A COMPARATIVE STUDY OF CAD DATA EXCHANGE BASED ON THE STEP STANDARD

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This paper presents an example of applicability of the STEP standard in a design process of a new specific sub-assembly product in the heavy machinery industry in Serbia. We tested the interoperability of the STEP standard in several CAD systems and their compatibility and quality of conversion process. Results obtained in performed experiments demonstrate that the STEP standard offers simplified data exchange between heterogeneous CAD systems and serves not only as a tool for integration and exchange the data of products and processes, but subsequently supports the integration of companies by increasing the global competitiveness, more efficient archiving, evolving and reusing the data in a solving complex entrepreneurial and production problems worldwide.

Keywords: STEP, CAD data exchange, Interoperability, CAD tools.

1. Introduction

Modern production, based on a strict demand of the global market, requires time reduction in all phases of the product realization from idea to disposal, influencing the entire product lifecycle and indirectly introduces evolution in all aspects of the product management and, consequently, to decreasing cost of production and increasing the quality. With new information and communication technologies ICT (ICT) technologies this process has been accelerated towards new industrial paradigms, like industrial revolution 4.0 [1]. Thus, multinational production companies require new approaches and PLM solutions to integrate all information and data generated from each phase, to adopt and allow interoperability among different and several software solutions developed for different sectors and based on a different exchange standard.

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Most common way of translating and transferring the data about product from one to another CAD system is based on neutral formats, where the original system exports data in this format, while the importing system reads it. Some of the neutral formats are defined by international organizations (such as STEP, STL, IGES), while others have become widely accepted, for common use (such as DWG and DXF). The idea of universal standard for sharing and exchange the CAD content exists for more than 40 years, but even today there are several directions of its development, to make exchange more flexible, manageable and adoptable for different software solution and users. One of these is STEP standard [2], based on ISO 10303 [3]. STEP data format has been widely discussed and applied to export a CAD data, typically to be applied in different CAx systems and PLM stages (design, production process, product usage, maintenance, etc.), but it is not taken as a major lightweight model representation.

In this paper we performed a test of the STEP standard utilization during the development of a new product in heavy machinery industry in Serbia which uses different software solution and receives CAD content from dislocated collaborator. Through the case study we demonstrated several problems that can arise during the conversion of CAD files to STEP format, by analyzing the quality of the exchanged models and size of the generated files. Some of the noticed errors after observations are: problems of accuracy during conversion, problems with quality of the model, misalignment of solids, missing features, etc.

2. STEP solutions for CAD/CAM data exchange

STEP - „STandard for the Exchange of Product model data" is an ISO standard (ISO 10303) for exchange the data of products and processes with purpose to provide a support to automation of the design process in industrial environment [2]. The basic methodology of STEP is in modular structure and open architecture that enable high flexibility and possibility of extension and improvement of existing protocols, if needed.

STEP format covers a wide range of different types of product (electronic and electro-mechanic products, ship construction, production plants layouts, etc.), but also entire product lifecycle from product design and production to its usage and disposal. As the STEP standard is being improved, each of the covered sub-standards is named with a specific name in format ISO 10303-XXX, where the XXX stands for the code of each sub-standard [3], [5].

In a modern PLM environment, data exchange between CAD and CAM systems must also provide reliable information about product manufacturing, besides the information of product 3D model. Specific application protocols were developed with this dedicated purpose (e.g. STEP-NC AP 238 – CAD, CAM and NC data of the machining process) [4].

In smaller SMEs (small and medium enterprises) where the employees use different and cheaper software solutions for different stages and there is no uniform and reliable platform for collaborative product management like 3DExperience or SmarTeam, it is crucial to maintain the quality and accuracy of the exported data and virtual content of the product from previous to the next stage. Here we are focused on a CAD content of parts and assemblies.

2.1 Conversion to STEP - accuracy and advantages comparing to translators

The exchange process from one CAD system to another starts by exporting files in neutral format (conversion) and ends by its importing (extraction and transferring) to another CAD software. This can be done directly between the two software solutions, or by using independently developed ones, third party translators, which exist for the most common CAD systems in use today. Some of the independent software are: Cimfotek; Compunix (for CATIA, Unigraphics and Pro/Engineer); PTC (for CATIA, PDGS, CADAM); Theorem Solutions (for CADDS, CATIA, SolidEdge, SolidWorks, Unigraphics, Mechanical Desktop, Pro/Engineer); Spatial (for CATIA, Pro/Engineer, Parasolid, ACIS). Besides these locally run solutions, there are also numerous online services offering the same, basically intended to the SMEs.

