PNEUMATIC CYLINDER DESIGN USING CAD-BASED PROGRAMMING

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ABSTRACT: *Engineers and designers may notice an increased productivity in their work using the API (Application Programming Interface) of a general purpose CAD system. Using this API, macros as well as complete applications can be easily developed. The current paper presents an application, called Cylin3Der, that makes use of these capabilities. The application deals with the automation of the design process of a pneumatic double acting cylinder. The aim is to provide to the user a simple form that will act as a starting point for the cylinder design process, based on a set of parameters that follow the relevant ISO 15552 standard. The design process is supported by a group of tools incorporated. These tools deal with: the creation of the components (solid models) of the cylinder, the assembly of the finished product, rendering and finally the creation of the various related documents.*

KEYWORDS: *product design, application programming interface (API), systems integration*

1   INTRODUCTION

Usage of the application programming interface (API) is the key factor for automating almost every feature that is available in modern CAD systems. Therefore, the API is a precious and extremely helpful tool, when developing CAD based engineering applications. Using API based applications, enhances the productiveness of enterprises that use CAD systems for designing and manufacturing their products.

The benefits that derive from the use of a CAD API during the design process are remarkably vast. For example, product development procedures can be automated and many CAD features can be combined into one user-friendly application. In addition to these, many mechanical processes can be easily simulated through a safe, computer based, environment; an interface between software modules as well as between a software module and a programmable machine can also be achieved, thus increasing the production capability of modern industries.

2   LITERATURE SURVEY

Researchers use the API of modern CAD systems in a variety of areas to improve accuracy and effectiveness, while at the same time drastically reduce the necessary processing time and cost for most of their engineering procedures. Kyratsis et al. developed the DRILL3D software routine that creates parametrically controlled drilling tool geometries. They use these tool models and cutting conditions, while achieving the generation of 3D solid models using all the relevant data involved in the drilling manufacturing process simulation (drilling tool, cut workpiece, undeformed chip etc.). The application derives solid models of the undeformed chip by applying boolean operations between the 3D solid model of the tool and the 3D models of the workpieces (Kyratsis et al., 2011a, b). Miao et al. investigated the drilling tool geometry effects on the deposition residual stresses when diamond coated carbide drills are used. This approach combines the solid modeling of diamond-coated two-flute twist drills using a commercially available CAD system with the finite element analysis of the solid models, in order to simulate the residual stresses which are generated during the deposition process due to the mismatched thermal expansion coefficients (Miao et al., 2009).

Haba and Oancea investigate how manufacturing technologies are used in the assembly of a single cylinder engine block. A series of design activities proposed, were implemented based on the GENgine subroutine developed in the Open DCL and Visual

LISP programming environments (Haba and Oancea, 2015a, b). Dul et al. developed an intelligent assembly methodology based on features of standard parts using CATIA, in order to contribute towards the assembly of large number of parts. The methodology, tests the shape feature of the assembly base in order to implement the intelligent assembly of standard parts. The results presented, showed improved efficiency of the designer's work and reduced work intensity in the assembly design (Dul et al., 2014).

Pan et al. presented the application CADWE (Computer Aided Design While Engineering) which aims at merging CAD and CAE under a unified platform. An initial model is adopted to build up consistent and shareable integration information, instead of focusing on programming to link different models for keeping consistency. In more detail, generalized features of the initial geometry are used to achieve synchronous and automatic change propagation from initial geometry to detailed model and FEM analysis. As a result, numerous duplications, human errors in complex simplification operation and incompatible data exchange may be prevented. The application was developed in NX/Open API (Pan et al., 2016).

Xiaoshu and Xichen created an off-line programming system for laser remanufacturing. The application included solid modeling in 3D space, direct slicing for determining the path data, and the creation of the manufacturing process control. Robot models, workpieces, nozzles and workspaces were designed and the motion of the end-effector was determined (Xiaoshu and Xichen, 2009).

