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Data Exchange with Support for the Neutral Processing of Formats in Computer-Aided Design/Computer-Aided Manufacturing Systems

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Abstract: This paper presents an analysis and the research results of system couplers in CAx systems that ensure the correct transfer of product data from the "supplier" system to the "recipient" system. This study presents the results of the compatibility tests between the systems based on the verification of geometric models and their attributes after importing/exporting 2D and 3D objects using neutral data exchange formats. The purpose of the research was to identify neutral formats that do not introduce errors after data transversion in certain types of geometry. Tests and analyses were carried out on selected formats depending on the type of object, such as 2D objects (DXF), solid objects (IGES, STEP, SAT, and PARASOLID), and surface objects (IGES, STEP, SAT, and X_T). One of the results of our research and analysis is the need for continuous development of CAD/CAE systems because current neutral formats are not 100% effective.

Keywords: CAD; data exchange; data format; CAx; DXF; IGES; STEP; SAT; X_T

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1. Introduction

The evolution of manufacturing systems follows the development of technology and their immediate environment in the form of machines, devices, methods, and tools supporting the work on the preparation of technical documentation [1–3]. This documentation describes the product model, processes, and means of production in use. The obvious fact is that computer tools help to meet the production challenges of a rapidly changing world economy, especially in the work of engineers [4,5]. Chen [6] discussed the use of various CAD systems to simplify the product design process. A solution based on commands was proposed to address the problems identified in the development of such a collaborative design system. The authors Chiu and Chlebus designed a representation scheme for multiple material objects in a CAD system [7,8].

In modern production engineering, the phase of design and technical preparation of production is of particular importance, the cost of which comprises 70% of the entire production project [8–10]. Therefore, the designer, regardless of whether they are a constructor, technologist, or planner, must have an efficient tool and technique that enables a comprehensive analysis of the adopted concepts, models, and variants of solutions [11–13].

Previous studies have dealt with the portrayal of the ISO STEP tolerance standard as an enabler of intelligent manufacturing systems [14]. Gargallo [15,16] investigated a method of smoothing and untangling surface meshes independent of CAD parameterization, where he proposed a method to optimize triangular and quadrilateral meshes on

parameterized surfaces. Kaščák et al. [17] revealed that data exchange is important in all manufacturing processes, including plastic injection molding, machining simulations, and other technologies [18–21]. Jauhar [22] addressed the heterogeneous environment of CAx systems that have problems in revising and reimporting neutral models, due to the missing required identification information. Jian [23] designed an improved NBA algorithm to optimize the BP neural network algorithm for semantic feature recognition. Xiaoxia et al. [24] revealed that feature-based data exchange (FBDE) between CAD systems is one of the most important issues in collaborative product development and presented a solution based on procedure recovery for FBDE. The result was the functioning of FBDE between different commercial CAD systems and improvements in previous approaches to architecture and procedure searches. Modern CAD systems are used to build models of product shapes based on functions with parameterization and constraints [25].

Integrated computer-aided design/computer-aided manufacturing/computer-aided engineering (CAD/CAM/CAE) were studied by Kirkwood [26,27] in his research, resulting in a set of practices for the effective adaptation of sustained CAD/CAE integration. In their studies, Varga [28] and Kirkwood [29] described the importance of data validation in the CAE process of systems in more detail. Krause [30] highlighted the significance of the rapid transfer of CAD data between different CAD systems to speed up and reduce the number of iterative steps in product development processes. Kreis' research [31] involved the use of a large volume of data in the development process, especially in the applied CAx environment, and an examination of the exchange of data between different areas of design, analysis, and production in the automotive field. The assessment of data formats for layered manufacturing was discussed by Kumar [32] and by Varga in terms of the machining process [33]. Lee [34,35] explained data processing for CAD/CAM/CAE systems. Lipman [36] studied PMI representation in CAD model STEP data exchange files.

Zhang [37] analyzed the corresponding relations of topological entities. He proposed a method for comparing topological entities in the integration of heterogeneous CAD systems. Lu [38] presented data modeling for the integration between specifications and verification for GPS-compliant cylindricity based on category theory. The importance of data transfer was similarly explored by Kumar and Martin [39,40] but in the area of additive manufacturing. In another study [41], the importance of data in assessing the accuracy of machined parts was highlighted. Several other studies involved investigations of data transfer [42–44].

Data exchange between different systems is of particular importance in the process of planning. In the production process, this can be in the field of machining or the case of the production of complex-shaped parts, such as the sintering of powders. In all these cases, data processing and the correct exchange of CAD data are important for an efficient production method. Most often, the team making "strategic" decisions to start production must have a universal data exchange format that ensures the least loss of data from the "giver", i.e., the customer, to the "recipient", i.e., the contractor. They must also have appropriate software that is fully compatible with other systems so that the exchange between them is possible. From this point of view, a full compliance analysis consisting of the verification of geometric models and their attributes after import/export is needed to understand all the issues related to data exchange in CAD/CAE systems [10,21,29,45]. The integration of methods, models, tools, and data is the basis for the comprehensive implementation of computer techniques in a modern manufacturing company.

The overriding goal is to increase awareness of the existence of errors in the transmission of data between CAx systems. Additionally, the existing research aims to highlight the current level of application of neutral formats and assess the compatibility of converted objects [46–48]. Errors at the design stage, which include the transfer and transfer of objects using CAx systems, can significantly increase the cost of implementing the production process [49,50]. This paper draws attention to the possibility of product inconsistency due to incorrect translation between the available programming languages. One of the systems that compares and determines the level of compliance is NIST software 2022.

Measuring compliance includes, among other aspects, geometrical dimensioning and tolerancing, surface specification, annotations regarding finishing, welded joints, and material specification [51].

Generating and optimizing section profile data for rapid prototyping and manufacturing was studied in [52], which presents a slicing algorithm based on a geometric continuity model. Poya [53] analyzed a unified approach to a posteriori generation of arbitrary high-order curvilinear meshes via the solid mechanics analogy, and Pratt [54] described a STEP standard for product data exchange. Different forms of data processing were studied by Adrian et al. [55], who examined the data in heat transfer processes in a heat pipe heat exchanger utilizing a working fluid R134A. Other authors [56–58] examined transfer data in additive processes and its effect on the evaluation of geometrical performance, while Ryou [59] was involved in the development of a data interface for rapid prototyping in STEP-NC. With the use of model-based definitions, most of the data required to design, analyze, and manufacture a product are stored in digital format [60–62]. An effective communication model between CAD, CAPP, and CAM applications in a distributed manufacturing planning environment is considered a key component of CIM [63]. Data handling in NURBS models based on the STEP format was studied by Starly [64]. Staves [65] and Venkiteswaran [66] investigated the associations between CAD models, as well as their geometric dimensions and tolerances.