As it has been already seen in the past, the use of specific translators for direct conversion between individual CAD systems is safe for simple parts, but also slow and practically unfeasible in complex geometry or resulting anomalies on the model, missing solids or surfaces features.

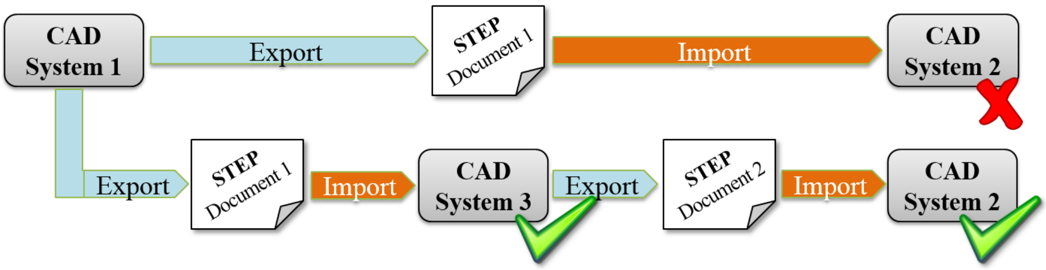

Fig. 1. Data exchange process between two CAD systems

For example, by exporting a CAD model to STEP and later importing the same model from STEP to CAD software, basic geometrical features from the originating format can be lost, which decreases the value of the archived content and unable further deployment of the model, consequently increasing the cost of development and production. In order to avoid or prevent this problem, one of the solutions is to use last application protocols, the 3rd CAD system as a compatible bridge, as presented in fig. 1, or to use CAD software from the same PLM brand.

Beside the accuracy of the exchanged model, it is important to offer a compact form of the converted CAD data for the fast and reliable transfer to dislocated units of the enterprise.

2.2 STEP compression – 3D Model export and import

Since the STEP is widely used in industry, there is a need to reduce the size of converted data and optimize the process of data exchange between different CAD systems.

From this need, compressed STEP is developed and now it is in use in all relevant application protocols (AP203, AP209, AP214, AP242, etc.), with the focus on accuracy of 3D geometry, manufacturing geometry and information, 3D polygonal geometry, etc. In the most cases, the STEP format will take a less space compared to other CAD formats because of the simplified geometry of part and assembly models.

The logic behind compression of STEP documents is defined by IFCZIP agreement [6]. In order to differentiate compressed from native STEP files, their extension is changed from *.stp to *.stp.Z, and from *.xml to *.xml.Z (Fig. 2).

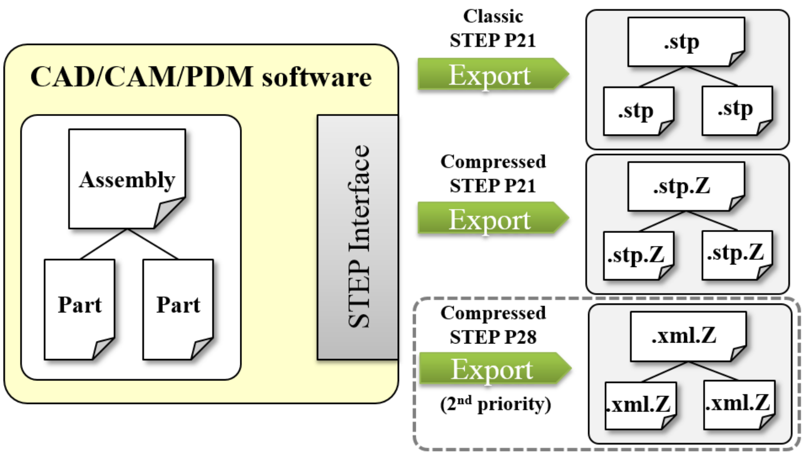

Fig. 2. Compression of the STEP files

Compression and decompression of STEP files is a process applicable on the whole structure of the STEP file, from individual parts to the parts of assembly (Fig. 3). Same parts of an assembly will be compressed once enabling reduction of the STEP file size.