Neto et al. presented a CAD-based application for robot programming, allowing users with basic CAD skills to develop off-line robot controlling programs. They take advantage of the relevantly low cost of general purpose CAD systems, in order to extract the appropriate pieces of information for the creation of a series of robot commands. A series of experiments were performed using the proposed application, and, as a result, users with limited training were able to perform specific robot programming tasks. In addition, there is no need for purchasing extra high price commercial robot simulation software (Neto et al., 2010). Baizid et al. have created IRoSim (Industrial Robotics Simulation Design Planning and Optimization), a platform which is based on SolidWorks API and aims to offer an intuitive environment for designing and simulating tasks related to robots. The concept was to integrate features of mechanical CAD and robotics CAD under the same application offering a user-friendly interface. A series of 3D models are included for the implementation of several tasks

related to robots and their simulation. IRoSim provides the tools for: defining the task, CAD learning of the end-effector's trajectory, checking the manipulator's reachability to perform the task, simulating the motion and finally validating the manipulator's trajectory to avoid possible collisions (Baizid et al., 2016).

Chun et al. developed a web based material database in order to assist designers to compare different materials. It provides the functionality of material selection and creation of comparison graphs based on their properties. A combination of Web services and SolidworksTM API was used, so that the designers can intuitively select materials for their products, while appropriate performance indexes are used (Chun et al., 2007; Ashby, 2003).

Mathew and Rao presented a system for processing assembly models and extracting assembly related data. These data were used to identify the relationships between different components of an assembly, thus encouraging the generation of feasible assembly sequences. Instead of having a human interpretation of the assembly design, a direct CAD database interface extracts the relationship parameters, requiring minimum manual involvement. The information extracted, is used to generate a list describing the links between the assembled parts, the involved features and the type of link explicitly to facilitate assembly analysis and planning. With the use of the API the assembly mate information is captured (Mathew and Rao, 2010).

Kumar et al. dealt with feature-based modeling and process parameters selection in a computer-aided process planning system for prismatic micro parts. The system maps the Extensible Markup Language (XML) data from the feature-based model and produces the corresponding process parameters required for the micro part manufacturing. It follows two directions; the first one uses the feature-based modeling and automatically extracts manufacturing feature information, while the second selects the appropriate process parameters for the given micro features using a knowledge-based system (KBS) approach. The application of the proposed system was verified with a case study implemented in Autodesk Inventor (Kumar et al., 2014).

The present paper describes the Cylin3Der application which was developed using the API of the Solidworks CAD system through the embedded VBA (Visual Basic for Applications) programming interface (Tzotzis and Kyratsis, 2016). Cylin3Der is a collection of CAD based routines that automate the design of a pneumatic double-acting cylinder according to the ISO 15552 (2004) standard. The

cylinder's components are built based on the specified parameters and all components are modeled and assembled automatically. At the same time, all the necessary mechanical drawings, a component process plan and a complete high quality render of the product are established.

3   CYLIN3DER APPLICATION

The Cylin3Der application consists of a group of tools that are used to calculate the necessary parameters for the design of a pneumatic double-acting cylinder. The model of the cylinder conforms to the ISO 15552 (2004) standard, which refers to specific design parameters i.e. dimensions, materials and rated pressure (up to 10bar).This pneumatic cylinder is one of the most common mechanical elements used in a wide range of machinery.

3.1 Cylin3Der flowchart

Figure 1 depicts the flowchart of the Cylin3Der application. The workflow is an one way process that begins with the dimensioning of the cylinder. In order for the dimensioning to be completed, the user must set the required parameters, i.e., the force applied to the piston (N), the number of duty cycles (c/min) that the piston must perform and the pressure of the compressed air (bar). Following the calculation of the theoretical diameter, the materials

of the components can be determined based on the cylinder's working environment. This means that each component is associated to specific attributes and properties based on the selected material. Next, the detail-design begins with the creation of the solid models of every cylinder component. Finally, these components are assembled to form the cylinder.

