre of an asymmetric tolerance field. The aim was to develop the software independently of a CAD/CAM system based on the OpenCascade open source kernel. However, OpenCascade proved to be lacking in functionality. To speed up development, it was decided to develop the software as a plug-in to PTC Creo Parametric. During the development of the software, a number of issues were identified, such as the limitations of the different methods of creating annotations and the limitations of the available APIs. The different ways of applying 3D annotations offered to the designer in PTC Creo Parametric are presented as equivalent, but they are not. This has already been discussed in section 9.2. Limitations of the APIs were for example, the inability to retrieve semantic references in sufficient detail. These issues made it impossible to achieve the original goal within the given time frame. As a result, the objective of the software to be developed was modified. The aim was now to provide the best possible support for designers and manufacturers. The software helps designers to deliver MBD models that can be transferred via STEP AP242 with minimal problems. The software does this by listing the annotations present in the MBD model, indicating which annotations have no semantic references or have semantic references that cannot be retrieved, and also indicating which annotations will lose data (values and semantic references) when the model is exported to STEP AP242. For these annotations, the software also allows the designer to automatically convert them, where possible, to another annotation type that avoids data loss. The software helps manufacturers by allowing them, on the one hand to list all the annotations present in the model, classifying them by type (linear, diameter, asymmetric tolerance field present, GD&T, . . .) and, on the other hand, to highlight them in the CAD model with the corresponding semantic references when they are selected in the list. In this way, no annotations are overlooked and additional costs due to manufacturing errors caused by overlooked annotations are avoided. The software was tested by a number of people in local companies. Based on their input, the GUI and functionality of the software were optimised. Two problems that arose during the development are worth mentioning here. The first one is that listing all the annotations present in the MBD CAD model proved to be not so simple. When the APIs were used to scan the CAD model for 3D annotations, annotations that had previously been deleted appeared in the list. This was not the case when the CAD system was simply used by a designer, but it did occur when annotations were retrieved via software. To solve this problem, a special detection routine had to be programmed. The second one is that APIs cannot be considered stable across multiple releases of the CAD system. This means that with each new release, the source code must be re-examined and adapted to the changed or new APIs.

9.5 Contribution to knowledge

Based on the literature review (see chapter 1, chapter 2 and chapter 3), it can be concluded that following assumptions are made when applying the MBD philosophy.:

1. the 3D CAD model is the authority
2. only dimensions with tolerances that deviate from the general tolerance are applied to the model
3. the collaboration between the various parties involved in the realisation of a product, such as designers, manufacturers, quality control, can be done, if necessary, via neutral exchange formats such as STEP AP242 and QIF
4. MBD provides direct benefits to all these stakeholders. To get the most out of MBD, all annotations should be representation PMI. This means that they are machine-readable and provided with the correct semantic references when required.

The logical consequence of these assumptions is that everyone involved in the manufacture of a product must have absolute confidence that the CAD geometry can be transferred error-free from one party to another. Error-free does not mean that a CAD system will not produce error messages when the CAD model is opened. It does mean that the transfer is done in such a way that the deviations in the geometry are significantly smaller than the imposed tolerances. Publications such as Gerbino 2003 and H. Ma et al. 2006 indicate that discrepancies can occur due to, among other things, differences in the mathematical models used in different CAD systems. However, it is not specified how large these deviations can be locally in a CAD model imported into another CAD system. This thesis proposes a method to investigate this and applies it to the exchange of CAD models between a number of common CAD systems. This involved distinguishing between so-called analytical shapes and splines-based shapes and checking whether the deviations could affect the manufacture of products within the specified tolerance. The result of this study shows that exchanging geometry between CAD systems using neutral exchange formats is not as trivial as some publications make it out to be. The study also shows that almost all CAD systems do not comply with the full specification of the STEP standard. The concept of "`UNCERTAINTY_MEASURE_WITH_UNIT`", which specifies the absolute accuracy of the CAD model, is ignored. The concept of "`VALUE_FORMAT_TYPE_QUALIFIER`", which specifies the number of decimal places of a tolerance field, may contradict the value of "`UNCERTAINTY_MEASURE_WITH_UNIT`". The configuration of one CAD system must be matched to the characteristics of another CAD system in order to achieve the best possible data exchange. It is not just a matter of pressing the export and import buttons.

As mentioned above, all annotations in an MBD model are best created as a PMI representation. The literature review showed that little or no consideration was given to the procedures used to create the annotations in the CAD model. No distinction was made between the different functions provided by a CAD system for applying the annotations. This PhD study investigates whether this is actually the case in different CAD systems. The study shows that this is not entirely true. With the exception of manufacturing, this statement is true for everyone involved in the realisation of a product if they all use the same CAD system. There are two reason for this. The first is that using the same CAD system avoids data loss problems. The second reason is that, with the exception of manufacturing, the other people involved do not need to make any changes to a CAD model in order to complete their task. If manufacturing is involved or if it is necessary to use neutral exchange formats such as STEP AP242, the statement is no longer correct. In this case, the procedure used and the function used to create the annotation is important, as it makes it easier or harder to modify the model to meet manufacturing needs.

As mentioned above, the available literature argues that MBD benefits all stakeholders. Manufacturing is explicitly mentioned here, with the possibility of automatic toolpath generation given as an example. However, there is no mention of how this can be achieved. The research for this thesis has shown that the few concrete examples given by MBD proponents are based on features of a specific CAD/CAM system and cannot be clearly attributed to the MBD philosophy. One of the barriers to realising the full potential of an MBD model is the loss of data that occurs when exporting to STEP AP242 if the specific annotation features of the CAD system used are not taken into account. In this thesis, a software package has been developed that aims to remove this obstacle as far as possible by providing the user with an overview of the annotations that have been applied to the CAD model. It also indicates which annotations are incomplete or have been created in a way that causes data loss. In the latter case, the user is offered the option of automatically converting these annotations to another method that does not cause data loss.

9.6 Future work

The software development work carried out as part of this PhD ultimately resulted in a plug-in for PTC Creo Parametric. If the software wants to be independent of a specific CAD system, it would be preferable if it could be ported to other CAD systems or become a stand-alone software package. To this end, a comparative study of the APIs of the software libraries of various CAD systems such as Inventor, CATIA v5, Siemens NX and independent CAD kernels such as ACIS, Parasolid, SMLib, C3D Modeler should be carried out. The aim is to check that they provide the functionality required for the software.

Future research could also focus on adjusting toolpaths based on imposed (asymmetric) tolerance fields.

References

Articles

Abuhaiba, Ibrahim S.I. (2006). 'Efficient OCR using simple features and decision trees with backtracking'. In: *Arabian Journal for Science and Engineering* 31.2 B, pp. 223–243. ISSN: 21914281.

Agovic, A., T. Trautner and F. Bleicher (2022). 'Digital Transformation - Implementation of Drawingless Manufacturing: A Case Study'. In: *Procedia CIRP* 107. Publisher: Elsevier B.V., pp. 1479–1484. ISSN: 22128271. DOI: 10.1016/j.procir.2022.05.178. URL: https://doi.org/10.1016/j.procir.2022.05.178.

Alemanni, M., F. Destefanis and E. Vezzetti (2011). 'Model-based definition design in the product lifecycle management scenario'. In: *International Journal of Advanced Manufacturing Technology* 52.1-4, pp. 1–14. ISSN: 02683768. DOI: 10.1007/s00170-010-2699-y.

American Society of Mechanical Engineers (2019). 'Model Organization Practices'. In: Y14.47. ISBN: 9780791871768, p. 3.

Antonelli, Michele et al. (2013). 'Subdivision surfaces integrated in a CAD system'. In: *CAD Computer Aided Design* 45.11. Publisher: Elsevier Ltd ISBN: 3404867866, pp. 1294–1305. ISSN: 00104485. DOI: 10.1016/j.cad.2013.06.007. URL: http://dx.doi.org/10.1016/j.cad.2013.06.007.

Barbero, Basilio Ramos, Carlos Melgosa Pedrosa and Raúl Zamora Samperio (2017). 'Learning CAD at university through summaries of the rules of design intent'. In: *International Journal of Technology and Design Education* 27.3. Publisher: Springer ISBN: 1079801693, pp. 481–498. ISSN: 15731804. DOI: 10.1007/s10798-016-9358-z.

Boy, Jochen et al. (2014). 'CAx-IF Recommended Practices For the Representation and Presentation of Product Manufacturing Information (PMI)'. In: p. 102.

Buscei, Colette (2018). 'DMDII FINAL PROJECT REPORT'. en. In: *DMDII*, p. 79. URL: https://apps.dtic.mil/sti/pdfs/AD1077258.pdf.

Cheney, Douglas C and Bryan R Fischer (2015). 'NIST GCR 15-997 Measuring the PMI Modeling Capability in CAD Systems'. In: *National Institute of Standards and Technology, NIST-GCR* 15. Publisher: NIST, p. 77.

Chern, Shiing-Shen (1990). 'What is geometry?' In: *The American Mathematical Monthly* 97.8, pp. 679–686. URL: https://www.jstor.org/stable/2324574.

Conover, Jonathan S. and Ibrahim Zeid (2006). 'Development of a prototype for transfer of drawing annotations into the ASME Y14.41 standard'. In: *American Society of Mechanical Engineers, Computers and Information in Engineering Division, CED*. ISBN: 0791837904, pp. 1–8. ISSN: 10716947. DOI: 10.1115/IMECE2006-15323.

Ding, Shuhui et al. (2021). 'MBD Based 3D CAD Model Automatic Feature Recognition and Similarity Evaluation'. In: *IEEE Access* 9. Publisher: IEEE, pp. 150403–150425. ISSN: 21693536. DOI: 10.1109/ACCESS.2021.3126333.

Fang, Frank Zhigang et al. (2016). 'Closed Loop PMI Driven Dimensional Quality Lifecycle Management Approach for Smart Manufacturing System'. In: *Procedia CIRP* 56. Publisher: The Author(s), pp. 614–619. ISSN: 22128271. DOI: 10.1016/j.procir.2016.10.121. URL: http://dx.doi.org/10.1016/j.procir.2016.10.121.

Feeney, Allison Barnard, Simon P. Frechette and Vijay Srinivasan (2015). 'A Portrait of an ISO STEP Tolerancing Standard as an Enabler of Smart Manufacturing Systems'. In: *Journal of Computing and Information Science in Engineering* 15.2, pp. 1–5. ISSN: 15309827. DOI: 10.1115/1.4029050.

Fischer, Kevin, Phil Rosche and Asa Trainer (2015). 'Investigating the Impact of Standards-Based Interoperability for Design to Manufacturing and Quality in the Supply Chain'. In: *NIST Grants and Contracts Report* NISTGCR 15-1009. Publisher: NIST. DOI: 10.6028/NIST.GCR.15-1009.

Gallaher, Michael P et al. (2004). 'Cost Analysis of Inadequate Interoperability in the U.S. Capital Facilities Industry'. In: *Nist*. Publisher: NIST ISBN: NIST GCR 04-867, pp. 1–210. ISSN: <null>. URL: papers2://publication/uuid/69C8B354-4830-4874-929E-ACBCC00E3204.

Garcia, Chris and Paul Perreault (June 2011). 'Who needs 2-D Drawings'. In: *Appliance Design*, pp. 27–29. ISSN: 15525937. URL: http://search.proquest.com/docview/877006751/.

Giachetti, Ronald E. (1999). 'Standard manufacturing information model to support design for manufacturing in virtual enterprises'. In: *Journal of Intelligent Manufacturing* 10.1. Publisher: National Institute of Standards and Technology, pp. 49–60. ISSN: 09565515. DOI: 10.1023/A:1008916530350. URL: http://dx.doi.org/10.1023/A:1008916530350.

Gielingh, Wim (2008). 'An assessment of the current state of product data technologies'. In: *CAD Computer Aided Design* 40.7. Publisher: Elsevier, pp. 750–759. ISSN: 00104485. DOI: 10.1016/j.cad.2008.06.003.

Gopalakrishnan, Saikiran, Nathan W. Hartman and Michael D. Sangid (2020). 'Model-Based Feature Information Network (MFIN): A Digital Twin Framework to Integrate Location-Specific Material Behavior Within Component Design, Manufacturing, and Performance Analysis'. In: *Integrating Materials and Manufacturing Innovation* 9.4. Publisher: Springer International Publishing, pp. 394–409. ISSN: 21939772. DOI: 10.1007/s40192-020-00190-4. URL: https://doi.org/10.1007/s40192-020-00190-4.

Gu, Hong et al. (2001). 'Identifying, correcting, and avoiding errors in computer-aided design models which affect interoperability'. In: *Journal of Computing and Information Science in Engineering* 1.2, pp. 156–166. ISSN: 15309827. DOI: 10.1115/1.1384887.

Hardwick, M. et al. (2013). 'A roadmap for STEP-NC-enabled interoperable manufacturing'. In: *International Journal of Advanced Manufacturing Technology* 68.5-8, pp. 1023–1037. ISSN: 14333015. DOI: 10.1007/s00170-013-4894-0.

Hedberg, Thomas et al. (2016). 'Testing the digital thread in support of model-based manufacturing and inspection'. In: *Journal of Computing and Information Science in Engineering* 16.2, pp. 1–10. ISSN: 15309827. DOI: 10.1115/1.4032697.

Hedberg Jr, Thomas and Mark Carlisle (2019). 'Proceedings of the 10th model-based enterprise summit (MBE 2019)'. In: *NIST Advanced Manufacturing Series* Mbe. Publisher: NIST, pp. 100–124. DOI: 10.6028/NIST.AMS.100-24.

Hudspeth, Mike (2006). 'Transition from 2D to 3D'. In: *Cadalyst* 23.9, pp. 39–41. ISSN: 08205450.

Huhtala, Merja, Mika Lohtander and Juha Varis (2012). 'Confusing of terms PDM and PLM : examining issues from the PDM point of view .' In: *Flexible Automation and Intelligent Manufacturing, FAIM2012* November 2016.

Husted, Ernie (2019). 'GD&T, Manufacturing Imperative'. In: *Quality* 58.3. Publisher: BNP Media, pp. 24–25.

International, ECMA (2007). 'Universal 3D File Format'. In: *ECMA Standards CN - 0000* June. ISBN: ECMA-363, p. 237.

Islam, M. N. (2004). 'Functional dimensioning and tolerancing software for concurrent engineering applications'. In: *Computers in Industry* 54.2. Publisher: Elsevier, pp. 169–190. ISSN: 01663615. DOI: 10.1016/j.compind.2003.09.006.

ISO (1989). '2768-1'. In: Publisher: ISO. URL: https://www.iso.org/standard/7748.html.

— (2006). '16792:2006 Technical product documentation – Digital product definition data practices'. In: *Iso*. ISBN: 978 0 539 03769 2.

Jack, Hugh (2013). 'Manufacturing Design'. In: *Engineering Design, Planning, and Management*. Publisher: Academic Press ISBN: 978-0-12-397158-6, pp. 419–468. DOI: 10.1016/b978-0-12-397158-6.00011-5. URL: https://www.sciencedirect.com/science/article/pii/B9780123971586000115.

Jankauskas, Kestutis (2010). 'Time-Efficient NURBS Curve Evaluation Algorithms'. In: *Evaluation* 1.5, pp. 60–69.

Jian, Chengfeng et al. (Dec. 2023). 'QSCC: A Quaternion Semantic Cell Convolution Graph Neural Network for MBD Product Model Classification'. en. In: *IEEE Transactions on Industrial Informatics* 19.12, pp. 11477–11486. ISSN: 1551-3203, 1941-0050. DOI: 10.1109/TII.2023.3246066. URL: https://ieeexplore.ieee.org/document/10050112/ (visited on 24/06/2024).

Jing, Xuwen et al. (2020a). 'Research on the intelligent generation method of MBD model 3D marking using predefined features'. In: *Concurrent Engineering Research and Applications* 28.3, pp. 222–238. ISSN: 15312003. DOI: 10.1177/1063293X20958920.

— (2020b). 'Research on the intelligent generation method of MBD model 3D marking using predefined features'. In: *Concurrent Engineering Research and Applications* 28.3. Publisher: SAGE Publications Sage UK: London, England, pp. 222–238. ISSN: 15312003. DOI: 10.1177/1063293X20958920.

La Course, Don (2001). 'Developer roundtable : STEP vs . IGES'. In: *Cadalyst*, pp. 24–30.

Lenne, Dominique, Indira Thouvenin and Stéphane Aubry (2009). 'Supporting design with 3D-annotations in a collaborative virtual environment'. In: *Research in Engineering Design* 20.3. Publisher: Springer, pp. 149–155. ISSN: 09349839. DOI: 10.1007/s00163-009-0071-8.

Lipman, Robert (2017). 'STEP file analyzer software'. In: *Journal of Research of the National Institute of Standards and Technology* 122.16, pp. 1–2. ISSN: 21657254. DOI: 10.6028/jres.122.016.

Lipman, Robert and Joshua Lubell (2015). 'Conformance checking of PMI representation in CAD model STEP data exchange files'. In: *CAD Computer Aided Design* 66. Publisher: Elsevier Ltd, pp. 14–23. ISSN: 00104485. DOI: 10.1016/j.cad.2015.04.002. URL: http://dx.doi.org/10.1016/j.cad.2015.04.002.

Liu, Fang and Li Hong Qiao (Apr. 2012). 'Product Information Modeling and Organization with MBD'. en. In: *Applied Mechanics and Materials* 163, pp. 221–225. ISSN: 1662-7482. DOI: 10.4028/www.scientific.net/AMM.163.221. URL: https://www.scientific.net/AMM.163.221 (visited on 26/07/2024).

Ma, Homan et al. (2006). 'TESTING SEMANTIC INTEROPERABILITY'. en. In: URL: https://itc.scix.net/pdfs/w78-2006-tf193.pdf.

Maghsoodloo, Saeed and Ming Hsien Li (2000). 'Optimal asymmetric tolerance design'. In: *IIE Transactions (Institute of Industrial Engineers)* 32.12. Publisher: Springer ISBN: 0740817000, pp. 1127–1137. ISSN: 15458830. DOI: 10.1080/07408170008967467.

Mattei, David (1993). 'New version of IGES supports B-Rep solids'. In: *Mechanical Engineering* 115.1. Publisher: American Society of Mechanical Engineers ISBN: 00256501, p. 50. URL: http://search.proquest.com/docview/230147336?accountid=15533.

Menin, Stefan and Kenneth J. Korane (2012). 'Working with dimensional tolerances'. In: *Machine Design* 84.7, pp. 64–66. ISSN: 00249114.

Michaloski, John (2016). 'First Article Inspection Requirement Report Generation from QIF Using C ++ , CodeSynthesis , and Mozilla Xerces'. In: Publisher: git hub usnistgov.

Morey, Bruce (2020). 'Reduce Cost, Increase Speed with Model-Based Definition'. In: *Manufacturing Engineering* 165.12. Publisher: SOC MANUFACTURING ENGINEERS, pp. 46–51. ISSN: 0361-0853. URL: https://www.sme.org/technologies/articles/2020/december/reduce-cost-increase-speed-with-model-based-definition/?ite=2522&ito=2433&itq=e7e28286-9020-4706-8ec4-21d7883bf037&itx%5Bidio%5D=1144843.

Morris, Robert (2007). 'Enhance First Article Inspection'. In: *Quality* 46.7, S8.

Nnaji, Bart O., Yan Wang and Kyoung-Yun Kim (2004). 'Cost-Effective Product Realization: Service-Oriented Architecture for Integrated Product Life-cycle Management'. In: *IFAC Proceedings Volumes* 37.5. Publisher: Elsevier, pp. 1–12. ISSN: 14746670. DOI: 10.1016/s1474-6670(17)32337-6.

Omar, Mohamed K., Sharmeeni Murugan and Rohana Abdullah (2011). 'The impact of tolerance limit on cost of quality'. In: *IEEE International Conference on Industrial Engineering and Engineering Management*. Publisher: IEEE ISBN: 9781457707391, pp. 1500–1504. ISSN: 21573611. DOI: 10.1109/IEEM.2011.6118167.

Page, Cover (2006). 'Us pro'. In: *Exchange Organizational Behavior Teaching Journal* 12.

Peak, Russell S. et al. (2004). 'STEP, XML, and UML: Complementary technologies'. In: *Journal of Computing and Information Science in Engineering* 4.4, pp. 379–390. ISSN: 15309827. DOI: 10.1115/1.1818683.

Peng, Zhiguo et al. (2020). 'A new method for interoperability and conformance checking of product manufacturing information'. In: *Computers and Electrical Engineering* 85. Publisher: Elsevier Ltd. ISSN: 00457906. DOI: 10.1016/j.compeleceng.2020.106650.

Pfouga, Alain and Josip Stjepandić (2018). 'Leveraging 3D geometric knowledge in the product lifecycle based on industrial standards'. In: *Journal of Computational Design and Engineering* 5.1. Publisher: Oxford University Press, pp. 54–67. ISSN: 22885048. DOI: 10.1016/j.jcde.2017.11.002.

Pratt, Michael J. (2001). 'Introduction to iso 10303—the step standard for product data exchange'. In: *Journal of Computing and Information Science in Engineering* 1.1. Publisher: American Society of Mechanical Engineers New York, NY, USA, pp. 102–103. ISSN: 15309827. DOI: 10.1115/1.1354995.

Pratt, Michael J. and Bill D. Anderson (2001). 'A shape modelling applications programming interface for the STEP standard'. In: *CAD Computer Aided Design* 33.7, pp. 531–543. ISSN: 00104485. DOI: 10.1016/S0010-4485(01)00052-5.

Qin, Yuchu et al. (2017). 'Status, Comparison, and Issues of Computer-Aided Design Model Data Exchange Methods Based on Standardized Neutral Files and Web Ontology Language File'. In: *Journal of Computing and Information Science in Engineering* 17.1. Publisher: American Society of Mechanical Engineers Digital Collection. ISSN: 15309827. DOI: 10.1115/1.4034325.

Quintana, Virgilio, Louis Rivest, Robert Pellerin and Fawzi Kheddouci (2012). 'Re-engineering the Engineering Change Management process for a drawing-less environment'. In: *Computers in Industry* 63.1. Publisher: Elsevier B.V., pp. 79–90. ISSN:

01663615. DOI: 10.1016/j.compind.2011.10.003. URL: http://dx.doi.org/10.1016/j.compind.2011.10.003.

Quintana, Virgilio, Louis Rivest, Robert Pellerin, Frédérick Venne et al. (2010). 'Will Model-based Definition replace engineering drawings throughout the product life-cycle? A global perspective from aerospace industry'. In: *Computers in Industry* 61.5, pp. 497–508. ISSN: 01663615. DOI: 10.1016/j.compind.2010.01.005.

Ramnath, Satchit et al. (2020). 'Interoperability of CAD geometry and product manufacturing information for computer integrated manufacturing'. In: *International Journal of Computer Integrated Manufacturing* 33.2. Publisher: Taylor & Francis, pp. 116–132. ISSN: 13623052. DOI: 10.1080/0951192X.2020.1718760. URL: https://doi.org/10.1080/0951192X.2020.1718760.

Rinos, Konstantinos et al. (2021). 'Implementation of model-based definition and product data management for the optimization of industrial collaboration and productivity'. In: *Procedia CIRP* 100. Publisher: Elsevier B.V., pp. 355–360. ISSN: 22128271. DOI: 10.1016/j.procir.2021.05.082. URL: https://doi.org/10.1016/j.procir.2021.05.082.

Saal, Christopher et al. (2021). 'New Methods for Continuous Communication within the 3D-Model-Based Process Chain'. In: *ISSE 2021 - 7th IEEE International Symposium on Systems Engineering, Proceedings*. Publisher: IEEE ISBN: 9781665431682, pp. 1–7. DOI: 10.1109/ISSE51541.2021.9582500.

Sarkar, Arkopaul and Dušan Šormaz (2019). 'On semantic interoperability of model-based definition of product design'. In: *Procedia Manufacturing* 38.Faim 2019. Publisher: Elsevier B.V. ISBN: 1740593154, pp. 513–523. ISSN: 23519789. DOI: 10.1016/j.promfg.2020.01.065. URL: https://doi.org/10.1016/j.promfg.2020.01.065.

Schleich, Benjamin et al. (2018). 'Geometrical Variations Management 4.0: Towards next generation geometry assurance'. In: *Procedia CIRP* 75. Publisher: Elsevier B.V., pp. 3–10. ISSN: 22128271. DOI: 10.1016/j.procir.2018.04.078. URL: https://doi.org/10.1016/j.procir.2018.04.078.

Steinbrenner, J. P., N. J. Wyman and J. R. Chawner (2001). 'Procedural CAD model edge tolerance negotiation for surface meshing'. In: *Engineering with Computers* 17.3. Publisher: Springer, pp. 315–325. ISSN: 01770667. DOI: 10.1007/PL00013392.

Unver, Ertu (2006). 'Strategies for the Transition to CAD Based 3D Design Education'. In: *Computer-Aided Design and Applications* 3.1-4, pp. 323–330. ISSN: 16864360. DOI: 10.1080/16864360.2006.10738470.

Venne, Frédérick, Louis Rivest and Alain Desrochers (2010). 'Assessment of 3D annotation tools as a substitute for 2D traditional engineering drawings in aerospace product development'. In: *Computer-Aided Design and Applications* 7.4, pp. 547–563. ISSN: 16864360. DOI: 10.3722/cadaps.2010.547-563.

Weill, R (1988). 'R. weill'. In: *Robotics and Computer-Integrated Manufacturing* 4.1-2. Publisher: Elsevier, pp. 41–48.

Xu, Tongming et al. (2015). 'Automatic tool path generation from structuralized machining process integrated with CAD/CAPP/CAM system'. In: *International Journal of Advanced Manufacturing Technology* 80.5-8, pp. 1097–1111. ISSN: 14333015. DOI: 10.1007/s00170-015-7067-5.

Yoders, Jeff (2007). 'Acrobat 3D 8 makes 3D/CAD file-sharing easier'. In: *Building Design \& Construction VO - 48* 10, pp. 1–3. ISSN: 0007-3407.

Zhao, Yaoyao Fiona et al. (2012). 'Quality Information Framework - Integrating metrology processes'. In: *IFAC Proceedings Volumes (IFAC-PapersOnline)* 14.PART 1. Publisher: Elsevier ISBN: 9783902661982, pp. 1301–1308. ISSN: 14746670. DOI: 10.3182/20120523-3-RO-2023.00113. URL: http://dx.doi.org/10.3182/20120523-3-RO-2023.00113.

Zhong, Wenhao et al. (2021). 'A MBD Model-driven Automatic Generation Method of Onmachine Measuring Program for CNC Machining'. In: *Journal of Physics: Conference Series* 2095.1. ISSN: 17426596. DOI: 10.1088/1742-6596/2095/1/012045.

Books

Beckers, Raphael et al. (2016). *Interoperability and visualization of complex products based on JT standard*. Vol. 4. Publication Title: Advances in Transdisciplinary Engineering. IOS Press. ISBN: 978-1-61499-702-3. DOI: 10.3233/978-1-61499-703-0-828.

Childs, P.R.N. (2021). 'Tolerancing and Precision Engineering'. en. In: *Mechanical Design*. Elsevier, pp. 413–449. ISBN: 978-0-12-821102-1. URL: https://linkinghub.elsevier.com/retrieve/pii/B9780128211021000111 (visited on 22/07/2024).

Henzold, Georg (2020). *Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection: A Handbook for Geometrical Product Specification Using ISO and ASME Standards, Third Edition*. Publication Title: Geometrical Dimensioning and Tolerancing for Design, Manufacturing and Inspection: A Handbook for Geometrical Product Specification Using ISO and ASME Standards, Third Edition. Butterworth-Heinemann. ISBN: 978-0-12-824061-8. DOI: 10.1016/C2020-0-01081-9.

Herron, J B (2013). *Re-use your CAD : the model-based CAD handbook*. Action Engineering. ISBN: 978-1-4948-7717-0.

Nasr, Emad Abouel and Ali K. Kamrani (2007). *Initial Graphics Exchange Specifications (IGES)*. Publication Title: Computer-Based Design and Manufacturing. Springer US. ISBN: 978-0-387-23324-6. DOI: 10.1007/978-0-387-23324-6_6. URL: https://doi.org/10.1007/978-0-387-23324-6_6.

Nyffenegger Felix et al. (2020). 'Product Lifecycle Management Enabling Smart X'. In: *Product Lifecycle Management Enabling Smart X*. Ed. by Felix Nyffenegger et al. Vol. 594. Springer International Publishing, pp. 604–617. ISBN: 978-3-030-62806-2. DOI: 10.1007/978-3-030-62807-9. URL: http://link.springer.com/10.1007/978-3-030-62807-9.

Ram, Peethani Sai and K. Deepak Lawrence (2021). 'Implementation of Quality Information Framework (QIF): Towards Automatic Generation of Inspection Plan from Model-Based Definition (MBD) of Parts'. en. In: *Industry 4.0 and Advanced Manufacturing*. Ed. by Amaresh Chakrabarti and Manish Arora. Series Title: Lecture Notes in Mechanical Engineering. Singapore: Springer Singapore, pp. 127–138. ISBN: 9789811556883 9789811556890. DOI: 10.1007/978-981-15-5689-0_12. URL: http://link.springer.com/10.1007/978-981-15-5689-0_12 (visited on 15/11/2023).

Saxena, Anupam and Birendra Sahay (2005). *Computer aided engineering design*. en. New York: Springer. ISBN: 978-1-4020-2555-6.

Ullmann, David G (2003). *Th mechanical design process*. ISBN: 978-0-9993578-0-4.

Zhou, Jiwei, Jorge D. Camba and Travis Fuerst (2022). *A Comparative Study on the Use and Interpretation of Annotated 3D Models*. Vol. 640 IFIP. Publication Title: IFIP Advances in Information and Communication Technology ISSN: 1868422X. Springer International Publishing. ISBN: 978-3-030-94398-1. DOI: 10.1007/978-3-030-94399-8_23. URL: http://dx.doi.org/10.1007/978-3-030-94399-8_23.

Proceedings

Alducin-Quintero, Gerardo et al. (2012). '3D model annotation as a tool for improving design intent communication: A case study on its impact in the engineering change process'. In: *Proceedings of the ASME Design Engineering Technical Conference*. Vol. 2.