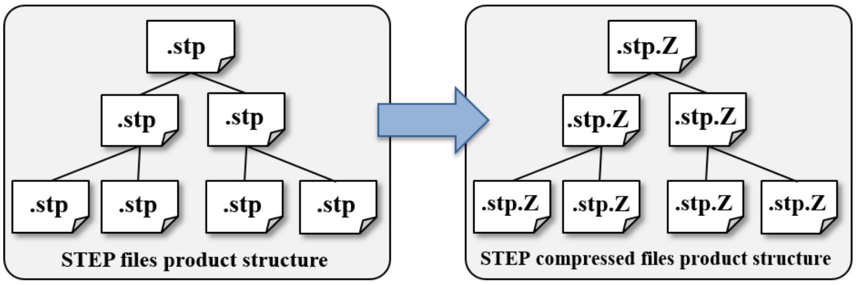

Fig. 3. Compression of the assembly

Export and import of the compressed STEP files are done by selected pre/postprocessor, which can automatically recognize if the file was previously compressed (usually by the file extension). During the process of export, the whole structure of the assembly is formed as it illustrates fig. 4.

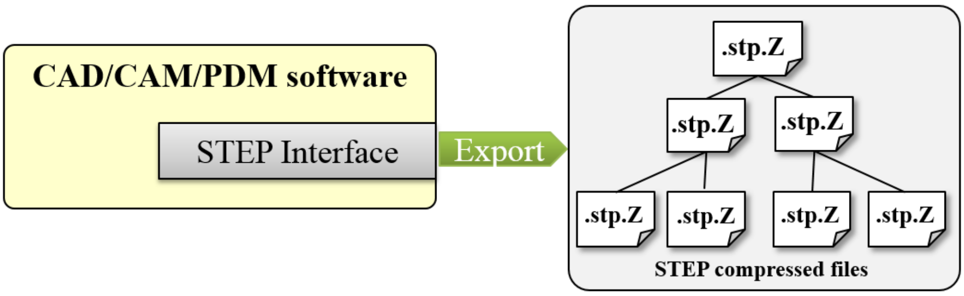

Fig. 4. Export of the compressed STEP file

During the importing process, it is necessary to select the tree of the whole assembly and the system will subsequently automatically connect, unpack and import all related files as it is shown in fig. 5.

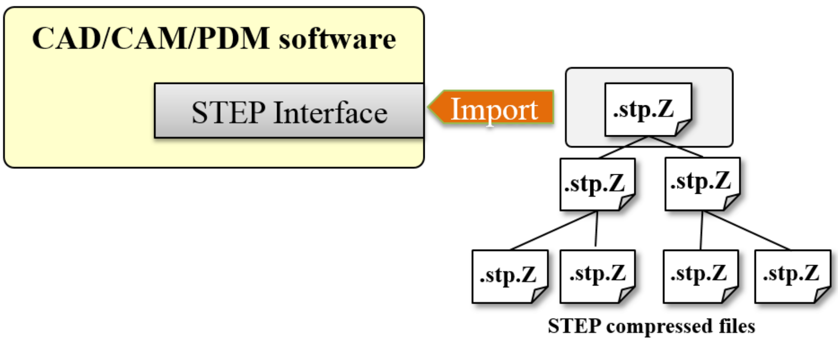

Fig. 5. Import of the compressed STEP file

The benefit of using compression on STEP files is reduction of space from 60% to 85%, comparing to uncompressed STEP files. This enables a faster transfer and visualization of the data in dislocated development centers or production facilities.

Users also should be aware that history of the design after converting original files in STEP is usually lost, as well as constraints implemented in assemblies, while some PLM solutions enable visibility of embedded knowledge, sketches, product tree and additional elements of the product virtual representation.

3. Assemble data integration among multiple CAD systems

In this experiment we considered the application of the STEP standard in a supplier Serbian company ("Wacker Neuson" from Kragujevac, Serbia) on a concrete example from heavy machinery industry a sub-assembly of excavator Wacker Neuson model EW65. The design team received documentation about the sub-assembly: 1) 2D technical drawings of the supporting assembly; 2) 2D technical drawings with basic dimensional requirements, as .dxf files, with the purpose of quality and dimensions inspection of the individual parts of the assembly; 3) other documentation (time schedule, production volumes, etc.).

(c)

Fig. 6. Excavator Wacker Neuson model EW65 - 3D model of part Loffestiel Lang E14-102: a) CATIA V5R20, b) Autodesk Inventor Professional 2015, c) ZWSoft ZW3D 2015 SP

Based on delivered documents, team had the task to make a 3D design of the sub-assembly of excavator, to define the production process, and the cost of production, and to disseminate data about design in STEP format to other teams.

