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Engineering

ANALYSIS OF IMPLEMENTATION OF MBD AND STEP AP242 STANDARD INTO MODERN CAD TOOLS

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ABSTRACT:
THIS PAPER PRESENTS AN OVERVIEW OF RECORDING DATA ON TOLERANCES AND PRODUCT SURFACES, USING THE MODEL BASED DEFINITION (MBD) METHOD, AND THE TRANSFER BETWEEN VARIOUS MODERN CAD MODELERS, USING THE STEP AP242 STANDARD. WITH MBD IMPLEMENTED IN DESIGN PROCESS, WE PLACE 3D MODEL IN THE CENTER OF DEVELOPMENT. REFERRING TO THAT, THERE IS NO NEED FOR SEPARATE 2D DRAWING FILES AND THAT MAKES SIMPLER MANAGEMENT WITH ALL PROJECT DOCUMENTATION. USING MBD INSTEAD OF 2D DRAWINGS CAN BE TIME SAVING, WHEN WE ARE DESIGNING COMPLEX MODELS WITH A LOT OF PRODUCT MANUFACTURING INFORMATION (PMI). BY PRODUCING THE SAME MODEL IN DIFFERENT MODELERS, ADDING PMI TO MODELS AND CONVERTING MODELS INTO STEP AP242, WE REVIEWED THE RELEVANCE OF CONVERSIONS AND MODELERS. THE RESULTS HAVE SHOWN THAT MODERN MODELERS OF THE LATEST GENERATIONS ARE COMPATIBLE WITH THE ISO 10303-242: 2014 (STEP AP242) STANDARD AND ALLOW RELIABLE TRANSFER OF PMI BETWEEN DIFFERENT MODELERS USING THE MBD METHOD.

KEY WORDS: MBD, MODEL BASED DEFINITION, PMI, PRODUCT MANUFACTURING INFORMATION, STEP AP242

INTRODUCTION

Model based definition (MBD) stands for cataloging components or assemblies, using only a 3D model. This includes geometry, dimensional and geometric tolerances, materials and surface quality. Traditionally, cataloging these data included, and still does, transforming 3D models into 2D views and making technical drawings. With MBD the designers place PMI directly on 3D model in the process of designing. An example of a combination of a 2D drawing and a 3D model is shown in Figure 1. Being the most accurate and universal language for providing information on what to do and how to do it, 2D technical drawings are generally still being used for industrial communication. The basic purpose of technical drawings is to transfer, control and preserve

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product information in a way that eliminates misinterpretations and false assumptions[3]. The purpose of MBD is to place the 3D model at the center of development. The model can be accessed by anyone who needs information about this model, editing is allowed only to the users, that have access to the editing and that is set by PDM (Product Data Management) system. This allows better tracking and control and easier understanding of the product for those who are less skilled in reading 2D documentation.

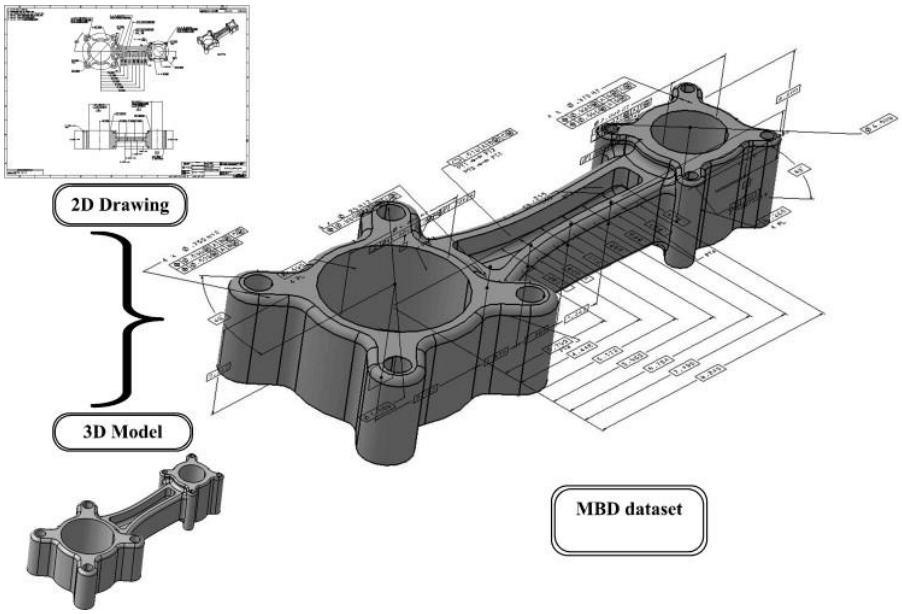

Figure 1: MBD as a combination between a 2D technical drawing and a 3D model[4]