Issue: PARTS A AND B. American Society of Mechanical Engineers, pp. 349–356. ISBN: 978-0-7918-4501-1. DOI: 10.1115/DETC2012-70872.

Bijnens, John, Karel Kellens and David Cheshire (2018). 'Accuracy of geometry data exchange using STEP AP242'. In: *Procedia CIRP*. Vol. 78. ISSN: 22128271. Elsevier, pp. 219–224. ISBN: 978-1-5108-7569-2. DOI: 10.1016/j.procir.2018.09.048.

Cam, C a D (2000). 'Geometric Modeling Problems in Industrial CAD / CAM / CAE'. In: *Simulation*, pp. 1–10.

Chambolle, Frédéric (2013). 'The usage of the standards into the long term archiving and retrieval, and the exchange of engineering design data'. In: *IFIP Advances in Information and Communication Technology*. Vol. 409. ISSN: 1868422X. Springer, pp. 685–694. ISBN: 978-3-642-41500-5. DOI: 10.1007/978-3-642-41501-2_67.

Gerbino, S (2003). 'Tools for the interoperability among CAD systems'. In: *XIII ADM - XV INGEGRAF International Conference on "Tools and methods evolution in engineering design"*, p. 13.

Hedberg, Thomas (2017). 'Engineering a $100B Paradigm Shift: Economic and System Drivers to Interoperability Innovation'. In: *Global Product Data Interoperability Summit*, p. 47.

Herron, Jennifer, Lionel Andujar and Ryan Gelotte (2019). 'A qif case study-maintaining the digital thread from oem to supplier'. In: *NIST MBE Summit April 1-4*, p. 7.

Lachenmaier, Jens F., Heiner Lasi and Hans Georg Kemper (2015). 'A concept for extracting and sharing technical data from digital product models for subsequent processing'. In: *Proceedings of the Annual Hawaii International Conference on System Sciences*. Vol. 2015-March. ISSN: 15301605. IEEE, pp. 987–996. ISBN: 978-1-4799-7367-5. DOI: 10.1109/HICSS.2015.122.

Ma, Weiyin and Nailiang Zhao (2000). 'Catmull-Clark Surface Fitting for Reverse Engineering Applications Weiyin Ma and Nailiang Zhao'. In: *Proceedings Geometric Modeling and Processing 2000. Theory and Applications*. IEEE, pp. 274–283.

Miralbes-Buil, Ramón et al. (2020). 'The Model Based Definition Features - MBD Applied to the Technical Computer Aid Design and Drawing'. In: *Lecture Notes in Mechanical Engineering*. ISSN: 21954364. Logroño, Spain: Springer, pp. 105–114. ISBN: 978-3-030-41199-2. DOI: 10.1007/978-3-030-41200-5_12.

Otey, Jeffrey M. et al. (2014). 'A review of the design intent concept in the context of CAD model quality metrics'. In: *ASEE Annual Conference and Exposition, Conference Proceedings*. ISSN: 21535965, pp. 24–100. DOI: 10.18260/1-2--19992.

Sandberg, Martin, Satyanarayana Kokkula and Gerrit Muller (2019). 'Transitioning from technical 2D drawings to 3D models: a case study at defense systems'. In: *INCOSE International Symposium*. Vol. 29. Issue: 1 ISSN: 2334-5837, pp. 879–894. DOI: 10.1002/j.2334-5837.2019.00641.x.

Sohlenius, G. (1992). 'Concurrent Engineering'. In: *CIRP Annals - Manufacturing Technology*. Vol. 41. Issue: 2 ISSN: 17260604, pp. 645–655. DOI: 10.1016/S0007-8506(07)63251-X.

Trainer, Asa et al. (June 2016). 'Gaps Analysis of Integrating Product Design, Manufacturing, and Quality Data in the Supply Chain Using Model-Based Defintion'. en. In: *Volume 2: Materials; Biomanufacturing; Properties, Applications and Systems; Sustainable Manufacturing*. Blacksburg, Virginia, USA: American Society of Mechanical Engineers, V002T05A003. ISBN: 978-0-7918-4990-3. DOI: 10.1115/MSEC2016-8792. URL: https://asmedigitalcollection.asme.org/MSEC/proceedings/MSEC2016/49903/Blacksburg,%20Virginia,%20USA/269036 (visited on 27/07/2024).

TTF Group (2003). 'TTF Showcases PRC , the New Intelligent Compressed Format , at DMS in Tokyo from June 25 th to 27 th , 2003'. In: vol. 33. Issue: 0. TTF Group.

Wardhani, Rivai and Xun Xu (2016). 'Model-based manufacturing based on STEP AP242'. In: *MESA 2016 - 12th IEEE/ASME International Conference on Mechatronic* *and Embedded Systems and Applications - Conference Proceedings*. IEEE, pp. 1–5. ISBN: 978-1-5090-6190-7. DOI: 10.1109/MESA.2016.7587187.

Other Sources

Adobe (2023). *Adding 3D models to PDFs (Acrobat Pro)*. (Visited on 24/04/2023).

AIF (2010). *AEROSPACE: First Article Inspection*. Pages: 1-18. URL: https://www.sae.org/standards/content/as9102b/ (visited on 06/04/2022).

Bergen, Ron (2002). *Color coding*. Pages: 90 Volume: 32 Issue: 2 ISSN: 15388689. DOI: 10.1097/00152193-200202000-00078. (Visited on 16/07/2023).

CADimensions (2022). *MBD is Here: 10 Reasons Why Everyone is Moving Away From Traditional 2D Drawings*. URL: https://www.cadimensions.com/knowledge-base/why-the-industry-is-transitioning-to-mbd-from-drawings/ (visited on 25/05/2023).

CAPVidia (Jan. 2014). *MBDVidia*. Company. URL: http://www.solution-lab.co.kr/down/MBDVidia_brochure.pdf?PHPSESSID=187e7cef7018f3c0272bf34da6512a32 (visited on 19/11/2023).

— (2022). *MBDVidia, Easy & Powerful MBD Workflow CAD Translation Software*. URL: https://www.capvidia.com/products/mbdvidia (visited on 22/04/2022).

Capvidia (2016). *Auto Generation of AS9102 First Article Inspection (FAI) Form in MBDVidia*. URL: https://www.capvidia.com/blog/333-auto-generation-of-as9102-first-article-inspection-fai-form-in-mbdvidia (visited on 22/08/2022).

Dassault Systèmes (2022). *Setting Small Scale in CATIA V5 Session*. URL: http://catiadoc.free.fr/online/cfyuggen_C2/cfyuggen01.htm (visited on 19/07/2022).

Doorakkers, Rob (2019). *Model Based Definition: nice-to-have or must-have for an automated, flawless industry 4.0 production environment?* Publication Title: IGS GeboJagema.

EACPDS, Team (Aug. 2017). *Getting Started with PTC Creo Web.Link*. (Visited on 09/08/2023).

Fridman, Michael (2019). *MODEL-BASED DEFINITION (LifeWorx '19 Presentation)*.

Gregorio, J, C Lorch and J Hughes (Apr. 2023). *Model Based Definition – Recommendations for Supply Chain Integration*. en. Tech. rep. Edition: 2. National Physical Laboratory. DOI: 10.47120/npl.MS46. URL: http://eprintspublications.npl.co.uk/id/eprint/9640 (visited on 31/07/2024).

Guthrie CAD/GIS Software (2022). *First Article Inspection (FAI) | The Beginner's Guide for 2022*. URL: https://www.guthcad.com/blog/first-article-inspection.html (visited on 05/04/2022).

IBM (2003). *Hd13614: Step / Export / Accuracy Can'T Be Changed : 0.005 By Default / Uncertainty_Measure_With_Unit(Length_Measure(0.005)*. URL: https://www.ibm.com/support/pages/apar/HD13614 (visited on 11/11/2022).

Insight Team (2020). *What is First Article Inspection and Why is it important?* URL: https://insight-quality.com/what-is-first-article-inspection/ (visited on 05/04/2022).

Insize (2023). *Plain plug gages*. URL: http://www.insize.com/page-58-590.html (visited on 08/07/2023).

ISO 10303-203 (2022). URL: https://www.iso.org/search.html?q=ISO%2010303-203&hPP=10&idx=all_en&p=0 (visited on 09/09/2022).

ISO 10303-214 (2022). URL: https://www.iso.org/search.html?q=ISO%2010303-214&hPP=10&idx=all_en&p=0 (visited on 09/09/2022).

Jimmy Nguyen (June 2023). *Top 8 Neutral 3D CAD File Formats*. URL: https://www.capvidia.com/blog/top-neutral-3d-cad-file-formats (visited on 02/11/2023).

Loffredo, Dave (2017). *Version 17 Software Released*. URL: https://www.steptools.com/blog/20170420_release_v17/ (visited on 05/03/2023).

LOTAR (2023). *LOTAR*. URL: https://lotar-international.org/why-lotar/ (visited on 06/03/2023).

Made-in Europe (2022). *Zo kunnen jobbers aanhaken bij de MBD trend in de maakindustrie*. URL: https://www.made-in-europe.nu/2022/06/zo-kunnen-jobbers-aanhaken-bij-de-mbd-trend-in-de-maakindustrie/ (visited on 09/03/2023).

Metrology News (2020). *New ISO Standard Offers Integrated Model for Manufacturing Quality Information*. URL: https://metrology.news/new-iso-standard-offers-integrated-model-for-manufacturing-quality-information/ (visited on 22/04/2023).

Mohammed, Shafi Khurieshi (2023). 'Model Based Definition for Robotic Assembly'. en. PhD thesis. Norwegian University of Science and Technology. URL: https://ntnuopen.ntnu.no/ntnu-xmlui/bitstream/handle/11250/3111307/Shafi%20Khurieshi%20Mohammed.pdf?sequence=3.

Nascimento, Raphael (2017). *MBD: Standalone Annotations vs. Annotation Features*. (Visited on 09/03/2023).

Open CASCADE Technology (2023). *STEP Translator*. URL: https://dev.opencascade.org/doc/overview/html/occt_user_guides__step.html (visited on 19/08/2023).

OPEN MIND Technologies (2019). *HyperMILL, CAM strategies and functions for efficient manufacturing*. Pages: 80.

PDES (1998). *Recommended Practices for AP 203*. Tech. rep. Publication Title: Representations. PDES, Inc. Industry consortium.

PTC (2023a). *TPI 32869 : Detailed information regarding model accuracy*. Pages: 1-5. URL: https://support.ptc.com/appserver/cs/view/solution.jsp?n=32869 (visited on 31/01/2023).

— (2023b). *About Family Tables*. URL: https://support.ptc.com/help/creo/creo_pma/r8.0/usascii/index.html#page/fundamentals/fundamentals/fund_ten_sub/About_Family_Tables_1.html%23 (visited on 04/09/2023).

— (2023c). *About RuleCHECK*. English. URL: https://support.ptc.com/help/creo/creo_pma/r9.0/usascii/index.html#page/model_analysis/modelcheck/About_RuleCHECK.html#wwID0ESSZ0 (visited on 18/09/2023).

— (2022a). *About STEP Export Profile Options and the Corresponding Configuration Options*. URL: https://support.ptc.com/help/creo/creo_pma/r9.0/usascii/index.html#page/data_exchange/interface/step_export_profile_and_config_options.html (visited on 16/10/2022).

— (2023d). *About User-Defined Features*. URL: https://support.ptc.com/help/creo/creo_pma/r8.0/usascii/index.html#page/part_modeling/part_modeling/part_five_sub/About_User_Defined_Features_part.html# (visited on 06/09/2023).

— (2023e). *Conditional Statements in Relations*. URL: https://support.ptc.com/help/creo/creo_pma/r8.0/usascii/index.html#page/fundamentals/fundamentals/fund_seven_sub/Conditional_Statements_in_Relations.html%23wwID0EOM52 (visited on 04/09/2023).

— (2023f). *Creo Object TOOLKIT Java User's Guide 9.0.2.0*. URL: https://www.ptc.com/support/-/media/support/refdocs/Creo_Parametric/9,-d-,0/otk_javaug_Creo9000.pdf?sc_lang=en&source=search (visited on 09/09/2023).

— (2022b). *Which versions of STEP can be import or export in Creo Parametric*. URL: https://www.ptc.com/en/support/article/cs57599 (visited on 16/10/2022).

PTC Inc. (2021). *Creo ® Parametric Web.Link™ User's Guide 7.0.4.0*. English. User manual. PTC, p. 481.

Quality Magazine (2013). *DMSC Submits QIF v1.0 for ANSI Standardization*. URL: https://www.qualitymag.com/articles/91313-dmsc-submits-qif-v10-for-ansi-standardization (visited on 22/04/2023).

Ramesh, Madhavi (2017). *Creo Combination States and Model-Based Definition (MBD)*. URL: https://www.ptc.com/en/blogs/cad/combination-states-and-mbd (visited on 29/05/2023).

Rowe, Jeff (2017). *A Conversation With Oboe Wu On the Future Of SOLIDWORKS MBD*. URL: https://www10.mcadcafe.com/blogs/jeffrowe/2017/03/09/a-conversation-with-oboe-wu-on-the-future-of-solidworks-mbd/ (visited on 24/05/2023).

Sandvik Coromant (2022). *CoroReamer 835 835.T-0600-A1-PF 1024*. URL: https://www.sandvik.coromant.com/en-gb/tools/boring-and-reaming-tools/reamers/cororeamer-835 (visited on 08/07/2023).

Sangole, Archana (2000). 'Investigation of Microgeometry Flaws in Design Data Exchange'. PhD thesis. The University of Western Ontario.

Smith, Tony (2004). *Intel touts 'MP3 for 3D' universal graphics format*. URL: https://www.theregister.com/2004/04/21/intel_u3d/ (visited on 25/04/2023).

Tetra4D (2023). *Tetra4D Converter Help*. Pages: 1-67. URL: http://www.tetra4d.com/tetra-4d-converter/.

TransMagic (2022). *TransMagic STEP Viewer*. URL: https://transmagic.com/step-viewer/ (visited on 11/04/2022).

Appendix A

A.1 Handling of model accuracy (IGES)

A.1.1 A. Export from Inventor 2022

A cube is created with dimensions 0.004 mm × 0.004 mm × 0.004 mm and exported to IGES (see Figure A.1).

The smallest distance between two vertices is 0.004 mm. Thus, the expected absolute accuracy is at most 0.004. However, as shown in Figure A.2, the file has an accuracy of 0.01 mm.

To study the effect of this incorrect accuracy (0.01 mm instead of 0.004 mm), the IGES file was read into all the CAD systems for this test.

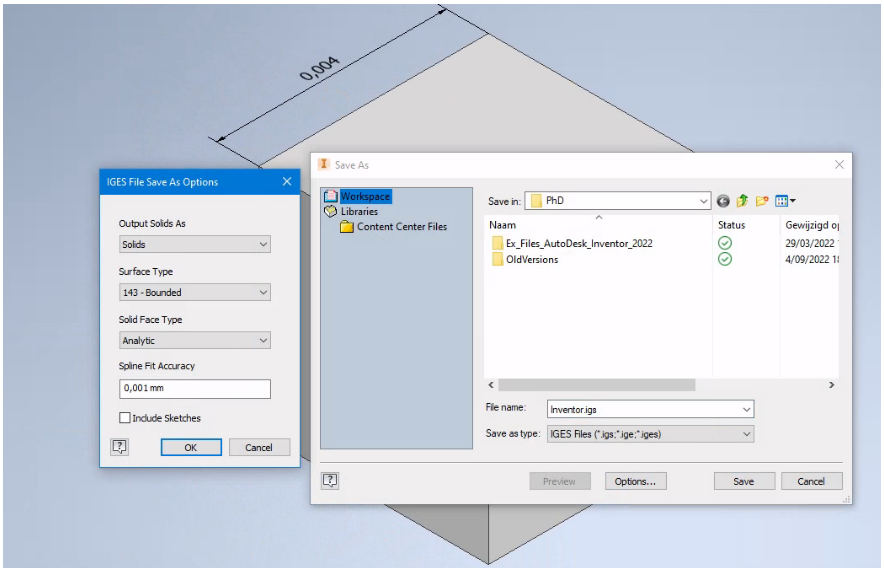
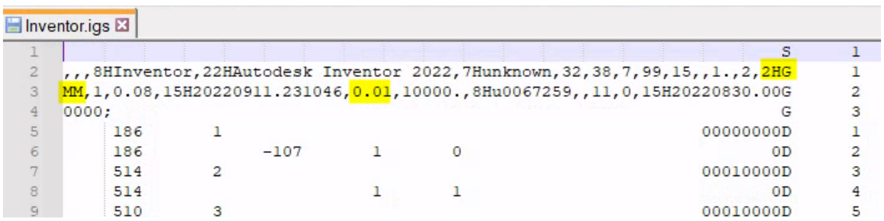


Figure A.1: Inventor 2022 model is exported to IGES


Inventor.igs

```
1                                                                          S      1
2  ,,,8HInventor,22HAutodesk Inventor 2022,7Hunknown,32,38,7,99,15,,1.,2,2HG      1
3  MM,1,0.08,15H20220911.231046,0.01,10000.,8Hu0067259,,11,0,15H20220830.00G      2
4  0000;                                                                    G      3
5       186       1                                                 00000000D      1
6       186            -107        1        0                             0D      2
7       514       2                                                 00010000D      3
8       514                        1        1                             0D      4
9       510       3                                                 00010000D      5
```

Figure A.2: IGES file created by Inventor 2022 has an absolute accuracy of 0.01 mm

Import in CATIA V5-6R2022 SP1

The default options for IGES import were used (see Figure A.3).

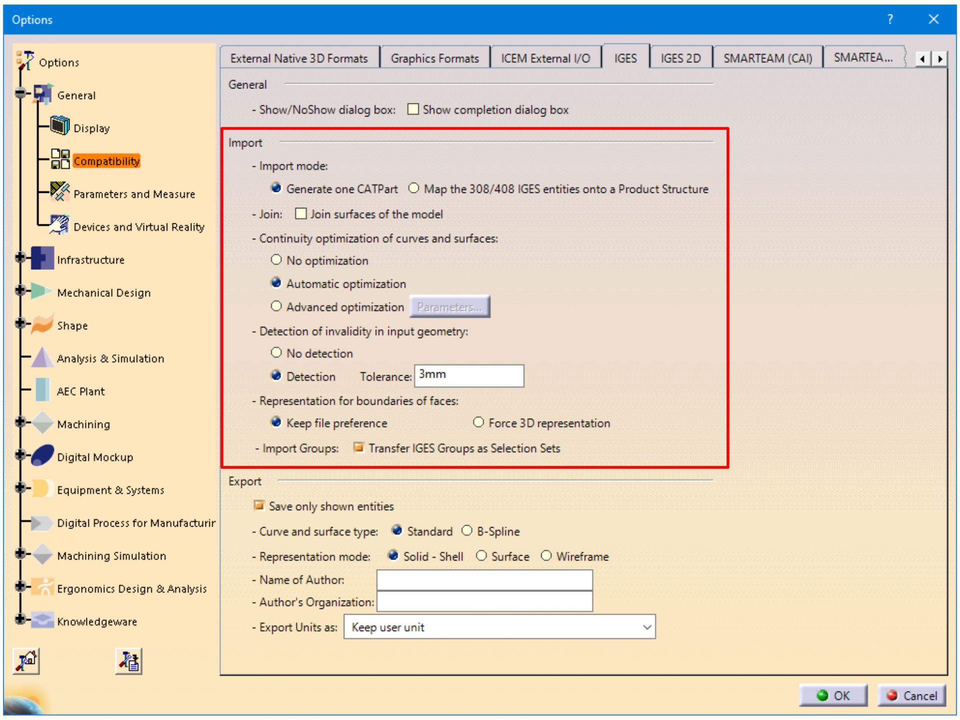


Figure A.3: Default settings for importing IGES files in CATIA v5

Reading in the IGES file fails completely (see Figure A.4).

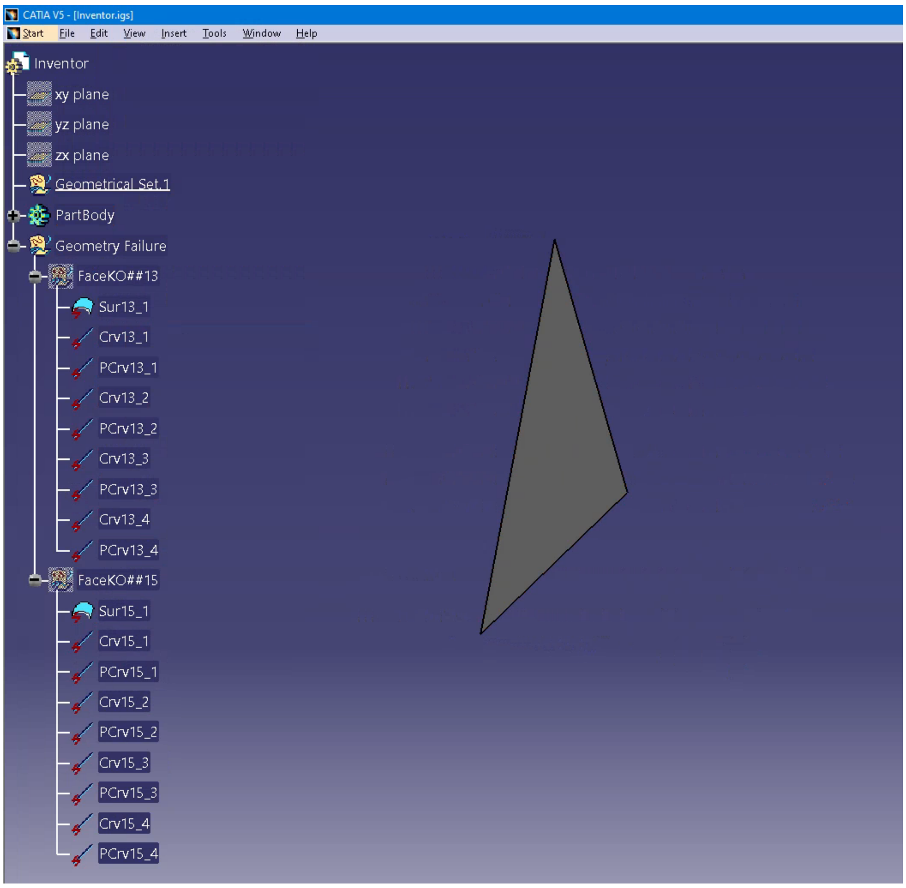


Figure A.4: Importing the IGES file in CATIA V5 fails completely

To check if this is really caused by an incorrect accuracy in the IGES file, this is changed manually from 0.01 to 0.004. Then the file is re-read in CATIA V5. The reconstruction of the geometry still fails completely. So there is another reason for this failure. To find out the accuracy used internally, the model was exported back to IGES and the accuracy defined here was checked. This accuracy is 0.001 mm, which is the internal accuracy CATIA uses for ordinary (medium-sized) models. By default, Inventor exports a solid in IGES to “Solid” (see Figure 6.3). Although this is allowed by the IGES specification (Mattei 1993; Page 2006), this is an option not supported by all CAD systems. The cube is again exported to IGES in Inventor 2022, but this time the “Output Solids As” option is set to “Surfaces” (see Figure A.5). The result can be seen in Figure A.6

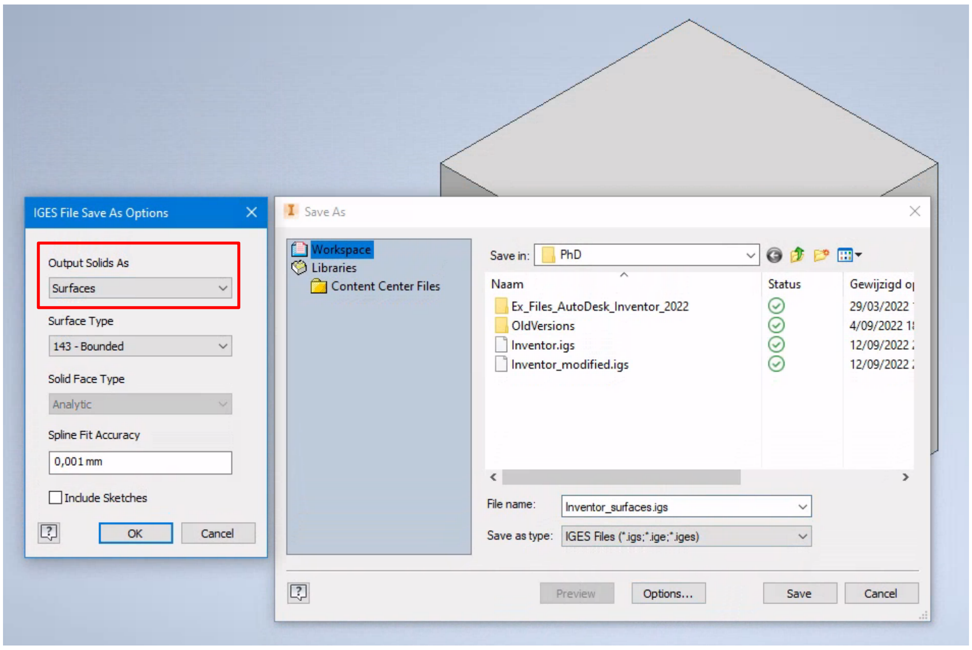


Figure A.5: Exporting solids as surfaces by Inventor 2022

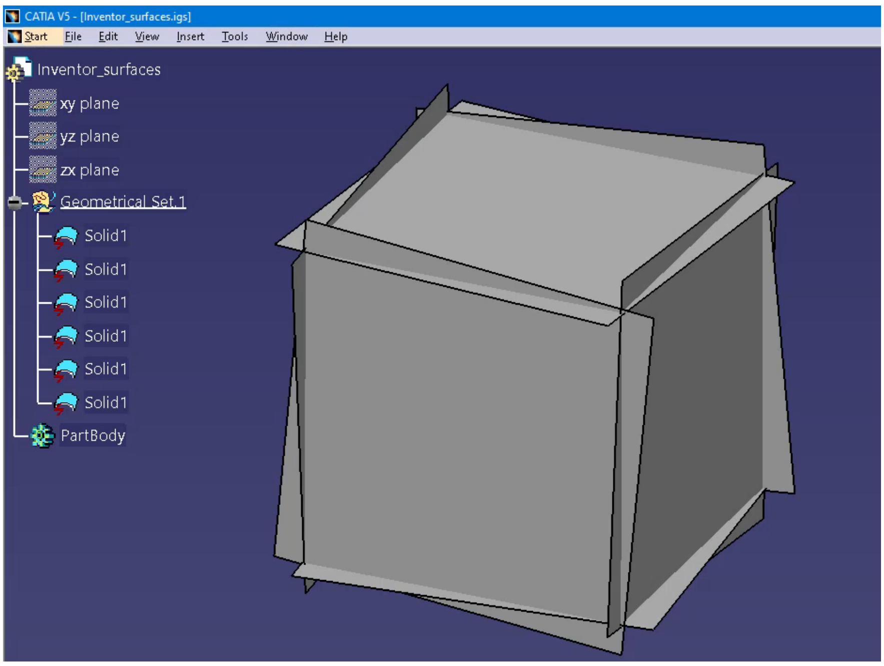


Figure A.6: After exporting the solid as surfaces, there is an improvement, but still not good enough

The fact that the surfaces do not match may be caused by the accuracy being incorrectly specified in the IGES file or by the way the surfaces are defined in the IGES file. Manually changing the accuracy from 0.01 to 0.004 gives the same result as in Figure A.6. In contrast, when the surfaces are defined as "Trimmed" instead of "Bounded" (see Figure A.7), the result is correct. Six square surfaces are now obtained (see Figure A.8). These still need to be converted into a solid in CATIA V5. Clearly, the accuracy specified in the IGES file is ignored.

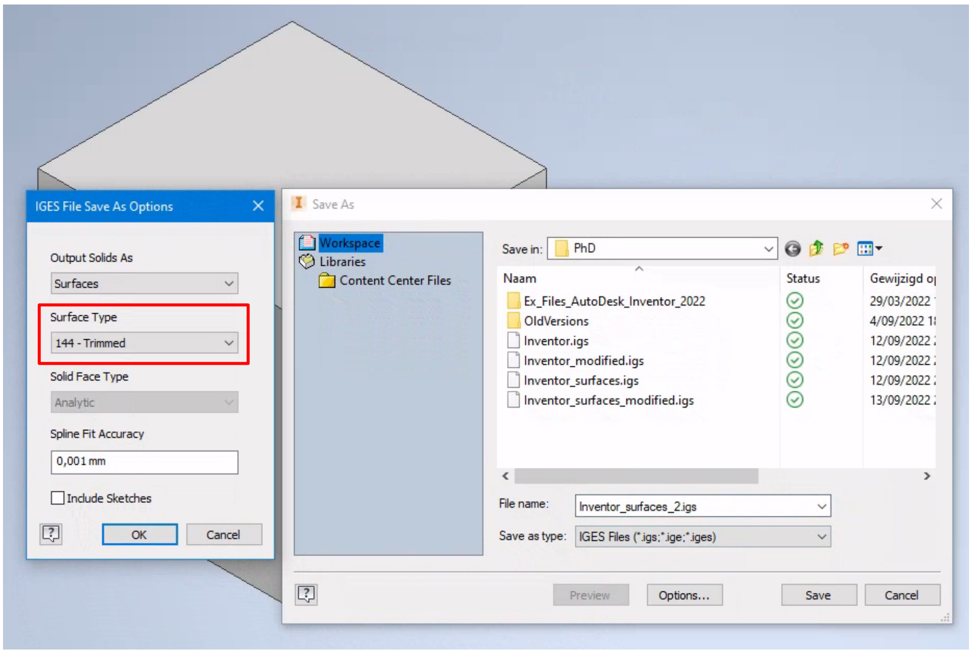


Figure A.7: Defining surfaces as "trimmed surfaces" when exporting by Inventor 2022

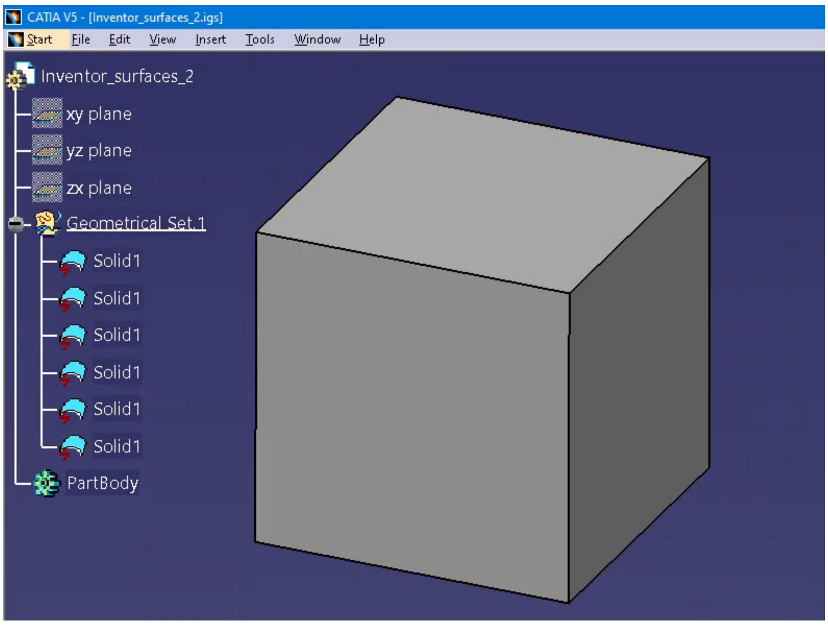


Figure A.8: After exporting the solid as trimmed surfaces, the result is correct

Table A.1: A summary of the results for CATIA V5 for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	No	No	No	No	Yes	Yes
Export[1] accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Solid recognised	No	No	-	-	-	-

[1] The imported model is re-exported to IGES and the accuracy defined in this IGES file is examined.