A number of different routines can be used following the cylinder assembly. The choice is given to the user. First, for each component the mechanical drawing is created and saved for future use. These drawings depict all the necessary views of the components with the appropriate dimensions and annotations. Second, the list of components together with pieces of information about their size and volume becomes available (bill of material – BOM). Third, the cylinder with all the components is rendered; a useful feature for documentation purposes. Finally, the process plan for the sheet metal component is created automatically, combined with written work orders to the technicians about the manufacturing processes involved. At this instance, the application integrates the CAD based pieces of information with documentation software, i.e., Microsoft Office Word and Excel.
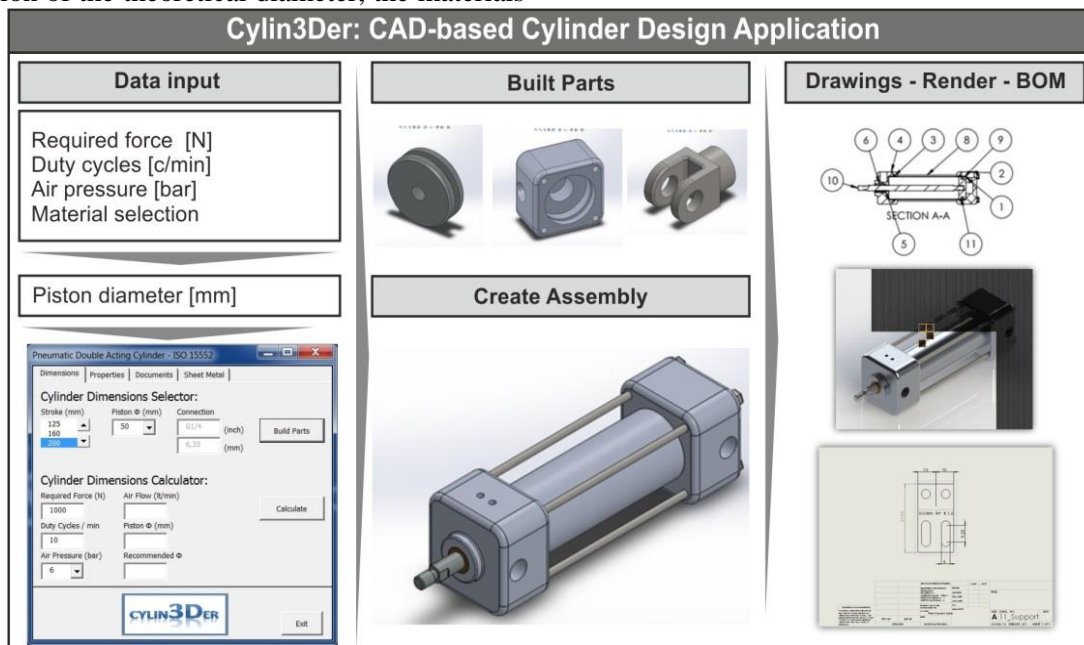

Figure 1. Flow chart of Cylin3DerCAD-based application.

3.2 Cylin3Der flowchart

During the first stage of the design process, the application performs a number of calculations in order to determine the dimensions of the cylinder. The necessary equations involve a set of parameters, i.e., compressed gas pressure, piston

diameter, force generated, friction, piston stroke and air pressure.

The forces developed on both sides of the piston (F1 and F2) can be calculated using equation (1) and (2), respectively.

$$F_1 = [(P * \pi * d^2)/4] * \eta \qquad (1)$$

$$F_2 = \left[\left(P * \pi * (d^2 - d_1^2)\right)/4\right] * \eta \qquad (2)$$

where:
P: is the pressure of the compressed air in bars,
d: is the outside diameter of the piston in mm
η: is the theoretical coefficient that determines the loss of force due to friction.

Additionally, equations 3 and 4 are used by the application to calculate the amount of air needed for the piston to actuate.

$$C_1 = \left[\frac{\pi}{4} * d^2 * l * \left(\frac{P + P_{atm}}{P_{atm}}\right)\right]/1000 \qquad (3)$$

$$C_2 = \left[\frac{\pi}{4} * (d^2 - d_1^2) * l * \left(\frac{P + P_{atm}}{P_{atm}}\right)\right] \\ /1000 \qquad (4)$$

where:
P: is the pressure of the compressed air in bars,
Patm: is the pressure of the atmosphere in bars,
d: is the outside diameter of the piston in mm
l: is the stroke of the piston in mm.