MBD is standardized by several standards. It has been standardized by both ISO and ASME and the United States Department of Defense. The American Society of Mechanical Engineers (ASME) standardized MBD for the first time in ASME Y14.41-2003 in 2003. In 2012, the standard was improved and re-published as ASME Y14.41-20012. The International Organization for Standardization (ISO) standardized MBD with the ISO-16792: 2006 standard.

Advantages of using MBD

The method is appreciated in engineering environments where people deal with complex products that are incorporated into even more complex structures, while facing a wide variety of product and manufacturing information. Typical examples of this area are the aviation and automotive industries, as they need more than just a description of geometries (e.g. geometric and dimensional tolerances) and surface treatment and other properties. Combined with the development and design process, excluding 2D documentation saves a lot of time. On the other

[3] Quintana, V.; Rivest, L.; Pellerin, R.; Venne, F.; Kheddouci, F.; *Will Model-Based Definition replace engineering drawings throughout the product lifecycle? A global perspective from aerospace industry*, Computers in Industry, Volume 61, Issue 5, p. 497-508, 2010

[4] Quintana, V.; Rivest, L.; Pellerin, R.; Venne, F.; Kheddouci, F.; *Will Model-Based Definition replace engineering drawings throughout the product lifecycle? A global perspective from aerospace industry*, Computers in Industry, Volume 61, Issue 5, p. 497-508, 2010

hand, the MBD approach is not practical when developing simple products, e.g. reinforcing ribs for structures, supports, etc. Such products are usually simple because no high precision is required, and geometric tolerances and surface quality are not specified, and therefore there are no time savings. Most development time is saved by the fact that engineers are no longer required to choose between different sections and views, as was the case on 2D technical drawings, because all the necessary information is given on the model. This does not exclude choosing views onto which they would like to place their dimensions; it only means that their choice is not limited by placing these views on a limited paper format. In practical terms, dividing the views is similar to drawing each view on a separate drawing and thus not limiting the engineer by space. By eliminating the need to generate 2D drawings, the costs and time of development are reduced[5]. Savings are also reflected in immediate information transfer. Because anyone in need of product information has access to one single model, this means that they can view the current status of the project without delay. This reduces the information transfer time between the stages from development to sale.

Another advantage is that with appropriate approaches to MBM (Model-Based Manufacturing), CAD-CAM transformation can automatically create CAM manufacturing strategy, which has so far been done by production engineers. With an appropriate database of cutting tools and their properties, and with the data on processing, provided on the 3D model, appropriate decision algorithms can be used to automatically generate the CNC code, where all the necessary processing characteristics, including the choice of tools, are specified[6].

Since tolerances can be determined during the development of a product in a 3D model, discrepancies can be detected more quickly, as opposed to converting to 2D views, where the sense of functionality of the model is lost. Often it also happens that a 2D technical drawing lacks some information. For example, if in the case of a blind hole with a certain depth you forget to specify this dimension on a 2D drawing, a through hole will be drilled.

In assemblies, errors occur in the preparation of 2D documentation for larger systems, also due to incorrect recording of the number of elements on 2D technical drawings and the fact that component drawings are not updated together with changes to the CAD model. Since additional communication is needed between production engineers and designers, a lot of time is wasted. According to the Scania company, using MBD saved them approximately 18% of the time precisely due to the reduction of the extra communication, caused by errors[7].