Import in Siemens NX Version 2019 Build 2501

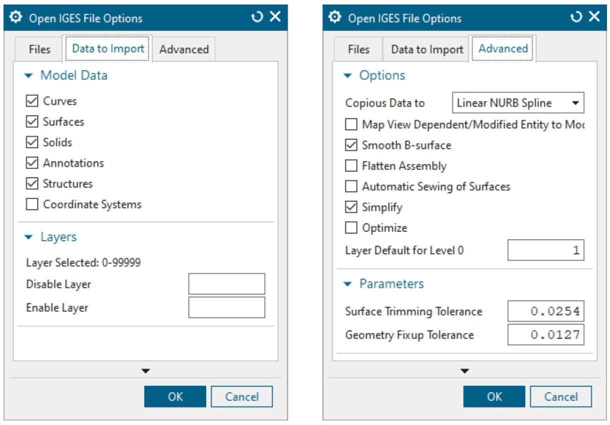


Figure A.9: Settings used to import IGES file in Siemens NX

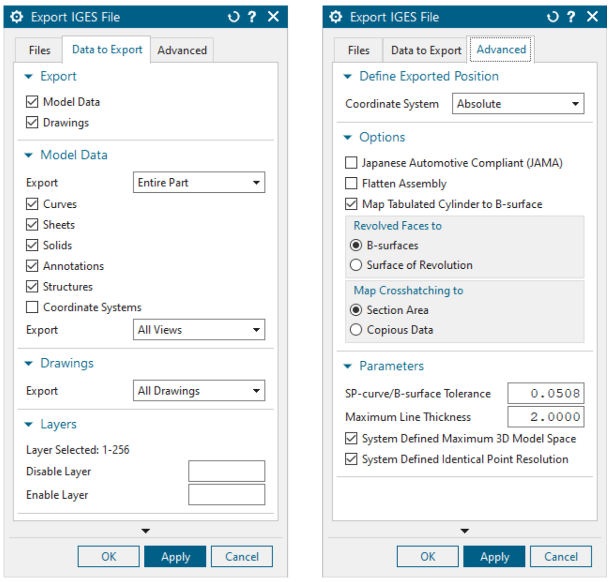


Figure A.10: Settings used to export IGES file in Siemens NX

Table A.2: A summary of the results for Siemens NX Version 2019 for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	Yes	Yes	Yes	Yes	Yes	Yes
Export accuracy	1.0E-05	1.0E-05	1.0E-05	1.0E-05	1.0E-05	1.0E-05
Solid recognised	Yes	Yes	-	-	-	-

Import in PTC Creo Parametric 8.0.4.0

PTC Creo Parametric has several configuration options (see Figure A.11) for importing neutral exchange files:

- template : the accuracy defined within the template is used
- automatic : PTC Creo tries to determine the required accuracy automatically
- external : the incoming model accuracy will be used even if a template is used
- internal : this sets the internally preferred value. For most formats, this is the default relative accuracy (see subsection 2.2.2) which is 0.0012.

There is an option to request the current model accuracy. This is co-displayed in Table A.3 below.

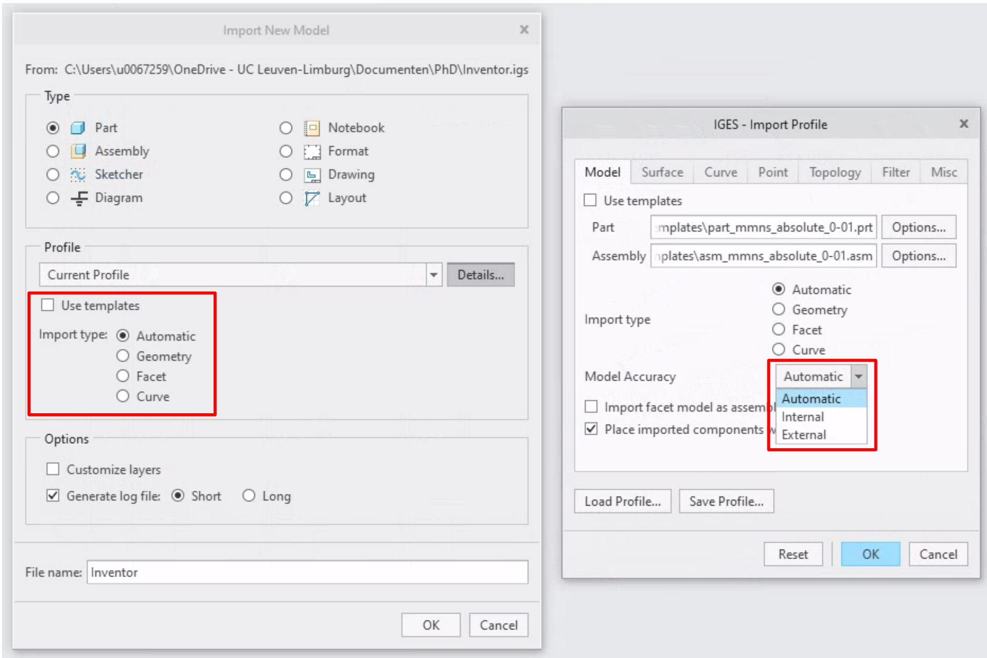


Figure A.11: Configuration options for importing neutral exchange files in PTC Creo

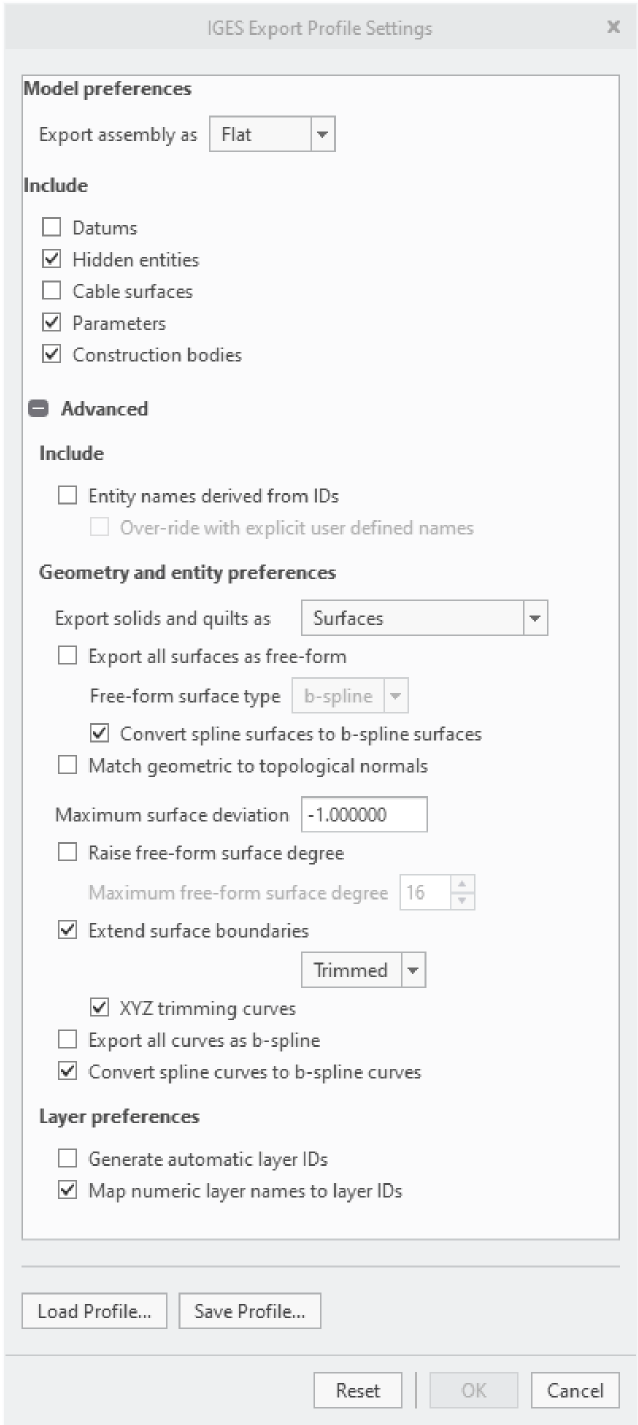


Figure A.12: Settings used to export IGES file in PTC Creo

Table A.3: A summary of the results for PTC Creo Parametric 8.0.4.0 (“No template - Automatic”) for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	Yes	Yes	Yes	Yes	Yes	Yes
Export accuracy	6.93e-07	6.93e-07	6.96e-07	6.96e-07	6.96e-07	6.96e-07
Solid recognised	Yes	Yes	-	-	-	-
Resulting accuracy	6.93-07	6.93e-07	6.96e-07	6.96e-07	6.96e-07	6.96e-07

Table A.4: A summary of the results for PTC Creo Parametric 8.0.4.0 (“No template - Internal”) for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	Yes	Yes	Yes	Yes	Yes	Yes
Export accuracy	6.93e-07	6.93e-07	6.96e-07	6.96e-07	6.96e-07	6.96e-07
Solid recognised	Yes	yes	-	-	-	-
Resulting accuracy	6.93-07	6.93e-07	6.96e-07	6.96e-07	6.96e-07	6.96e-07

Table A.5: A summary of the results for PTC Creo Parametric 8.0.4.0 (“No template - External”) for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	Yes	Yes	Yes	Yes	Yes	Yes
Export accuracy	5.77e-06	5.77e-06	5.77e-06	5.77e-06	6.12e-06	6.12e-06
Solid recognised	Yes	Yes	-	-	-	-
Resulting accuracy	5.77e-06	5.77e-06	5.77e-06	5.77e-06	6.12e-06	6.12e-06

Table A.6: A summary of the results for PTC Creo Parametric 8.0.4.0 ("Template (absolute accuracy 0.01 mm) - Automatic") for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	No	No	No	No	No	No
Export accuracy	0.01	0.01	0.01	0.01	0.01	0.01
Solid recognised	No	No	-	-	-	-
Resulting accuracy	0.01	0.01	0.01	0.01	0.01	0.01

Table A.7: A summary of the results for PTC Creo Parametric 8.0.4.0 ("Template (absolute accuracy 0.001 mm) - Automatic") for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	Yes	Yes	No	No	No	No
Export accuracy	0.001	0.001	0.01	0.00	0.001	0.001
Solid recognised	Yes	Yes	-	-	-	-
Resulting accuracy	0.001	0.001	0.001	0.001	0.001	0.001

Import in Inventor 2022

The default settings for IGES import were used (see Figure A.13).

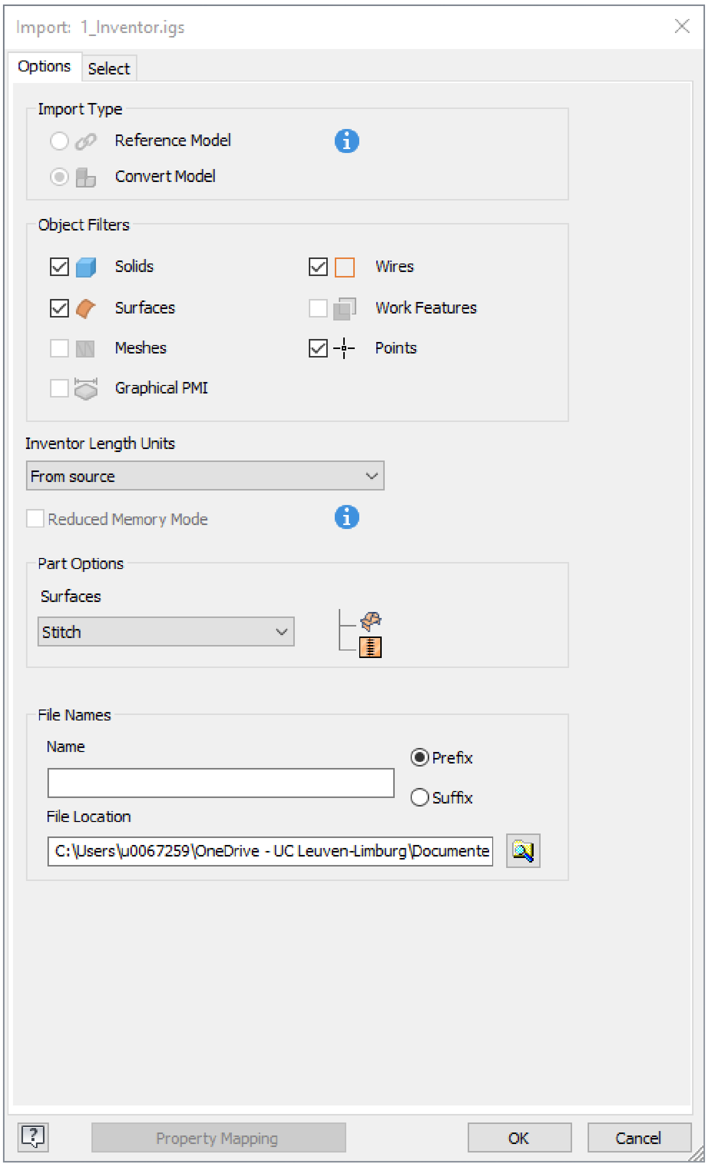


Figure A.13: Default settings for importing IGES files in Inventor 2022

Table A.8: A summary of the results for Inventor 2022 for the import of the IGES file generated by Inventor 2022

IGES settings						
Output Solids	Solids	Solids	Surfaces	Surfaces	Surfaces	Surfaces
Surface Type	Bounded	Bounded	Bounded	Bounded	Trimmed	Trimmed
Solid Face Type	Analytic	Analytic	Analytic	Analytic	Analytic	Analytic
Spline Fit Accuracy	0.001	0.001	0.001	0.001	0.001	0.001
Accuracy IGES file	0.01	0.004	0.01	0.004	0.01	0.004
Import successful	Yes	Yes	Yes	Yes	Yes	Yes
Solid recognised	Yes	Yes	-	-	-	-

A.1.2 B. Export from CATIA V5-6R2022 SP1

A cube is created with the aforementioned dimensions and exported to IGES. As in Inventor 2022, the designer can activate specific options in CATIA for export to IGES (see Figure A.14). Two representation modes will be tested “Solid - Shell” and “Surface”.

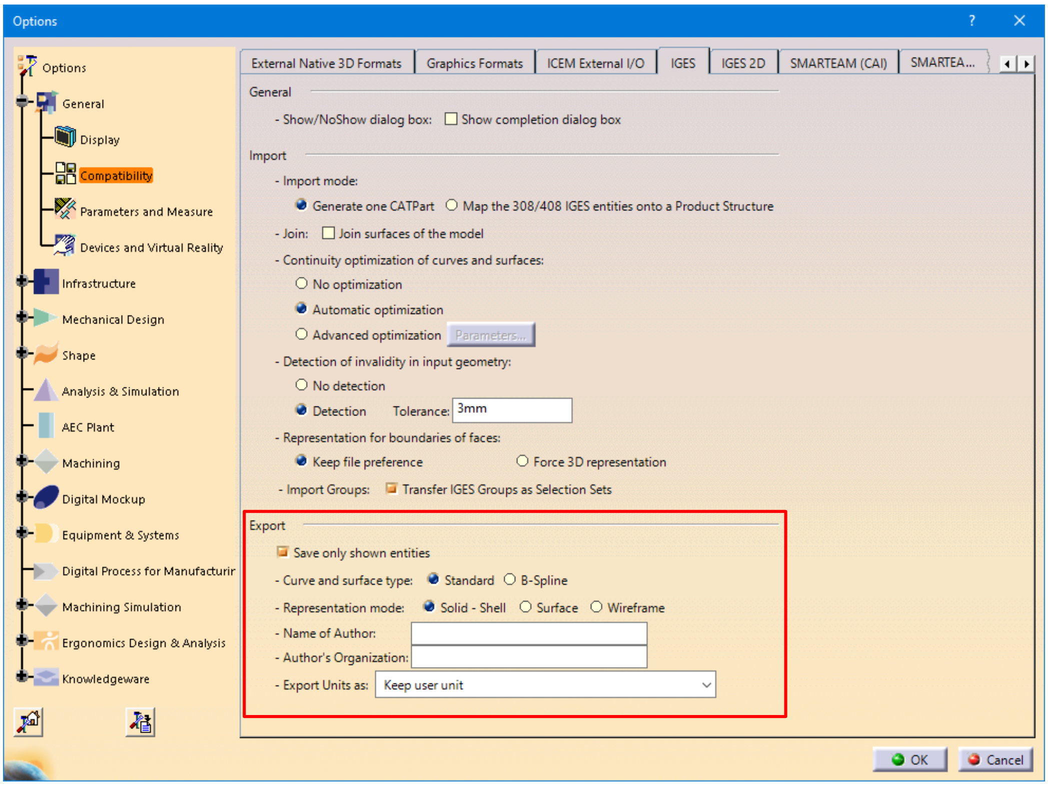


Figure A.14: Possible settings for exporting IGES files in CATIA v5

As the dimensions of the cube are 0.004 mm × 0.004 mm × 0.004 mm, the smallest distance between two vertices is 0.004 mm. Thus, the expected absolute accuracy is at most 0.004. As shown in Figure A.15, the file has an accuracy of 0.001 mm. This is the default accuracy used by CATIA to create a model (see Figure 2.1). So in this case, the accuracy specified in the IGES file matches that of the native CAD model.

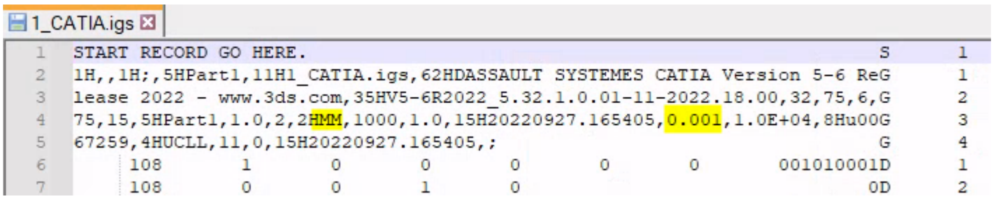

```
1_CATIA.igs
1  START RECORD GO HERE.                                               S      1
2  1H,,1H;,5HPart1,11H1_CATIA.igs,62HDASSAULT SYSTEMES CATIA Version 5-6 ReG      1
3  lease 2022 - www.3ds.com,35HV5-6R2022_5.32.1.0.01-11-2022.18.00,32,75,6,G      2
4  75,15,5HPart1,1.0,2,2HMM,1000,1.0,15H20220927.165405,0.001,1.0E+04,8Hu00G      3
5  67259,4HUCLL,11,0,15H20220927.165405,;                                G      4
6       108       1       0       0       0       0       0       001010001D      1
7       108       0       0       1       0                               0D      2
```

Figure A.15: IGES file created by CATIA v5 has an absolute accuracy of 0.001 mm

Import in Inventor 2022

The default settings for IGES import were used (see Figure A.13).

Table A.9: A summary of the results for Inventor 2022 for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	0.01	0.01
Solid recognised	Yes	-

Import in Siemens NX Version 2019 Build 2501

The default settings for IGES import were used (see Figure A.9)

Table A.10: A summary of the results for Siemens NX Version 2019 Build 2501 for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	1.0E-05	1.0E-05
Solid recognised	No	-

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, PTC Creo has several configuration options for importing neutral exchange files.

Table A.11: A summary of the results for PTC Creo Parametric 8.0.4.0 ("No template - Automatic") for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	6.93e-07	6.93e-07
Solid recognised	Yes	-
Resulting accuracy	6.93e-07	6.93e-07

Table A.12: A summary of the results for PTC Creo Parametric 8.0.4.0 ("No template - Internal") for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	6.93e-07	6.93e-07
Solid recognised	Yes	-
Resulting accuracy	6.93e-07	6.93e-07

Table A.13: A summary of the results for PTC Creo Parametric 8.0.4.0 ("No template - External") for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	5.77e-06	5.77e-06
Solid recognised	Yes	-
Resulting accuracy	5.77e-06	5.77e-06

Table A.14: A summary of the results for PTC Creo Parametric 8.0.4.0 ("Template (absolute accuracy 0.01 mm) - Automatic") for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	No	No
Export accuracy	0.01	0.01
Solid recognised	-	-
Resulting accuracy	0.01	0.01

Table A.15: A summary of the results for PTC Creo Parametric 8.0.4.0 ("Template (absolute accuracy 0.001 mm) - Automatic") for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	0.001	0.001
Solid recognised	Yes	-
Resulting accuracy	0.001	0.001

Import in CATIA V5-6R2022 SP1

The default settings for IGES import were used (see Figure A.3).

Table A.16: A summary of the results for CATIA v5 for the import of the IGES file generated by CATIA v5

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	No	Yes
Export accuracy	0.001	0.001
Solid recognised	-	-

A.1.3 C. Export from Siemens NX Version 2019 Build 2501

A cube is created with the aforementioned dimensions and exported to IGES. To make a cube with such small dimensions, the designer must specify an appropriate accuracy before making the model (see Figure A.16). The active absolute accuracy is determined by the formula

$$accuracy = Distance\ Tolerance \cdot Optimize\ Curve\ Distance\ Tolerance\ Factor = 0.001$$

As in Inventor 2022, the designer can activate specific options in Siemens NX for export to IGES. The default settings that can be seen in Figure A.10 were used. To limit exports to surfaces, the "Solids" option is unchecked.

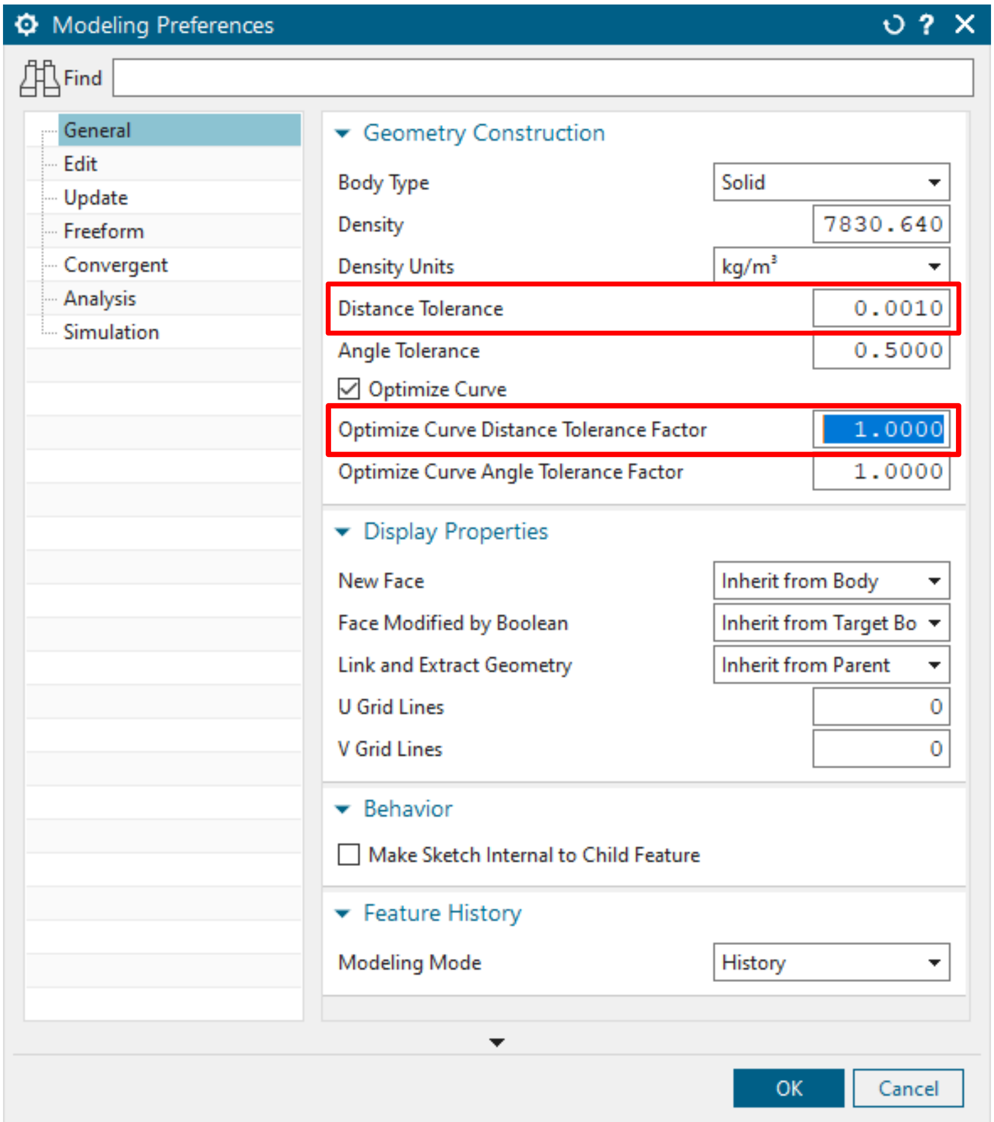


Figure A.16: Preferences specified for the creation of a new model in Siemens NX

Import in Inventor 2022

The default settings for IGES import were used (see Figure A.13).

Table A.17: A summary of the results for Inventor 2022 for the import of the IGES file generated by Siemens NX Version 2019

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	Yes	No
Export accuracy	0.01	-
Solid recognised	Yes	-

Import in CATIA V5-6R2022 SP1

The default options for IGES import were used (see Figure A.3).

Table A.18: A summary of the results for CATIA V5 for the import of the IGES file generated by Siemens NX Version 2019

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	Yes	No, the end result is only one single point
Export accuracy	0.001	0.001
Solid recognised	No	-

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, PTC Creo has several configuration options for importing neutral exchange files.

Table A.19: A summary of the results for PTC Creo Parametric 8.0.4.0 ("No template - Automatic") for the import of the IGES file generated by Siemens NX Version 2019

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	Yes	No, the end result is only one single point
Export accuracy	6.96e-07	0.087
Solid recognised	Yes	-
Resulting accuracy	6.96e-07	0.087

Table A.20: A summary of the results for PTC Creo Parametric 8.0.4.0 ("No template - Internal") for the import of the IGES file generated by Siemens NX Version 2019

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	Yes	No, the end result is only one single point
Export accuracy	6.96e-07	0.087
Solid recognised	Yes	-
Resulting accuracy	6.96e-07	0.087

Table A.21: A summary of the results for PTC Creo Parametric 8.0.4.0 ("No template - External") for the import of the IGES file generated by Siemens NX Version 2019

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	Yes	No, the end result is only one single point
Export accuracy	6.12e-06	0.007
Solid recognised	Yes	-
Resulting accuracy	6.12e-06	0.007

Table A.22: A summary of the results for PTC Creo Parametric 8.0.4.0 ("Template (absolute accuracy 0.01 mm) - Automatic") for the import of the IGES file generated by Siemens NX Version 2019

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	No, the end result is only one single point	No, the end result is only one single point
Export accuracy	0.01	0.01
Solid recognised	-	-
Resulting accuracy	0.01	0.01

Table A.23: A summary of the results for PTC Creo Parametric 8.0.4.0 ("Template (absolute accuracy 0.001 mm) - Automatic") for the import of the IGES file generated by Siemens NX Version 2019

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	Yes	No, the end result is only one single point
Export accuracy	0.001	0.001
Solid recognised	Yes	-
Resulting accuracy	0.001	0.001

Import in Siemens NX Version 2019 Build 2501

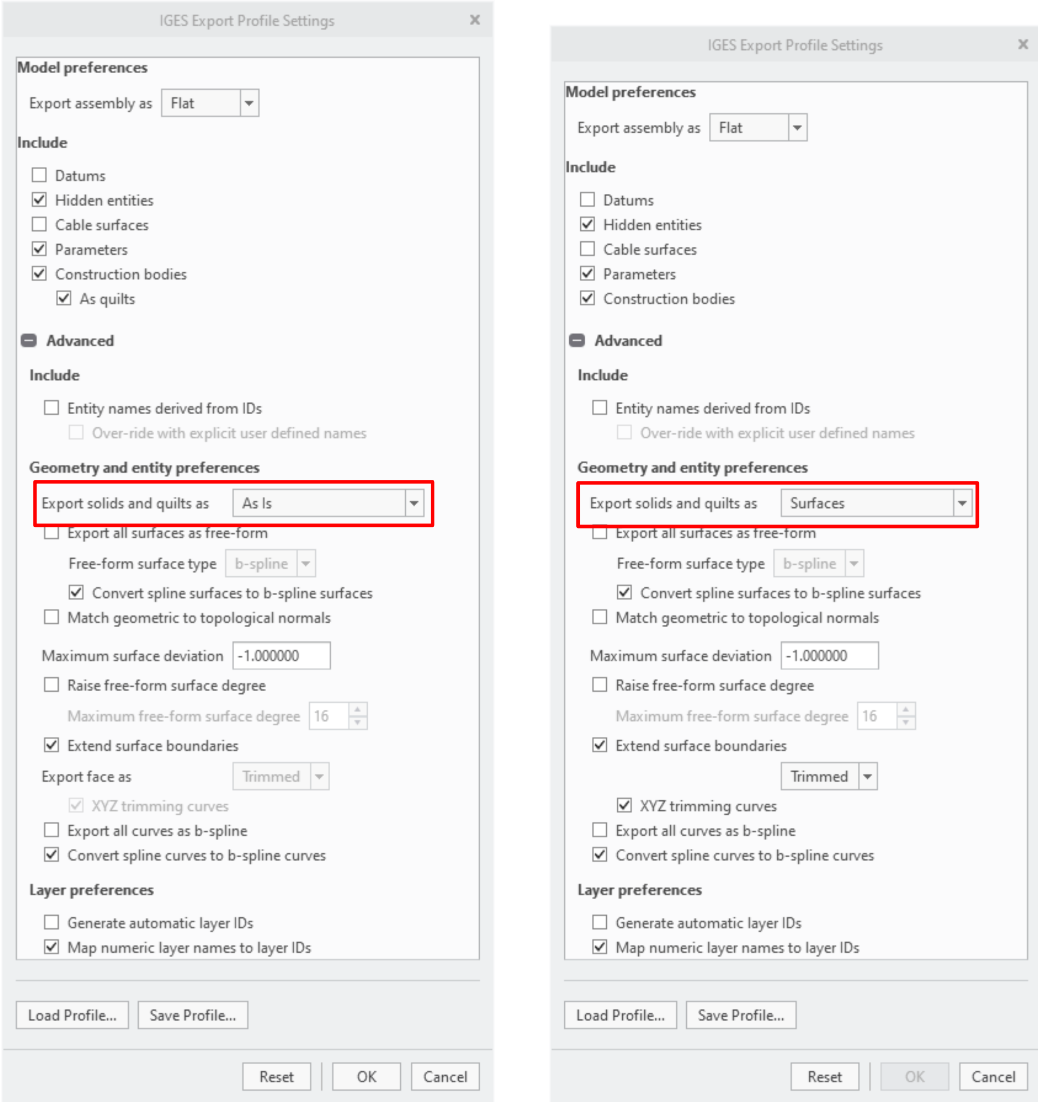
The default settings for IGES import were used (see Figure A.9)

Table A.24: A summary of the results for Siemens NX Version 2019 Build 2501 for the import of the IGES file generated by Siemens NX Version 2019 Build 2501

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	1.0E-05	1.0E-05
Import successful	Yes	No, the end result is only one single point
Export accuracy	1.0E-05	1.0E-05
Solid recognised	No	-

A.1.4 D. Export from PTC Creo Parametric 8.0.4.0

To create the cube in PTC Creo, a part template with an absolute accuracy of 0.001 mm is used.