Figure 2 depicts the main window of the application in which the user can insert all the necessary data for the piston calculation. Employing the aforementioned equations, the application performs the calculations in order to determine the recommended piston diameter and the air flow required.
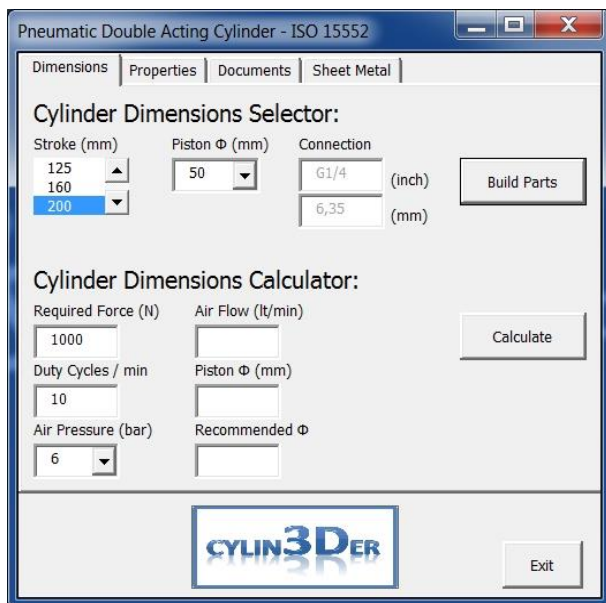

Figure 2. Main dialogue window of the application.

Further to the initial equations, the application makes use of additional geometric relations to determine the rest of the cylinder's geometric parameters, for example, the connection holes on the cylinder's ends, which are directly related to the piston diameter. Those relations are detailed in ISO 15552.

3.3 The component design process

The design process of the cylinder components starts after its initial dimensioning. The process involves the solid modeling of every component of the cylinder assembly. The necessary data for accomplishing the design process is based on the initially inserted piston diameter and the piston stroke. The application takes advantage of the component standardization and derives all the secondary dimensions from the primary ones. In the case of the ISO 15552 standard, the basic dimensions are the piston diameter and the piston stroke, whereas the secondary dimensions are related in a specific manner to the primary dimensions and can be found in charts and technical manuals.

Figure 3 depicts the 'Properties' tab of the application, which is used for setting the template path, the file saving path and the determination of the materials used. After setting the template path, the application has access to the necessary templates required for the modeling of a new part or a new assembly or a new drawing document. Similarly, setting the file saving path, every document created is stored automatically in the specified folder. Finally, by setting the selected materials, all the components obtain specific mechanical properties during the design process. The selected materials depend on the compressed air pressure.
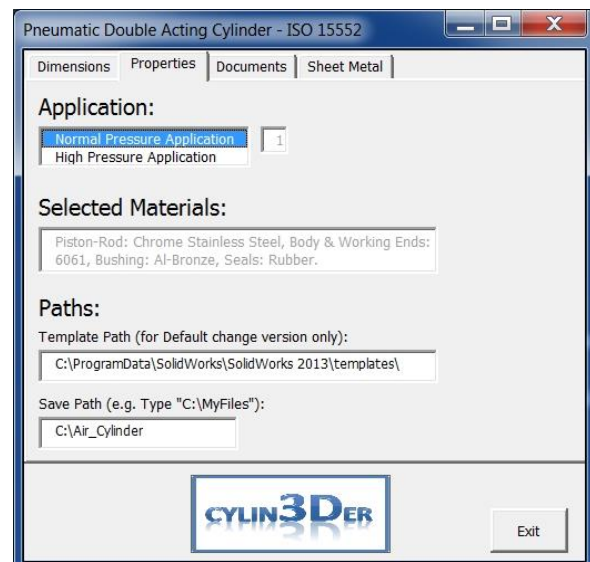

Figure 3. The Properties tab of the application.

The strategy required for the design process to be performed follows six steps that are repeated each time a new 3D model/component is built (Figure 4). In the first two steps the application uses the inserted data to declare the necessary variable types and to set the initial values of the variables. During the third step, the application connects with