In the manufacturing of products, constant control over the quality of production is of vital importance for competitiveness reasons. Much of this is done by sensor-equipped instruments, capable of measuring dimensions by multiaxial movement, which cannot be measured with conventional instruments; such instruments are also very accurate. Being guided similarly to CNC machines, the path of the sensor is determined according to the product. In a traditional way, the instrument would measure the product and display the readings, while the engineer would verify

[5] Quintana, V.; Rivest, L.; Pellerin, R.; Venne, F.; Kheddouci, F.; *Will Model-Based Definition replace engineering drawings throughout the product lifecycle? A global perspective from aerospace industry*, Computers in Industry, Volume 61, Issue 5, p. 497-508, 2010

[6] Wardhani, R.; Xu, X.; *Model-based manufacturing based on STEP AP242*, 12th IEEE/ASME International Conference on Mechatronic and Embedded Systems and Applications (MESA), Auckland, New Zealand, p.p. 1-5, 2016

[7] Lundqvist, T.; Phillips, F.; *Model Based Definition: The Main Effects of Implementing Model Based Definition in an Automotive Manufacturing Industry*, master thesis, Stockholm, Sweden, 2016

that the dimensions meet the required regulations. The MBD method has already provided the instrument with the requirements that the product should meet, so you can immediately know whether or not the machine is working within the appropriate tolerance limits[8].

The process from development to sale involves people who are not skilled in reading and understanding technical documentation, however, they are important for transferring the product from the idea to the market. These are people from the sales, purchasing and other financial areas. The ability to observe a product in a 3D view is very suitable for them, since it can give them a proper idea of it. The user can easily understand geometry as well as geometric and dimensional tolerances by simply manipulating the model (translation, rotation, zooming, etc.)[9].

Archiving data

It is also important for the proper management of product data to keep all product data and to be able to access them freely throughout the life cycle and beyond. On the other hand, CAD tools are constantly changing and being upgraded, so it often happens that after a few years, the tools that can read old documents properly are no longer available. The purpose of standardization is also to ensure the continuity and sustainability of a record for archiving purposes. Converting 3D models with all the PMI data into the neutral STEP AP242 format will result in a computer format that will be readable in the future and will allow independence from the user environment in which the model was made. By doing so, companies can make sure that their current knowledge will not become useless even in the case of a new software provider. They will be able to open everything they have available today in the future if they only keep the STEP file.

Drawbacks and challenges of MBD

The key disadvantage of using MBD is that it is first necessary to teach the engineers how to properly transfer information onto a 3D model. The main thing is to learn how to master the software and get used to a new approach of communicating information. It is also important that engineers learn the transparency of communicating information and selecting appropriate views.

A suitable method should be provided that will allow proper communication and notification of possible changes on the model, so that everyone who has access to the model will be informed of any changes. This is crucial in order not to make any changes that would lead to the purchase of wrong elements in the assembly, because people in charge of purchasing would not know that changes were made. This is also important in order to avoid mismanagement, as production engineers would not know that changes in surface quality and other things were introduced on the 3D model. Informing the production is important, as it is necessary to create appropriate CAD-CAM transformation once a change in the model has been approved.

[8] Fang, F. Z.; Li, Z.; Arokiam, A.; Gorman, T.; *Closed Loop PMI Driven Dimensional Quality Lifecycle Management Approach for Smart Manufacturing System*, 9th International Conference on Digital Enterprise Technology – DET 2016 – »Intelligent Manufacturing in the Knowledge Economy Era«, Nanjing, China, p.p. 614-619, 2016

[9] Carvajal, A.; *Quantitative comparison between the use of 3D vs 2D visualization tools to present building design proposals to non-spatial skilled end users*, Ninth International Conference on Information Visualisation (IV'05), London, UK, p.p. 291-296, 2005

STEP AP242

Neutral CAD formats have been developed in order to provide a single language that can be read by different CAD systems (CAD packages) and other CAx systems (Figure 2). Because companies use different CAD systems, it is a neutral form of some sort that allows the exchange of data, which is important both for data exchange within companies as well as between different companies. Neutral formats are necessary because, for example, a company develops a CAD model, another company carries out analyses, and a third party produces end products.

The neutral format should include a description of all product information over the life cycle, it should be independent of hardware and software, it should allow upgrading and updating, it should be stable over a long period of time, and in addition to reliable file sharing, we need shared access to a common database[10].

