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MODERN ANNOTATE AND EXPORT INSTRUMENTS IN 3D PDF FILES

Abstract: *The paper presents two novelties with respect to INVENTOR 2018, namely: direct 3D model sizing, adding prescriptions on roughness, shape and position deviations, including the ability to export the 3D model to PDF files to allow 3D rotation commands , obtaining sections, shade views, wireframe, etc. The modernity of these tools leads to the obtaining of PDF files independent of the IPT native file, which further allow for an efficient communication between the designer and the respective operator.*

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

1. INTRODUCTION TO ANNOTATIONS

At present, the annotation issue is different from software to software, but there is a high level of performance achieved by almost all existing software in the field of design. If over 15-20 years ago the design was mostly 2D and in some 3D, nowadays most design software are primarily in 3D and get the 2D documentation later, through commands that project the solid according to the plans work defined in three-dimensional space. To these projections of view or section type, quotas/sizes, deviations of shape and position, rugosities, thermal treatments, technical notes, etc. are added. The current differences [1], between the various software on the market, are not very numerous, but there is still a differentiation, namely how to pass on the information gathered to the next stage of the design process. Specifically, there is middle - class software that use two types of files, with different extensions [2], one for 3D type information, and another for 2D annotation information that has the property that the 3D file automatically determines the 2D file. Thus the correspondence from 3D to 2D is univocated. Changing the name of one of the so-called "linked" files necessarily involves restoring the connection, otherwise the software displays warning messages that it cannot find the link between 2D and 3D.

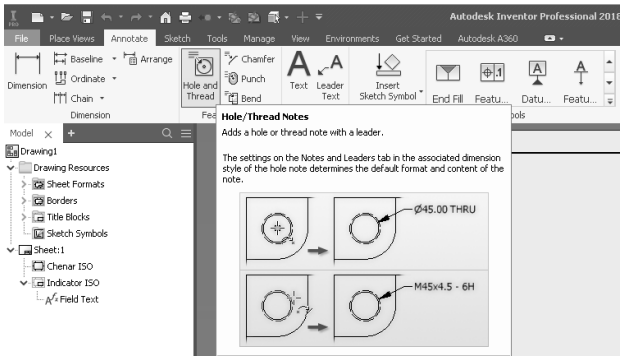

Fig. 1 The main 2D annotation commands

This category includes software like Solid Edge, Autodesk Inventor. The most common 2D annotation commands in Inventor 2018 [3] are: Dimension, Baseline, Ordinate, Chain, Hole and Thread, Text, Text Leader, Symbol, etc. shown in figure 1.

Figure 2 shows an example of a 2D drawing with all the necessary annotations.

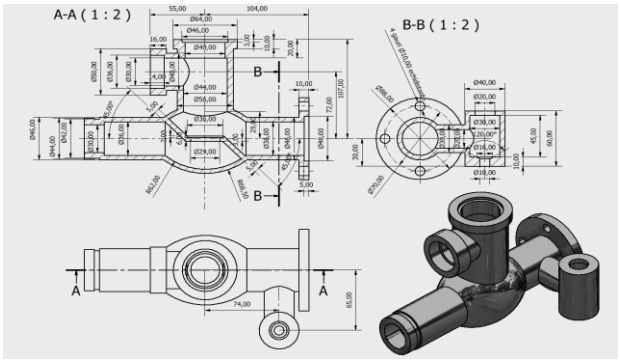

Fig. 2 Example of 2D drawing with annotations [4]

Unlike the above case, there are exceptional cases when 3D and 2D information is found in a single file, as is the case with NX Siemens software. The most important fact in this situation is that this unique file contains 3D, 2D, CAE and CAM data.

2. DESCRIPTION OF THE MODERN METHOD OF 3D FILE ANNOTATION

Theoretically speaking, the axonometric representation, which is a 2D and half-type representation, manages to transmit much easier and faster information about the shape and dimensions of the designed piece compared to an execution drawing that contains 1-3 orthogonal projections of view or section type.