Other analyses of data exchange formats can be found in [67–69], which examine product data exchange and product model sharing. In [70], the authors described and presented the concept of internal assembly models and the associated set of geometry processing operators. Wu [71] solved the computer modeling of geometric variations in mechanical parts and assemblies. Another data exchange method in additive processing was described in [72–74], which analyzed slice data representation and formats for multi-material objects. Zhao, Zhou, Zhou, and Zhu [75–78] described data processing in the field of geometric tolerancing, which involves the integration of CAD or CAE applications. Stelmach [79] described the important role of data processing in the influence of hydrodynamic changes in a system with a pitched-blade turbine on mixing power. Some studies presented digital induction motor models based on the finite element method or investigated the possibility of modeling parts in different production technologies [80–83], in which the authors analyzed the importance of individual data in these processes. Runde [84] and Kurylo [85] analyzed the data exchange format for the engineering of building automation systems and data work in the design of an automated system for car body measurement.

The importance of data exchange in CAD was discussed by Rappoport [86], who argued that parametric-feature-based data exchange is one of the most important open problems in reliable modeling. In his results, he presented a general outline of the Universal Product Representation (UPR) architecture, providing universal support for all the data levels used by current CAD systems.

The possibility of further data exchange using different methods was discussed by other authors who focused on the FBDE function, the method of topological comparison of entities, or the quantitative optimization of interoperability during data exchange. Data exchange is also encountered with functions (FBDE), which has been one of the key issues in the history of CPD and should be accommodated in the latest CBDM-enabled CPD. In One study, it was concluded that, for the FBDE function, the same geometric shape would be constructed by different modeling procedures, meaning that no unique modeling procedure can be used for the same geometric shape in FBDE. Therefore, for the exchange of the modeling sequence, mapping between source and target elements should be considered, which includes five cases: 1:1 mapping, 1:n mapping, 1:0 mapping, 0:1 mapping, and in the last case, a singular element [87]. Zhang [88] analyzed the sharing of CAD models, which is, in most cases, a complex problem when dealing with heterogeneous CAD systems. The author introduced a new asymptotic strategy to enrich the interoperability theory based on features in singularity solving. The results of his research showed that a properly designed approach can ensure that the exchanged model of the target CAD system can

be parametrically adjusted. Varga [89] also focused on the application of CAE systems for machining in the production of shape-complex surfaces.

The analysis of the available literature indicates a lack of studies that specify the percentage level of object compliance between the selected neutral formats. This manuscript is part of the effort to fill the gap in the current state-of-the-art research and is the basis for similar cases of object conversion or testing software that examine compatibility levels.

2. CAx Class Systems

In the modern world, an inherent attribute of an engineer is a computer and the corresponding software. Of the many types of engineering software, the most important are the various CAx systems, which make it possible to significantly accelerate and facilitate (and thus reduce the cost of) the transition from idea to practice [4,10,34,54,77].

Computer-aided systems (CAx) involve the use of computer technology to aid in the design, analysis, and manufacture of products. Advanced CAx tools merge many different aspects of product lifecycle management (PLM), including design, finite element analysis (FEA), manufacturing, production planning, product, and more. CAx systems enable the identification and correction of design defects as early as the virtual model design phase, thereby speeding up the product design and manufacturing process.

There are many CAD/CAE/CAx systems in the engineering software market. Among the many tools used in integrated manufacturing are CAD systems (computer-aided design), which are computer systems supporting the design of structures, details, and tools and machines: SolidWorks, AutoCAD (Mechanical, Electrical, LT), Autodesk (Fusion 360, Inventor, Product Design Suite, and Revit), PTC Cero, CATIA, Siemens NX, Civil3D, MicroStation, Rhino, SolidEdge, Sketchup, Alias, Vectorworks, OpenSCAD, FreeCAD, Draftsight, Spaceclaim, Tekla, Geomagic Design, ArtCam, Inroads, ArchiCAD, BricsCAD, CorelCAD, CADStd Lite, and CorelDRAW Technical. The listed software programs provide applications for all industries including construction, architecture, energy, and automotive.

2.1. Overview of CAx Systems

The following CAEs (computer-aided engineering systems, defined as computer systems supporting calculations and engineering analyses related to strength requirements and the simulation of the complex stress states of products and machines) should be noted: NX Nastran® for, Ansys Design Space, Pro/Engineer® Wildfire, Catia, and Unigraphics® NX.

The many tools in integrated manufacturing belonging to CAx systems include CAD, CAE, and CAM (computer-aided manufacturing), which are used to facilitate resource management (raw materials and semi-finished products) and enable the use of production machines. PDM (product data management) is widely used to support the management of resource planning and engineering processes and, once implemented, to support production management. EDM (engineering data management) and PPC (product planning and control) systems support the data processing facilities in CAx systems and are useful for complex information flow management: production requirements, design data, product versions and engineering changes, product structure, product properties, and development innovations.

2.2. Data Exchange between CAx Systems

In modern industry, it is quite common to encounter a situation where one person creates a model of an element in a CAD system, and another person creates a form for it along with tooling. As a result, the programs in which they work are very likely to be different. This is quite important for the form designer because they need to have a detailed model from another program to perform further modeling. There would be no problem with file exchange if the two systems were the same or if they worked based on an identical data format. The problem with file compatibility only occurs when the systems are different, and worse, when files for one program cannot be directly loaded in another program. In such a situation, it is necessary to translate them into a different

format [2,6,12,75,78]. The given potential problem is one of many that require the use of neutral saving formats.

3. Neutral Formats for Data Exchange

Neutral formats (often interchangeable terms such as standards, couplers, converters, norms, or translators are used) of data exchange ensure communication between systems of a similar class, such as CAx.

These systems are often built on the basis of various representation schemes. The best-known translators are DXF, IGES, and STEP [5,34,49].

3.1. DXF Standard

DXF (Drawing Exchange Format) is a format for exchanging drawing (graphic) data and was developed by Autodesk [1,7–9,15,54,70]. Initially, this translator was intended for the AutoCAD program, but over time, it has become a generally available standard used for 2D data exchange.

3.2. IGES Standard

IGES (Initial Graphics Exchange Specification) is a neutral format that was created in 1979 to enable the exchange of information between CAD/CAM systems [49]. Moreover, since 2003, it is also fully compatible with the standard contained in NISTIR 6972 (USA).

The classical IGES converter can be divided into two parts: a preprocessor and a postprocessor. The preprocessor enables the conversion of information stored in the internal CAD database into the IGES format, while the postprocessor enables the conversion of information from the loaded IGES format into the internal CAD format. All CAD programs are now equipped with the IGES post- and preprocessor. However, the use of this format is limited due to the freedom to define component entries, the accuracy of geometry creation, and the type of tool library used [50].