The basic conversion process involves geometric information (shape and dimensions), while newer versions also allow the storage of non-graphic data, information on manufacturing, tolerances and assemblies in the structure of a STEP file. These are also basic requirements that should be fulfilled if the neutral format is to be used for the MBD method for communication between different CAD packages.

STEP or ISO 10303 is an international standard that prescribes a neutral file transfer format. It was developed by the International Organization for Standardization. Different areas use different versions of the standard, distinguished by the number that comes after STEP. At the beginning, only a number was used behind the acronym, and later the AP symbol, standing for Application Protocol, was added. Each application protocol is intended for a particular type of industry and is adapted to its requirements. STEP thus covers different areas, but basically it is divided into three major parts – development, manufacturing and support of a product life cycle.

STEP is used in industry for sharing 3D CAD geometric models with organizations that use different CAD software with various primary file formats. More importantly, it is used to transfer files to the control, manufacturing and assembly departments in the companies themselves[11].

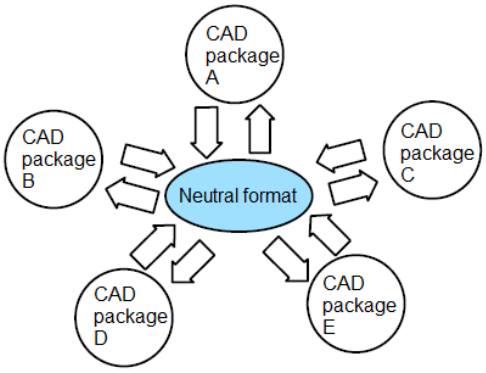

Figure 0: A neutral format is the common point between different CAD packages[12]

[10] Duhovnik, J.; Tavčar, J.; Elektronsko poslovanje in tehnični informacijski sistemi: PDMS-Products data management systems, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, 2000

[11] Fischer, B. R.; *A step up: a neutral file format brings more information into play*, Mechanical Engineering-CIME, Vol. 137, Issue 3 p.p. 42-45, 2015

[12] Duhovnik, J.; Tavčar, J.; Elektronsko poslovanje in tehnični informacijski sistemi: PDMS-Products data management systems, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, 2000

STEP AP242 is a standard for the exchange of product data, introduced by the International Organization for Standardization in 2014 in ISO10303-242. It is referred to as Managed Model-Based 3D Engineering. ISO10303-242 was created as a product of joint development of two previous standards. These are standards for the automotive industry (STEP AP214) and the aviation industry (STEP AP203)[13]. STEP AP214 is managed by ProSTEP iViP and SASIG, while PDES is responsible for AP203. Both standards are used in a wide range of global industries and have similar basic structures, however, there are also differences. In order to reduce the costs of parallel development and maintenance, joint development was proposed. The new STEP AP242 standard also offers the possibility of unified increase in STEP support from stronger industries. The basic purpose of the standard was to combine the functionalities of AP214 and AP203 into a model that would define a structured general-purpose approach, develop a semantic data model that would represent product and manufacturing information and other external references to elements, assembled parts and kinematics in assemblies[14]. Development also focused on the ASME Y14.41 and ISO16792 standards, which describe MBD. In order to make it possible, it is important to keep data on the 3D model even after conversion into a neutral format. STEP AP242 allows preserving semantic data on dimensions and dimensional tolerances, geometric tolerances and other data on surface treatment. This makes STEP AP242 a step towards eliminating 2D formats from the industrial environment, and constant use of data, defined on 3D models, as it allows information to be provided along the product development chain by means of a uniform binding to a 3D model.

The STEP AP242 standard focuses on the presentation of data to support CAD/CAE (Computer Aided Engineering) and PDM in data exchange, system integration, visualization and long-term storage of data and information.

The main areas covered by AP242:

- MBD (Model Based Development),
- integration of PDM,
- LTA (Long Term Archiving),
- integration of the supply chain,
- exchange of development data with included composite elements,
- advanced product and manufacturing information (PMI).

AP242 records product and manufacturing information in two ways. The first one is graphical, which is also referred to as presentation. Since the standard allows storing graphic information, the information given on the model is displayed graphically as lines and points. This was already possible in the STEP AP214 standard. The graphic part is intended for visual presentation and is not the key to subsequent processing, however, it allows the modeler to graphically display the processing information on the screen.

