

ASPECTS ON 3D DESIGN OF ASSEMBLIES AND PART LISTS CUSTOMIZING WITH INVENTOR 2016

Abstract: *The paper aims to present some aspects of the 3D design of assemblies obtained with Inventor 2016 referring to the constraints to pieces placed in the overall file, achievement of new items directly in the assembly concerned, inserting standardized elements in "Content Center" like standardized wedges, safety washers, bearings, etc. The second part of the paper describes how to customize the parts list/table in the drawing.*

Key words: *Design, CAD, modelling, place views, intelligent border, bill of material.*

1. INTRODUCTION TO SOFTWARE – HARDWARE COMPATIBILITY

The design and computer -aided design stage where we are is the result of gradual transition from the work on a generalized drawing board at an earlier time, to the computer -aided drawing stage. Basically the 3 moments: drawing on the drawing board, computer -aided drawing, and computer -aided designs that are natural stages due to the technical evolution not only in the IT field but also in the associated software. Current benefits could be achieved only when both hardware and software components have been synchronized and offered designers extremely powerful tools.

For example in the 90s the graphic stations of the moment used to contain highly performant hard components but if we compare those characteristics with those of today we find that there is a huge difference as follows: a video card in the past had 2MB of memory allocated, while today it usually has 2GB, a server from the 90s had 2MB RAM and now an average PC generally has 4 to 8 GB RAM and so on. In other words, there is bi-univocal correspondence between the hardware platform and software version used.

Specialists' recommendation is that the version of software to be installed should be launched on the market at a maximum 2 years' difference compared to the launch of the hardware of the computer used. Otherwise commissioning delays occur in the execution of orders placed and sometimes even deadlocks. To say nothing of the fact that, in addition to the operating system which is also subject to constant continuous evolution, there is a need to install a version as close as possible to the current year of the software used in design, both variants involving additional resources that the operating system should make available. What happens if the costs of acquiring licenses and work equipment can not be supported? There is only one compromise solution, namely upgrading that component or those components that delay completion of the orders placed.

For example it is recommended the upgrade of RAM, video card and recently even the replacement of old hard drive with a SSD able to speed up 5-8 times data reading and saving. This latest upgrade version has become the main solution today due to lower acquisition costs in the last year. Finally we conclude that the design and engineering work is quite expensive and require particularly special performance training. In case of using Autodesk Inventor 2016 software, the manufacturing company recommends the following minimum system features: 64-bit Microsoft Windows 7 with Service Pack 1, CPU type 64-bit Intel® or AMD, 2 GHz or faster, memory min. 8 GB RAM, min. disk space 100 GB free, graphics Microsoft® Direct3D 10® capable graphics card or higher, 1,280 x 1,024 or higher screen resolution, Microsoft .NET Framework 4.5, etc.

2. INSERTING THE COMPONENTS OF A 3D ASSEMBLY

The 3D assembly [1] is a file containing links to each three dimensional item and is characterized by certain 3D constraints on important parts.

There are two methods for obtaining the assemblies [2]: the first relates to shaping all parts separately and inserting followed by imposing constraints and the second according to which experienced designers and planners generate directly sub-assemblies within certain assembly -type file by generation "piece by piece "and" step by step ". Regardless of the method, the first step in generating a 3D assembly/overall file is to define a new project with the command "Projects" which sets a variety of aspects related to patterns, workplace, location of standardized elements, items purchased, drawings, etc.

For example Figure 1 illustrates 5 variants of insertion in an assembly: piece insertion performed in an earlier time, insertion of standard part in "Content Center", insertion of parts through import [3] using other specialized software (technology AnyCAD), inserting iLogic components and inserting electrical components. In the same figure it can be seen the variant of creating a new piece with the "Create" without closing the assembly file but distinctly saving in a certain location and under a name imposed by the designer. It should be

emphasized that the first piece being inserted, after having been fixed to the reference plane, must be declared of "Grounded Component" type or the appropriate setting could be active, namely "Place and Ground first component at origin". All other items accessed by command "Place" shall be constrained primarily in relation to the first component and between them. All components inserted [4] can be viewed as a name in the Browser from the left side of the modeling space in Inventor. The size of the ASSEMBLY file is not the sum of the PART files because, as I already said above, the 3D assembly works with links to the files related to distinct parts.