3.3. STEP Standard

STEP (Standard for the Exchange of Product Model Data) is a standard that defines the rules for writing a product model based on geometric, topological, technological, and material data. The product model and its mapping data record are created based on the ISO 10303 Standard, including the STEP standard specification [23–25,27,30,32,41–44,74].

This translator was developed through a program funded by large industrial corporations wishing to bring about standardization in production, mainly to reduce intermediate production costs. When developing the STEP standard, the best features of standards developed in different countries were selected. The IGES standard played the greatest role here [60,82].

The STEP standard defines the rules for creating a product model in a way that allows for the processing and storage of information about the product and its manufacturing processes. As the STEP standard evolves, it becomes possible to flexibly configure CAx software and hardware components while avoiding incompatibility between data and network communication protocols.

All data in this standard are transported (similar to IGES) in the form of ASCII files. This facilitates their transfer between computers and CAx systems with different hardware and software platforms. The application of the STEP standard is based on application protocols, two of which are very important for the typically cooperative automotive industry. These are AP 214, for applications in the mechanical area, and AP 212 for applications in the electrical engineering area, as described in Table 1.

Table 1. The advancement of STEP standardization [26,28,29,59,64].

Protocol No.	Application	Status	Remarks
AP203	Structural design	IS	Structural 3D models in the manufacturing industry. Functional record in AP214
AP212	Description of the structure of electrotechnical devices	CDC	Structure of electrical equipment record in the construction of ships, cars, airplanes, and in the space industry. Compliance with AP214
AP214	Product data structure in the design of manufacturing processes for automotive components	IS	Structure development and record in the machine industry

In 2014, the STEP AP 242 application protocol was introduced, which combines the common features of the above protocols, as shown in Table 1. The main advantages of Step AP 242 are the functionality and capability to support all AP 203 and AP214 protocols, as well as 3D semantic PMI, high-quality 3D shapes, 3D kinematic assemblies, and 3D wire harnesses and pipelines. Due to its complex functionality, the AP242 application protocol is the industry's preferred STEP development protocol.

The main tool (programming language) for modeling the product structure in the STEP application is EXPRESS-M (Modeler) and EXPRESS, which is a text language for describing the data model.

The EXPRESS language is the basic element of the comprehensive, standardized description of a product model in the scope of the ISO 10303 Standard. Apart from the EXPRESS language, the EXPRESS-G graphic module has also been developed. An important benefit resulting from the consistent use of formal language is the unambiguous definition of the data transferred in the application. This makes it possible to test the specification of the data model by means of correct syntax and to generate the so-called adaptation programs, e.g., CAD pre- and postprocessors [14,19,36,70].

3.4. Formats Compatible with the CAx Kernel

In addition to neutral data exchange formats, there are also others ensuring proper data exchange between systems of a specific type.

Parasolid Transmit File Format

Parasolid Transmit file: This format enables data exchange between systems built on the Parasolid kernel. It allows for storing topological and geometric information of a created shape [7,11,18,19,21,40].

There are two types of translators:

(1) Parasolid X_T—text (saves information in a text file);
(2) Parasolid X_B—binary (saving data using symbols 0 and 1).

The purpose of the X_B format is usually limited to data exchange between the same type of systems (older or newer versions). The X_T format has a wider application for other systems as well. It does not mean that the file exported in the X_B format cannot be imported to other systems with the above-mentioned kernel.

ACIS Format

ACIS: This is a translator that enables data exchange between systems built on the kernel of the same name. Like Parasolid, it has the ability to save in text (SAT (*Standard ACIS Text*)) and binary (SAB (*Standard ACIS Binary*)). The format commonly used is SAT only, which uses the ASCII code for writing information. The saved file can be viewed using a simple text editor [54,60,62].

4. Translation Errors

Universal formats, apart from their undoubted advantages, also have disadvantages. Whenever the translation process is performed, there is a possibility of incorrect shape interpretation, which may result in geometry errors.

Common mistakes include [16,49,63,72] the following errors:

- Surface shortcomings—a gap is created between the adjacent surfaces, which prevents the connection of both surfaces;
- Holes—areas where, for example, the surface has disappeared and a void is thus generated;
- No surface trim—fragments of the surface protrude from the outline of the solid or the surface that was to be created, the cutting of which was omitted during translation;
- Overlapping geometry elements—when surfaces or edges intersect at some point, creating incorrect geometry;
- Unnecessary "garbage" in the file—when there is other, unnecessary information in the file in addition to the necessary data (e.g., chunks).

Data transfer, interoperability, and unreliability of selected operations

In the article entitled "*Product data exchange and product model sharing*" [13,33,41,55,83], the content of which was presented at one of the lectures at Delft University of Technology in June 2001, Joris Vergeest mentions, among others, the reasons for data transfer between systems, interoperability problems, and symptoms of defective operations.

The main reasons for data exchange between CAx systems are as follows [53,72,76]:

- The need to transfer models from systems that are unable to perform specific operations to systems with the desired functions;
- The continuation of work in another system;
- Communicating with other, often distant colleagues;
- Archival goals.

If the exchange occurs between the same type of systems (when the software modules are the same and of the same version), then the most effective way of communication is direct transfer (Figure 1), which does not cause data loss.

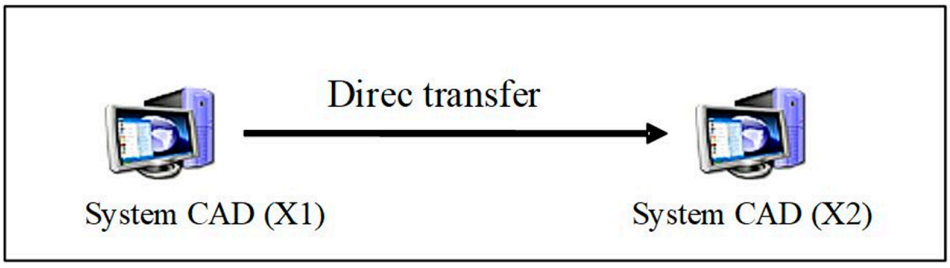

Figure 1. Direct data transfer between systems of the same type.

If the two systems are of a different type, it is usually not possible to transfer data from system A to system B (Figure 2). The file exported by system X is not understood by system Y. So, some kind of processing (conversion) is needed to enable this exchange.