The second one, referred to as semantic mode, is more important. However, it is usually not visible to the user but the software can read it. It is called representation. This is a record in a file that determines the geometric and dimensional tolerances and surface treatment. It is in fact

[13] STEP AP242 Project: http://www.ap242.org, last accessed: 19. June, 2019

[14] Fischer, B. R.; *A step up: a neutral file format brings more information into play*, Mechanical Engineering-CIME, Vol. 137, Issue 3 p.p. 42-45, 2015

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bound to the elements of the solid model. On solid models that we open in modelers, this information does not appear.

METHODOLOGY

The computer programs were compared from the viewpoint of MBD support and their ability to export models into STEP AP242. They were selected on the basis of the availability of software package licenses and their usability in industry. The selected programs include Siemens NX 11, SolidWorks 2018, Autodesk Inventor 2018 and PTC Creo 5.0.

In order to be able to compare different modelers with respect to their ability to perform MBD, it was necessary to first create a reference model. For the purpose of describing the shape, the reference model was required to allow capturing as many dimensional and geometric data and tolerances as possible. The model used is shown in Figure 3.

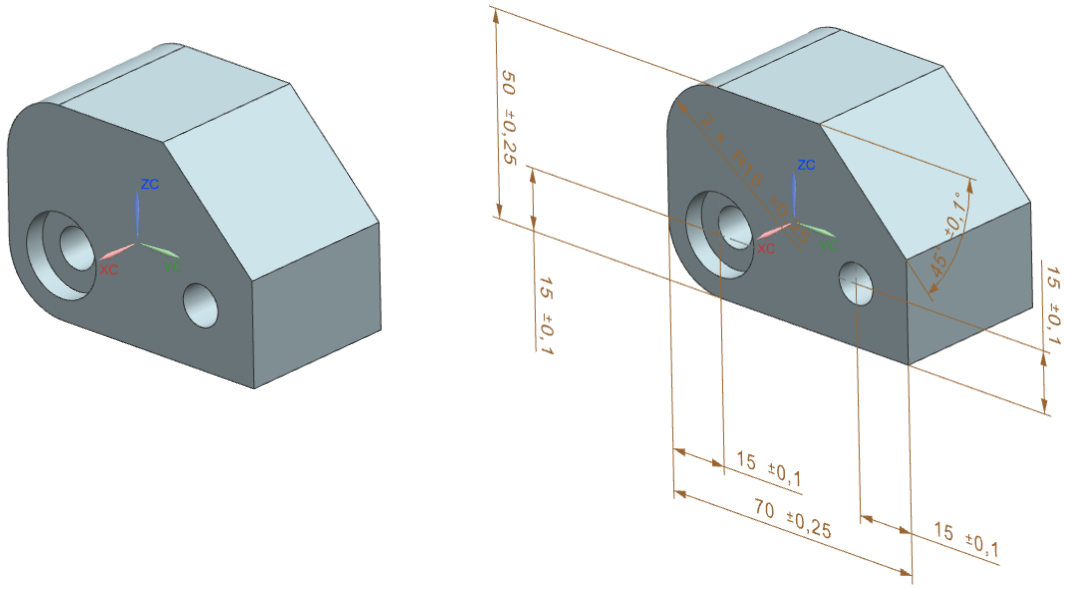

Figure 0: Reference model and model with added PMI (dimensions only)

RESULTS

Table 1 shows whether or not we get a solid model when importing STEP files. The far left column shows from which modeler the file was exported, and the adjacent one shows which extension the exported file received. Because modelers allow for different file exports, the number of options varies from modeler to modeler. Beginning with the second column, you can see for each modeler whether a solid model appeared upon opening, whether the file could not be opened, or the file could be opened but there was no solid model.