Fig. 3 A 3D annotation tool

Even if achieving an axonometric projection is extremely easy using the tools existing in 3D design software, providing the allowances and adding the necessary annotations may become quite difficult.

2.1. Types of tools

Working with annotations for 3D files is totally different from adding annotations to 2D files. Figure 3 shows the tools for annotations in 3D files. When comparing to annotation commands related to 2D files, the difference between them becomes obvious. The tools that allow work with annotations on 3D files are: *„Tolerance Feature", „DRF", „Tolerance Advisor", „Dimension", ‚Hole/Thread Note", „Surface texture", „Leader Text", „General Note", „General Profile Note"*. For example, figure 4 shows how to start launching the *Dimension* command, at which time the software asks the user what standard should apply to use this command.

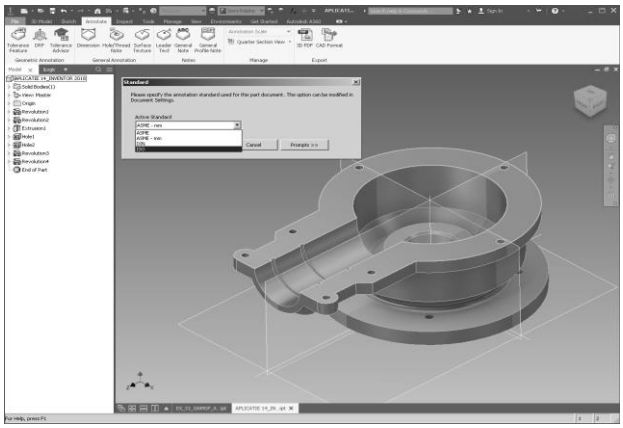

Fig. 4 Start of Dimension command in 3D [4]

Depending on the selected element, the response of the command is a linear dimension of the distance or arc type, the diameter of the circle. The quotas are added with "*Dimension*", specifying that both the cylindrical surface and the circle generating cylindrical surface can be selected, the effect being in this case the diameter of the circle or the diameter of the cylinder. If a flat surface is selected followed by another, then the quote/size refers to the angle between the surfaces, and if only one edge is selected, the dimension refers to the length of that edge. If a hole is desired to be sized, the "*Hole / Thread Note*" command can be selected to provide information on both the hole diameter and its type, i.e. whether it is punched or *Counterbore* with the addition of specific quotas/sizes.

Adding the shape or position deviations is done with the "*Tolerance Feature*" command, as shown in figure 5. To add a position deviation, it is important to ensure that at a previous stage the reference surface has been marked with a capital letter. For example, if the surface selected and marked with "A" in figure 6 must have the longitudinal axis parallel to the sliding path at the bottom of the mobile device [5], the "*Tolerance Feature*" will be used, the "//" symbol will be selected from the command set and "*Toggle Date Feature*" will ticked to define a new reference base to be used further. Depending on the position tolerance type chosen, the software has the ability to automatically select the correct baseline option, and if there is more than one correct option, they are available to the designer to select one of them.

By adding some shape deviations [6] and the implicit creation of a set of reference bases, they can be used, as the case may be, for the inscription of the position deviations.