(a) Settings used for export to solids

(b) Settings used for export to surfaces

Figure A.17: Settings used to export model to IGES in PTC Creo Parametric

Import in Inventor 2022

The default settings for IGES import were used (see Figure A.13).

Table A.25: A summary of the results for Inventor 2022 for the import of the IGES file generated by PTC Creo Parametric 8.0.4.0

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	0.01	0.01
Solid recognised	Yes	-

Import in CATIA V5-6R2022 SP1

The default options for IGES import were used (see Figure A.3).

Table A.26: A summary of the results for CATIA V5 for the import of the IGES file generated by PTC Creo Parametric 8.0.4.0

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	No, the end result is a blend between a square and a triangle	Yes
Export accuracy	0.001	0.001
Solid recognised	Yes	-

Import in Siemens NX Version 2019 Build 2501

The default settings for IGES import were used (see Figure A.9)

Table A.27: A summary of the results for Siemens NX for the import of the IGES file generated by PTC Creo

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	1.0E-05	1.0E-05
Solid recognised	Yes	-

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, PTC Creo has several configuration options for importing neutral exchange files.

Table A.28: A summary of the results for PTC Creo ("No template - Automatic") for the import of the IGES file generated by PTC Creo

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	6.96e-07	6.96e-07
Solid recognised	Yes	-
Resulting accuracy	6.96e-07	6.96e-07

Table A.29: A summary of the results for PTC Creo ("No template - Internal") for the import of the IGES file generated by PTC Creo

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	6.96e-07	6.96e-07
Solid recognised	Yes	-
Resulting accuracy	6.96e-07	6.96e-07

Table A.30: A summary of the results for PTC Creo ("No template - External") for the import of the IGES file generated by PTC Creo

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	Yes	Yes
Export accuracy	5.77e-06	6.12e-06
Solid recognised	Yes	-
Resulting accuracy	5.77e-06	6.12e-06

Table A.31: A summary of the results for PTC Creo ("Template (absolute accuracy 0.01 mm) - Automatic") for the import of the IGES file generated by PTC Creo

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	No	No
Export accuracy	0.01	0.01
Solid recognised	-	-
Resulting accuracy	0.01	0.01

Table A.32: A summary of the results for PTC Creo ("Template (absolute accuracy 0.001 mm) - Automatic") for the import of the IGES file generated by PTC Creo

IGES settings		
Output Solids	Solids	Surface
Surface Type	Standard	Standard
Accuracy IGES file	0.001	0.001
Import successful	No	No
Export accuracy	0.001	0.001
Solid recognised	-	-
Resulting accuracy	0.001	0.001

A.2 Handling of model accuracy (STEP AP203 and AP214)

A.2.1 A. Export from Inventor 2022

A cube is created with the aforementioned dimensions and exported to STEP AP214 (see Figure A.18).

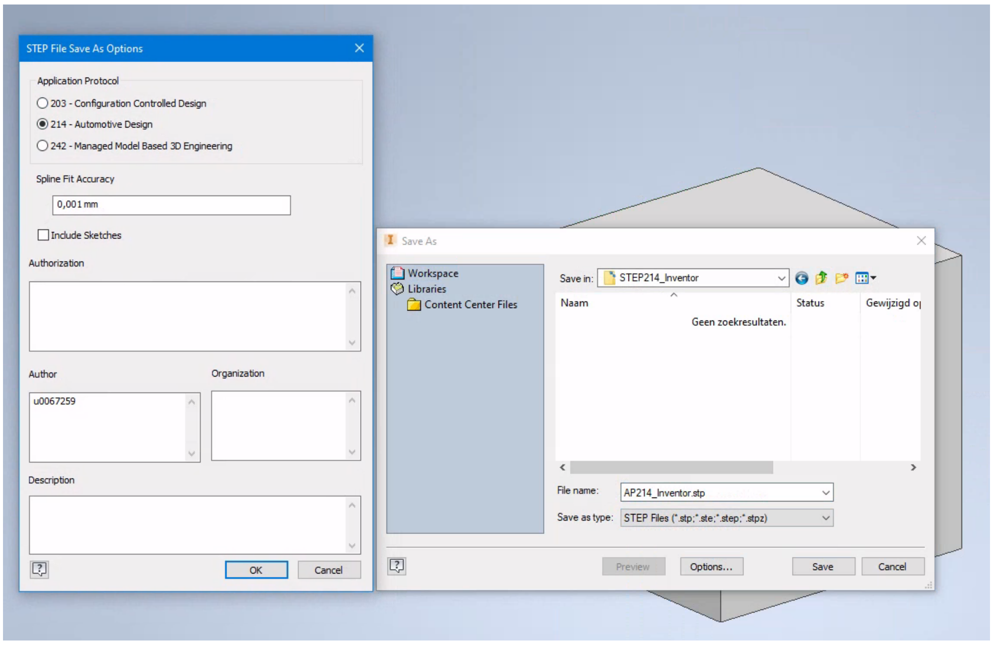


Figure A.18: Settings used to export model to STEP AP214 in Inventor 2022

As the dimensions of the cube are 0.004 mm × 0.004 mm × 0.004 mm, the smallest distance between two vertices is 0.004 mm. Thus, the expected absolute accuracy is at most 0.004. However, as shown in Figure A.19, the file has an accuracy of 0.01 mm.

```
#188=CARTESIAN_POINT('Origin',(1.82267025552143,1.86728318492768,0.002));
#189=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#193,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
#190=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#193,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
```

Figure A.19: STEP AP214 file created by Inventor 2022 has an absolute accuracy of 0.01 mm

To study the effect of this incorrect accuracy (0.01 mm instead of 0.004 mm), the STEP file was read into all the CAD systems and re-exported to STEP AP214 to see the impact on accuracy in the newly generated STEP file.

Import in CATIA V5-6R2022 SP1

The default options for STEP import were used (see Figure A.20).

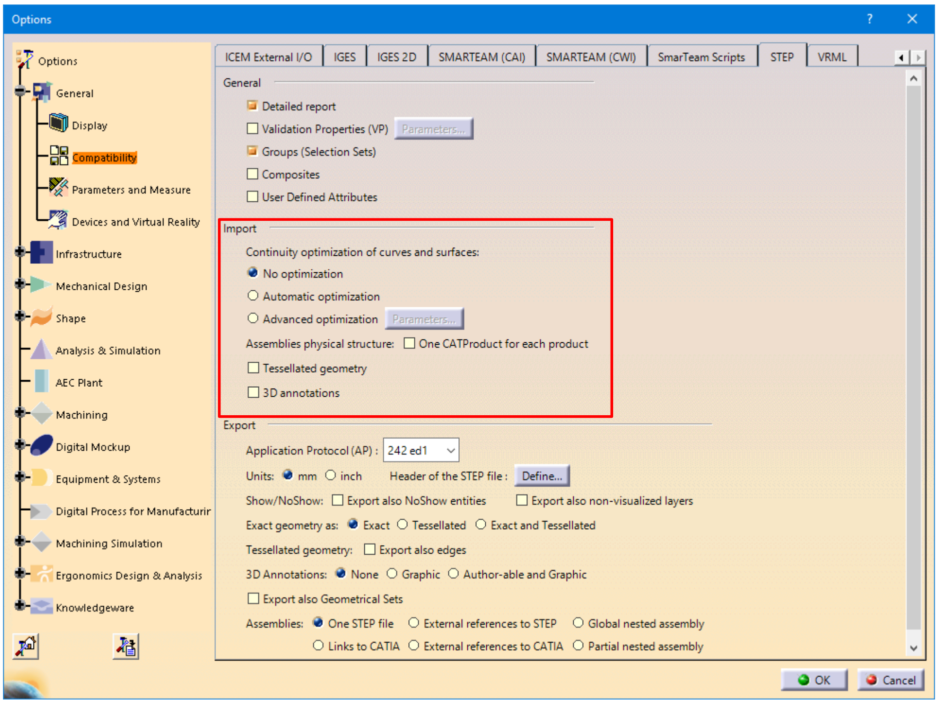


Figure A.20: Default settings for importing STEP files in CATIA v5

CATIA has an option to change the accuracy with which a CAD model is created to a limited extent. This is the “Scale” option (see Figure A.21). Three settings are possible. The first is “Small range”. The absolute accuracy of the CAD model is set to 1×10^{-5} mm. The second is “Normal range” . The absolute accuracy of the CAD model is set to 1×10^{-3} mm. The third is the "Large range". The absolute accuracy of the CAD model is then set to 0.1 mm.

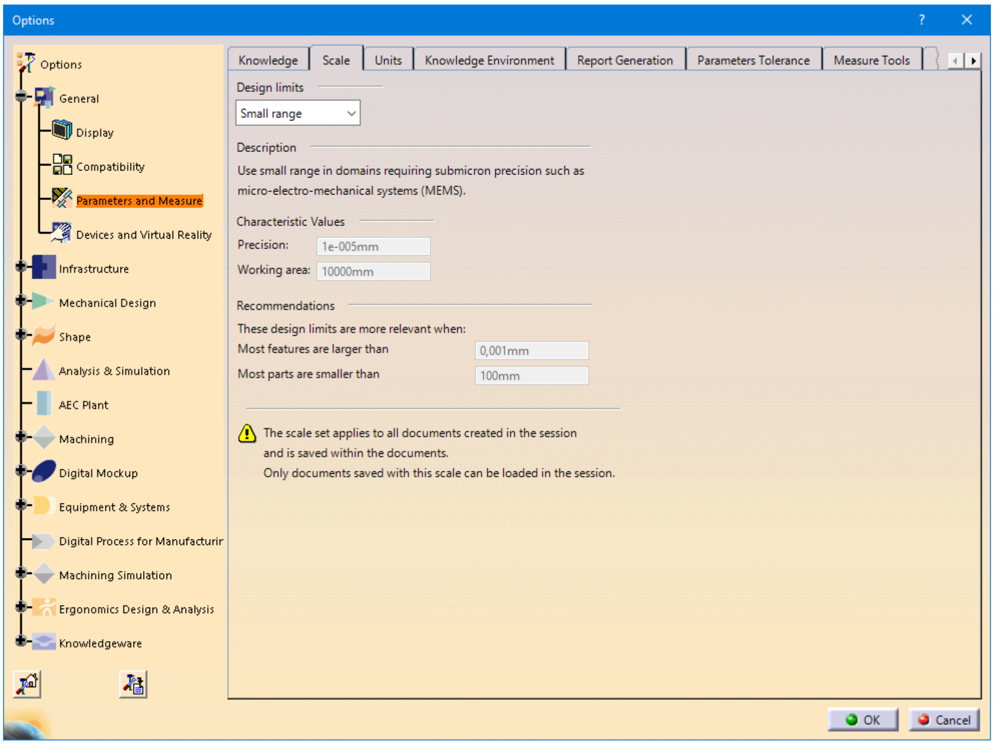


Figure A.21: “Scale” configuration in CATIA to modify the absolute accuracy

Table A.33: A summary of the results for CATIA V5 for the import of the STEP AP214 file generated by Inventor 2022

	Large range	Normal range	Small range
Accuracy STEP file	0.01	0.01	0.01
Import successful	No	Yes	Yes
Export accuracy	0.5	0.005	5×10^{-5}
Solid recognised	-	Yes	Yes

According to documentation available from IBM (IBM 2003), the value 0.005 mm is a fixed value that is independent of the actual accuracy used and cannot be changed by the user. This is based on the default setting for the design limits which is "Normal range". Based on the results in Table A.33, it is concluded that for CATIA, the accuracy of a STEP file exported by CATIA is equal to 5 $\times$ the set model accuracy for newly created CAD models.

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.22).

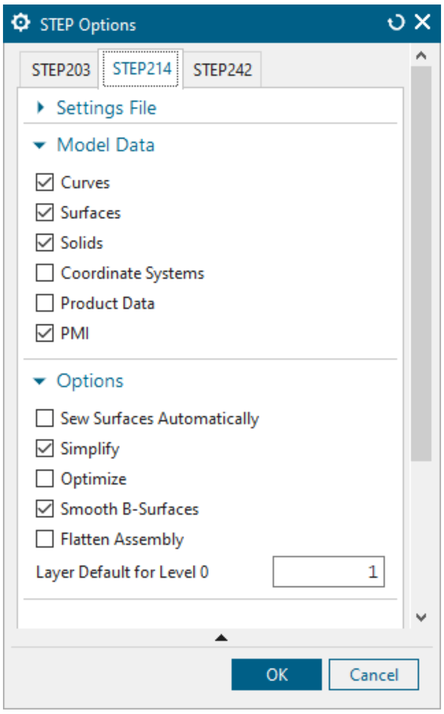


Figure A.22: Default settings for importing STEP files in Siemens NX

Table A.34: A summary of the results for Siemens NX for the import of the STEP AP214 file generated by Inventor 2022

Accuracy STEP file	0.01
Import successful	Yes
Export accuracy	0.001
Solid recognised	Yes

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, Creo has several configuration options for importing neutral exchange files.

Table A.35: A summary of the results for PTC Creo for the import of the STEP AP214 file generated by Inventor 2022

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	0.01	0.01	0.01	0.01
Import successful	Yes	Yes	Yes	No
Export accuracy	6.928×10^{-7}	5.774×10^{-6}	6.928×10^{-7}	0.01
Solid recognised	Yes	Yes	Yes	No

Import in Inventor 2022

The default settings for STEP import were used (see **??**).

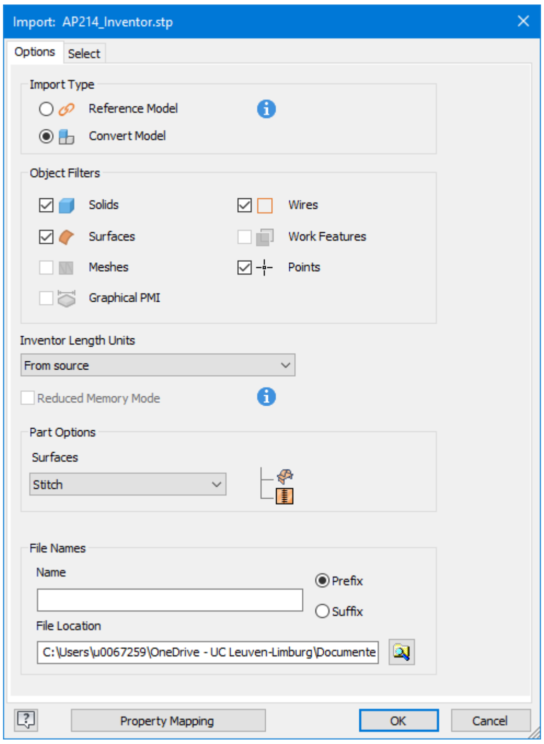


Figure A.23: Default settings for importing STEP files in Inventor 2022

Table A.36: A summary of the results for Inventor 2022 for the import of the STEP AP214 file generated by Inventor 2022

Accuracy STEP file	0.01
Import successful	Yes
Export accuracy	0.01
Solid recognised	Yes

A.2.2 B. Export from CATIA V5-6R2022 SP1

A cube is created with the aforementioned dimensions and exported to STEP AP214 (see Figure A.24).

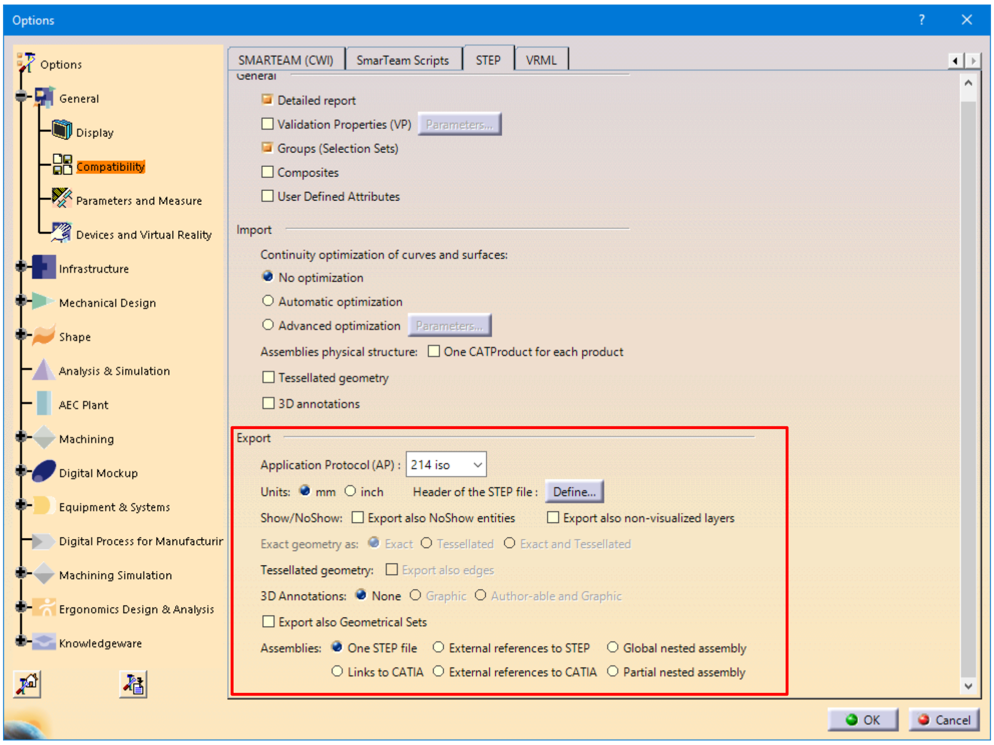


Figure A.24: Settings used to export model to STEP AP214 in CATIA V5

Import in Inventor 2022

The default settings for STEP import were used (see Figure A.23).

Table A.37: A summary of the results for Inventor 2022 for the import of the STEP AP214 file generated by CATIA V5

Accuracy STEP file	0.005
Import successful	Yes
Export accuracy	0.01
Solid recognised	Yes

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.22).

Table A.38: A summary of the results for Siemens NX for the import of the STEP AP214 file generated by CATIA V5

Accuracy STEP file	0.005
Import successful	Yes
Export accuracy	0.001
Solid recognised	Yes

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, Creo has several configuration options for importing neutral exchange files.

Table A.39: A summary of the results for PTC Creo for the import of the STEP AP214 file generated by CATIA V5

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	0.005	0.005	0.005	0.005
Import successful	Yes	Yes	Yes	No
Export accuracy	6.928×10^{-7}	5.774×10^{-6}	6.928×10^{-7}	0.01
Solid recognised	Yes	Yes	Yes	No

Import in CATIA V5-6R2022 SP1

The default options for STEP import were used (see Figure A.20).

Table A.40: A summary of the results for CATIA V5 for the import of the STEP AP214 file generated by CATIA V5

	Large range	Normal range	Small range
Accuracy STEP file	0.005	0.005	0.005
Import successful	No	Yes	Yes
Export accuracy	0.5	0.005	5×10^{-5}
Solid recognised	-	Yes	Yes

A.2.3 C. Export from Siemens NX Version 2019 Build 2501

A cube is created with the aforementioned dimensions and exported to STEP AP214 (see Figure A.25). To make a cube with such small dimensions, the designer must specify an appropriate accuracy before making the model (see Figure A.16). The active absolute accuracy is determined by the formula

$$accuracy = Distance\ Tolerance \cdot Optimize\ Curve\ Distance\ Tolerance\ Factor = 0.001$$

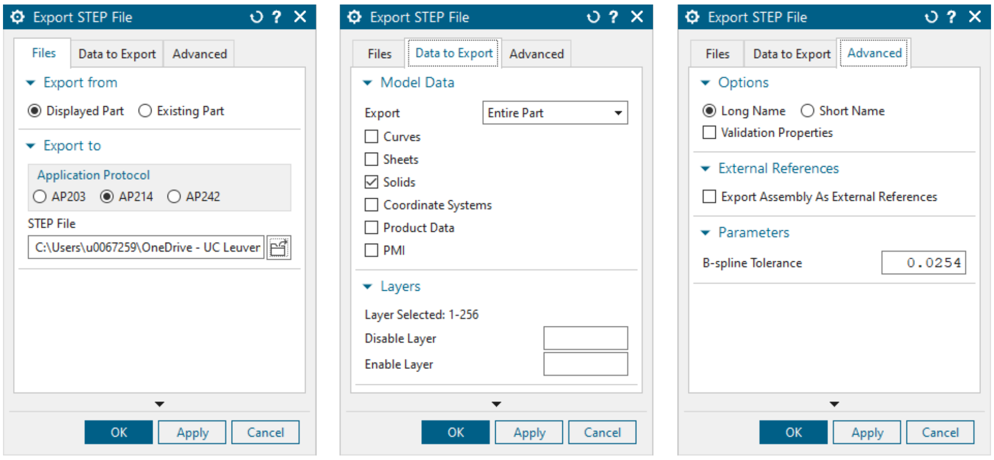


Figure A.25: Settings used to export model to STEP AP214 in Siemens NX

Import in Inventor 2022

The default settings for STEP import were used (see Figure A.23).

Table A.41: A summary of the results for Inventor 2022 for the import of the STEP AP214 file generated by Siemens NX

Accuracy STEP file	2×10^{-5}
Import successful	Yes
Export accuracy	0.01
Solid recognised	Yes

Import in CATIA V5-6R2022 SP1

The default options for STEP import were used (see Figure A.20).

Table A.42: A summary of the results for CATIA V5 for the import of the STEP AP214 file generated by Inventor 2022

	Large range	Normal range	Small range
Accuracy STEP file	2×10^{-5}	2×10^{-5}	2×10^{-5}
Import successful	No	Yes	Yes
Export accuracy	0.5	0.005	5×10^{-5}
Solid recognised	-	Yes	Yes

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page page A7, Creo has several configuration options for importing neutral exchange files.

Table A.43: A summary of the results for PTC Creo for the import of the STEP AP214 file generated by CATIA V5

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	2×10^{-5}	2×10^{-5}	2×10^{-5}	2×10^{-5}
Import successful	Yes	Yes	Yes	No
Export accuracy	6.928×10^{-7}	5.774×10^{-6}	6.928×10^{-7}	0.01
Solid recognised	Yes	Yes	Yes	No

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.22).

Table A.44: A summary of the results for Siemens NX for the import of the STEP AP214 file generated by CATIA V5

Accuracy STEP file	2×10^{-5}
Import successful	Yes
Export accuracy	0.001
Solid recognised	Yes

A.2.4 D. Export from PTC Creo Parametric 8.0.4.0

To create the cube in PTC Creo, a part template with an absolute accuracy of 0.001 mm is used. The default export settings for STEP AP214 are used (see Figure A.26).

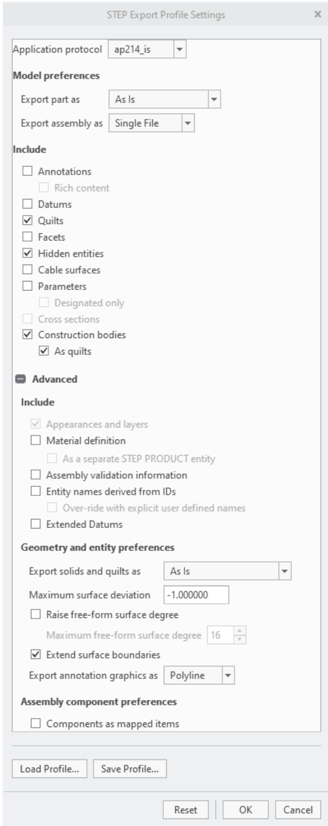


Figure A.26: Settings used to export model to STEP AP214 in PTC Creo

Import in Inventor 2022

The default settings for STEP import were used (see Figure A.23).

Table A.45: A summary of the results for Inventor 2022 for the import of the STEP AP214 file generated by PTC Creo

Accuracy STEP file	0.001
Import successful	Yes
Export accuracy	0.01
Solid recognised	Yes

Import in CATIA V5-6R2022 SP1

The default options for STEP import were used (see Figure A.20).

Table A.46: A summary of the results for CATIA V5 for the import of the STEP AP214 file generated by PTC Creo

	Large range	Normal range	Small range
Accuracy STEP file	0.001	0.001	0.001
Import successful	No	Yes	Yes
Export accuracy	0.5	0.005	5×10^{-5}
Solid recognised	-	Yes	Yes

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.22).

Table A.47: A summary of the results for Siemens NX for the import of the STEP AP214 file generated by PTC Creo

Accuracy STEP file	0.001
Import successful	Yes
Export accuracy	0.001
Solid recognised	Yes

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, Creo has several configuration options for importing neutral exchange files.

Table A.48: A summary of the results for PTC Creo for the import of the STEP AP214 file generated by PTC Creo

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	0.001	0.001	0.001	0.001
Import successful	Yes	Yes	Yes	No
Export accuracy	6.928×10^{-7}	5.774×10^{-6}	6.928×10^{-7}	0.01
Solid recognised	Yes	Yes	Yes	No

A.3 Handling of model accuracy (STEP AP242)

A.3.1 A. Export from Inventor 2022

In Inventor 2022, a beam model is created with the dimensions 100 mm × 100 mm × 50 mm. In it, $100^{+0.035}_{-0.000}$, a dimension with a lower and an upper tolerance is created (see Figure A.27). This model is exported to STEP AP242 (see Figure A.28). Analysis of the STEP file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the dimension is stored as "representation PMI" (Figure A.29). An excerpt from the STEP file can be seen in Figure A.30.