Table 1: Opening solid models

		Import modeling software			
Export modeling software	Exported file extension	Siemens NX 11	SolidWorks 2018	Autodesk Inventor Professional 2018	PTC Creo 5.0
Siemens NX 11	.stp	+	+	+	+

	.stpz	+	-	+	-
	.stpx	o	-	-	-
	.stpxz	+	-	-	-
SolidWorks 2018	.stp	+	+	+	+
Autodesk Inventor Professional 2018	.stp	+	+	+	+
	.stpz	+	-	+	-
PTC Creo 5.0	.stp	+	+	+	+

Legend: + opens a solid model, o opens a file but no solid model, - does not open a file

Table 2 shows PMI presentation on the model. The table is structured in the same way as Table 1. PMI is shown only graphically and has no actual information connection with the model. An example is presented on Figure 4.

The .stp file overview shows whether or not the file contains product and manufacturing information. In addition, you could already see in the first part of the file (file information) whether the STEP AP242 standard was used for transformation. The results are shown in Table 3, where the modelers from which the STEP file was imported are presented on the far left, while the other two columns show separately whether the information was included and whether STEP AP242 was used.

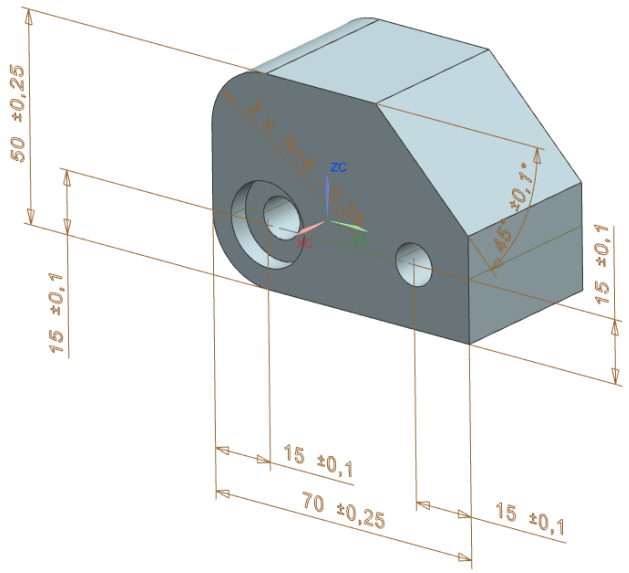

Figure 4: Part exported from Siemens NX 11 and imported back in Siemens NX 11

Table 2: Presenting PMI on a model

		Opening modeling software			
Export modeling software	Exported file extension	Siemens NX 11	SolidWorks 2018	Autodesk Inventor Professional 2018	PTC Creo 5.0
Siemens NX 11	.stp	+	-	-	+
	.stpz	-	o	-	o
	.stpx	o	o	o	o
	.stpxz	-	o	o	o
SolidWorks 2018	.stp	-	-	-	-

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Autodesk Inventor 2018	.stp	-	-	-	+
	.stpz	-	o	-	o
PTC Creo 5.0	.stp	+	-	-	+

Legend: + PMI displayed, o no solid model, - no PMI displayed

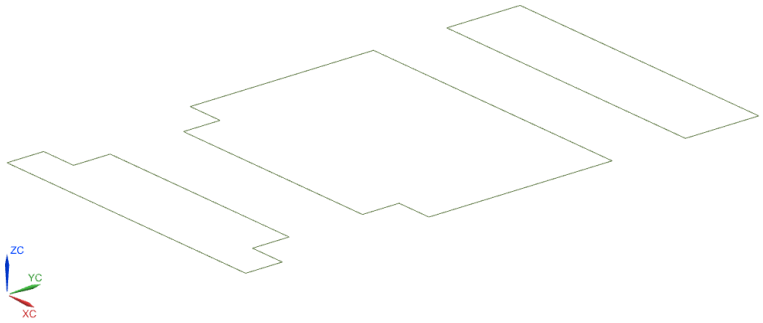

Figure 5: File in .stpx format opened in Siemens NX 11

Table 2: Results of .stp files validation

Modeling software	Information preservation	Use of AP242
Siemens NX 11	+	+
SolidWorks 2018	+	+
Autodesk Inventor Professional 2018	+	+
PTC Creo 5.0	+	+

Legend: + information included, - information not included

The size of the converted models on the disk is shown in Table 4. The left column shows the modeler from which the STEP file was exported, the adjacent column contains the file extension of the exported file, and the far right column shows file size on disk.