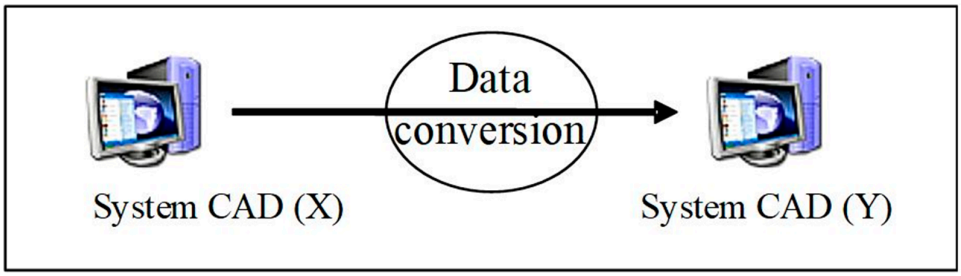

Figure 2. Data sent from system X to Y; the need to process information.

To improve the control of various exchange processes and reduce the number of converters, it is necessary to use CAD transfer standards. If a specific standard is used, the sending system should export the data in a format defined by it. The receiving system must be able to read and correctly interpret the data (Figure 3).

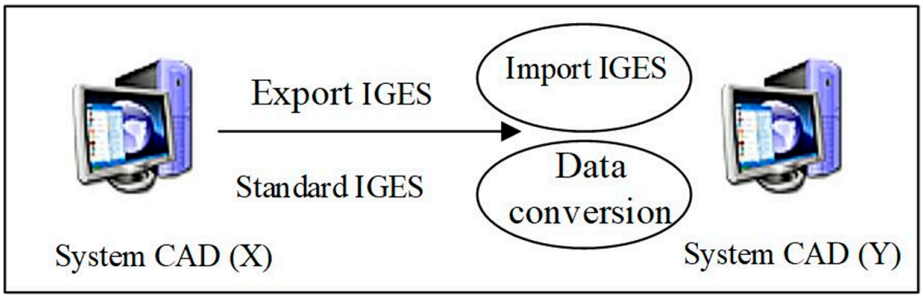

Figure 3. Using IGES data exchange standard.

STEP is the most developed standard. One of the advantages of using STEP is the semi-automatic data export/import resulting from the definition of the standard, which is "computer-readable" (Figure 4).

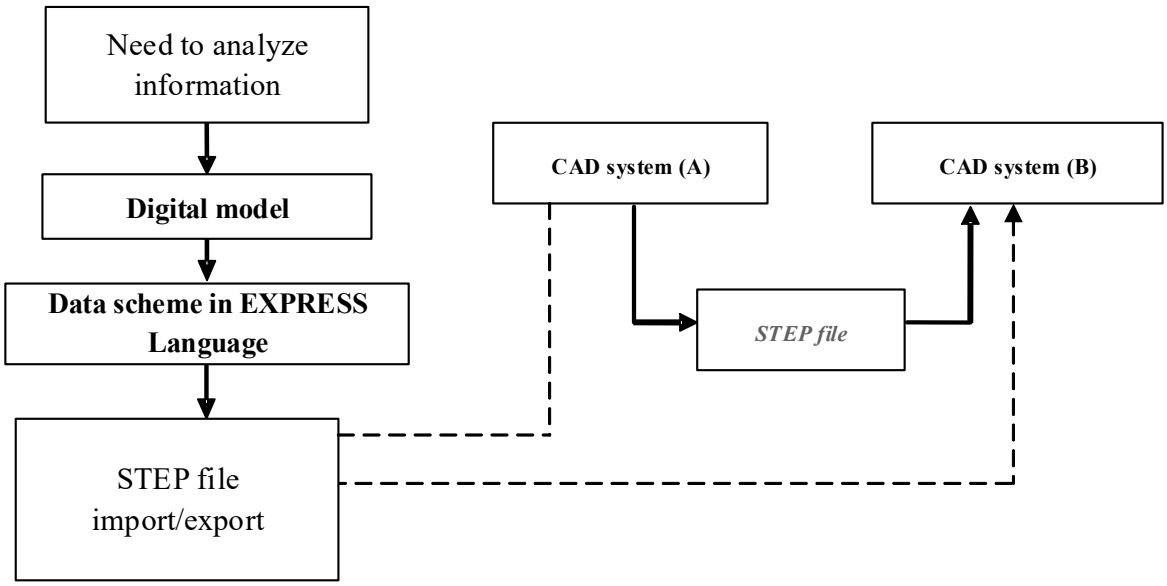

Figure 4. Data sent from system A to B using the STEP standard [14,34,35,66,76,80].

5. Interoperability Problem

With the advent of computer-aided design and quality engineering tools, the speed of software development has changed drastically. The integration of operations, data, models, and results from many disciplines has become the key to effective teamwork in industry. As a result of the high pace of development, the requirements for the information to be transferred and the methods for transferring knowledge, information, and data between designers and engineers have increased. Transfer issues have posed serious obstacles that have only been defined recently [31,67].

In collaborative design and engineering projects, most of the transferred data are reprocessed in CAD and other industrial systems. The increasing amount of information transferred between the CAx family systems is a kind of adoption of data exchange standards, and these practices are successfully supported using the PDM methodology. A significant fraction of transactions between systems fail due to the loss of semantics and/or the poor interoperability of software tools [61].

When formulating the problem of interoperability and operations between users X and Y, we can distinguish three concepts defined for each of the users, languages (Σ^X and Σ^Y) containing the possibility of coding objects, areas of representation (Q^X and Q^Y) containing representations of q^X and q^Y objects, and spaces of reflection (U^X and U^Y) defining the semantics of the objects. A reflection space is a collection of all entities that are relevant to a specific application or context.

Usually, the set of all possible representations of Q is a subset of U. In general, there is a very significant relationship between Q and U. The relation between the concepts is shown in Figure 5. The dashed lines show the process of mapping Q to the subset U. The relationship $a \in \Sigma$ causes the transition from the state q^1 to q^2, the output of y, and the transition from u^1 to u^2 [52,72,85].