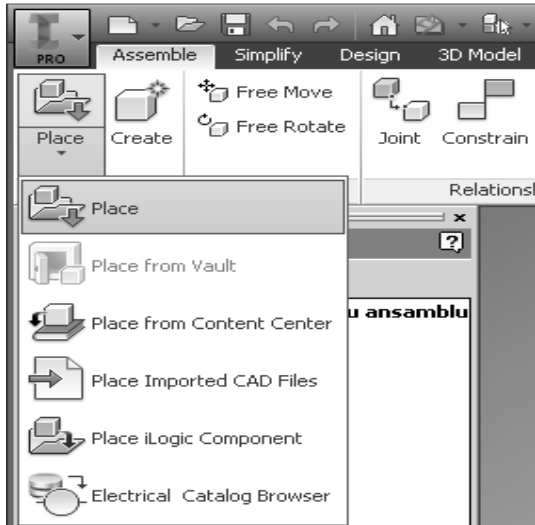


Fig. 1 Methods of inserting parts into the assembly

Also in Figure 1 it can be seen that in addition to the 5 ways of insertion, the „Assembly" file also contains the command "Create", "Free Move", "Free Rotate". The last two are required for preliminary positioning of 3D models before imposing constraints.

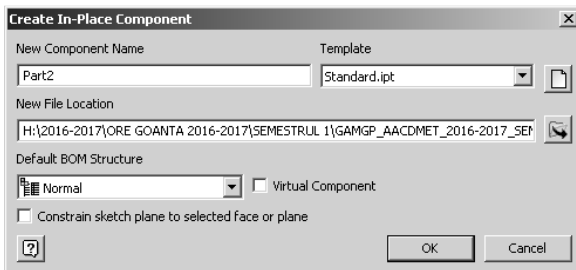


Fig. 2 „Create" window

Figure 2 presents the window of "Create" command which shows that the designer is able to impose the file name related to the component being modeled, determining the type of file (IPT, IAM) and the place in which to be saved and also the constraint with the rest of the assembly by the drawing plane to be used.

3. CONSTRAINING THE COMPONENTS INSERTED

As mentioned above each new component introduced into the assembly [5] is restricted onto the 6 degrees of

freedom with the command "Constrain" that opens the window in Figure 3.

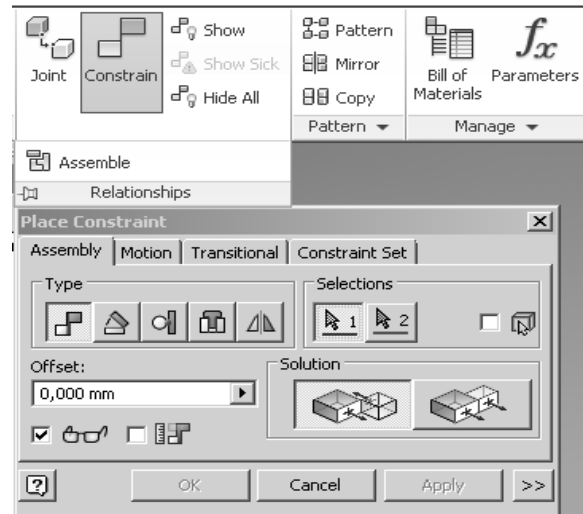


Fig. 3 „Constrain" window

As to the order of selecting the objects to be restricted, the first object is always moved to the second. In the "Constrain" window there are available to the designer the following constraints: "Mate," "Angle", "Tangent", "Insert", "Symmetry". Particularly important are the variants of "Solution" type which determine the orientation of normals to the surface selected. The much easier variant to impose constraints is the command "Assembly" that brings into contact the components of a 3D assembly.