Table 0: File size on disk

Modeling software	Exported file extension	File size on disk [kB]
Siemens NX 11	.stp	1161
	.stpz	270
	.stpx	7
	.stpxz	2
SolidWorks 2018	.stp	1758
Autodesk Inventor Professional 2018	.stp	478
	.stpz	141
PTC Creo 5.0	.stp	1587

CONCLUSION

Table 1 reveals that modelers have no problem opening ordinary .stp files. Other than the basic versions of STEP files cause problems. The biggest problem is that some modelers are unable to import specific file formats.

Considering that two modelers (Siemens NX 11 and Autodesk Inventor Professional 2018) out of the four compared have the option of exporting and opening a compressed .stpz file, this would be a desirable option also in other modelers. Especially due to the fact that in the case of the exported file from Siemens NX 11, the compressed file takes up only 25% of space, compared to the ordinary non-compressed STEP file.

.stpx and .stpxz files are XML types of STEP files. The difference between them is that the latter is a compressed version of the former. These files are capable of being both exported and opened only in the Siemens NX 11 modeler, where it is interesting that a solid model is obtained when opening a compressed file type, but when opening a non-compressed file, displayed are only the lines of the sections where the model was cut when views were created.

Looking at the results, presented in Table 2, you can see that the display of PMI on solid models is not so common. Similarly, when displaying this information, some confusion may occur, as all the data are displayed on one model, which makes them unclear. Displaying different graphic elements is one of the possibilities, provided by STEP AP242.

In principle, the display of PMI is not crucial for the manufacturing process to continue because CAM programs do not require graphical information, they only need the representative part. The graphical display is important for the production engineers to be able to see what the requirements on the model are, but for this purpose, it is easier to use some other dedicated file. One of the options includes a 3D .pdf file that can be easily opened in Adobe Acrobat Reader. It can be exported from all of the compared modelers, except Siemens NX 11, which uses JT files for this purpose. For viewing these files, Siemens also provides free software for computers, as well as for smartphones and tablets running Android or iOS.

Reviewing .stp files in the text format and the results obtained from Table 3 show that conversions from solid models into STEP files are appropriate. This is evident from the fact that the information, required by the software that is used in the phases that follow development, is preserved. Even if the information is invisible to the eye, a file still contains it. This fact makes it the key information transmitter between different software environments.

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REFERENCES

1. **Quintana, V.; Rivest, L.; Pellerin, R.; Venne, F.; Kheddouci, F.;** *Will Model-Based Definition replace engineering drawings throughout the product lifecycle? A global perspective from aerospace industry*, Computers in Industry, Volume 61, Issue 5, p. 497-508, 2010;
2. **Wardhani, R.; Xu, X.;** *Model-based manufacturing based on STEP AP242*, 12th IEEE/ASME International Conference on Mechatronic and Embedded Systems and Applications (MESA), Auckland, New Zealand, p.p. 1-5, 2016;
3. **Lundqvist, T.; Phillips, F.;** *Model Based Definition: The Main Effects of Implementing Model Based Definition in an Automotive Manufacturing Industry*, master thesis, Stockholm, Sweden, 2016;
4. **Fang, F. Z.; Li, Z.; Arokiam, A.; Gorman, T.;** *Closed Loop PMI Driven Dimensional Quality Lifecycle Management Approach for Smart Manufacturing System*, 9th International Conference on Digital Enterprise Technology – DET 2016 – »Intelligent Manufacturing in the Knowledge Economy Era«, Nanjing, China, p.p. 614-619, 2016;
5. **Carvajal, A.;** *Quantitative comparison between the use of 3D vs 2D visualization tools to present building design proposals to non-spatial skilled end users*, Ninth International Conference on Information Visualisation (IV'05), London, UK, p.p. 291-296, 2005.
6. **Duhovnik, J.; Tavčar, J.;** Elektronsko poslovanje in tehnični informacijski sistemi: PDMS-Products data management systems, University of Ljubljana, Faculty of Mechanical Engineering, Ljubljana, 2000;
7. **Fischer, B. R.;** *A step up: a neutral file format brings more information into play*, Mechanical Engineering-CIME, Vol. 137, Issue 3 p.p. 42-45, 2015;
8. STEP AP242 Project: http://www.ap242.org, last accessed: 19. June, 2019;