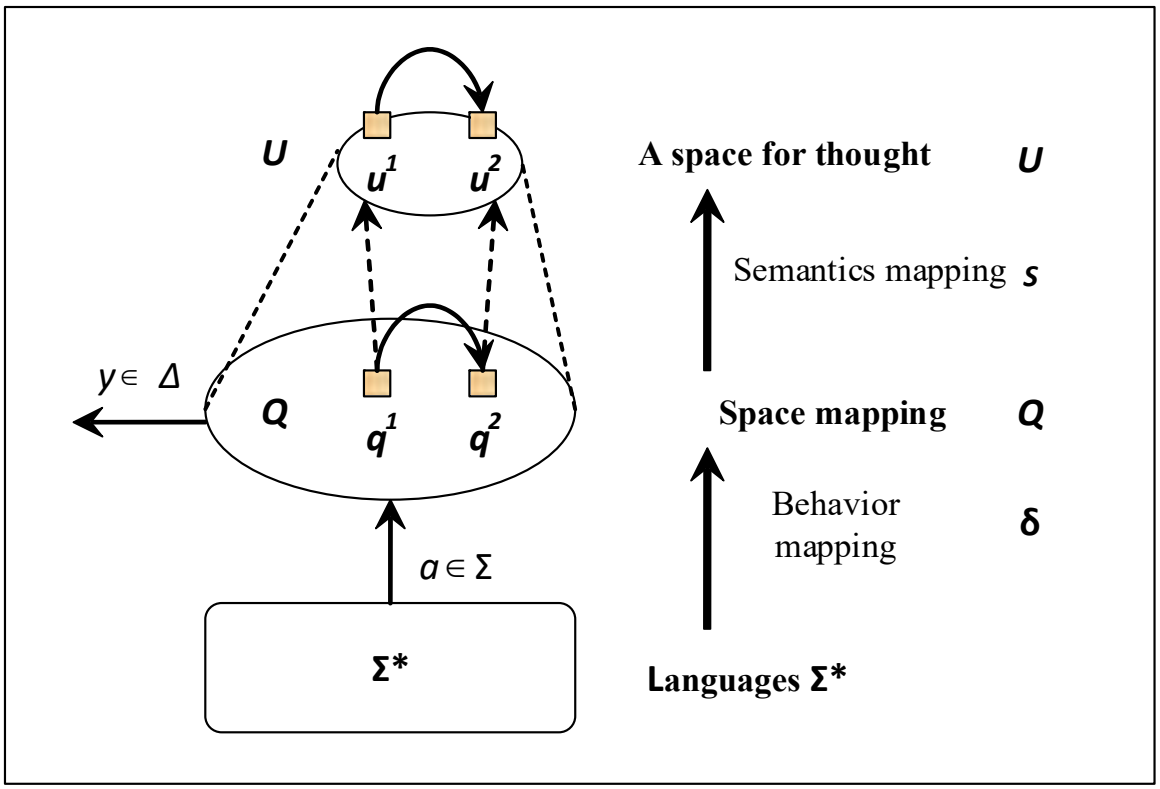

Figure 5. Language specificity Σ* for each user (X, Y, etc.).

In order to treat X and Y users separately, the definition of a reflection space (U) should be known to each side. Let us suppose that a member of team X, while working on a joint project, sends data to team member Y. The transfer initiated by X issuing the command $t \in \Sigma^X$ such as the string $y^X = \lambda^X(t, q^X)$ will be the serial code of the object q^X. Then, the string y^X will serve as an input string for the Y system, which will then be transferred to the q^Y state (Figure 6). The transfer can be considered successful if $u^X = u^Y$.

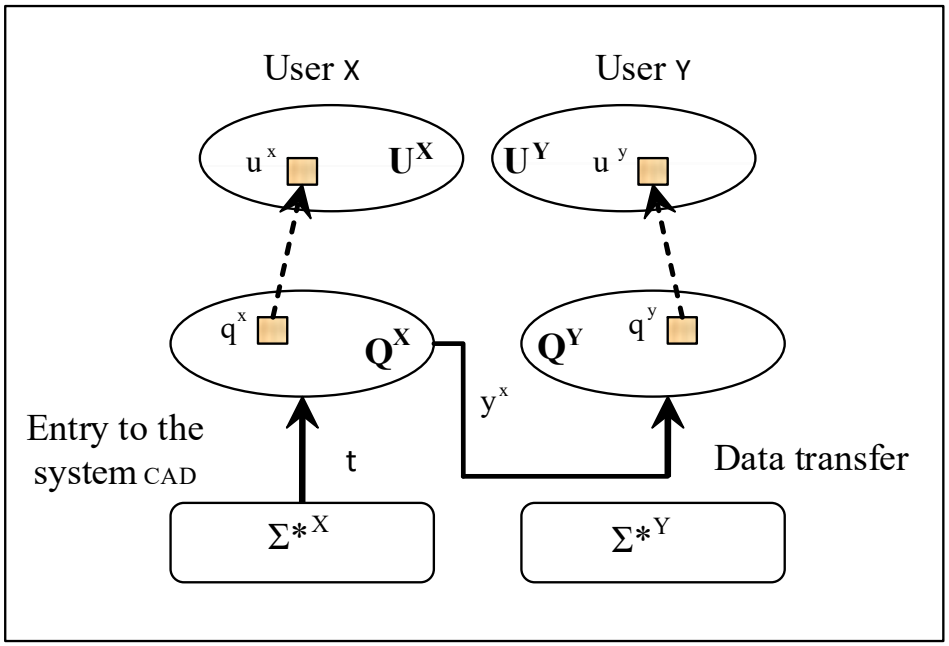

Figure 6. Data transfer from user X to Y.

6. Research

The preparation of this study required an analysis of the purpose, scope, course, and presentation of the results. A series of experiments based on a similar scheme and presentation method was planned.

The experiments consisted of exporting and importing 2D (flat documentation) and 3D (solids and surfaces) objects (Figure 7) to and from various CAD/CAE/CAx systems, in the following data exchange formats:

- For 2D, DXF objects (*.dxf);
- For 3D objects, IGES (*.igs; *.iges); STEP (*.stp; *.step); and ACIS, including SAT (*.sat) for systems with the ACIS kernel and PARASOLID (*.x_t) for systems with the PARASOLID kernel.

The aims of this research were as follows:

(1) To verify the compliance of CAD systems with selected neutral data exchange formats;
(2) To select the translator that is most suitable for the type of transferred entries (offering the lowest number of errors);
(3) To become familiar with the effects of applying certain import/export options.

The research plan included the following steps:

(1) Designing 2D drawings and 3D objects in individual systems;
(2) Saving the document in the default format offered by the program;
(3) Exporting (using the "Save as. . ." or "Export" command) the file to the specified format;
(4) Planning the course of the experiment;
(5) Defining the evaluation criteria;
(6) Determining the importance of each criterion;
(7) Importing the file into other systems;
(8) Analyzing the results;
(9) The presentation of the results;
(10) The comparison of the tested translators.