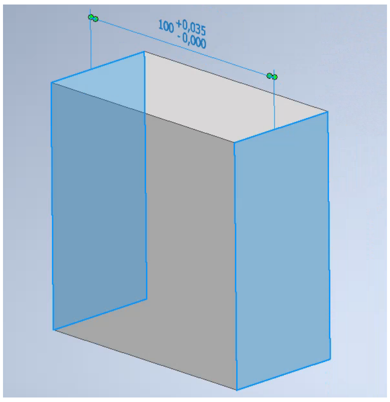


Figure A.27: Beam model with a dimension with a lower and an upper tolerance (Inventor 2022)

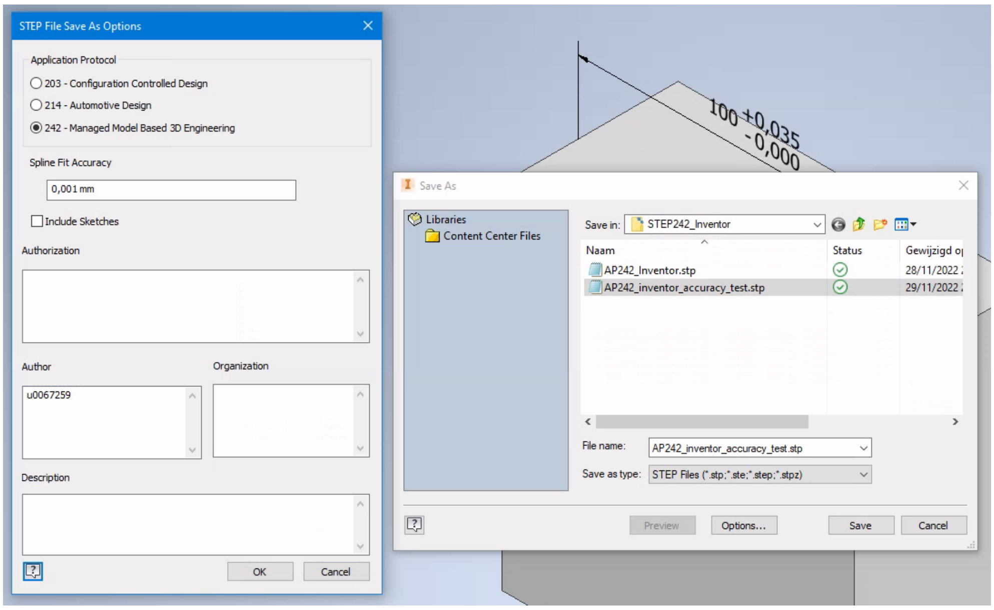
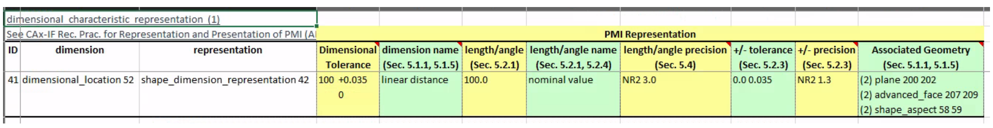


Figure A.28: Settings used for export to STEP AP242 by Inventor 2022



dimensional_characteristic_representation (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PMI (A

			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
41	dimensional_location 52	shape_dimension_representation 42	100 +0.035 0	linear distance	100.0	nominal value	NR2 3.0	0.0 0.035	NR2 1.3	(2) plane 200 202 (2) advanced_face 207 209 (2) shape_aspect 58 59

Figure A.29: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file exported by Inventor 2022

```
#39=DRAUGHTING_MODEL_ITEM_ASSOCIATION(
'PMI representation to presentation link','',#52,#70,#75);
#40=DRAUGHTING_MODEL_ITEM_ASSOCIATION('','',#38,#70,#75);
#41=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#52,#42);
#42=SHAPE_DIMENSION_REPRESENTATION('',(#49),#356);
#43=MEASURE_QUALIFICATION('','upper bound',#45,(#51));
#44=MEASURE_QUALIFICATION('','lower bound',#46,(#51));
#45=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#358);
#46=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#358);
#47=TOLERANCE_VALUE(#46,#45);
#48=PLUS_MINUS_TOLERANCE(#47,#52);
#49=(
LENGTH_MEASURE_WITH_UNIT()
MEASURE_REPRESENTATION_ITEM()
MEASURE_WITH_UNIT(LENGTH_MEASURE(100.),#358)
QUALIFIED_REPRESENTATION_ITEM((#50))
REPRESENTATION_ITEM('nominal value')
);
#50=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.0');
#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');
#52=DIMENSIONAL_LOCATION('linear distance','',#59,#58);
#53=ID_ATTRIBUTE('ShapeAspect.1',#58);
```

```
#354=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
#355=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,
'DISTANCE_ACCURACY_VALUE',
'Maximum model space distance between geometric entities at asserted c
onnectivities');
#356=(
GEOMETRIC_REPRESENTATION_CONTEXT(3)
GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#354))
GLOBAL_UNIT_ASSIGNED_CONTEXT((#358,#360,#361))
REPRESENTATION_CONTEXT('','3D')
);
#357=(
GEOMETRIC_REPRESENTATION_CONTEXT(3)
GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#355))
GLOBAL_UNIT_ASSIGNED_CONTEXT((#358,#360,#361))
REPRESENTATION_CONTEXT('','3D')
);
#358=(
LENGTH_UNIT()
NAMED_UNIT(*)
SI_UNIT(.MILLI.,.METRE.)
);
```

Figure A.30: Excerpt from the STEP AP242 file created by Inventor 2022

The dimension can be found in the following format in this STEP file (Table A.49). The tolerances are assigned to a linear distance as indicated by `DIMENSIONAL_LOCATION` where #59 and #58 identify the semantic references (Boy et al. 2014, p. 9-11) .

Table A.49: Excerpt from the definition of the dimension in the STEP AP242 file created by Inventor

Dimension component	STEP
100	#41=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#52,#42); #42=SHAPE_DIMENSION_REPRESENTATION(”,(#49),#356); #47=TOLERANCE_VALUE(#46,#45); #48=PLUS_MINUS_TOLERANCE(#47,#52); #49=(LENGTH_MEASURE_WITH_UNIT() MEASURE_REPRESENTATION_ITEM() MEASURE_WITH_UNIT(LENGTH_MEASURE(100.),#358) QUALIFIED_REPRESENTATION_ITEM((#50)) REPRESENTATION_ITEM(’nominal value’)); #50=VALUE_FORMAT_TYPE_QUALIFIER(’NR2 3.0’); #52=DIMENSIONAL_LOCATION(’linear distance’,”,#59,#58); #56=GEOMETRIC_ITEM_SPECIFIC_USAGE(”,”,#58,#364,#207); #57=GEOMETRIC_ITEM_SPECIFIC_USAGE(”,”,#59,#364,#209); #58=SHAPE_ASPECT(’ShapeAspect.1’,”,#363,.T.); #59=SHAPE_ASPECT(’ShapeAspect.2’,”,#363,.T.); #354=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358, ’DISTANCE_ACCURACY_VALUE’, ’Maximum model space distance between geometric entities at asserted c onnectivities’); #358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.));

+0.035	`#44=MEASURE_QUALIFICATION('','lower bound',#46,(#51));`
	`#46=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#358);`
	`#47=TOLERANCE_VALUE(#46,#45);`
	`#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');`
	`#354=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,`
	`'DISTANCE_ACCURACY_VALUE',`
	`'Maximum model space distance between geometric entities at asserted c`
	`onnectivities');`
	`#358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.));`

-0.000	`#43=MEASURE_QUALIFICATION('','upper bound',#45,(#51));`
	`#45=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#358);`
	`#47=TOLERANCE_VALUE(#46,#45);`
	`#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');`
	`#354=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,`
	`'DISTANCE_ACCURACY_VALUE',`
	`'Maximum model space distance between geometric entities at asserted c`
	`onnectivities');`
	`#358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.));`

As stated in Boy et al. 2014, p. 24-25, the accuracy of the display of the dimension is specified by

```
#50=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.0');
```

This indicates that the dimension value should be displayed with three digits before the decimal point and no digits after it.

The accuracy of the display of the upper and lower tolerance is determined by

```
#51=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3')
```

This indicates that the value of the upper and lower tolerance should be displayed with one digit before the decimal point and three digits after it.

The absolute accuracy of the CAD model in this STEP AP242 file is 0.01 mm (Table A.50) as indicated by `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

Table A.50: Model accuracy applied within the STEP file created by Inventor 2022

Model accuracy	STEP

0.01	`#354=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.01),#358,`
	`'DISTANCE_ACCURACY_VALUE',`
	`'Maximum model space distance between geometric entities at asserted c`
	`onnectivities');`

mm	`#358=(LENGTH_UNIT()NAMES_UNIT(*)SI_UNIT(.MILLI.,.METRE.));`

The display accuracy of the dimension (0.001) is higher than the absolute accuracy (0.01) used for the CAD model.

Import in CATIA V5-6R2022 SP1

The following options for STEP import were used (see Figure A.31).

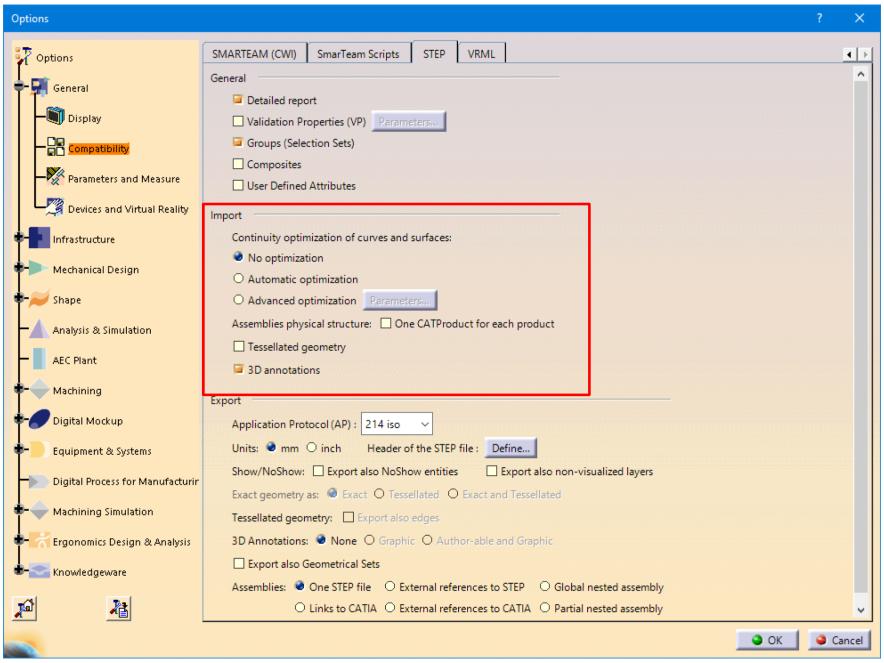


Figure A.31: Settings for importing STEP files in CATIA v5

The “Scale” option used within this CATIA model is “Normal range”. The absolute accuracy of the CAD model is hereby set to 1×10^{-3} mm (see page A25). The feature tree in Figure A.32 shows that the annotation is imported as “presentation PMI”. When the imported model is re-exported to a STEP AP242 file, analysis of the resulting STEP file with the “NIST STEP File Analyzer and Viewer”(Lipman 2017) shows that the annotation is saved as “presentation PMI” (see Figure A.33). This “presentation PMI” is specified as “general tolerance” (see Table A.51).

The absolute accuracy specified in this STEP file is 0.005 mm as indicated by the parameter `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

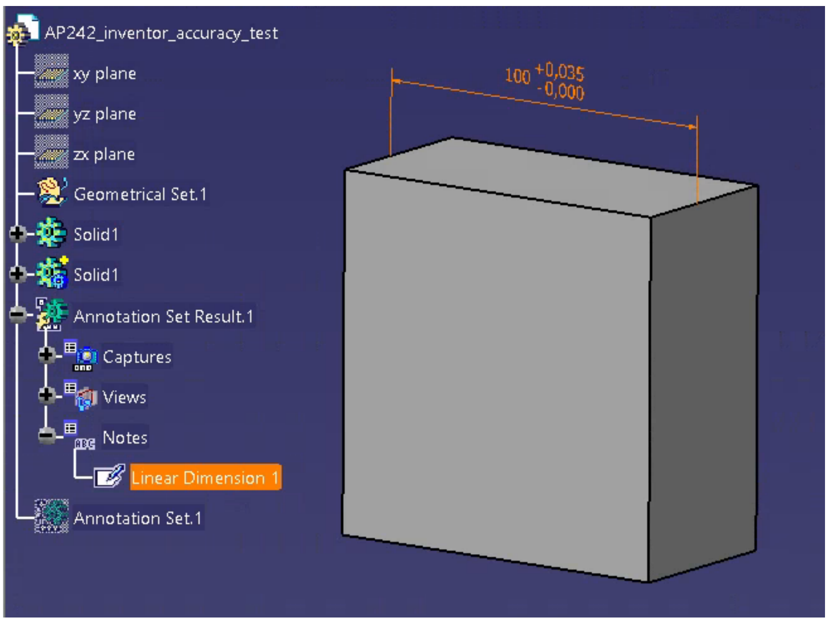


Figure A.32: Result of importing the STEP AP242 file generated by Inventor 2022 in CATIA V5 draughting_callout (1)

ID	name	contents
376	Linear Dimension 1	(1) tessellated_annotation_occurrence 375

Figure A.33: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by CATIA V5

Table A.51: Excerpt from the definition of the dimension in the STEP AP242 file created by Inventor and imported in and re-exported by CATIA V5

STEP

```
#374=PRESENTATION_STYLE_ASSIGNMENT((#373)) ;
#373=CURVE_STYLE(’ ’,#371,POSITIVE_LENGTH_MEASURE(0.129999995232),#372) ;
#376=DRAUGHTING_CALLOUT(’Linear Dimension 1’,(#375)) ;
#375=TESSELLATED_ANNOTATION_OCCURRENCE(’Linear Dimension 1’,(#374),#366) ;
#372=DRAUGHTING_PRE_DEFINED_COLOUR(’white’) ;
#371=DRAUGHTING_PRE_DEFINED_CURVE_FONT(’continuous’) ;
#366=(GEOMETRIC_REPRESENTATION_ITEM()REPOSITIONED_TESSELLATED_ITEM(#370)
REPRESENTATION_ITEM(’general tolerance’)TESSELLATED_GEOMETRIC_SET((#399,#409,#419,
#429,#439,#449,#459,#469,#479,#489,#499,#509,#519,#529,#539,#549,#559,#569))
TESSELLATED_ITEM()) ;
#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,
’distance_accuracy_value’,’CONFUSED CURVE UNCERTAINTY’) ;
#12=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;
```

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.34).

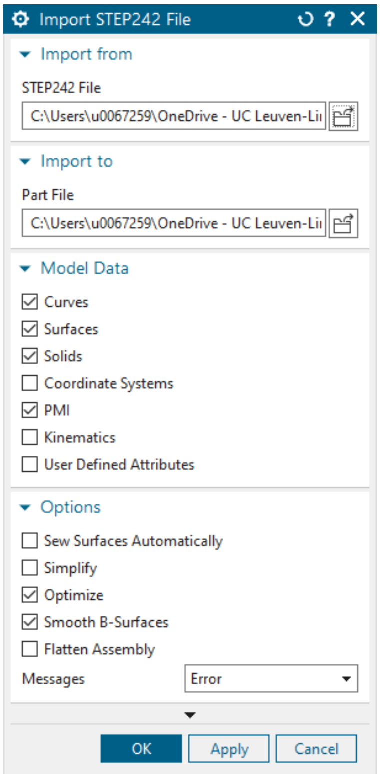


Figure A.34: Settings for importing STEP AP242 files in Siemens NX

The MBD navigator in Figure A.35 shows that the annotation is imported as "representation PMI". When the imported model is re-exported to a STEP AP242 file using the settings shown in Figure A.36, the analysis with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows the annotation is saved as "representation PMI" (see Figure A.37). However, two things differ from the original STEP AP242 file. The first one is that the display accuracy of the dimension and its assigned tolerances has changed. The accuracy of the dimension has changed from three digits before and no digits after the decimal point to three digits before and one digit after the decimal point. The accuracy of the upper and lower tolerances has changed from one digit before and three digits after the decimal point to one digit before and two digits after the decimal point (see Table A.52). However, this is not displayed when the re-exported STEP file is re-imported into Siemens NX (see Figure A.38). It can be concluded that the value of `VALUE_FORMAT_TYPE_QUALIFIER` is ignored by the CAD system. The second one is that the parameter `DIMENSIONAL_LOCATION='linear distance'` is replaced by `DIMENSIONAL_SIZE='diameter'`. Only the name (linear dimension 1) is retained (see Table A.52).

The absolute accuracy specified in this STEP file is 0.005 mm as indicated by the parameter `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

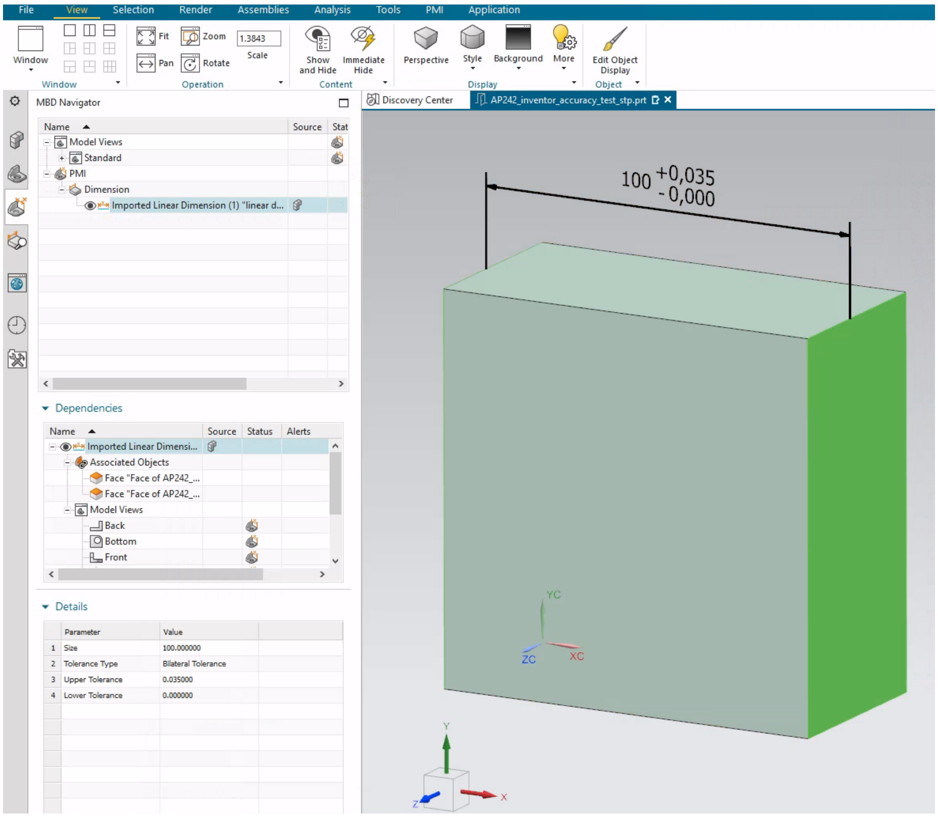


Figure A.35: Result of importing the STEP AP242 file generated by Inventor 2022 in Siemens NX Version 2019

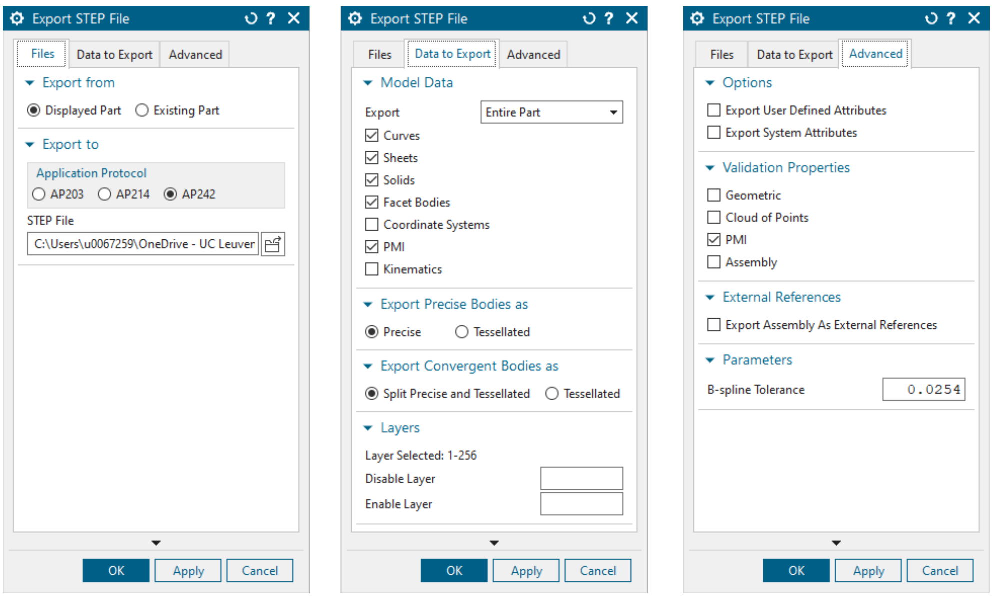
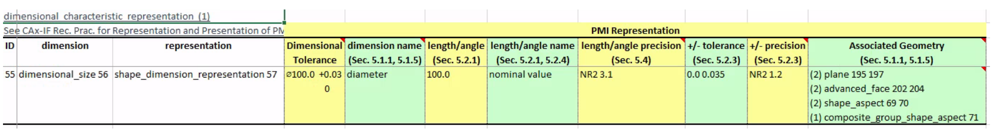


Figure A.36: Settings used for export to STEP AP242 by Siemens NX Version 2019



dimensional_characteristic_representation (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PM

ID	dimension	representation	PMI Representation							
			Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
55	dimensional_size 56	shape_dimension_representation 57	⌀100.0 +0.03 0	diameter	100.0	nominal value	NR2 3.1	0.0 0.035	NR2 1.2	(2) plane 195 197 (2) advanced_face 202 204 (2) shape_aspect 69 70 (1) composite_group_shape_aspect 71

Figure A.37: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by Siemens NX

Table A.52: Excerpt from the definition of the dimension in the STEP AP242 file created by Inventor and imported in and re-exported by Siemens NX

Dimension component	STEP

100.	`#49=PLUS_MINUS_TOLERANCE(#50,#56);`
	`#50=TOLERANCE_VALUE(#53,#54);`
	`#55=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#56,#57);`
	`#56=DIMENSIONAL_SIZE(#71,'diameter');`
	`#57=SHAPE_DIMENSION_REPRESENTATION('',(#58),#363);`
	`#58=(`
	`LENGTH_MEASURE_WITH_UNIT()`
	`MEASURE_REPRESENTATION_ITEM()`
	`MEASURE_WITH_UNIT(POSITIVE_LENGTH_MEASURE(100.),#367)`
	`QUALIFIED_REPRESENTATION_ITEM((#59))`
	`REPRESENTATION_ITEM('nominal value')`
	`);`
	`#59=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.1');`
	`#363=(`
	`GEOMETRIC_REPRESENTATION_CONTEXT(3)`
	`GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#364))`
	`GLOBAL_UNIT_ASSIGNED_CONTEXT((#367,#366,#365))`
	`REPRESENTATION_CONTEXT('AP242_inventor_accuracy_test_stp',`
	`'TOP_LEVEL_ASSEMBLY_PART')`
	`);`
	`#364=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#367,`
	`'DISTANCE_ACCURACY_VALUE','Maximum Tolerance applied to model');`
	`#367=(`
	`LENGTH_UNIT()`
	`NAMED_UNIT(*)`
	`SI_UNIT(.MILLI.,.METRE.)`
	`);`

0.035	`#50=TOLERANCE_VALUE(#53,#54);`
	`#52=MEASURE_QUALIFICATION('','',#54,(#60));`
	`#54=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#367);`
	`#60=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.2');`
	`#367=(`
	`LENGTH_UNIT()`
	`NAMED_UNIT(*)`
	`SI_UNIT(.MILLI.,.METRE.)`
	`);`

0.	`#50=TOLERANCE_VALUE(#53,#54);`
	`#51=MEASURE_QUALIFICATION('','',#53,(#60));`
	`#53=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#367);`
	`#60=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.2');`
	`#367=(`
	`LENGTH_UNIT()`
	`NAMED_UNIT(*)`
	`SI_UNIT(.MILLI.,.METRE.)`
	`);`

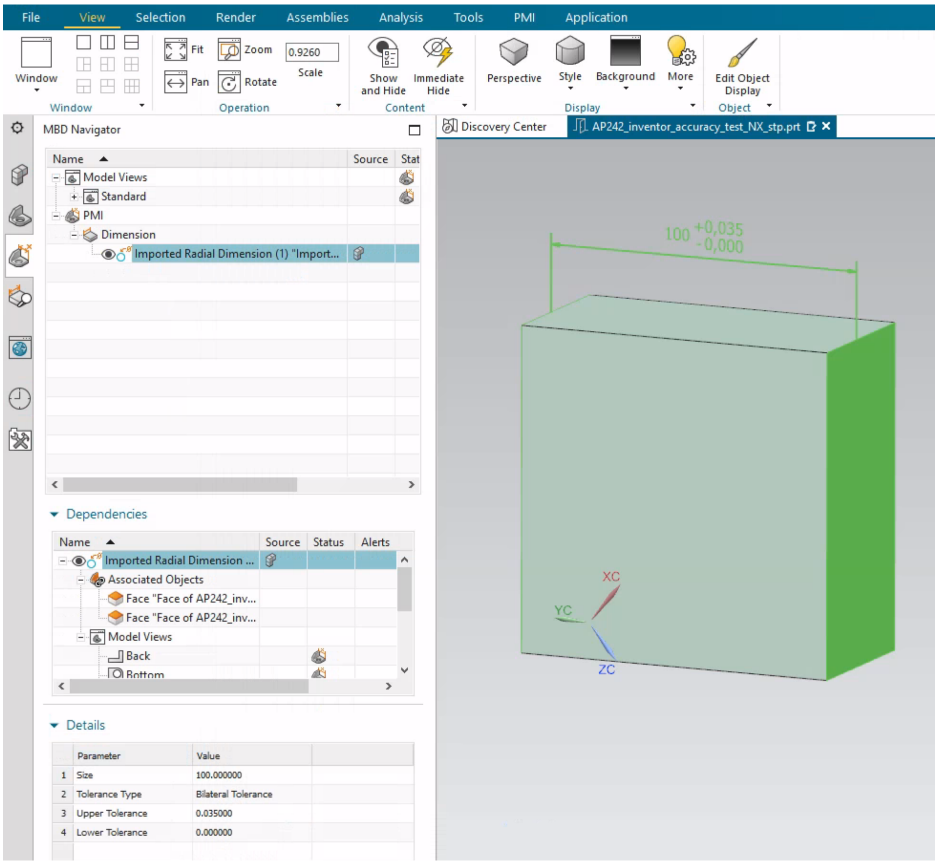


Figure A.38: Result of re-importing the re-exported STEP AP242 Inventor file in Siemens NX

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, Creo has several configuration options for importing neutral exchange files. For each configuration option, when the STEP file is imported, the annotation is defined as "representation PMI" (see Figure A.39 and Table A.53). When the imported model is re-exported to a STEP AP242 file (see Figure A.40), the annotation is saved as "presentation PMI" and is defined as a "weld symbol" (see Table A.54).

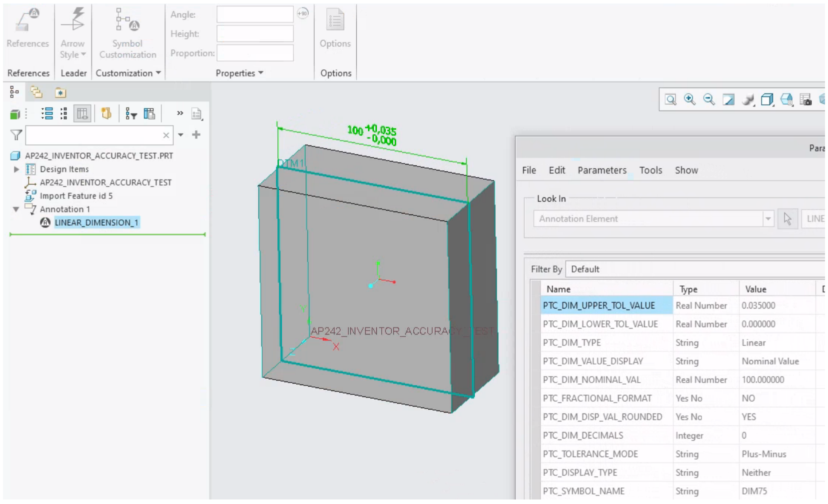


Figure A.39: Result of importing the STEP AP242 file generated by Inventor 2022 in PTC Creo (accuracy set to automatic)

Table A.53: A summary of the results for PTC Creo for the import of the STEP AP242 file generated by Inventor 2022

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	0.01	0.01	0.01	0.01
Import successful	Yes	Yes	Yes	Yes
Representation PMI recognised	Yes	Yes	Yes	Yes
Export accuracy	0.0149994	0.01	0.0149994	0.01
Representation PMI retained	No	No	No	No

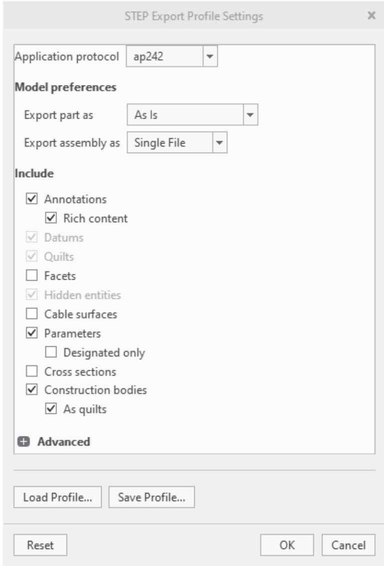


Figure A.40: Settings used to re-export the imported model to a STEP AP242 file in PTC Creo

Table A.54: Excerpt from the definition of the dimension in the STEP AP242 file created by Inventor and imported in and re-exported by PTC Creo 8.0.4.0

STEP
`#4=COLOUR_RGB('',2.156862745098E-1,0.E0,3.725490196078E-1);`
`#5844=CURVE_STYLE('',#23,POSITIVE_LENGTH_MEASURE(2.E-2),#4);`
`#5845=PRESENTATION_STYLE_ASSIGNMENT((#5844));`
`#5846=ANNOTATION_CURVE_OCCURRENCE('LINEAR_DIMENSION_1',(#5845),#5843);`
`#5843=GEOMETRIC_CURVE_SET('weld symbol',(#310,#313,#316,#319,#322,#325,#328,`
`...`
`#5782,#5787,#5792,#5797,#5802,#5807,#5812,#5817,#5822,#5827,#5832,#5837,#5842));`

Import in Inventor 2022

The default settings for STEP import were used. Only the "graphical PMI" option was additionally checked (see Figure A.41). The dimension is imported as "presentation PMI" (Figure A.42). This is consistent with the terminology in the "graphical PMI" option. When the imported model is re-exported to a STEP AP242 file, the annotation is no longer present in the file, either as representation PMI or presentation PMI (see Figure A.43).