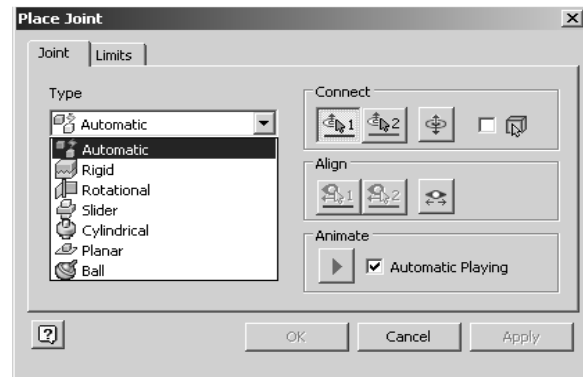


Fig. 4 „Joint" window.

Figure 4 illustrates in detail the options of the command "Join" which allows the introduction of a new type of constraint called joints such as "Rigid" "Rotational" "Slider", "Cylindrical", "Planar" "Ball" and "Automatic".

4. INSERTING STANDARDIZED COMPONENTS

There are very frequent situations in which an assembly needs standardized elements like: screws, nuts, washers, bearings, safety washers, bolts, pins, "O" ring seals or rotation sockets, pipes, fittings, etc. . All these can be placed in an assembly with the command "Place

from Content Center" which, when activated, opens the window in Figure 5.

4.1. Inserting bearings

The bearings library is in accordance with these standards: ANSI, BSI, GB, GOST, ISO, JIS, DIN, PARKER and contain types of radial, radial-axial bearings, axial variants with balls in a row or two rows, with roller or needles.

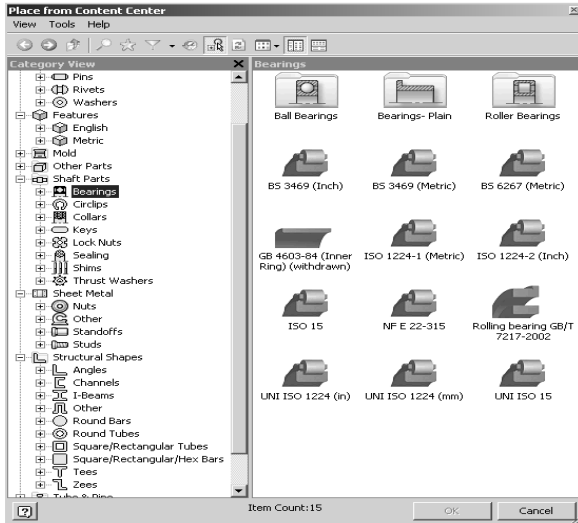


Fig. 5 „Place from Content Center” window

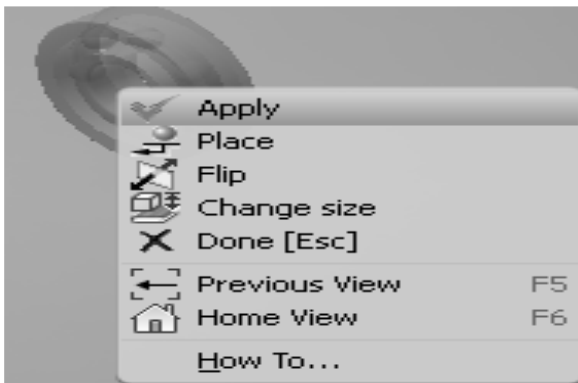


Fig. 6 Inserting Standardized Bearings

Exemplifying in Figure 6 the insertion of a standardized bearing, it should be noted that, after choosing the standard and type of bearing (radial, radial axial, axial with roller or ball), it is selected the cylindrical section where the bearing is to be fitted and the command extract automatically its inner diameter, while the outer diameter will be selected from a table of different sizes.