the CAD system in order to gain access to the objects of the Solidworks API. By gaining access to these objects, the application performs these required actions: creation of a new part document, drawing of the necessary sketches, and creation of the solid model of the component with all its features.
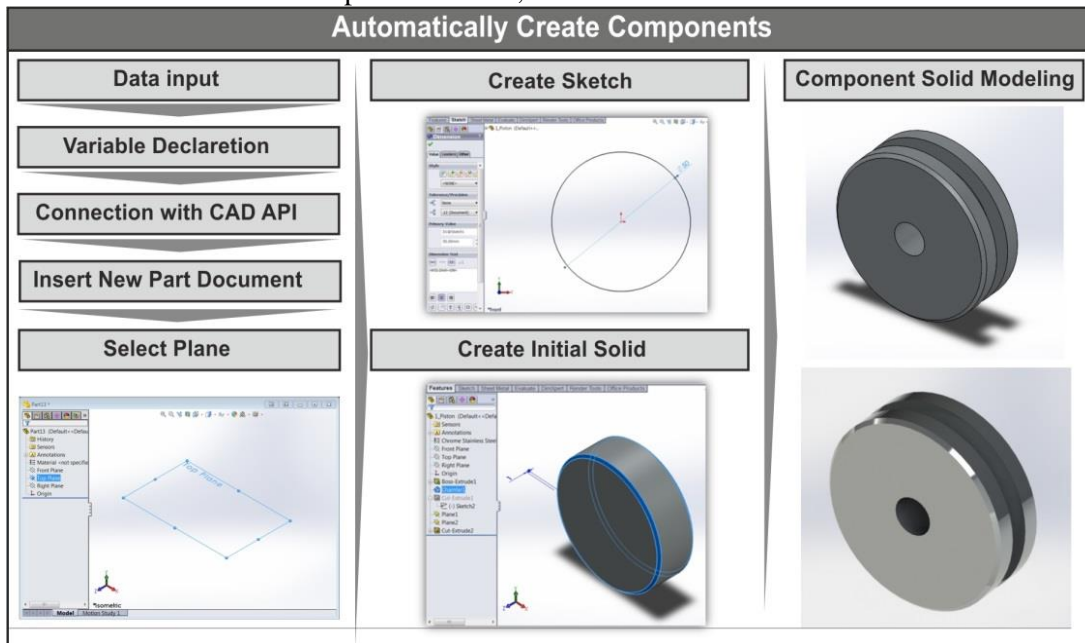

Figure 4. Six step strategy required for the solid modeling of each component
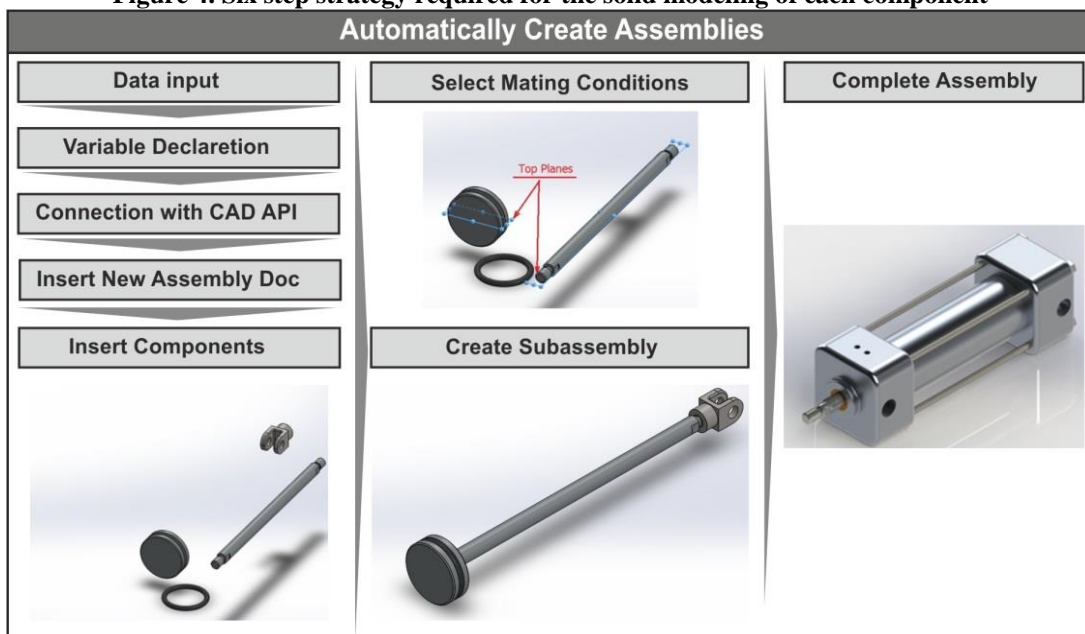

Figure 5. Flow of the assembly design process

3.4 The manufacturing processes and the product appearance

The complete model of the cylinder is an assembly that includes twelve components. Each component is modeled according to the ISO 15552 standard as detailed in Sections 3.1, 3.2 and 3.3. Most importantly, the component material is the parameter that defines the mechanical properties and thus the behavior of the pneumatic cylinder under stress. The present application obtains the values of the mechanical properties from an embedded library. The user is allowed to enrich this library by installing extra materials.