In order to fulfil all requirements, design team performed the following stages and experiment:

I) 3D modeling of the parts of the assembly (Fig. 6) in three independent and available CAD software (CATIA V5 R20, Autodesk Inventor 2015, ZWSoft ZW3D 2015 SP);

II) Modeling the assembly from individual parts (Bottom-Up) and conversion to STEP format, with consideration of the export options available in selected software (selection of application protocol, accuracy, space reduction on the disk/storage, etc.);

III) Opening the STEP files from all three CAD software in two neutral programs for reviewing STEP files - STP Viewer 2.3.0 and AB Viewer 11 and inspection a model accuracy and completeness;

IV) Importing of the STEP files from one to other CAD software, with the check of the conversion results, the level of preserved information and the possibility of errors during conversion.

3.1 Modeling the assembly in CATIA V5 R20

Firstly, we modelled the assembly model in CATIA [7, 8, 9, 10]. Total size of the assembly model is 3.25MB. It consists of individual files of each component (CATPart) and one assembly file (CATProduct). Exporting options of the model from CATIA's data format to .stp format are: tolerances (0.001mm); application protocols (AP203, AP203 ed2, AP214); units of measure (millimeters), export of invisible and hidden entities (disabled), defining the heading of the file (author, organization, additional description), selecting the option of exporting the assembly (as a single file and as individual files forming the assembly).

We selected export application protocol AP203 ed2 and the assembly as individual file was 822KB in size, making a space reduction of 74.6%. When exported as a single file (assembly with all parts in one file), exported model was 784 KB, achieving reduction of 76.4% compared to the native CATIA files.

3.2 Modeling the assembly in Autodesk Inventor Professional 2015

Modeling performed in Autodesk Inventor Professional 2015 resulted in an assembly model with size of 3.82 MB, consisting of one main assembly file, one project file and a set of individual part files. When exporting from native Inventor format to .stp format, application protocol AP203 was selected, and the resulting .stp file was with size of 690 KB, achieving a reduction of 82.3%, compared to the original Inventor files.

3.3 Modeling the assembly in ZWSoft ZW3D 2015 SP

Modeling of the same assembly was also done in CAD software ZWSoft ZW3D 2015 SP and its model with the files of individual parts has a total size of 3.41MB. Firstly, we exported the assembly as a set of individual files from the native ZW3D format in STEP format with application protocol AP203, with the resulting output of 704 KB, achieving a space reduction of 79.8% of disk space. When exporting the model as a single part (creating 1 part from the assembly), output file had 1.02 MB, achieving a reduction of 70.1% in disk space.

3.4 Inspection of STEP files in STEP viewers and compatibility crosschecking between used CAD systems

The files created by conversion in AP203 application protocol from three used software are checked regarding compatibility in two independent programs for viewing STEP-formatted documents (STP Viewer and AB Viewer) and crosschecked in all three CAD systems.

Table 1 shows an overview of file sizes (in bytes) that were created by exporting to STEP, original file sizes, and the achieved reduction by converting to STEP, as well as additional reduction achieved by ZIP compression of STEP documents.

Modern computers have large storage capabilities. So, reducing the files' size is not a matter of storing them on HDDs or optical disks. In fact, a smaller file is easier to be uploaded on cloud (if needed), opened by a CAD system or downloaded from a server.

Table 1

Overview on file sizes of native software format, STEP format, and achieved reduction by conversion to STEP and additional ZIP file compression

CAD system and STEP output type	Application protocol used	Native file size	.stp file size	Space reduction	WinZIP compressed file size	ZIP space reduction
CATIA V5R20 / single document	AP203 ed2	3.414.742	802.734	76.4%	137.432	95.9%
CATIA V5R20 / indiv. documents	AP203 ed2		866.328	74.6%	188.813	94.5%
Inventor 2015 / single document	AP203	4.002.816	705.642	82.3%	133.307	96.7%
ZW3D 2015 / single document	AP203	3.581.429	721.433	79.8%	192.991	94.6%
ZW3D 2015 / assembly as part	AP203		1.072.190	70.1%	132.570	96.3%

By visual examination we checked a presence of all model entities, absence of errors, missing features, missing surfaces, etc., as well as accuracy and quality of splines and complex surfaces and its representation in exported STEP files (Table.2).