4.2. Inserting safety rings

Another illustration is the insertion of an elastic safety ring. For this purpose, command "Place from Content Center" will be activated while selecting „Circlips", only DIN standard will be filtered, the section for installation will be indicated and the position from which to begin installation will be selected. The command automatically selects the section diameter and provides the designer

with the constructive version of the nearest diameter, while the user is to choose the ring width from the table in figure 7.

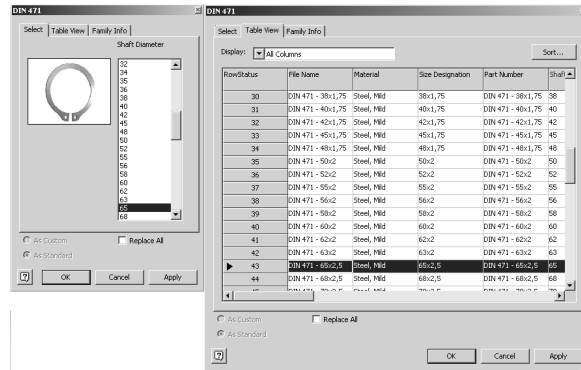


Fig. 7 Selecting the size of the safety ring

4.3. Inserting standardized wedges

In assemblies containing mechanical transmissions it is very often used assemblies with parallel wedges of type A, B, or C. For this reason Inventor 2016 is provided with the command "Key" whose window is shown in Figure 8.

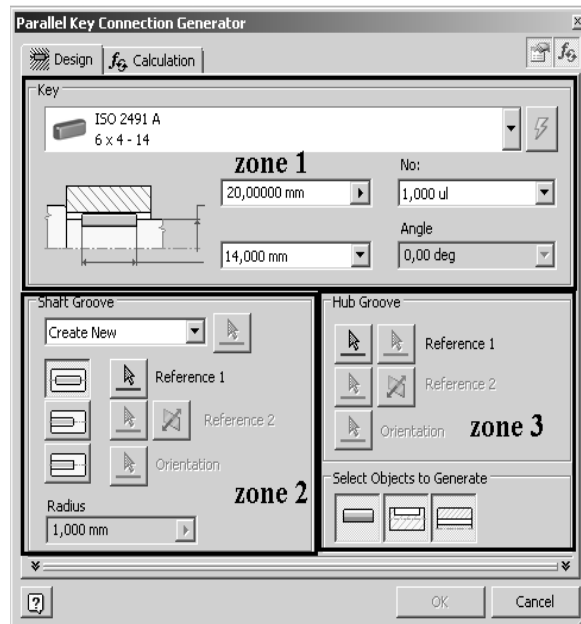


Fig. 8 „Key” window

The window contains three areas where it will be set the various modeling parameters as follows:

- Zone 1 - is the area that requires the type of parallel wedge, the standard applied and wedge size. These initial sizes may change during the execution of the command.
- Zone 2 - is the area where there are buttons for generating the wedge channel in the shaft.
- Zone 3 - is the area where there are buttons for generating the hub channel and the set of buttons establishing right from the beginning of the command implementation what is to be done.

5. INSERTING AND CUSTOMIZING THE PARTS LIST

After inserting view projections or the sections made in the assembly, it will be introduced the component table with command "Part List". This table is default, meaning with the table head written in English, with a much smaller number of columns than that according to current standards and dimensions different from those desired by the designer. For this reason it will be changed with command "Edit Part List Style". To activate this command, make click on the table, then with the right mouse button select the command mentioned whose window is shown in Figure 9.