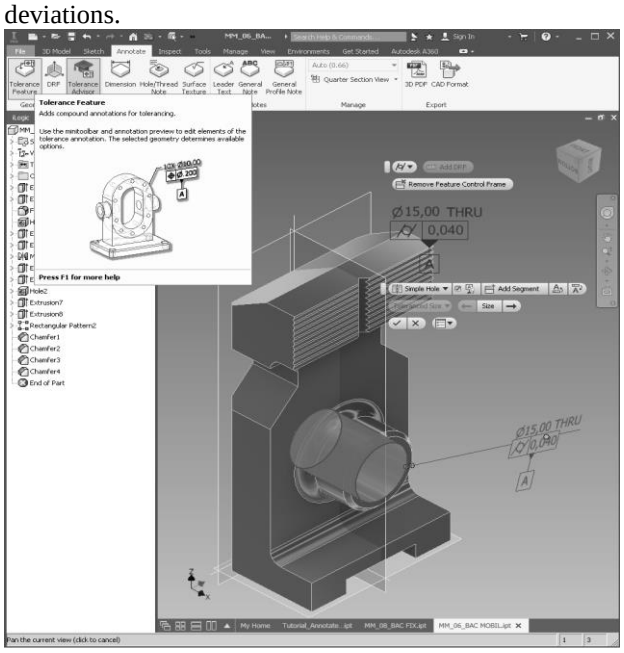

Fig. 5 Adding shape deviations

Working with these tools basically consists in the initial definition of the reference bases.

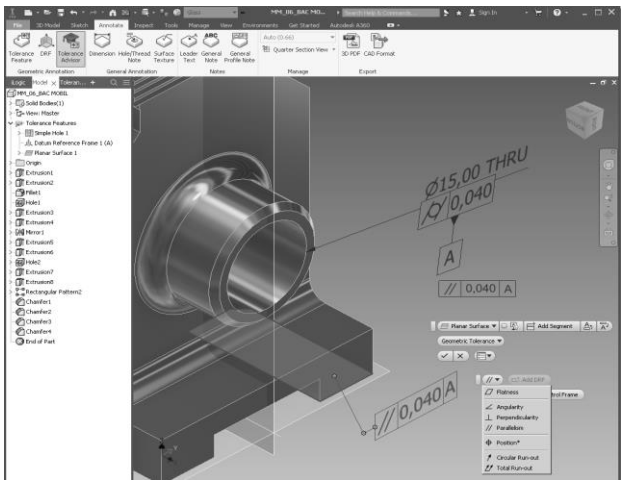

Fig. 6 Adding a position deviation to base "A"

If it is needed to add inscriptions on thermal treatments [7] or other special prescriptions, use can be made of the *Leader Text* or *General Notes*. In conclusion, this toolkit allows obtaining 3D representations that contain all the information about the manufacture of the respective piece exactly as in the 2D documentation. The advantage is that data interpretation is done much easier when the file reader's view in space is not very developed. All these annotations are added in the "*Model*" Browser as shown in figure 7.

2.2. Insertion and editing features

Each tool has its insertion peculiarity, but only a few of them appear in the paper. For example, when using the "Dimension" command, the user is allowed to either select the plane in which the quota/size is to be placed, or

by successively pressing the "Spacebar" button one of the planes where the size is placed is selected in turn.

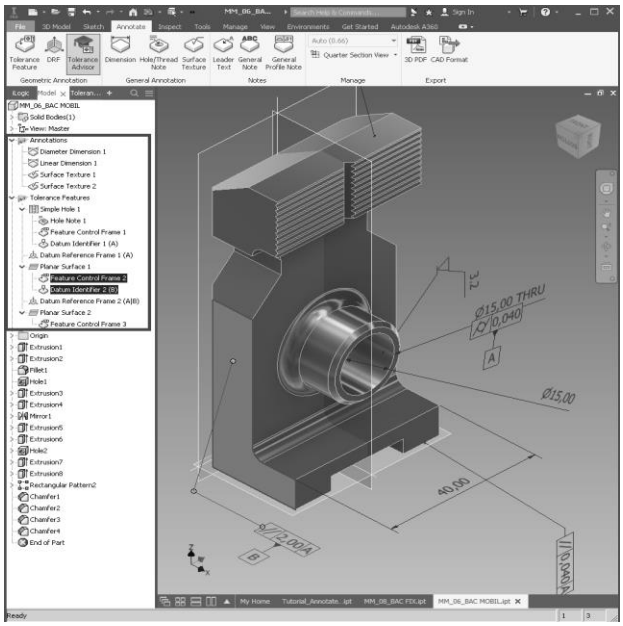

Fig. 7 Adding the annotations used in the "Model"

This way of working is accessed by pressing the right mouse button and is shown in figure 8.