In order to create the assembly model, the bottom-up method is used. It is the most common method and is preferred when the design process is straightforward; it begins with the modeling of the parts and ends up with the insertion of the built part-models into an assembly document. The result of

the assembly process is a CAD model that represents the physical model of the cylinder.

Similarly to the component design process, the flow of their assembly is divided in six steps (Figure 5). First of all, the application connects with the SolidWorks API and opens the already built components in silent mode (components are not visible to user). Then the application creates a new assembly document and inserts all components into the assembly document. Following that, the mating procedure includes the selection of the appropriate mating conditions and entities and completes the assembly of the components (Figure 6).

The generation of the BoM (Bill of Materials) is an important routine that supplements the application and can be used at any time if the assembly file is prepared. By using the assembly model, the application is capable of automatically extracting the custom properties (material, weight, manufacturing cost etc.) of the model, which then can be exported to a Microsoft Excel file. This task is completed with the interconnection of the Cylin3Der application with Microsoft Excel.

In order to begin the manufacturing process of a product, it is necessary that all mechanical drawings are available. The procedure for creating these drawings is always time-consuming. Despite the improved set of tools that the modern CAD systems provide, it is still a challenge for the engineers to quickly create large number of engineering drawings.
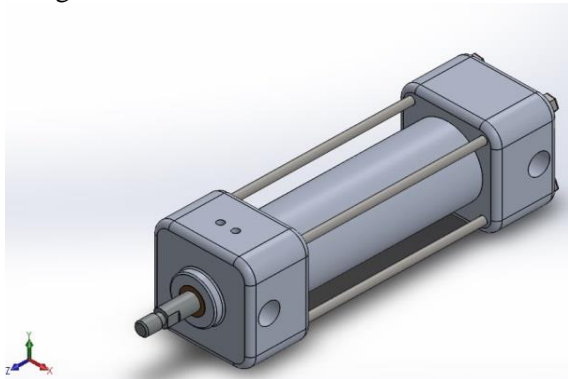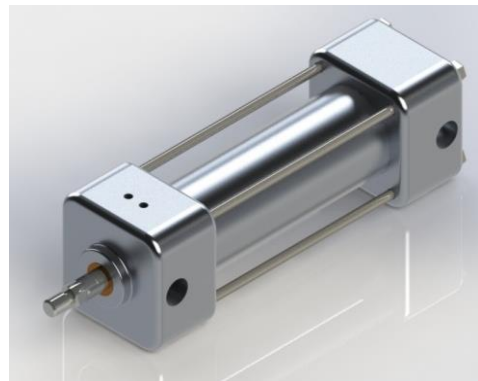

Figure 6. The Documents tab of the application.

The Cylin3Der application provides the user with the ability to automatically create the engineering drawings of all components and assemblies involved. This includes the standard three-view drawing of each component of the assembly, a sectioned view drawing of the assembly itself, along with an isometric view and the bill of materials. It is worth mentioning that even though this application is used to automate the design process for a specific mechanical system, it is possible to use it for other types of mechanical systems with limited modifications.

Figure 7. Depiction of the designed model and the rendered model of the cylinder.

The rendering function is used when a realistic representation of the designed model is needed, mainly for presentation or commercial purposes. The application uses pre-defined parameters (resolution, image type, image dimensions etc.) in order to create a realistic representation of the final product. The process consumes a lot of computer resources and usually takes time to complete depending on the available computer hardware resources. The result is a photorealistic image of the model that can be stored in the specified folder (Figure 7). Depending on the complexity of the

model and the computing capacity available, it is possible to change the rendering parameters for different quality results.