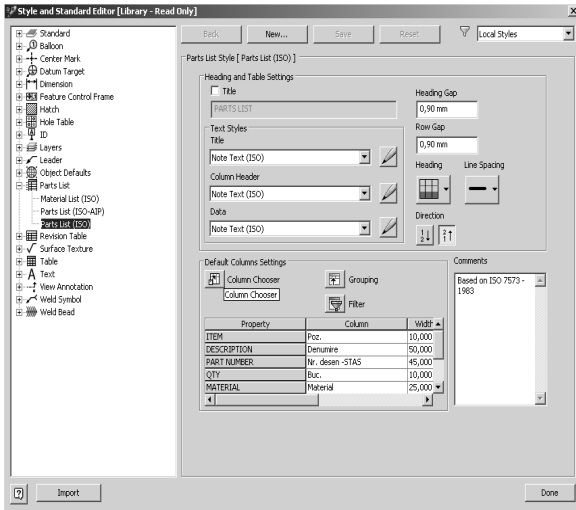


Fig. 9 „Edit Part List Style” window

All changes refer to the definition of the table head and the size of each column. Regarding changes to table contents use should be made of command "Edit Part List" with the right mouse button. The window of this new command is shown in Figure 10.

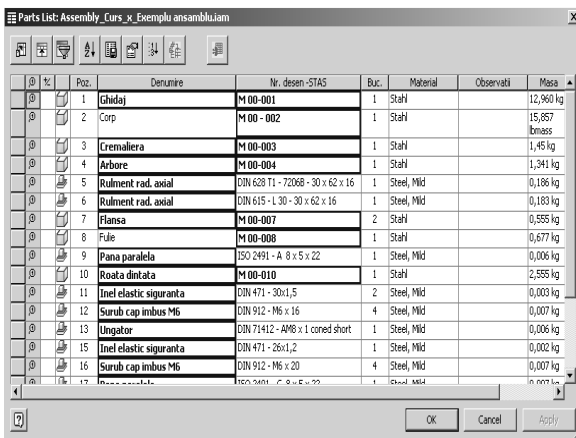


Fig. 10 „Edit Part List” window

All these changes of table content apply mainly to the number coding of the execution drawings and the name of each and every standardized item, which will be renamed in Romanian.

6. CONCLUSIONS

The conclusions regarding the aspects dealt with in this paper refer to the following advantages to the design activity brought about by the tools made available to the engineer by Inventor 2016:

- Possibility of inserting files created with other software in terms of benefiting from AnyCAD technology which modification directly from the native software automatically take over the changes to the Inventor-type assembly;
- 3D constraints were optimized;
- The possibility of constraints to joints.
- The software shows a library of extremely varied standardized items covering the most common standards;
- The windows for introducing standardized items in the 3D assembly are optimized;
- Changing the style of the component table/parts list and its content is very easy.

7. REFERENCES

- [1] Petru Dumitrache - *3D parametric modelling dedicated to group technology* – JIDEG , Volume 10 Special Issue 2015, fascicle 3, ISSN1843-3766
- [2] Haraga G., Ghelase D., “ *Creating of the primitive Shapes using AutoCAD*”, The Annals of “Dunarea de Jos” University of Galati, Fascicle XIV, Mechanical Engineering, CNCISIS code 220, „B+”, Indexed Journal BDI-CSA (Cambridge Scientific Abstracts) ISSN 1224-5615, pp. 95-98, 2009.
- [3] Goanta A.M. – “ Aspects on Transfer of Aided – Design Files” International Conference ModTech 2016 - Modern Technologies in Industrial Engineering, 15-18 June 2016, Iasi, Romania, publish by IOP Conf. Series: Materials Science and Engineering 145 (2016) 042001 doi: 10.1088/1757-899X/145/4/042001, pp. 1-6 electronic version.
- [4] Haraga G. “Modeling eco-friendly products in CAD systems”, ICEGD International conference on engineering graphics and design, Scientific Bulletin of the "POLITEHNICA" University of Timișoara, Transactions on Hydrotechnics, ISSN 1224- 6042, Tom 58 (72), Fascicola 1, CNCISIS code 251, B+, pp. 45-48, 2013, 13-15 June, Timisoara, Romania, 2013, http://www.ct.upt.ro/icegd2013/ICEGD_2013.pdf
- [5] Stancescu C. – Modelare parametrica si adaptiva cu Inventor – Editia a II-a, Bucuresti, Editura FAST, 2014, ISBN 978-973-86798-8-7.

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