Table 2

Results of STEP files inspection in STP Viewer and AB Viewer

CAD system and STEP output type	Application protocol used	STEP viewer used	STEP document opened normally?	Error free model?	Comments
CATIA V5R20 / single document	AP203 ed2	STP Viewer	YES	YES	Spline accuracy is low
		AB Viewer	YES	YES	Spline accuracy is low
CATIA V5R20 / indiv. documents	AP203 ed2	STP Viewer	YES	YES	Spline accuracy is low
		AB Viewer	YES	YES	Spline accuracy is low
Inventor 2015 / single document	AP203	STP Viewer	NO	-	Viewer shutdowns
		AB Viewer	NO	-	Model not opened
ZW3D 2015 / single document	AP203	STP Viewer	YES	YES	Spline accuracy is low
		AB Viewer	YES	YES	Spline accuracy is low
ZW3D 2015 / assembly as part	AP203	STP Viewer	YES	YES	Best quality achieved
		AB Viewer	YES	YES	Best quality achieved

After the testing of all combinations of exported STEP files provided by selected CAD systems, the STP Viewer and AB Viewer were able to open the exported models with success rate of 80%. For both exports from Inventor 2015, viewers were able to recognize the STEP files (by its extension *.stp), but were unable to open the model.

In all cases of successful model opening and visualization in STP and AB viewers, the representation was complete, without absence of model features or missing surfaces, but with low accuracy of splines and complex surfaces. The best result we found in the files exported by ZW3D. At the same time, this STEP document was also the largest in file size, comparing to the other.

In all cases, low accuracy of splines was present, which was especially visible in splines with larger radius. With the decrease of spline radius, the error visibility was also decreasing.

In order to achieve faster rendering of models, software producers often choose to decrease quality of the models “on screen”, which could explain this imperfection. Of course, this quality decrease is just for the user to see and to easily manipulate large/complex parts and assemblies, but those 3D models are kept and used in full quality by the CAD system.

The complexity of 3D projects increases every year (also of the CAD systems), but the producers often simplify the representation on screen for many features (e.g. circles are represented like polygons, 3D threads are usually replaced by a suggestive image, etc.).

Software compatibility cross-inspection has been done in the same way, by importing STEP files from one to other two software solutions and the results are presented in Table 3.

Table 3

Results of software compatibility cross-inspection between three CAD systems

CAD system and STEP output type	Application protocol used	Importing CAD system	STEP document opened normally?	Error free model?	Comments
CATIA V5R20 / single document	AP203 ed2	Inventor	YES	YES	
		ZW3D	YES	YES	
CATIA V5R20 / indiv. documents	AP203 ed2	Inventor	NO	-	Model not opened
		ZW3D	NO	-	Model not opened
Inventor 2015 / single document	AP203	CATIA	YES	YES	
		ZW3D	YES	YES	
ZW3D 2015 / single document	AP203	CATIA	YES	YES	
		Inventor	YES	YES	
ZW3D 2015 / assembly as part	AP203	CATIA	NO	-	Model not opened
		Inventor	YES	YES	

After cross-inspection of all combinations between used CAD systems in which the model was created and applied protocols, it is easy to notice that the 100% compatibility between the exported models is lacking.

4. Discussion

Conversion of the experimental model from CATIA V5 R20 was done by the newer AP203 ed2 application protocol, which enables the assembly to be exported identically as it is in the original format – as a single file representing the assembly and individual files for each of its parts but with missing constraints and model history. This was a problem for other CAD systems, not able to open and display the STEP file converted by a newer application protocol using novel capabilities.

From another perspective, the CATIA software was unable to open the model converted from ZW3D software, which prepared the assembly as a single part, also not offering the option to export the assembly as a single part.

Compared to STEP viewers, all three CAD systems achieved a much higher model visualization accuracy on screen, especially of splines and complex surfaces, leading to a conclusion that they are optimized for speed, achieved by decrease of quality.

When exporting the model from a CAD system to the STEP format, it is important to notice that the models' history of the parts and assembly constraints will be lost. Being so, the only way of recovering the parts' dimensions is by measuring them again. The data relating to geometrical constraints of the assembly is also lost, but the defined position of parts in the assembly is initially preserved, when the STEP file is imported to the selected CAD system. The structure of the models was preserved in all the tests where the import was possible, independently of the choice if the assembly was exported as an assembly, or as an assembly converted to a single part, where the parts of the assembly were presented as features of that part.

The different level of adjustability of export to STEP in all three CAD systems should also be taken into consideration, meaning that the application protocols were not implemented in the same way.