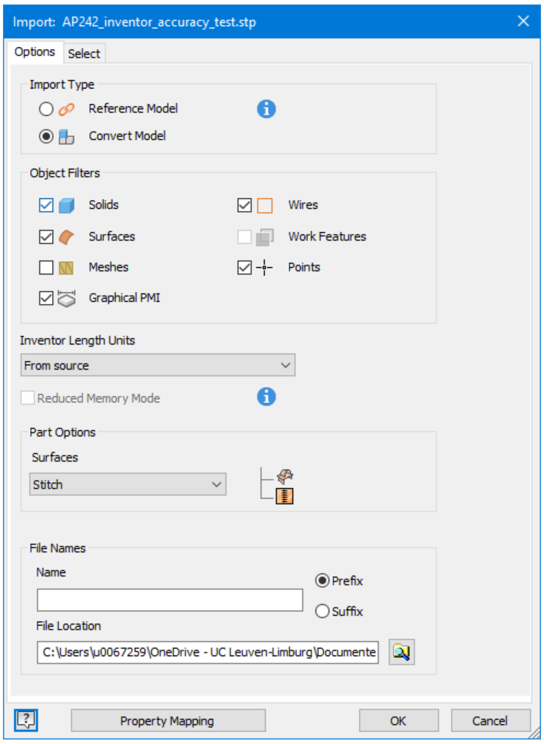


Figure A.41: Settings used to import the STEP AP242 file in Inventor 2022

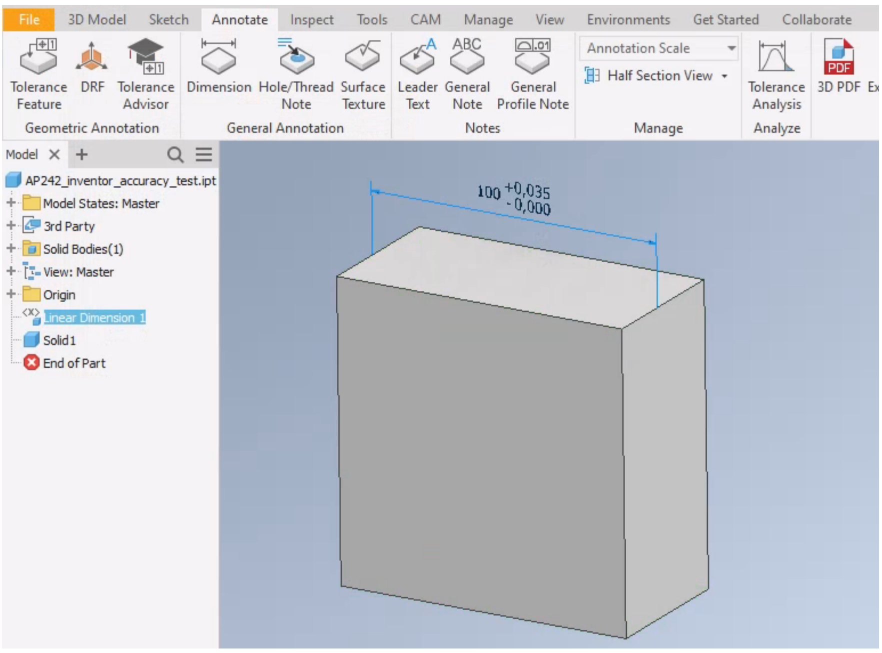


Figure A.42: Result of importing the STEP AP242 file generated by Inventor 2022 in Inventor 2022

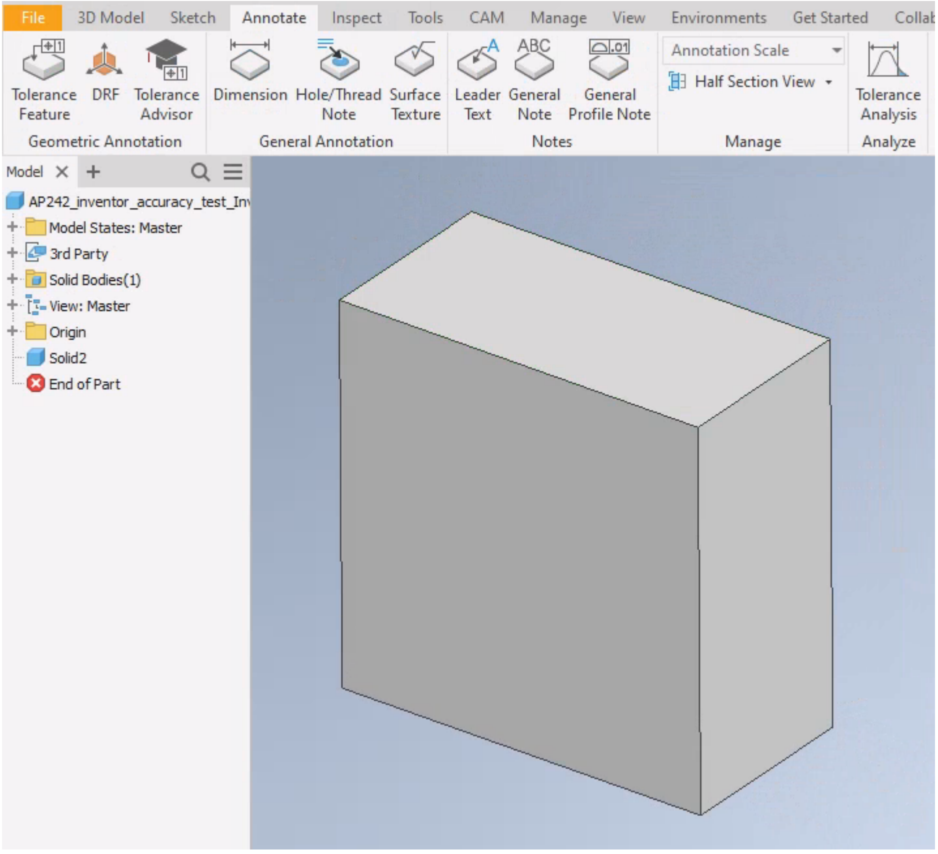


Figure A.43: Result of re-importing the re-exported STEP AP242 Inventor 2022 file in Inventor 2022

A.3.2 B. Export from CATIA V5-6R2022 SP1

In CATIA V5, a beam model is created with the dimensions 100 mm $\times$ 100 mm $\times$ 50 mm. In it, $100^{+0.035}_{-0.000}$, a dimension with a lower and an upper tolerance is created (Figure A.44). This model was exported to a STEP AP242 file. The settings used for this purpose are shown in Figure A.45. Analysis of the STEP file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the dimension is stored as "representation PMI" (Figure A.46).

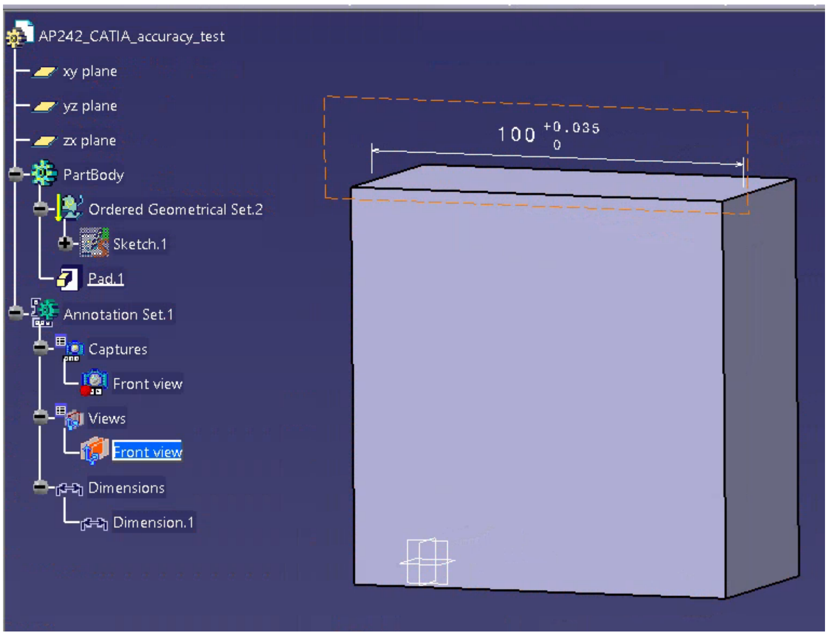


Figure A.44: Beam model with a dimension with a lower and an upper tolerance (CATIA V5)

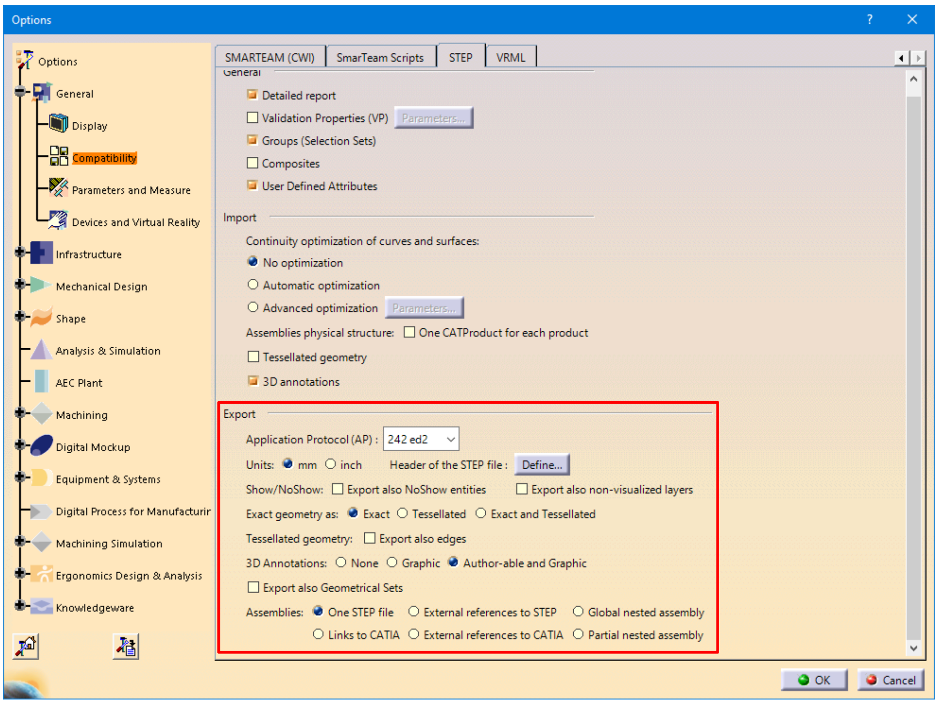
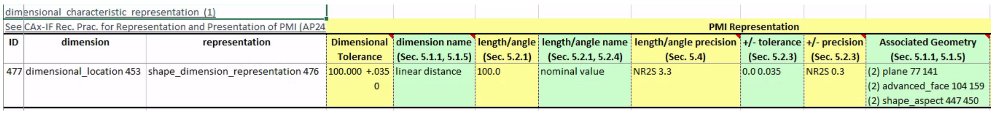


Figure A.45: Settings used for export to STEP AP242 by CATIA V5



dimensional_characteristic_representation (1)										
See CAx-IF Rec. Prac. for Representation and Presentation of PMI (AP24			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
477	dimensional_location 453	shape_dimension_representation 476	100.000 +.035 0	linear distance	100.0	nominal value	NR2S 3.3	0.0 0.035	NR2S 0.3	(2) plane 77 141 (2) advanced_face 104 159 (2) shape_aspect 447 450

Figure A.46: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file exported by CATIA V5

The dimension can be found in the following format in the resulting STEP AP242 file (Table A.55). The tolerances are assigned to a linear distance as indicated by the parameter `DIMENSIONAL_LOCATION` where #447 and #450 identify the semantic references (Boy et al. 2014, p. 9-11).

Table A.55: Excerpt from the definition of the dimension in the STEP file created by CATIA V5

Dimension component	STEP

100.000	`#480=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 3.3') ;`
	`#447=SHAPE_ASPECT('User Surface.1','single feature',#11,.T.) ;`
	`#450=SHAPE_ASPECT('User Surface.2','single feature',#11,.T.) ;`
	`#11=PRODUCT_DEFINITION_SHAPE(' ',' ',#10) ;`
	`#473=PROPERTY_DEFINITION('pmi validation property','',#453) ;`
	`#453=DIMENSIONAL_LOCATION('linear distance','',#447,#450) ;`
	`#476=SHAPE_DIMENSION_REPRESENTATION('',(#479),#16) ;`
	`#477=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#453,#476) ;`
	`#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,`
	`distance_accuracy_value','CONFUSED CURVE UNCERTAINTY') ;`
	`#478=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`
	`#479=(LENGTH_MEASURE_WITH_UNIT()MEASURE_REPRESENTATION_ITEM()`
	`MEASURE_WITH_UNIT(LENGTH_MEASURE(100.),#478)`
	`QUALIFIED_REPRESENTATION_ITEM((#480))`
	`REPRESENTATION_ITEM('nominal value')) ;`
	`#16=(GEOMETRIC_REPRESENTATION_CONTEXT(3)`
	`GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#15))`
	`GLOBAL_UNIT_ASSIGNED_CONTEXT((#12,#13,#14))`
	`REPRESENTATION_CONTEXT(' ',' ')) ;`

+0.035	`#488=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 0.3') ;`
	`#484=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#483) ;`
	`#490=PLUS_MINUS_TOLERANCE(#489,#453) ;`
	`#489=TOLERANCE_VALUE(#482,#484) ;`
	`#487=MEASURE_QUALIFICATION('','',#484,(#488)) ;`
	`#489=TOLERANCE_VALUE(#482,#484) ;`
	`#483=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`

0	`#486=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 0.3') ;`
	`#482=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#481) ;`
	`#490=PLUS_MINUS_TOLERANCE(#489,#453) ;`
	`#489=TOLERANCE_VALUE(#482,#484) ;`
	`#485=MEASURE_QUALIFICATION('','',#482,(#486)) ;`
	`#489=TOLERANCE_VALUE(#482,#484) ;`
	`#481=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`

As stated in Boy et al. 2014, p. 24-25, the accuracy of the display of the dimension is specified by

```
#480=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 3.3') ;
```

This indicates that the dimension value should be displayed with three digits before and after the decimal point.

The accuracy of the display of the upper and lower tolerance is determined by

```
#488=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 0.3') ;
```
and
```
#486=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 0.3') ;
```

This indicates that the value of the upper and lower tolerance should be displayed with no digit before the decimal point and three digits after it.

The absolute accuracy of the CAD model in this STEP AP242 file is 0.005 mm (Table A.56) as indicated by `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

Table A.56: Model accuracy applied within the STEP AP242 file created by CATIA V5

Model accuracy	STEP
0.005	`#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,` `'distance_accuracy_value','CONFUSED CURVE UNCERTAINTY') ;`
mm	`#12=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`

The display accuracy of the dimension (0.001) is higher than the absolute accuracy (0.005) used for the CAD model.

Import in Inventor 2022

The default settings for STEP import were used. Only the “graphical PMI” option was additionally checked (see Figure A.41). The dimension is imported as “presentation PMI” (Figure A.47). This is consistent with the terminology in the “graphical PMI” option. When the imported model is re-exported to a STEP AP242 file, the annotation is no longer present in the file, either as representation PMI or as presentation PMI.

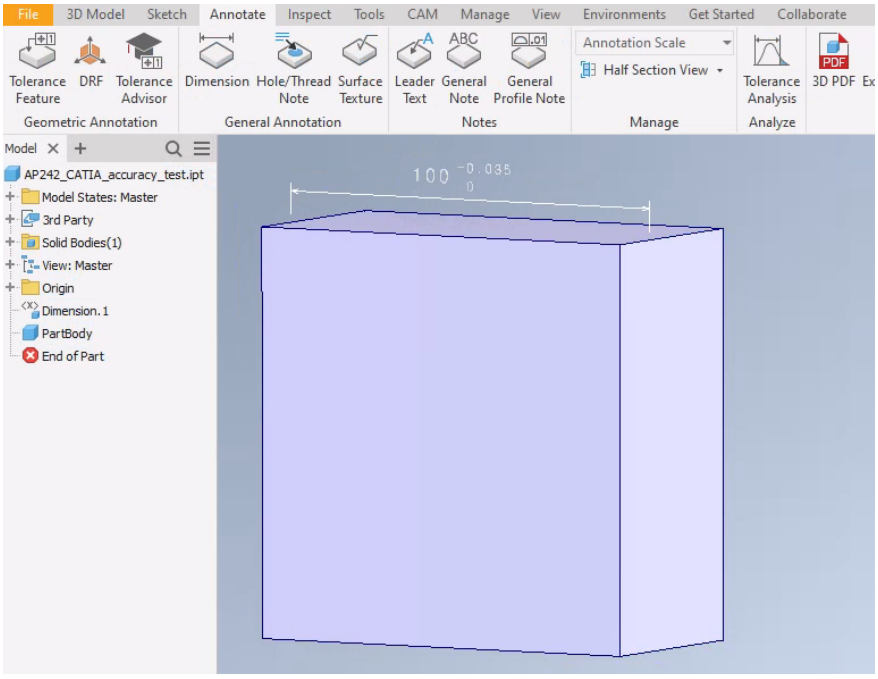


Figure A.47: Result of importing the STEP AP242 file generated by CATIA V5 in Inventor 2022

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.34).

The MBD navigator in Figure A.48 shows that the annotation is imported as "representation PMI". When the imported model is re-exported to a STEP AP242 file, the analysis with the "NIST STEP File Analyzer and Viewer" (Lipman 2017) shows the annotation is saved as "representation PMI". However, two things differ from the original STEP AP242 file. The first one is that the display accuracy of the assigned tolerances has changed from no digits before and three digits after the decimal point to one digit before and two digits after the decimal point (see Table A.57). However, this is not displayed when the re-exported STEP file is re-imported into Siemens NX (see Figure A.49). It can be concluded that the value of `VALUE_FORMAT_TYPE_QUALIFIER` is ignored by the CAD system. The second one is that `DIMENSIONAL_LOCATION='linear distance'` is replaced by `DIMENSIONAL_SIZE='diameter'`. Only the name (linear dimension 1) is retained (see Table A.57).

The absolute accuracy of the CAD model in this re-exported STEP file is 0.005 mm as indicated by `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

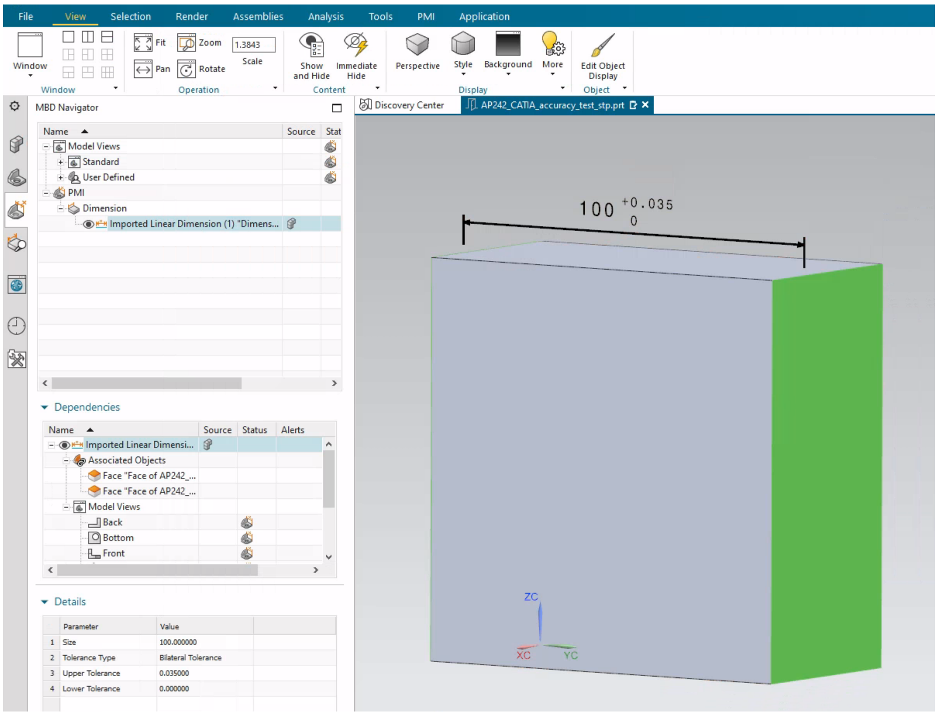


Figure A.48: Result of importing the STEP AP242 file generated by CATIA V5 in Siemens NX Version 2019

Table A.57: Excerpt from the definition of the dimension in the STEP AP242 file created by CATIA and imported in and re-exported by Siemens NX

Dimension component	STEP
100	#53=PLUS_MINUS_TOLERANCE(#54,#60); #54=TOLERANCE_VALUE(#57,#58); #59=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#60,#61); #60=DIMENSIONAL_SIZE(#75,’diameter’); #61=SHAPE_DIMENSION_REPRESENTATION(”,(#62),#379); #62=(LENGTH_MEASURE_WITH_UNIT() MEASURE_REPRESENTATION_ITEM() MEASURE_WITH_UNIT(POSITIVE_LENGTH_MEASURE(100.),#383) QUALIFIED_REPRESENTATION_ITEM((#63)) REPRESENTATION_ITEM(’nominal value’)); #63=VALUE_FORMAT_TYPE_QUALIFIER(’NR2 3.3’); #75=COMPOSITE_GROUP_SHAPE_ASPECT(’Imported Linear Dimension (1) "Dimension.1"’,’multiple elements’,#384,.T.); #379=(GEOMETRIC_REPRESENTATION_CONTEXT(3) GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#380)) GLOBAL_UNIT_ASSIGNED_CONTEXT((#383,#382,#381)) REPRESENTATION_CONTEXT(’AP242_CATIA_accuracy_test_NX’, ’TOP_LEVEL_ASSEMBLY_PART’)); #380=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#383, ’DISTANCE_ACCURACY_VALUE’,’Maximum Tolerance applied to model’); #383=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.));
+0.035	#53=PLUS_MINUS_TOLERANCE(#54,#60); #54=TOLERANCE_VALUE(#57,#58); #56=MEASURE_QUALIFICATION(”,”,#58,(#64)); #58=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#383); #64=VALUE_FORMAT_TYPE_QUALIFIER(’NR2 1.2’); #383=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.));
0	#53=PLUS_MINUS_TOLERANCE(#54,#60); #54=TOLERANCE_VALUE(#57,#58); #55=MEASURE_QUALIFICATION(”,”,#57,(#64)); #57=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#383); #64=VALUE_FORMAT_TYPE_QUALIFIER(’NR2 1.2’); #383=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.));

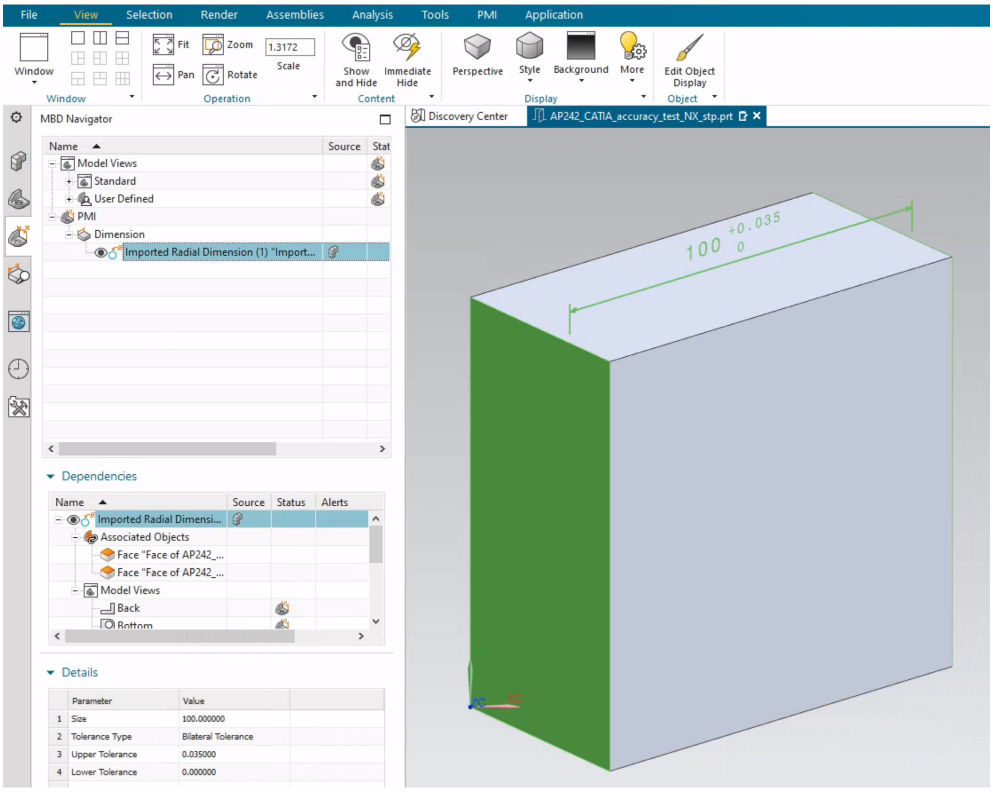


Figure A.49: Result of re-importing the re-exported STEP AP242 file in Siemens NX Version 2019

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, Creo has several configuration options for importing neutral exchange files. For each configuration option, when the STEP file is imported, the annotation is defined as “representation PMI” (see Figure A.39 and Table A.58). When the imported model is re-exported to a STEP AP242 file (see Figure A.40), the annotation is saved as “presentation PMI” and is defined as a “weld symbol” (see Table A.59).

Table A.58: A summary of the results for PTC Creo for the import of the STEP AP242 file generated by CATIA V5

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	0.005	0.005	0.005	0.005
Import successful	Yes	Yes	Yes	Yes
Representation PMI recognised	Yes	Yes	Yes	Yes
Export accuracy	0.0149994	0.005	0.0149994	0.01
Representation PMI retained	No	No	No	No

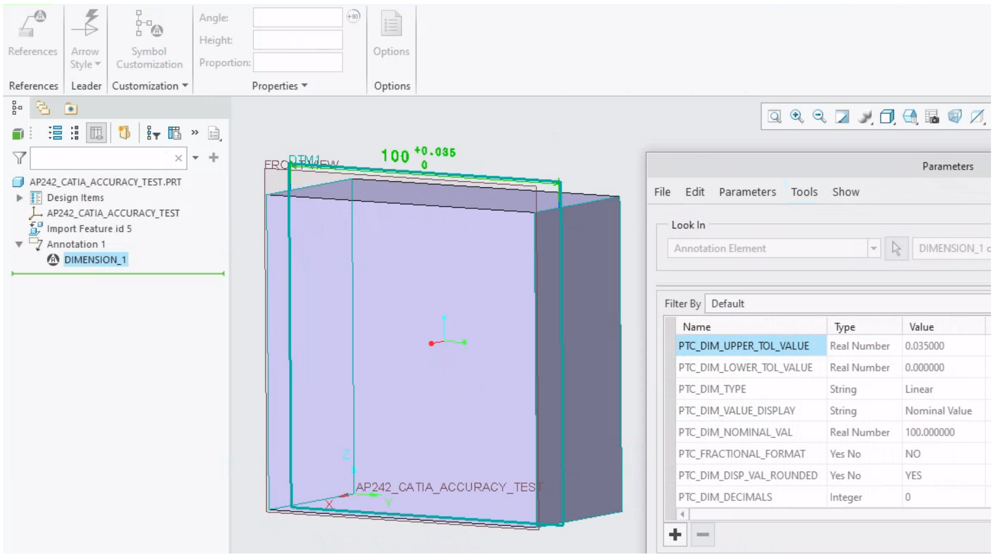


Figure A.50: Result of importing the STEP AP242 file generated by CATIA V5 in PTC Creo (accuracy set to automatic)

Table A.59: Excerpt from the definition of the dimension in the STEP AP242 file created by CATIA V5 and imported in and re-exported by PTC Creo 8.0.4.0

STEP

`#2559=CURVE_STYLE('',#23,POSITIVE_LENGTH_MEASURE(2.E-2),#4);`
`#2560=PRESENTATION_STYLE_ASSIGNMENT((#2559));`
`#2561=ANNOTATION_CURVE_OCCURRENCE('DIMENSION_1',(#2560),#2558);`
`#4=COLOUR_RGB('',2.156862745098E-1,0.E0,3.725490196078E-1);`
`#2558=GEOMETRIC_CURVE_SET('weld symbol',(#357,#360,#363,#366,#369,#372,#375,`
`...`
`#2510,#2515,#2520,#2525,#2530,#2535,#2538,#2541,#2544,#2547,#2552,#2557));`

Import in CATIA V5-6R2022 SP1

To import the STEP file, the options were used as shown in Figure A.31. The "Scale" option within this CATIA model is "Normal range". Th absolute accuracy of the CAD model is hereby set to 1×10^{-3} mm (see page A25). The feature tree in Figure A.51 shows that the annotation is imported as "representation PMI". When the imported model is re-exported to a STEP AP242 file, analysis of the resulting STEP AP242 file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the annotation is saved as "representation PMI" (see Figure A.52). All information of the original STEP file is retained.

The absolute accuracy specified in this STEP AP242 file is 0.005 mm as indicated by the parameter `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

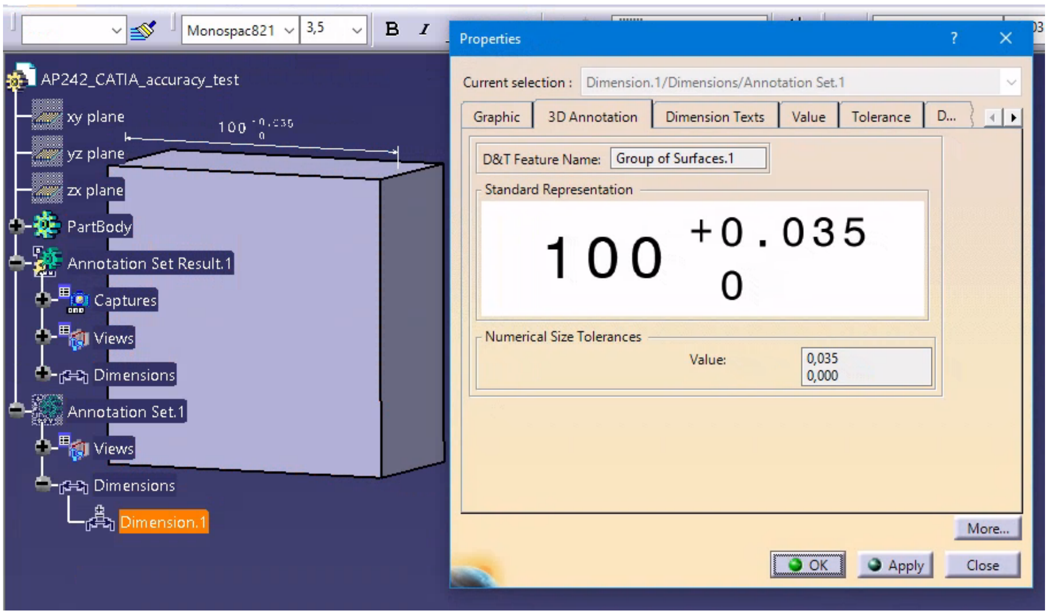
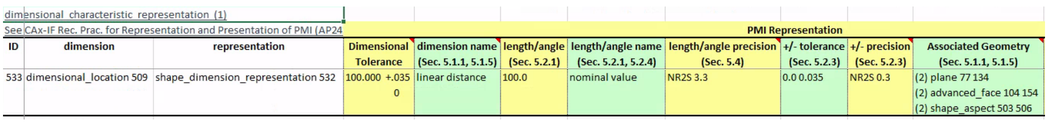


Figure A.51: Result of importing the STEP AP242 file generated by CATIA V5 in CATIA V5



dimensional_characteristic_representation (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PMI (AP24

			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
533	dimensional_location 509	shape_dimension_representation 532	100.000 +.035 0	linear distance	100.0	nominal value	NR2S 3.3	0.0 0.035	NR2S 0.3	(2) plane 77 134 (2) advanced_face 104 154 (2) shape_aspect 503 506

Figure A.52: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP file re-exported by CATIA V5

A.3.3 C. Export from Siemens NX Version 2019 Build 2501

In Siemens NX, a beam model is created with the dimensions 100 mm $\times$ 100 mm $\times$ 50 mm. In it, $100^{+0.035}_{-0.000}$, a dimension with a lower and an upper tolerance is created (Figure A.53). This model was exported to a STEP AP242 file. The settings used for this purpose are shown in Figure A.36. Analysis of the STEP file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the dimension is stored as "representation PMI" (Figure A.54).