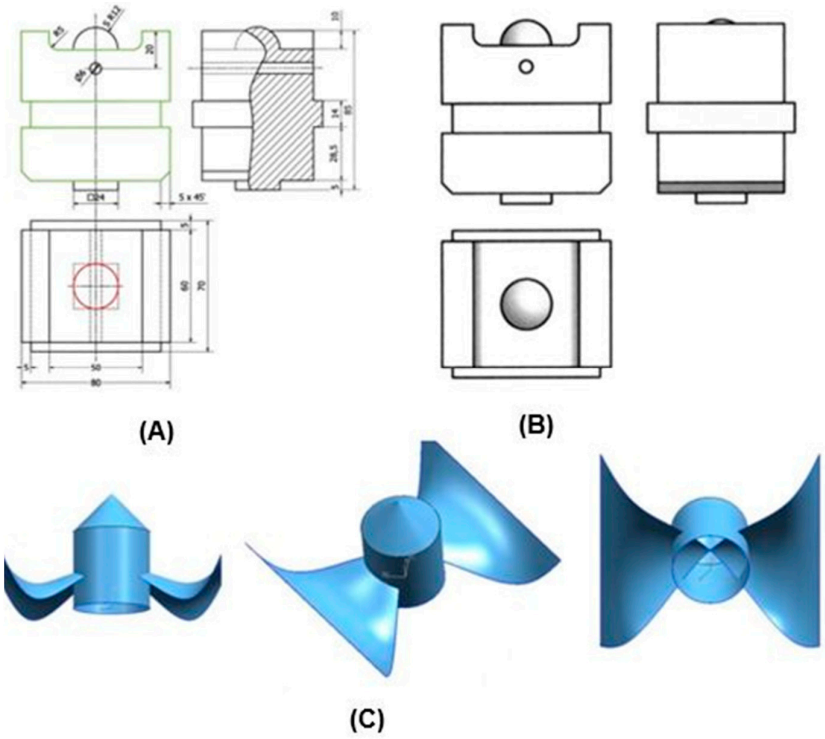

Figure 7. Example of transferred geometry: (**A**) an example of an object in 2D; (**B**) the 3D views of a solid rectangular object; (**C**) an example of a surface object.

The following assumptions were made:

(1) The exchange will be based on the "everyone with everyone" principle;
(2) CAE systems will act only as "recipients" of the file;
(3) Various possible options (if available) will be considered when exporting/importing data.

Ten systems (CATIA, SolidWorks, NX, AutoCAD, CADKey, Autodesk Inventor, Autodesk Fusion, FreeCAD, CADSoftTools, and BRL-CAD) were used for the tests, for which approval from distributors and technical support was obtained. Not every system participating in this study offers the function of saving/reading files in all the tested formats. There were cases in which import/export proved impossible, most often due to the limited license or version incompatibility of individual translators.

A more detailed description of the methodology applied to explain this is shown in Figure 8 for a 2D object, in Figure 9 for a solid object, and in Figure 10 for a surface object.

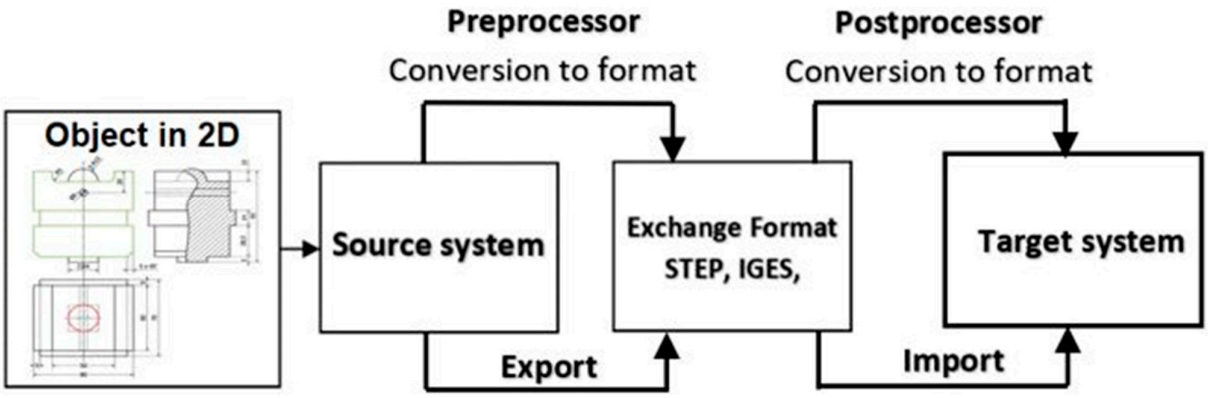

Figure 8. Methodology display example for object in 2D.

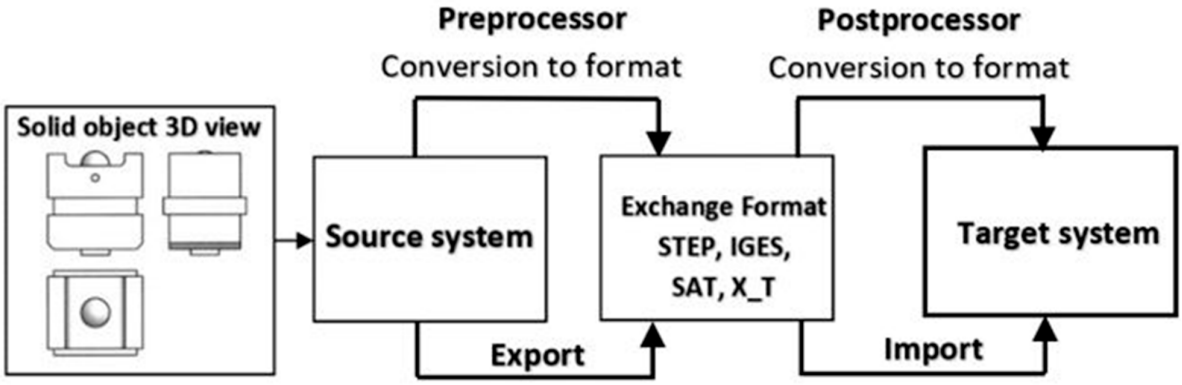

Figure 9. Methodology display example for solid object.

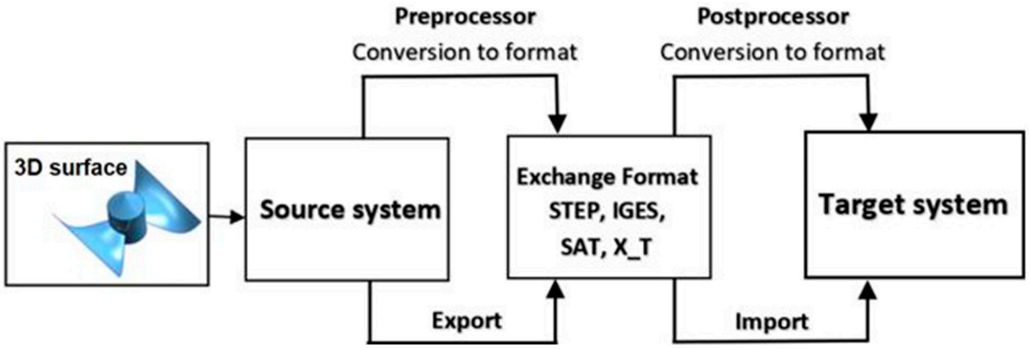

Figure 10. Methodology display example for surface object.