5. Conclusions

Data of the developed products presents an important asset of each company and being so the possibility of opening, editing, storing, redistribution and later re-opening the data is important element of the modern business. For this reason, the design and functionality modifications that find its way towards STEP standard should not jeopardize the technical integrity of data and the compatibility with earlier versions. Taking into consideration that STEP development is an ongoing process (also of the CAD/PLM systems), it is normal to expect more improvements in the process of ISO (STEP) standardization in the near future.

As an open standard, STEP has a very good support for further development for software with low hardware requirements, enabling everyone to work on a further development of application protocols. Nowadays, the global trend is to use mobile gadgets (tablets, mobile phones, etc.) for a fast 3D preview and visualization of CAD models on distant places, out of design offices, from conceptualization phase to the advertisement of new products. Thus, it is of the ultimate importance to have optimized, compatible, reduced and interchangeable 3D files available. The novelty of this study is cross-testing of a new STEP protocols in commercial software during large assembly design and investigation of the file size for the fast error-free loading and compact storage. To load satisfied and fast 3D visualization on an external medium, especially models of large assemblies and belonging parts and subassemblies it was particularly important to keep the initial structure of the STEP format and to optimize the values of representative geometry (coordinates, lengths, etc.) that do not affect quality.

In this paper we illustrate possible potential of the STEP standard in digitalization, exchange and conversion of CAD models and its compactness in distributed visualization which are important requirements in implementation of the concept of Industry 4.0. From some points of view, the CAx processes in the acclaimed Industry 4.0 cannot be seen without a strong standardization of files (parts, assemblies, simulations results, etc.).

When we think about the future of data exchange, it is important to mention that the fourth industrial revolution and creation of a cyber-physical production system are already ongoing. STEP standard will enable the implementation of cyber-physical systems concept that provides the objects to be identified, localized and accessible on a global level. Even though, the technology of STEP brings significant reduction of cost, increases the product quality and reduces the time of development and production to the companies using it. STEP facilitates the cooperation of different manufacturers, company suppliers, corporate partners and distributed systems using different PLM solutions. It enables a major reduction of the storage space and presents a high stake for the future to the companies that use it and invest in its development, because it is based on the principle to always remain backward compatible, and for this reason has a significant support in the CAD systems used today.

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R E F E R E N C E S

[1]. *R. Anderl,* "Industrie 4.0 – Advanced Engineering of Smart Products and Smart Production", Proceedings 19th International Seminar on High Technology Technological Innovations in the Product Development, Piracicaba, Brasil, October 9th, 1-14, 2014.

[2]. *S. J. Kemmerer,* "STEP - The Grand Experience, Manufacturing Engineering Laboratory", National Institute of Standards and Technology - NIST, Gaithersburg, 1999.

[3]. STEP Application Handbook ISO 10303 v3, SCRA, International Boulevard, SC, (2006). Link: https://pdesinc.org/downloadable_files/STEPapplicationhandbook63006BF.pdf.

[4]. *X. Xu, A.Y.C.N. Nee,* "Advanced Design and Manufacturing Based on STEP", University of Auckland, pp. 1-481, Springer-Verlag London, 2009.

[5]. *M. J. Pratt,* "Introduction to ISO 10303 – The STEP Standard for Product Data Exchange", J. Comp. Inform. Sci. Eng. 1, pp.102-103, 2001.

[6]. J. Boy, P. Roche, F. Darre, "Recommended practices for STEP file compression", CAx Implementor Forum, 2016.

[7]. *S. Ćuković, G. Devedžić, F. Pankratz, I. Ghionea, S. Karupppasamy,* "Praktikum za CAD/CAM – Augmented Reality –", University of Kragujevac, Faculty of Engineering, CIRPIS, Kragujevac, ISBN 978-86-6335-020-5, 2015.

[8]. *S. Ćuković, G. Devedžić, I. Ghionea,* "Automatic Determination of Grinding Tool Profile for Helical Surfaces Machining Using Catia/Vb Interface", U.P.B. Scientific Bulletin, Series D, Vol.72, No.2, pp. 85-96, ISSN 1454-2358, 2010.

[9]. *Radu Constantin Parpală*, "Virtual Design of a Machine Tool Feed Drive System", U.P.B. Sci. Bull., Series D, Vol. 71, No. 4, ISSN 1454-2358, 2009.

[10]. *Diana Popescu, Robert Iacob, Radu Parpala, Tiberiu Dobrescu*, "Virtual to Real in Robotic Assembly/Disassembly Tasks", U.P.B. Sci. Bull., Series D, Vol. 78, No. 2, ISSN 1454-2358, 2016.