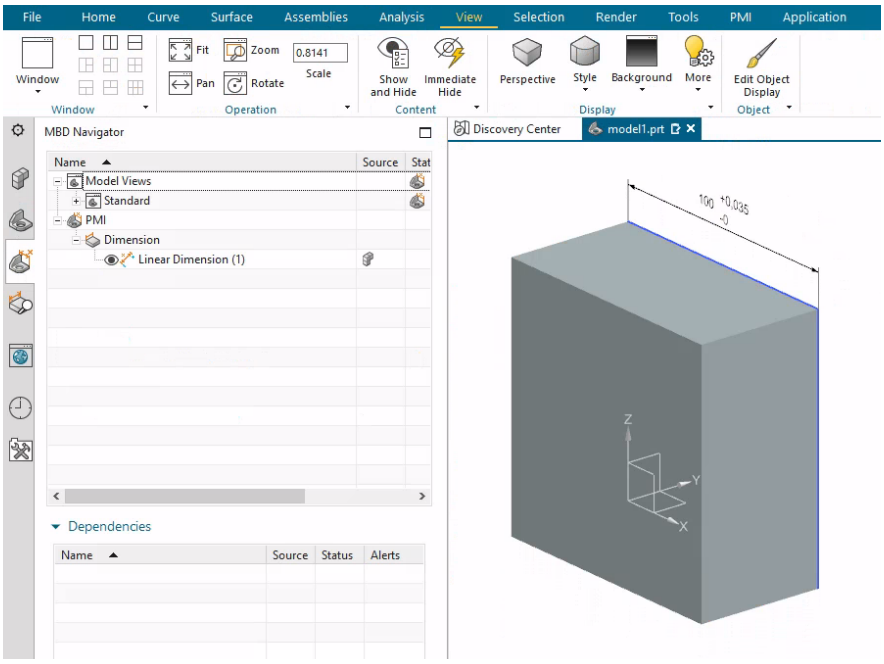
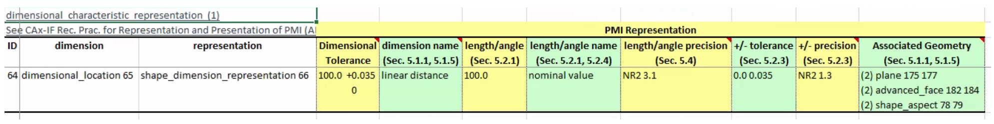


Figure A.53: Beam model with a dimension with a lower and an upper tolerance (Siemens NX Version 2019)



dimensional_characteristic_representation (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PMI (A

			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
64	dimensional_location 65	shape_dimension_representation 66	100.0 +0.035 0	linear distance	100.0	nominal value	NR2 3.1	0.0 0.035	NR2 1.3	(2) plane 175 177 (2) advanced_face 182 184 (2) shape_aspect 78 79

Figure A.54: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file exported by Siemens NX

The dimension can be found in the following format in the resulting STEP AP242 file (Table A.60). The tolerances are assigned to a linear distance as indicated by the parameter `DIMENSIONAL_LOCATION` (Boy et al. 2014, p. 9-11).

Table A.60: Excerpt from the definition of the dimension in the STEP AP242 file created by Siemens NX

Dimension component	STEP

100.000	`#51=PLUS_MINUS_TOLERANCE(#54,#58);`
	`#57=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#58,#59);`
	`#58=DIMENSIONAL_LOCATION('linear distance','',#190,#191);`
	`#59=SHAPE_DIMENSION_REPRESENTATION('',(#60),#475);`
	`#60=(`
	`LENGTH_MEASURE_WITH_UNIT()`
	`MEASURE_REPRESENTATION_ITEM()`
	`MEASURE_WITH_UNIT(POSITIVE_LENGTH_MEASURE(100.),#479)`
	`QUALIFIED_REPRESENTATION_ITEM((#61))`
	`REPRESENTATION_ITEM('nominal value'));`
	`#61=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.1');`
	`#190=SHAPE_ASPECT('','',#482,.T.);`
	`#191=SHAPE_ASPECT('','',#482,.T.);`
	`#475=(`
	`GEOMETRIC_REPRESENTATION_CONTEXT(3)`
	`GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#476))`
	`GLOBAL_UNIT_ASSIGNED_CONTEXT((#479,#478,#477))`
	`REPRESENTATION_CONTEXT('AP242_NX_accuracy_test',`
	`'TOP_LEVEL_ASSEMBLY_PART'));`
	`#476=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#479,`
	`'DISTANCE_ACCURACY_VALUE','Maximum Tolerance applied to`
	`model');`
	`#479=(`
	`LENGTH_UNIT()`
	`NAMED_UNIT(*)`
	`SI_UNIT(.MILLI.,.METRE.));`

`+0.035`	`#53=MEASURE_QUALIFICATION('','',#56,(#62));`
	`#54=TOLERANCE_VALUE(#55,#56);`
	`#56=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#479);`
	`#62=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');`
	`#479=(`
	`LENGTH_UNIT()`
	`NAMED_UNIT(*)`
	`SI_UNIT(.MILLI.,.METRE.)`
	`);`

`-0`	`#52=MEASURE_QUALIFICATION('','',#55,(#62));`
	`#54=TOLERANCE_VALUE(#55,#56);`
	`#55=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#479);`
	`#62=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');`
	`#479=(`
	`LENGTH_UNIT()`
	`NAMED_UNIT(*)`
	`SI_UNIT(.MILLI.,.METRE.)`
	`);`

The accuracy of the display of the dimension is specified by

```
#68=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.1');
```

This indicates that the dimension value should be displayed with three digits before and one after the decimal point.

The accuracy of the display of the upper and lower tolerance is determined by

```
#69=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.3');
```

This indicates that the value of the upper and the lower tolerance should be displayed with one digit before the decimal point and three digits after it.

The absolute accuracy of the CAD model in this STEP AP242 file is 0.005 mm (Table A.61) as indicated by `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

Table A.61: Model accuracy applied within the STEP AP242 file created by Siemens NX

Model accuracy	STEP
0.005	`#414=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#417,` `'DISTANCE_ACCURACY_VALUE','Maximum Tolerance applied to model');`

mm	`#417=(`
	`LENGTH_UNIT()`
	`NAMED_UNIT(*)`
	`SI_UNIT(.MILLI.,.METRE.)`
	`);`

Import in Inventor 2022

The default settings for STEP import were used. Only the "graphical PMI" option was additionally checked (see Figure A.41). The dimension is imported as "presentation PMI" (Figure A.55). This is consistent with the terminology in the "graphical PMI" option. When the imported model is re-exported to a STEP AP242 file, the annotation is no longer present in the file, either as representation PMI or presentation PMI.

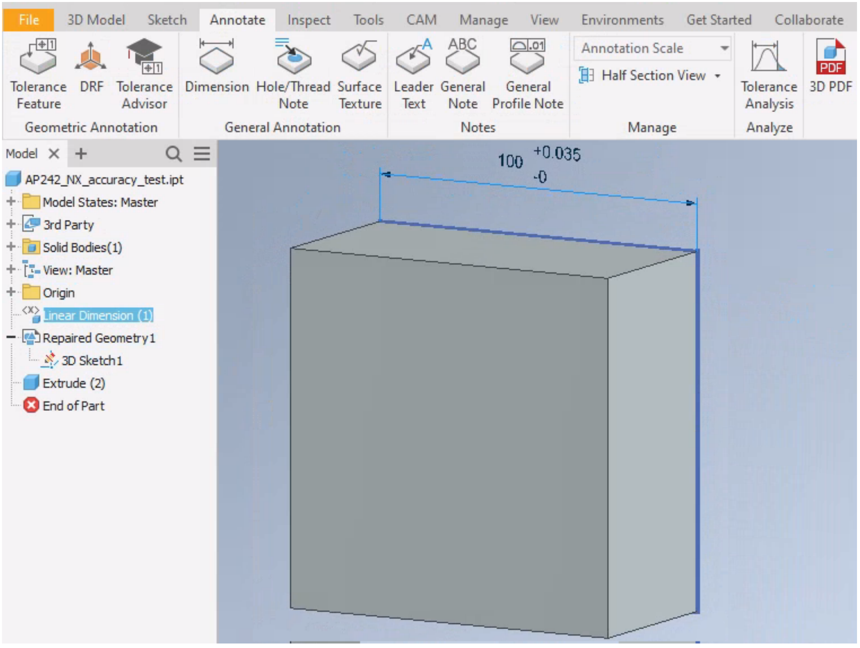


Figure A.55: Result of importing the STEP AP242 file generated by Siemens NX in Inventor 2022

Import in CATIA V5-6R2022 SP1

To import the STEP file, the options were used as shown in Figure A.31. The “Scale” option within this CATIA model is “Normal range”. Th absolute accuracy of the CAD model is hereby set to 1×10^{-3} mm (see page A25). The feature tree in Figure A.56 shows that the annotation is imported as “representation PMI”. However, this “representation PMI” is incomplete. The semantic references appear to be present, but the values of the dimension and of the assigned tolerances cannot be retrieved.

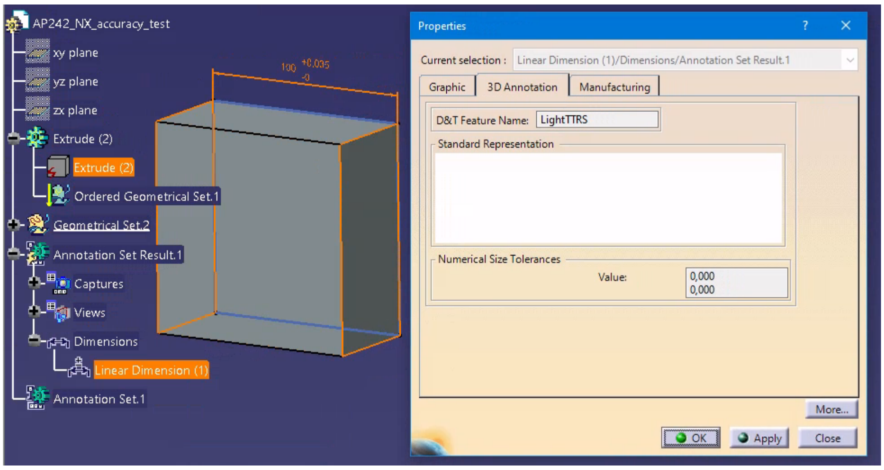


Figure A.56: Result of importing the STEP AP242 file generated by Siemens NX in CATIA V5

When the imported model is re-exported to a STEP AP242 file with the settings shown in Figure A.45, analysis of the resulting STEP file with the “NIST STEP File Analyzer and Viewer”(Lipman 2017) shows that the annotation is saved as “presentation PMI” (see Figure A.57). This “presentation PMI” is named “linear dimension” (see Table A.62).

The absolute accuracy specified in this STEP file is 0.005 mm as indicated by the parameter UNCERTAINTY_MEASURE_WITH_UNIT (Boy et al. 2014, p. 8).

draughting_callout (1)

ID	name	contents
268	Linear Dimension (1)	(1) tessellated_annotation_occurrence 267

Figure A.57: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by CATIA V5

Table A.62: Excerpt from the definition of the dimension in the STEP AP242 file created by Siemens NX and imported in and re-exported by CATIA V5

STEP

```
#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,'distance_accuracy_value',
'CONFUSED CURVE UNCERTAINTY') ;
#266=PRESENTATION_STYLE_ASSIGNMENT((#265)) ;
#265=CURVE_STYLE(' ',#31,POSITIVE_LENGTH_MEASURE(0.129999995232),#264) ;
#261=DIRECTION('Axis2P3D Direction',(0.,-1.,0.))  ;
#262=DIRECTION('Axis2P3D XDirection',(1.,0.,0.))  ;
#260=CARTESIAN_POINT('Axis2P3D Location',(100.,0.,0.))  ;
#263=AXIS2_PLACEMENT_3D('Linear Dimension (1)',#260,#261,#262) ;
#268=DRAUGHTING_CALLOUT('Linear Dimension (1)',(#267)) ;
#267=TESSELLATED_ANNOTATION_OCCURRENCE('Linear Dimension (1)',(#266),#259) ;
#286=COORDINATES_LIST(' ',2,((-100.,100.,0.),(-100.,115.22588,0.)))  ;
#296=COORDINATES_LIST(' ',2,((0.,100.,0.),(0.,115.22588,0.)))  ;
#306=COORDINATES_LIST(' ',2,((-100.,113.22587,0.),(0.,113.22587,0.)))  ;
#315=COMPLEX_TRIANGULATED_SURFACE_SET('Linear Dimension(1)',#316,786,((0.,-1.,0.)),(),
...
688),(694,695,697,698,693),(10,11,13,14,9))) ;
#264=DRAUGHTING_PRE_DEFINED_COLOUR('white') ;
#31=DRAUGHTING_PRE_DEFINED_CURVE_FONT('continuous') ;
#285=TESSELLATED_CURVE_SET('Linear Dimension (1)',#286,((1,2))) ;
#295=TESSELLATED_CURVE_SET('Linear Dimension (1)',#296,((1,2))) ;
#305=TESSELLATED_CURVE_SET('Linear Dimension (1)',#306,((1,2))) ;
#259=(GEOMETRIC_REPRESENTATION_ITEM()REPOSITIONED_TESSELLATED_ITEM(#263)REPRESENTATION_ITEM(
'linear dimension')TESSELLATED_GEOMETRIC_SET((#285,#295,#305,#315))TESSELLATED_ITEM()) ;
```

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, Creo has several configuration options for importing neutral exchange files. For each configuration option, when the STEP file is imported, the annotation is defined as "representation PMI" (see Figure A.58 and Table A.63). However the dimension value and the assigned tolerances are not legible. This is because PTC Creo doesn't recreate the annotation as a dimension with assigned tolerances, but as a graphic symbol.

Table A.63: A summary of the results for PTC Creo for the import of the STEP AP242 file generated by CATIA V5

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	0.005	0.005	0.005	0.005
Import successful	Yes	Yes	Yes	Yes
Representation PMI recognised	Yes	Yes	Yes	Yes
Export accuracy	0.0149994	0.005	0.0149994	0.01
Representation PMI retained	No	No	No	No

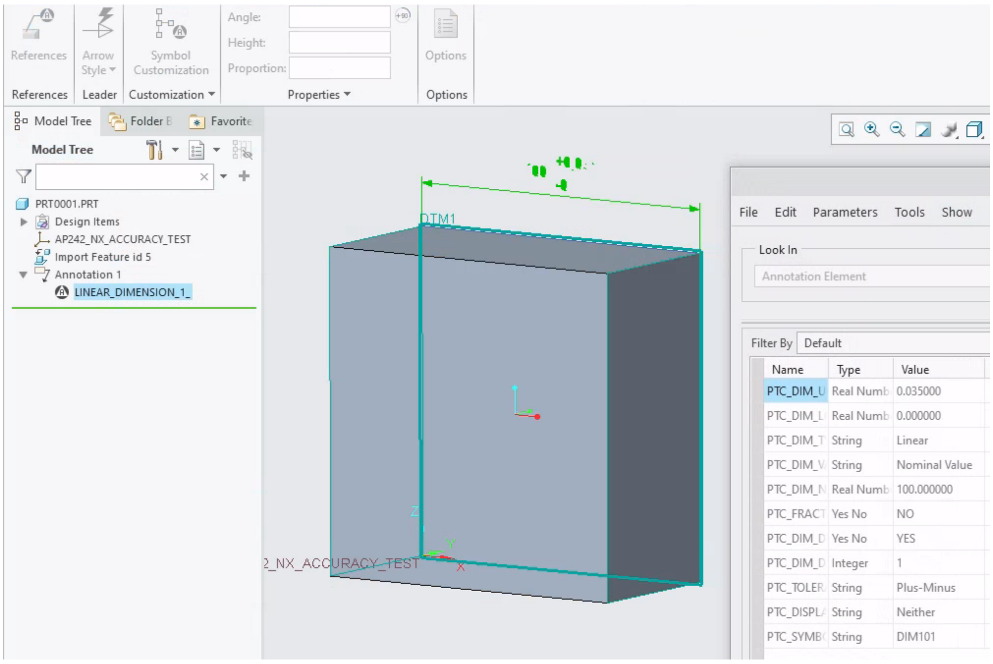


Figure A.58: Result of importing the STEP AP242 file generated by Siemens NX in PTC Creo (accuracy set to automatic)

When the imported model is re-exported to a STEP AP242 file with the settings shown in Figure A.40, analysis of the resulting STEP file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the annotation is saved as "presentation PMI" (Figure A.60) and is defined as a "weld symbol" (Table A.64).

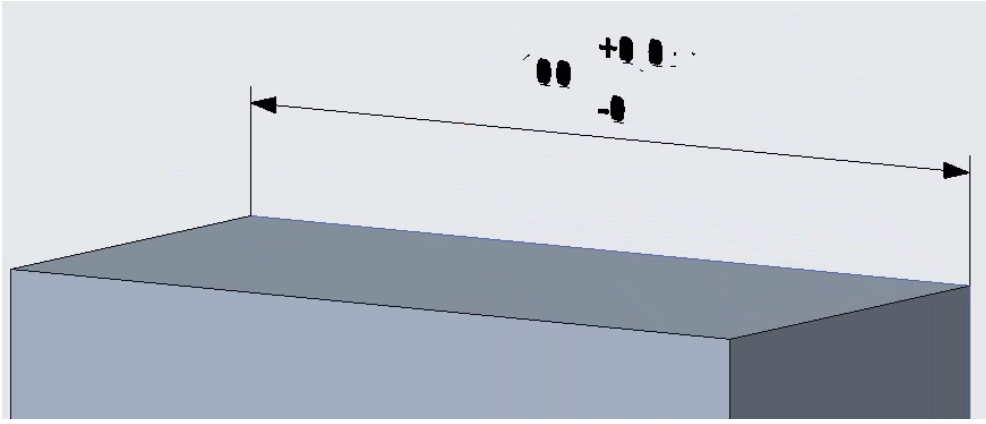
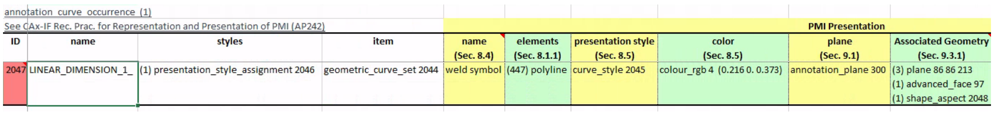


Figure A.59: The dimension value and the assigned tolerances are not legible in PTC Creo (accuracy set to automatic)



annotation_curve_occurrence (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PMI (AP242)

				PMI Presentation					
ID	name	styles	item	name (Sec. 8.4)	elements (Sec. 8.1.1)	presentation style (Sec. 8.5)	color (Sec. 8.5)	plane (Sec. 9.1)	Associated Geometry (Sec. 9.3.1)
2047	LINEAR_DIMENSION_1_	(1) presentation_style_assignment 2046	geometric_curve_set 2044	weld symbol	(447) polyline	curve_style 2045	colour_rgb 4 (0.216 0. 0.373)	annotation_plane 300	(3) plane 86 86 213 (1) advanced_face 97 (1) shape_aspect 2048

Figure A.60: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by PTC Creo (accuracy set to automatic)

Table A.64: Excerpt from the definition of the dimension in the STEP AP242 file created by Siemens NX and imported in and re-exported by PTC Creo 8.0.4.0

STEP
#172=DRAUGHTING_PRE_DEFINED_CURVE_FONT('continuous'); #2045=CURVE_STYLE('',#172,POSITIVE_LENGTH_MEASURE(2.E-2),#4); #2046=PRESENTATION_STYLE_ASSIGNMENT((#2045)); #2047=ANNOTATION_CURVE_OCCURRENCE('LINEAR_DIMENSION_1_',(#2046),#2044); #4=COLOUR_RGB('',2.156862745098E-1,0.E0,3.725490196078E-1); #2044=GEOMETRIC_CURVE_SET('weld symbol',(#303,#306,#309,#312,#315,#318,#321, ... #2029,#2032,#2035,#2038,#2043)); #2048=SHAPE_ASPECT('','aspect to capture model element for association',#243,.T.); #2049=DRAUGHTING_MODEL_ITEM_ASSOCIATION('','',#2048,#293,#2047); #2050=ITEM_IDENTIFIED_REPRESENTATION_USAGE('',$,#2048,#224, SET_REPRESENTATION_ITEM((#213,#97))); #2051=DRAUGHTING_MODEL_ITEM_ASSOCIATION('','',#2048,#293,#2047);

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.34). The MBD navigator in Figure A.61 shows that the annotation is imported as “representation PMI”.

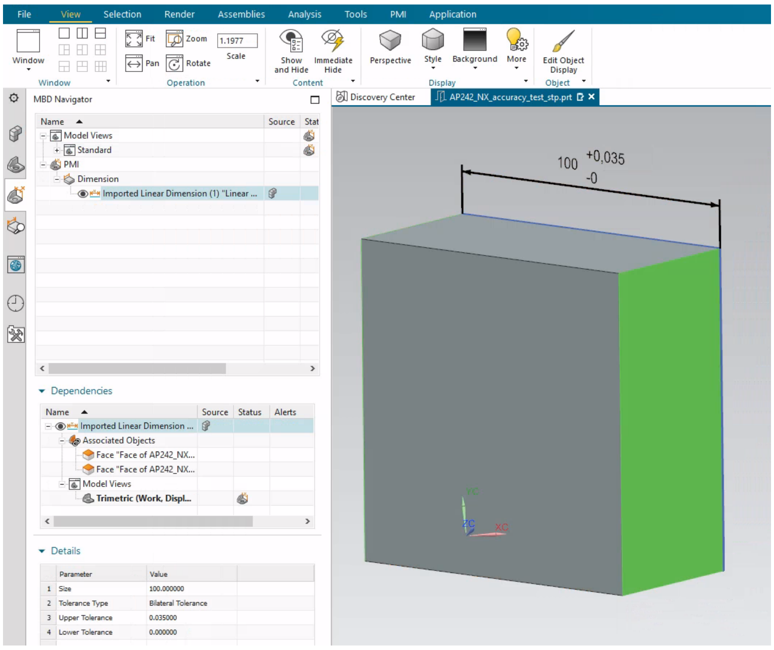


Figure A.61: Result of importing the STEP AP242 file generated by Siemens NX in Siemens NX Version 2019

When the imported model is re-exported to a STEP AP242 file, analysis of the resulting STEP AP242 file with the “NIST STEP File Analyzer and Viewer”(Lipman 2017) shows that the annotation is saved as “representation PMI” (Figure A.62).

dimensional_characteristic_representation (1)										
See CAx-IF Rec. Prac. for Representation and Presentation of PM			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
52	dimensional_size 53	shape_dimension_representation 54	⌀100.0 +0.03 0	diameter	100.0	nominal value	NR2 3.1	0.0 0.035	NR2 1.2	(2) plane 164 166 (2) advanced_face 171 173 (2) shape_aspect 185 186 (1) composite_group_shape_aspect 66

Figure A.62: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by Siemens NX

However, two things differ from the original STEP AP242 file. The first one is that the display accuracy of the assigned tolerances has changed from one digit before and three digits after the decimal point to one digit before and two digits after the decimal point (see Table A.65). However, this is not displayed when the re-exported STEP file is re-imported into Siemens NX (see Figure A.63). It can be concluded that the value of the parameter `VALUE_FORMAT_TYPE_QUALIFIER` is ignored by the CAD system. The second one is that the original parameter `DIMENSIONAL_LOCATION='linear distance'` is replaced by `DIMENSIONAL_SIZE='diameter'` (see Table A.65 and Figure A.63). This refers to the connection between the geometry and the dimension (Boy et al. 2014, p. 9, 15).

Table A.65: Excerpt from the definition of the dimension in the STEP AP242 file created by Siemens NX and imported in and re-exported by Siemens NX

Dimension component	STEP
100	#54=SHAPE_DIMENSION_REPRESENTATION('',(#55),#489); #55=(LENGTH_MEASURE_WITH_UNIT() MEASURE_REPRESENTATION_ITEM() MEASURE_WITH_UNIT(POSITIVE_LENGTH_MEASURE(100.),#493) QUALIFIED_REPRESENTATION_ITEM((#56)) REPRESENTATION_ITEM('nominal value')); #56=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.1'); #489=(GEOMETRIC_REPRESENTATION_CONTEXT(3) GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#490)) GLOBAL_UNIT_ASSIGNED_CONTEXT((#493,#492,#491)) REPRESENTATION_CONTEXT('AP242_NX_accuracy_test_NX', 'TOP_LEVEL_ASSEMBLY_PART')); #490=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#493, 'DISTANCE_ACCURACY_VALUE','Maximum Tolerance applied to model') #493=(LENGTH_UNIT() NAMED_UNIT(*) SI_UNIT(.MILLI.,.METRE.));
+0.035	#46=PLUS_MINUS_TOLERANCE(#47,#53); #47=TOLERANCE_VALUE(#50,#51); #49=MEASURE_QUALIFICATION('','',#51,(#57)); #53=DIMENSIONAL_SIZE(#66,'diameter'); #51=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#493); #57=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.2'); #66=COMPOSITE_GROUP_SHAPE_ASPECT('Imported Linear Dimension (1) "Linear Dimension (1)"', 'multiple elements',#494,.T.);
0	#46=PLUS_MINUS_TOLERANCE(#47,#53); #47=TOLERANCE_VALUE(#50,#51); #48=MEASURE_QUALIFICATION('','',#50,(#57)); #53=DIMENSIONAL_SIZE(#66,'diameter'); #50=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#493); #57=VALUE_FORMAT_TYPE_QUALIFIER('NR2 1.2'); #66=COMPOSITE_GROUP_SHAPE_ASPECT('Imported Linear Dimension (1) "Linear Dimension (1)"', 'multiple elements',#494,.T.);

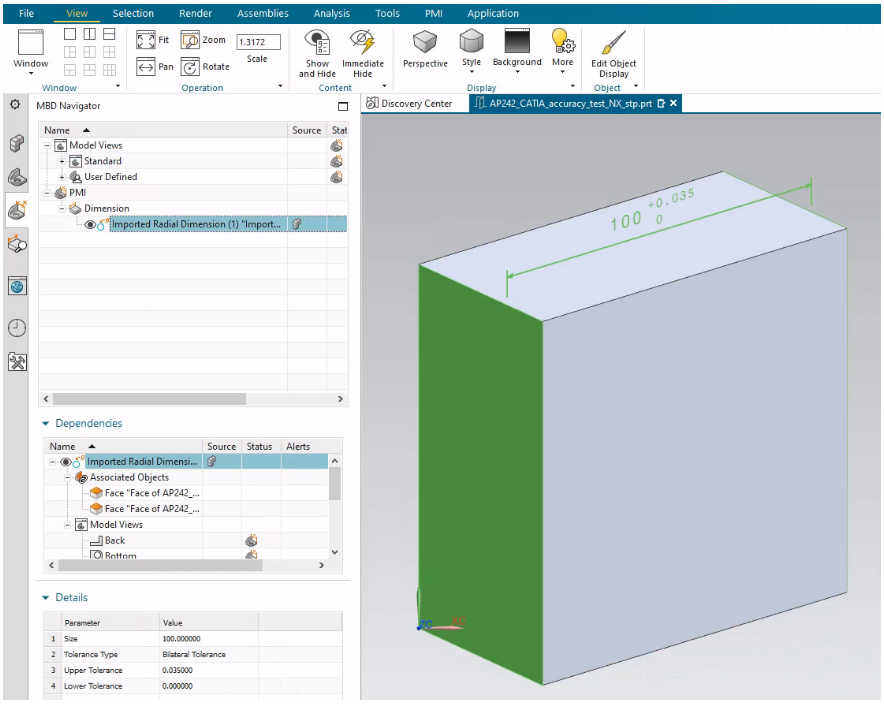


Figure A.63: Result of re-importing the re-exported STEP AP242 file in Siemens NX Version 2019

The absolute accuracy of the CAD model in this re-exported STEP file is 0.005 mm as indicated by `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

A.3.4 D. Export from PTC Creo Parametric 8.0.4.0

In PTC Creo 8.0.4.0, a beam model is created with the dimensions 100 mm $\times$ 100 mm $\times$ 50 mm. In it, $100^{+0.035}_{-0.000}$, a dimension with a lower and an upper tolerance is created (Figure A.64).