The research plan is schematically presented in Figure 11.

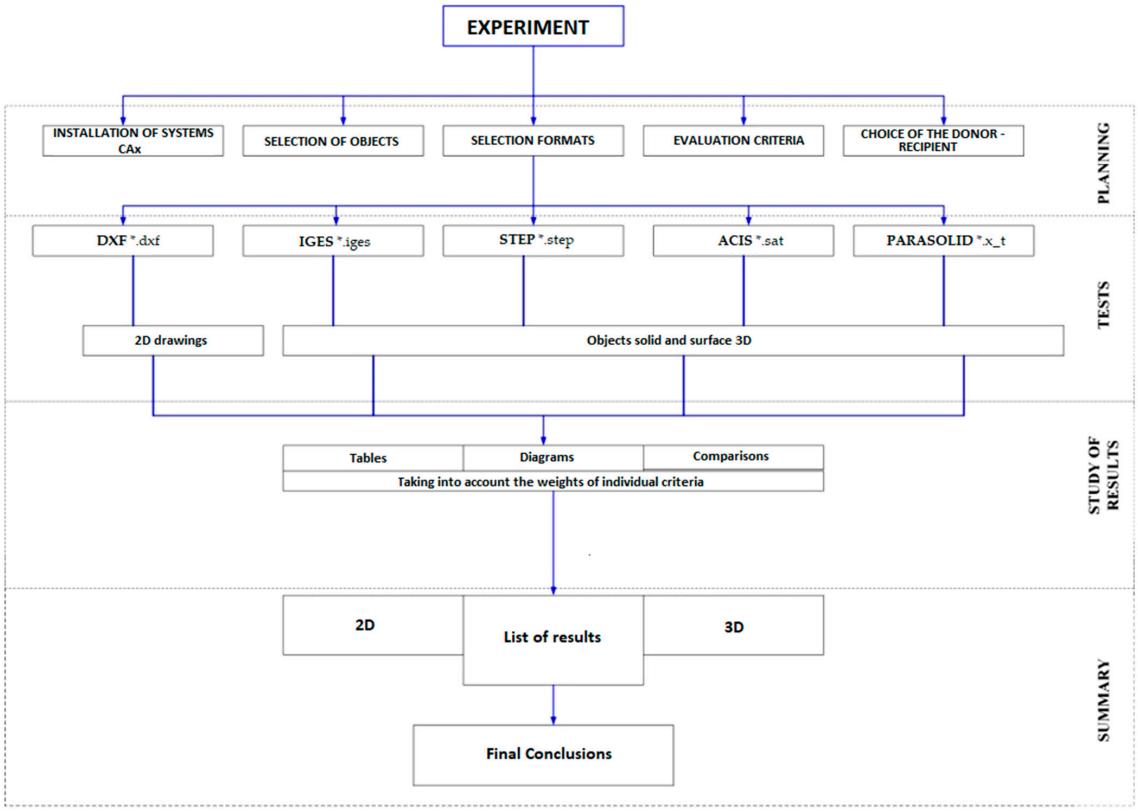

Figure 11. Diagram of the research plan.

7. Analysis of Import/Export of 2D and 3D Objects in Neutral Data Exchange Formats

7.1. Import/Export of 2D Objects in DXF Format

In the manufacturing process, we may encounter parts for which the final design can be implemented as a 2D or 3D model or a model with drawing documentation created in the CAD system. Thus, a model generated in any CAD system can be used for a variety of applications: for machining simulation and toolpath generation in a CAM system, finite element analysis, or documentation applications.

The reason for selecting neutral formats such as DXF, IGES, STEP, SAT, and X-T, which are the focus of this experiment, is their commercial availability. The first reason is that they are standards developed by the ANSI and ISO standards, and the second reason is their wide use in industrial sectors, such as the automotive and aerospace industries. They are also of interest in the field of CNC manufacturing, where data conversion, working with different processing formats, and data exchange between different CAD systems are frequently used. The evaluation criteria were selected and used to solve the problems of data processing between the customer's program and the program used for production, which can be applied to the machining process. In the production of parts, a large part is related to the program in which a given company works and in which a given CAD model is created. However, the range of programs that can be used in the field and the consequent work with them is limited by the format in which the CAD model is saved, sent, and subsequently opened. The selected formats were chosen based on the degree of compatibility between those most used for saving in different CAD systems. The DXF format is the most commonly used data format in current graphics and CAD programs. The choice of comparing DXF formats was based on their use in programming on CNC machines, which makes it possible to open these files and extract points and contours from them. This saves programming and testing time, and the resulting contour is an exact match to the drawing. DXF files usually contain several layers in which the part drawing is arranged.

When producing parts with the required dimensions and shapes, CNC programmers often encounter a problem called data conversion, especially in the case of shaped mold parts, where individual surfaces need to be manually modified using various tools in the CAD system. These errors appear as missing surfaces, gaps, holes, sharp edges, overlapping surfaces, or incorrect visualization of details. The reason for these errors is the format in which the CAD model is saved is different from the format in which it is subsequently opened in another CAD system. This error occurs when different CAD systems are used for different project teams.

Saving CAD models in a certain format, which is the starting point for its further production processing, as well as data processing conversion, has its own justification. Incorrect data processing not only increases the production time but also the time needed to make corrections in the CAD system. It is only when the model is repaired and meets the required properties that it is ready for the next process, e.g., programming toolpaths in the CAM system and generating NC data for a specified CNC machine.

The DXF translator, as a neutral format for the exchange of 2D objects between CAD/CAE/CAx systems, translated the transferred entries correctly in most cases. An imported object with a partial loss of geometry can be quickly repaired in many cases. Before exporting an object, it is always worth considering the "target" system for which it is intended. Some systems, or rather their translators, are not compatible with each other. In these cases, correct data exchange proved impossible. It is possible to experiment with different import options to find the right settings for the type of 2D data being transferred. It is also advisable to export the data several times with different export settings, if available. Sometimes, a change in one parameter (e.g., format version) can greatly improve the result of an import into another system. It is also worth noting which types of drawing entries transfer worse than others. The time spent on drawing them can often be wasted, as these entries are most likely to be lost during replacement [51,57,58,79,87–89].

7.2. Import/Export of 3D Solid Objects

During the analysis of data exchange in the form of a solid object entry in the IGES, STEP, SAT, and X_T formats, we aimed to select the format that was the most suitable for the type of geometry transferred. In order to compare the above translators, it was necessary to divide them into two groups:

(a) IGES and STEP—as neutral formats for data exchange between all CAx systems regardless of the kernel used;
(b) SAT and X_T—as formats enabling the exchange of information between systems using specific spatial modeling kernels.