Engineering drawings along with process plans are the documents that make the manufacturing process of any kind of product possible. A process plan contains information about the manufacturing process i.e. the quantity and the selected materials to be used, the machine operations etc. The Cylin3Der application creates automatically such a process plan for the sheet metal parts of the assembly, which are used to fasten the pneumatic

cylinder on a surface. The required information is extracted from the assembly model or its components and then it is inserted to a Microsoft Word document, which is the one that the workshop technician will be able to use for manufacturing purposes.

4   CONCLUDING REMARKS

The use of a CAD system API provides access to the functionality of a general purpose CAD system. Many time-consuming tasks can be automated, whereas large set of CAD features can get organized in an easy to use application. Furthermore, it is possible to supplement the developed applications with a user friendly interface. In the current paper, a CAD based application for designing and manufacturing an ISO15552 double acting pneumatic cylinder is presented. The Cylin3Der application produces automatically all the necessary models and pieces of information with a minimum amount of effort. The models of all components are used from a series of downstream tools and provide automatically product assemblies, BoMs, process plans, renders etc. facilities, while integrate the CAD system with Windows applications.

5   REFERENCES

►Ashby, M.F. and Johnson K. (2003) *Materials and Design*, Butterworth-Heinemann, NY.

►Baizid, K., Ćuković, S., Iqbal, J., Yousnadj, A., Chellali, R., Meddahi, A., Devedžić, G. and Ghionea I. (2016). *IRoSim: Industrial Robotics Simulation Design Planning and Optimization platform based on CAD and knowledgeware technologies*. Robotics and Computer-Integrated Manufacturing 42: 121–134

►Chun, D.M., Kim, H.J., Lee, J.C. and Ahm S.H. (2007). *Web-based material database for material selection and its application programming interface (API) for CAD*. Key Engineering Materials 345-346: 1593-1596.

►Du, B., Wang, X., Feng, Y., Yu, D. and Xu, G. (2014). *Intelligent Assembly Technology Based on Standard Parts Feature of CATIA*. Modern Applied Science 8(2): 49-55.

►Haba, S.A. and Oancea, G. (2015). *Digital manufacturing of air-cooled single-cylinder engine block*. International Journal of Advanced Manufacturing Technology 80(5): 747-759.

►Haba, S.A. and Oancea, G. (2015). *Studies on Thermal Contraction of Crankshaft Bearings under Extreme Low Temperatures*. Journal of Thermal Science 24(5): 96–501

►ISO 15552 *Pneumatic fluid power - Cylinders with detachable mountings, 1000 kPa (10 bar) series, bores from 32 mm to 320 mm - Basic, mounting and accessories dimensions* (2004)

►Kumar, S.P.L., Jerald, J. and Kumanan, S. (2014). *Feature-based modelling and process parameters selection in a CAPP system for prismatic micro parts*. International Journal of Computer Integrated Manufacturing 28(10): 1046-1062.

►Kyratsis, P., Bilalis, N. and Antoniadis, A. (2011). *CAD-based simulations and design of experiments for determining thrust force in drilling operations*. Computer Aided Design 43: 1879-1890.

►Kyratsis, P., Tapoglou N., Bilalis N. and Antoniadis A. (2011). Thrust force prediction of twist drill tools using a 3D CAD system application programming interface. International Journal of Machining and Machinability of Materials 10: 18-33.

►Mathew, A. and Rao, C.S.P. (2010). *A CAD system for extraction of mating features in an assembly*. Assembly Automation 30(2): 142 – 146.

►Miao, C., Qin, F., Sthur, G., Chou, K. and Thompson R. (2009). *Integrated Design and Analysis of Diamond-coated Drills*. Computer-Aided Design & Applications 6(2): 195-205.

►Neto P. and Mendes, N. (2013). *Direct off-line robot programming via a common CAD package*. Robotics and Autonomous Systems 61(8): 896-910.

►Pan, Z., Wang, X., Teng, R. and Cao, X. (2016). *Computer aided design while engineering technology in top-down modeling of mechanical product*. Computers in Industry 75: 151–161.

►Tzotzis A. and Kyratsis P. (2016). *CAD based design and manufacturing*, ISBN: 978-960-93-8570-1.

►Xiaoshu, J. and Xichen Y. (2009). *Off-line programming of a robot for laser re-manufacturing*. Tsinghua Science and Technology 14(1): 186-191.