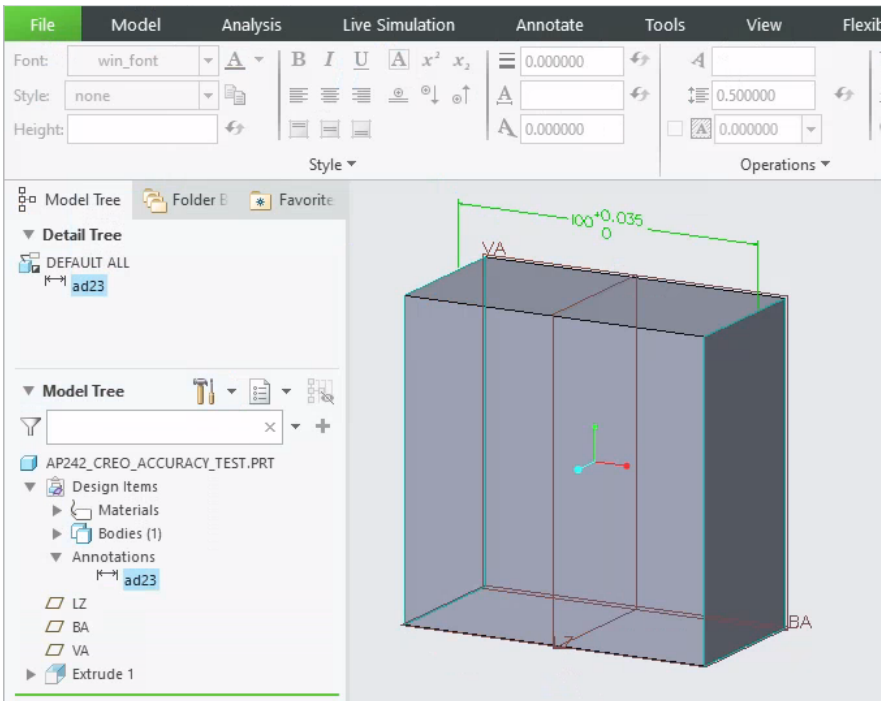
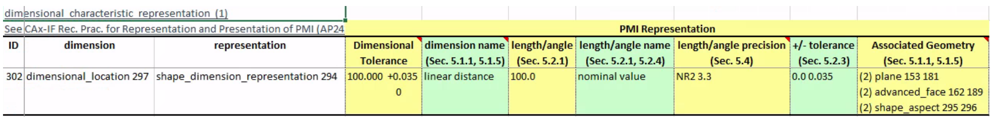


Figure A.64: Beam model with a dimension with a lower and an upper tolerance (PTC Creo 8.0.4.0)

Within PTC Creo 8.0.4.0, there are three different methods for creating a 3D annotation. The first is as an annotation element (driving dimension). The second is as a driven dimension. The third one is as an annotation feature. When the first method is used then the annotation is exported as presentation PMI. Only when the second and third method are used, the annotation is exported as representation PMI. For this test, the annotation is created as a driven dimension. The model was exported to a STEP AP242 file. The settings used for this purpose are shown in Figure A.40. Analysis of the STEP file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the dimension is stored as "representation PMI" (Figure A.65).



dimensional_characteristic_representation (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PMI (AP24

ID	dimension	representation	PMI Representation Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
302	dimensional_location 297	shape_dimension_representation 294	100.000 +0.035 0	linear distance	100.0	nominal value	NR2 3.3	0.0 0.035	(2) plane 153 181 (2) advanced_face 162 189 (2) shape_aspect 295 296

Figure A.65: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file exported by PTC Creo 8.0.4.0 (driven dimension)

The dimension can be found in the following format in the resulting STEP AP242 file (Table A.66). The tolerances are assigned to a linear distance as indicated by the parameter `DIMENSIONAL_LOCATION` (Boy et al. 2014, p. 9-11).

Table A.66: Excerpt from the definition of the dimension in the STEP AP242 file created by PTC Creo 8.0.4.0

Dimension component	STEP
100.000	#230=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(1.E-2),#225,'closure', 'Maximum model space distance between geometric entities at asserted connectivities'); #292=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.3'); #293=(LENGTH_MEASURE_WITH_UNIT()MEASURE_REPRESENTATION_ITEM() MEASURE_WITH_UNIT(LENGTH_MEASURE(1.E2),#225) QUALIFIED_REPRESENTATION_ITEM((#292))REPRESENTATION_ITEM #297=DIMENSIONAL_LOCATION('linear distance','',#295,#296); #301=PLUS_MINUS_TOLERANCE(#300,#297); #302=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#297,#294); #225=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)); #231=(GEOMETRIC_REPRESENTATION_CONTEXT(3) GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#230))GLOBAL_UNIT_ASSIGNED_CONTEXT ((#225,#228,#229))REPRESENTATION_CONTEXT('ID1','3')); #294=SHAPE_DIMENSION_REPRESENTATION('',(#293),#231); #295=SHAPE_ASPECT('','',#249,.T.); #296=SHAPE_ASPECT('','',#249,.T.);
+0.035	#299=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(3.5E-2),#225); #300=TOLERANCE_VALUE(#298,#299); #225=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.));
-0	#298=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.E0),#225); #300=TOLERANCE_VALUE(#298,#299); #225=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.));

The accuracy of the display of the dimension is specified by

```
#292=VALUE_FORMAT_TYPE_QUALIFIER('NR2 3.3');
```

This indicates that the dimension value should be displayed with three digits before and three after the decimal point.

No separate accuracy is specified for the lower and upper tolerances.

The absolute accuracy of the CAD model in this STEP AP242 file is 0.01 mm (Table A.67).

Table A.67: Model accuracy applied within the STEP AP242 file created by PTC Creo

Model accuracy	STEP
0.01	#230=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(1.E-2),#225,'closure', 'Maximum model space distance between geometric entities at asserted connectivities');
mm	#225=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.));

Import in Inventor 2022

The default settings for STEP import were used. Only the "graphical PMI" option was additionally checked (see Figure A.41). The dimension is imported as "presentation PMI" (Figure A.66). This is consistent with the terminology in the "graphical PMI" option. When the imported model is re-exported to a STEP AP242 file, the annotation is no longer present in the file, either as representation PMI or presentation PMI.

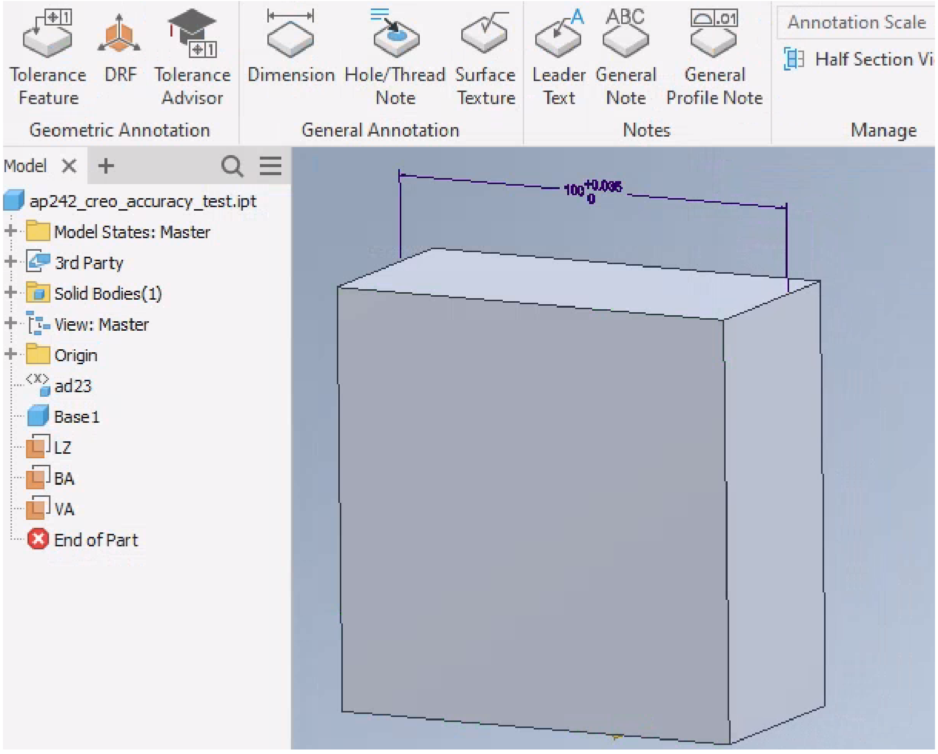


Figure A.66: Result of importing the STEP AP242 file generated by PTC Creo 8.0.4.0 in Inventor 2022

Import in CATIA V5-6R2022 SP1

To import the STEP file, the options were used as shown in Figure A.31. The “Scale” option within this CATIA model is “Normal range”. Th absolute accuracy of the CAD model is hereby set to 1×10^{-3} mm (see page A25). The feature tree in Figure A.67 shows that the annotation is imported as “representation PMI”.

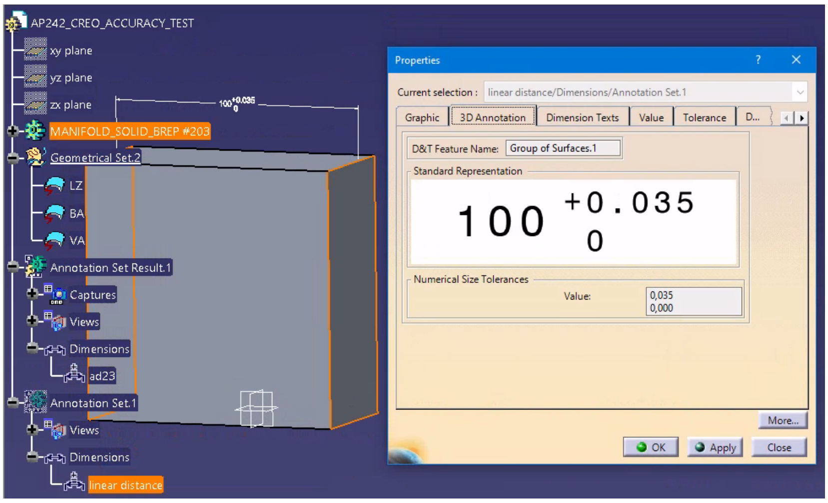


Figure A.67: Result of importing the STEP AP242 file generated by PTC Creo 8.0.4.0 in CATIA V5

When the imported model is re-exported to a STEP AP242 file with the settings shown in Figure A.45, analysis of the resulting STEP file with the “NIST STEP File Analyzer and Viewer”(Lipman 2017) shows that the annotation is saved as “representation PMI” (see Figure A.68). However, one thing differs from the original STEP AP242 file. Namely, the display accuracy of the assigned tolerances is now explicitly specified with no digits before the decimal point and three digits after. This was not the case in the original STEP AP242 file (Table A.68).

The absolute accuracy specified in this STEP file is 0.005 mm as indicated by the parameter `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

dimensional_characteristic_representation (1)										
See CAx-IF Rec. Prac. for Representation and Presentation of PMI (AP24			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
493	dimensional_location 469	shape_dimension_representation 492	100.000 +.035 0	linear distance	100.0	nominal value	NR2S 3.3	0.0 0.035	NR2S 0.3	(2) plane 116 175 (2) advanced_face 143 195 (2) shape_aspect 463 466

Figure A.68: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by CATIA V5

Table A.68: Excerpt from the definition of the dimension in the STEP AP242 file created by PTC Creo 8.0.4.0 and imported in and re-exported by CATIA V5

Dimension component	STEP

100.000	`#496=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 3.3') ;`
	`#463=SHAPE_ASPECT('','single feature',#11,.T.) ;`
	`#466=SHAPE_ASPECT('','single feature',#11,.T.) ;`
	`#469=DIMENSIONAL_LOCATION('linear distance','',#463,#466) ;`
	`#493=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#469,#492) ;`
	`#492=SHAPE_DIMENSION_REPRESENTATION('',(#495),#16) ;`
	`#15=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#12,`
	`'distance_accuracy_value','CONFUSED CURVE UNCERTAINTY') ;`
	`#506=PLUS_MINUS_TOLERANCE(#505,#469) ;`
	`#12=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`
	`#494=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`
	`#495=(LENGTH_MEASURE_WITH_UNIT()MEASURE_REPRESENTATION_ITEM()`
	`MEASURE_WITH_UNIT(LENGTH_MEASURE(100.),#494)`
	`QUALIFIED_REPRESENTATION_ITEM((#496))REPRESENTATION_ITEM`
	`('nominal value')) ;`
	`#16=(GEOMETRIC_REPRESENTATION_CONTEXT(3)`
	`GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#15))`
	`GLOBAL_UNIT_ASSIGNED_CONTEXT((#12,#13,#14))REPRESENTATION_CONTEXT`
	`(' ',' ')) ;`

+0.035	`#504=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 0.3') ;`
	`#500=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#499) ;`
	`#503=MEASURE_QUALIFICATION('','',#500,(#504)) ;`
	`#506=PLUS_MINUS_TOLERANCE(#505,#469) ;`
	`#505=TOLERANCE_VALUE(#498,#500) ;`
	`#499=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`

-0	`#502=VALUE_FORMAT_TYPE_QUALIFIER('NR2S 0.3') ;`
	`#498=LENGTH_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#497) ;`
	`#501=MEASURE_QUALIFICATION('','',#498,(#502)) ;`
	`#506=PLUS_MINUS_TOLERANCE(#505,#469) ;`
	`#505=TOLERANCE_VALUE(#498,#500) ;`
	`#497=(LENGTH_UNIT()NAMED_UNIT(*)SI_UNIT(.MILLI.,.METRE.)) ;`

When the re-exported STEP AP242 file is re-imported in CATIA V5, two things stand out. The first is that there are now two dimensions present in the model. The second one is that the original dimension that was representation PMI is now only presentation PMI. The additional dimension has been placed in another plane and is representation PMI. (Figure A.69).

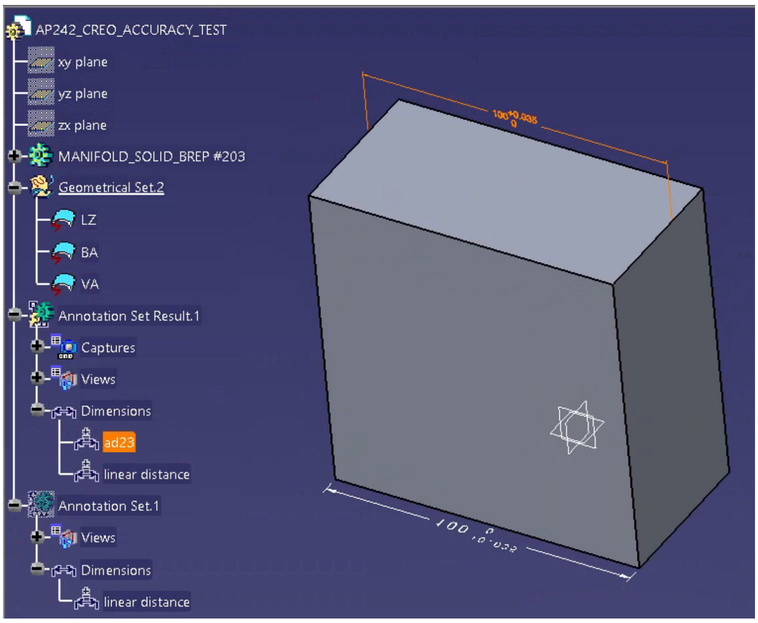


Figure A.69: Result of re-importing the re-exported STEP AP242 file in CATIA V5

Import in Siemens NX Version 2019 Build 2501

The default options for STEP import were used (see Figure A.34). The MBD navigator in Figure A.70 shows that the annotation is imported as "representation PMI".

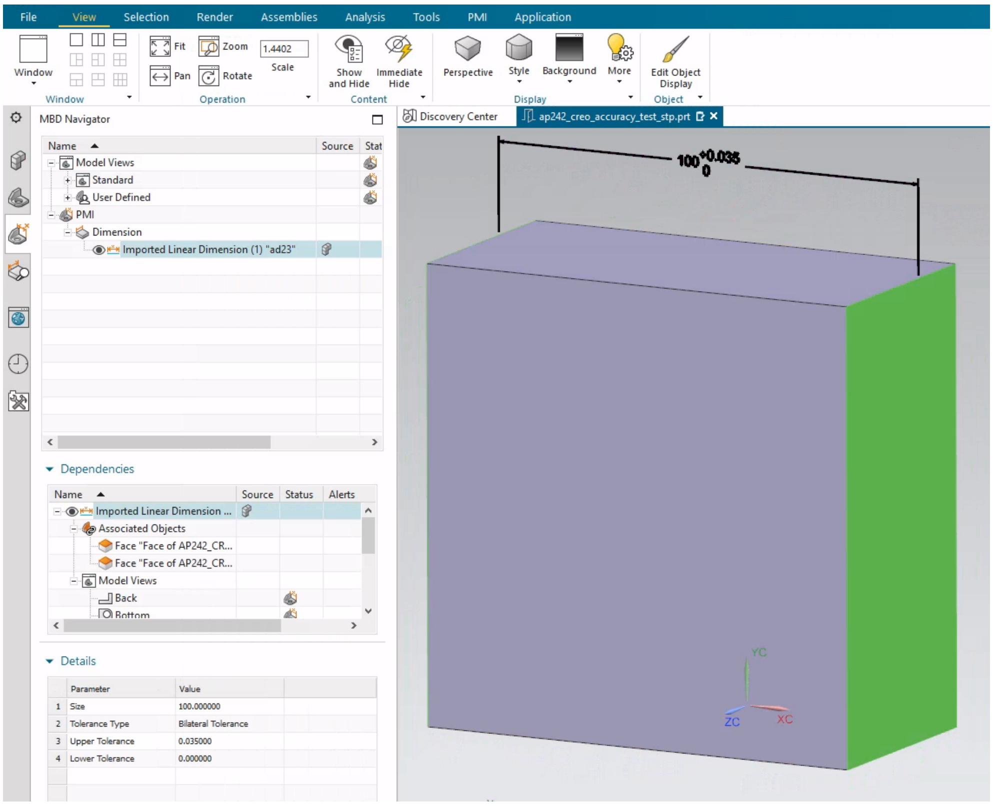


Figure A.70: Result of importing the STEP AP242 file generated by PTC Creo 8.0.4.0 in Siemens NX Version 2019

When the imported model is re-exported to a STEP AP242 file, analysis of the resulting STEP AP242 file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the annotation is saved as "representation PMI" (Figure A.71).

dimensional_characteristic_representation (1)										
See CAx-IF Rec. Prac. for Representation and Presentation of PM			PMI Representation							
ID	dimension	representation	Dimensional Tolerance	dimension name (Sec. 5.1.1, 5.1.5)	length/angle (Sec. 5.2.1)	length/angle name (Sec. 5.2.1, 5.2.4)	length/angle precision (Sec. 5.4)	+/- tolerance (Sec. 5.2.3)	+/- precision (Sec. 5.2.3)	Associated Geometry (Sec. 5.1.1, 5.1.5)
56	dimensional_size 57	shape_dimension_representation 58	⌀100.000 +0.03 0	diameter	100.0	nominal value	NR2 3.3	0.0 0.035	NR2 1.2	(2) plane 196 198 (2) advanced_face 203 205 (2) shape_aspect 111 112 (1) composite_group_shape_aspect 70

Figure A.71: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by Siemens NX

However, two things differ from the original STEP AP242 file. The first one is that the display accuracy of the assigned tolerances is now explicitly specified as one digit before the decimal point and two digits after. This was not the case in the original STEP AP242 file (Table A.69). However, this is not displayed when the re-exported STEP file is re-imported into Siemens NX (Figure A.72). It can be concluded that the value of the parameter `VALUE_FORMAT_TYPE_QUALIFIER` is ignored by the CAD system. The second one is that the original parameter `DIMENSIONAL_LOCATION='linear distance'` is replaced by `DIMENSIONAL_SIZE='diameter'` (see Table A.69 and Figure A.72). This refers to the connection between the geometry and the dimension (Boy et al. 2014, p. 9, 15).

Table A.69: Excerpt from the definition of the dimension in the STEP AP242 file created by PTC Creo and imported in and re-exported by Siemens NX

Dimension component	STEP
100	#50=PLUS_MINUS_TOLERANCE(#51,#57); #56=DIMENSIONAL_CHARACTERISTIC_REPRESENTATION(#57,#58); #57=DIMENSIONAL_SIZE(#70,’diameter’); #58=SHAPE_DIMENSION_REPRESENTATION(”,(#59),#429); #59=(LENGTH_MEASURE_WITH_UNIT() MEASURE_REPRESENTATION_ITEM() MEASURE_WITH_UNIT(POSITIVE_LENGTH_MEASURE(100.),#433) QUALIFIED_REPRESENTATION_ITEM((#60)) REPRESENTATION_ITEM(’nominal value’)); #60=VALUE_FORMAT_TYPE_QUALIFIER(’NR2 3.3’); #70=COMPOSITE_GROUP_SHAPE_ASPECT(’Imported Linear Dimension (1) "ad23"’,’multiple elements’,#434,.T.); #429=(GEOMETRIC_REPRESENTATION_CONTEXT(3) GLOBAL_UNCERTAINTY_ASSIGNED_CONTEXT((#430)) GLOBAL_UNIT_ASSIGNED_CONTEXT((#433,#432,#431)) REPRESENTATION_CONTEXT(’ap242_creo_accuracy_test_NX’, ’TOP_LEVEL_ASSEMBLY_PART’)); #430=UNCERTAINTY_MEASURE_WITH_UNIT(LENGTH_MEASURE(0.005),#433, ’DISTANCE_ACCURACY_VALUE’,’Maximum Tolerance applied to model’); #433=(LENGTH_UNIT() NAMED_UNIT(*) SI_UNIT(.MILLI.,.METRE.));
+0.035	#50=PLUS_MINUS_TOLERANCE(#51,#57); #51=TOLERANCE_VALUE(#54,#55); #53=MEASURE_QUALIFICATION(”,”,#55,(#61)); #55=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.035),#433); #61=VALUE_FORMAT_TYPE_QUALIFIER(’NR2 1.2’); #433=(LENGTH_UNIT() NAMED_UNIT(*) SI_UNIT(.MILLI.,.METRE.));
0	#50=PLUS_MINUS_TOLERANCE(#51,#57); #51=TOLERANCE_VALUE(#54,#55); #52=MEASURE_QUALIFICATION(”,”,#54,(#61)); #54=MEASURE_WITH_UNIT(LENGTH_MEASURE(0.),#433); #61=VALUE_FORMAT_TYPE_QUALIFIER(’NR2 1.2’); #433=(LENGTH_UNIT() NAMED_UNIT(*) SI_UNIT(.MILLI.,.METRE.));

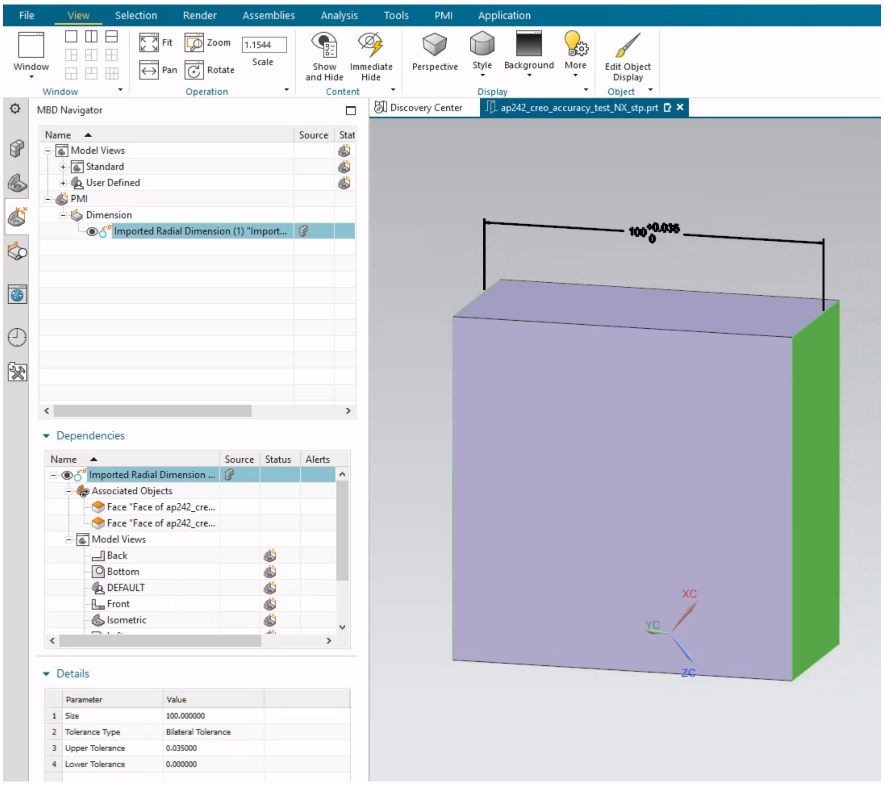


Figure A.72: Result of re-importing the re-exported STEP AP242 file in Siemens NX Version 2019

The absolute accuracy of the CAD model in this re-exported STEP file is 0.005 mm as indicated by `UNCERTAINTY_MEASURE_WITH_UNIT` (Boy et al. 2014, p. 8).

Import in PTC Creo Parametric 8.0.4.0

As mentioned on page A7, Creo has several configuration options for importing neutral exchange files. For each configuration option, when the STEP file is imported, the annotation is defined as "representation PMI" (see Figure A.73 and Table A.70).

Table A.70: A summary of the results for PTC Creo for the import of the STEP AP242 file generated by CATIA V5

	Automatic	External	Internal	Template (0.01)
Accuracy STEP file	0.01	0.01	0.01	0.01
Import successful	Yes	Yes	Yes	Yes
Representation PMI recognised	Yes	Yes	Yes	Yes
Export accuracy	0.0149994	0.01	0.0149994	0.01
Representation PMI retained	No	No	No	No

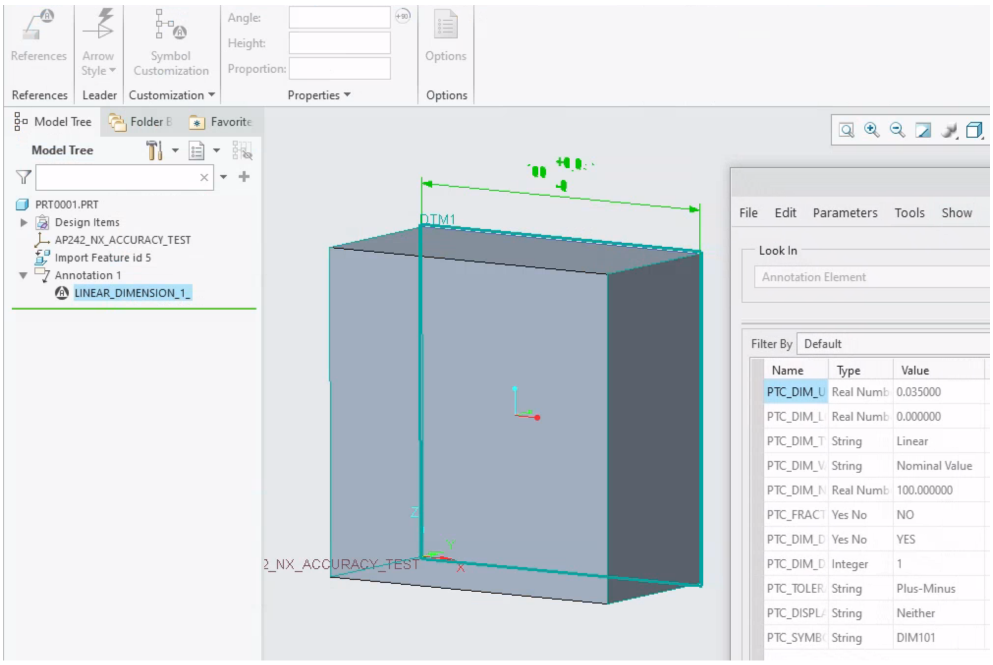
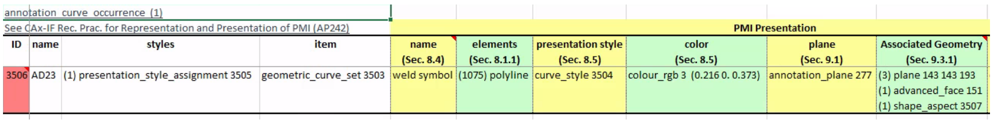


Figure A.73: Result of importing the STEP AP242 file generated by Siemens NX in PTC Creo (accuracy set to automatic)

When the imported model is re-exported to a STEP AP242 file with the settings shown in Figure A.40, analysis of the resulting STEP file with the "NIST STEP File Analyzer and Viewer"(Lipman 2017) shows that the annotation is saved as "presentation PMI" (Figure A.74) and is defined as a "weld symbol" (Table A.71).



annotation_curve_occurrence (1)

See CAx-IF Rec. Prac. for Representation and Presentation of PMI (AP242)

ID	name	styles	item	PMI Presentation name (Sec. 8.4)	elements (Sec. 8.1.1)	presentation style (Sec. 8.5)	color (Sec. 8.5)	plane (Sec. 9.1)	Associated Geometry (Sec. 9.3.1)
3506	AD23	(1) presentation_style_assignment 3505	geometric_curve_set 3503	weld symbol	(1075) polyline	curve_style 3504	colour_rgb 3 (0.216 0. 0.373)	annotation_plane 277	(3) plane 143 143 193 (1) advanced_face 151 (1) shape_aspect 3507

Figure A.74: Excerpt from the results of the NIST STEP File Analyzer and Viewer for the STEP AP242 file re-exported by PTC Creo (accuracy set to automatic)

Table A.71: Excerpt from the definition of the dimension in the STEP AP242 file created by PTC Creo 8.0.4.0 and imported in and re-exported by PTC Creo 8.0.4.0

STEP

`#170=DRAUGHTING_PRE_DEFINED_CURVE_FONT('continuous');`
`#3504=CURVE_STYLE('',#170,POSITIVE_LENGTH_MEASURE(2.E-2),#3);`
`#3505=PRESENTATION_STYLE_ASSIGNMENT((#3504));`
`#3506=ANNOTATION_CURVE_OCCURRENCE('AD23',(#3505),#3503);`
`#3=COLOUR_RGB('',2.156862745098E-1,0.E0,3.725490196078E-1);`
`#3503=GEOMETRIC_CURVE_SET('weld symbol',(#280,#283,#286,#289,#292,#295,#298,`
`...`
`#3472,#3475,#3478,#3481,#3484,#3487,#3490,#3493,#3496,#3499,#3502));`