The IGES translator, in comparison with the STEP translator, was less able to correctly recognize the type of an object, its topological conformity, and its division into integers. However, it was better at transferring sketches. The comparison of the obtained results for a solid object between two formats, IGES and STEP, is shown in Table 2.

Table 2. Comparison of the obtained results for a solid object (IGES and STEP).

Solid Object	IGES			STEP		
	Correct	Possible	Percentage Value	Correct	Possible	Percentage Value
Object type	45.8	56	82%	36	36	100%
Topological compliance	55.2	56	98%	36	36	100%
Division into integers	5	56	9%	4,5	36	13%
Sketches	21	56	38%	6	36	17%

Table 3 shows that the ACIS (*.sat) and PARASOLID translators performed well on the two most important parameters assessed. Unfortunately, they do not allow for the export of the division of an object into integers or a sketch.

Table 3. Comparison of the obtained results for a solid object (SAT and PARASOLID).

Solid Object	SAT			PARASOLID		
	Correct	Possible	Percentage Value	Correct	Possible	Percentage Value
Object type	2	2	100%	12	12	100%
Topological compliance	2	2	100%	12	12	100%
Division into integers	0	2	0%	0	12	0%
Sketches	0	2	0%	0	12	0%

If the CAD system offers object export in all four formats, then the STEP translator should be chosen. In contrast to IGES, it has fewer import/export options; therefore, it is more difficult to make a mistake.

7.3. Import/Export of 3D Surface Objects

By analyzing the results of data exchange in the form of surface object entries in IGES, STEP, SAT, and X_T formats between systems from the CAx family, an attempt was made to select the translator that performed best in the tests conducted.

As can be seen in Table 4, the IGES standard proved to be better than STEP this time. The translation of the type of object, its topological compliance, and a sketch could be performed more accurately using this standard. The data transferred to the Ansys Design Space system did not contain sketches. If this were not the case, the percentage of this type of transferred entries would not be 71% but 83%, as is shown in Table 4.

Table 4. Comparison of the obtained results for the surface object (IGES and STEP).

Surface Object	IGES			STEP		
	Correct	**Possible**	**Percentage Value**	**Correct**	**Possible**	**Percentage Value**
Object type	48	48	100%	34	36	94%
Topological compliance	46	48	96%	31,6	36	88%
Division into integers	48	48	100%	36	36	100%
Sketches	34 (40)	48 (42)	71% (83)	24	36	67%

The division of a surface object into integers depends on many factors and is less important here than in the case of solid objects. Unstitched surfaces always retain their division. By using the appropriate stitching sequence, it is possible to achieve the desired effect without major problems. As a result, we often achieve 100% compliance with the assessed parameter. The number of analyses performed in both formats is also worth considering. Typically, the higher the number of tests, the higher the probability of encountering a misinterpreted case, thus lowering the overall number of points obtained. As Table 4 shows, despite the greater number of analyses, the IGES format produced better results than STEP.

An analysis of the research results presented in Table 5 also reveals the relatively low number of tests performed. The systems tested often had limited licenses and did not always allow for this type of replacement. The results obtained with the SAT and X_T translators are very good; they would be even better if these formats also supported sketch import/export. To fully reveal the advantages and disadvantages of the SAT and X_T translators, a much larger number of tests should be carried out.

Table 5. Comparison of the obtained results for the surface object (SAT and X_T).

Surface Object	SAT			X_T		
	Correct	**Possible**	**Percentage Value**	**Correct**	**Possible**	**Percentage Value**
Object type	2	2	100%	6	6	100%
Topological compliance	2	2	100%	6	6	100%
Division into integers	2	2	100%	6	6	100%
Sketches	0	2	0%	0	6	0%

8. Conclusions

This paper presents detailed information and research results on computer systems supporting the work of an engineer–designer and provides a general method of solving the potential problems that may arise when working with different formats for 2D, solid, and 3D surfaces, comparing DXF, IGES, STEP, SAT, and X-T formats. In this way, this study allows for a better explanation of and approach to the efficiency of using these formats and makes the further processing of data easier to compare with each other. The models described in this paper can be generally applied to the user's needs only when using the formats outlined in the paper.

We conducted several data exchange studies to identify a neutral format that ensures the least data loss for a specific type of transferred geometry. The analysis was based on a carefully prepared experiment and defined evaluation criteria. All tests were carried out with great care and attention to avoid errors. The obtained results were presented in the form of tables and comparative graphs.

The following propositions can be suggested:

1. For the conversion of data in the form of geometric elements of a CAD model, the STEP file format is the most suitable. It can better recognize the type of object as well as its topological correspondence at a higher level compared with the IGES format. On the other hand, for 2D drawing formats, the IGES format is better.
2. Comparing the ACIS (*.sat) and Parasolid type translators, it can be stated that their main disadvantage is the insufficient export of the object division and the CAD model to integers or sketches. However, they are very good at defining the object type and topological correspondence, which plays a very important role when working with systems in which the main part of the project is meshing the model, i.e., the object. These are, for example, the Moldflow Plastic Insight systems or Simufact Additive, designed for metal additive manufacturing, where the CAD file is imported in Parasolid format, and the imported file is then used to generate a surface mesh of the model, containing data on the number of elements and the number of nodes, which is later important for the creation of the voxel mesh. Therefore, the correct choice of format is also important in this production process.
3. In the case of modeling surfaces, which are most often used in the field of automotive production or mold manufacturing, the IGES format proved to be better than the STEP format, which better handled the parameters of the project in the form of sketches, topological correspondence, object type, etc. Applied research in the form of the evaluation of neutral formats has its justification in the manufacturing process, such as the machining process, where it is necessary to work with different file formats from customers working in different CAE systems. These can be simple or complex surfaces, such as mold cavities, where the way all surfaces are rendered is very important. Therefore, the correct choice of format at the export or import stage of the part plays a crucial role in terms of correctly loading all surfaces, contours, and other geometric elements.
4. Consequently, file transfer and incorrect compatibility between different CAE systems can lead to erroneous data in the form of incorrectly loaded or displayed CAD models, or incorrect geometry elements. This means additional work and time spent on model editing in the form of stitching surfaces, drawing edges, problems with interpolation, the accuracy of drawing geometric elements, etc. All this then has an impact on the productivity of production, as well as on achieving the required accuracy and surface quality of the manufactured parts.

It is hoped that, in the future, design professionals will be able to choose, to some extent, neutral data exchange formats in their work. At the present stage of the development of CAD/CAE systems, neutral data formats do not work 100%; therefore, there is a need for further improvement and thus for the standardization of tools and principles used in design. The development of translators should be aimed at achieving full compatibility with CAx systems. Reaching this level will be largely possible through the acceleration of the introduction of a new product to the market and, consequently, the minimization of the costs associated with this process.

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