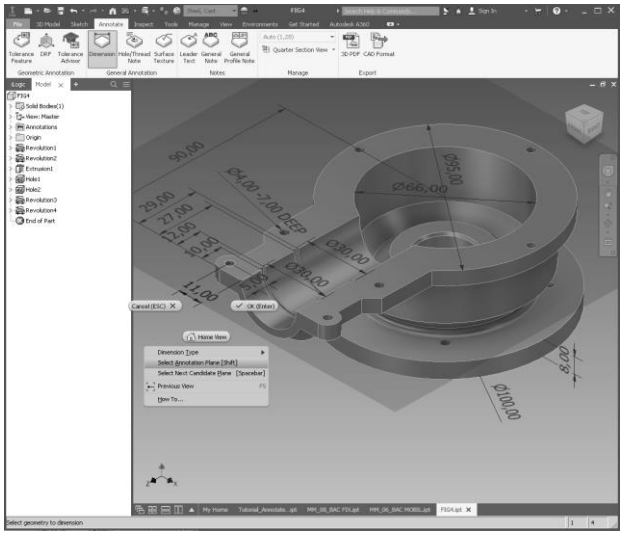

Fig. 8 Selection of quot/size layout

Another example is to edit the roughness value by using the right mouse button and selecting the "Edit" option that opens the window of both roughness symbol and its value. This detail is shown in figure 9. Also in this figure it is noted the quite large number of annotations in the "Model" browser.

3. EXPORTING INFORMATION BY PDF 3D FILES

Communication between different software users has been a priority for software developers, because since the first CAD works, it has been focused the problem of transferring information through exchange files between designers and designers and manufacturers.

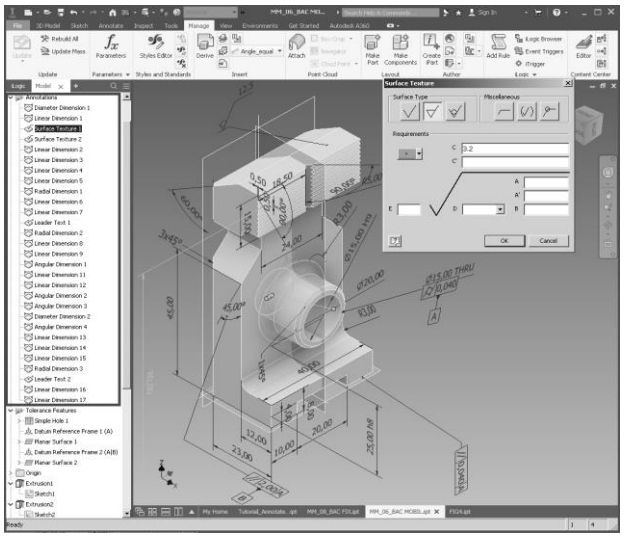

Fig. 9 Editing the roughness symbol and its value

The technical level and the different equipment standard of software or manufacturing equipment have always negatively affected the collaboration between the participants in the design or manufacture of new products.

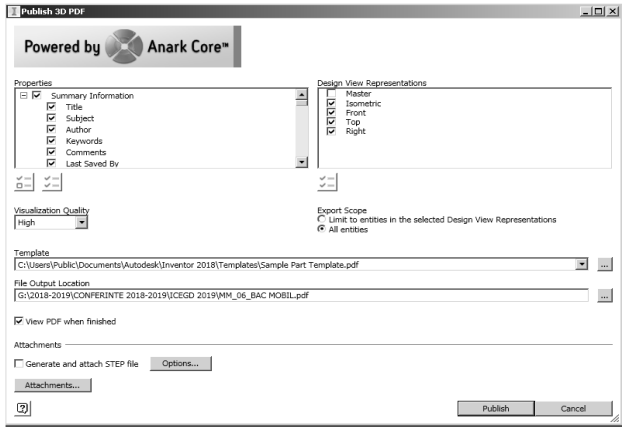

Fig. 10 Interface of „3D PDF" command

As a result, standard transfer files have been defined to allow for better collaboration between different software users. Out of these, it can be mentioned the most important IGES, SAT, STEP, etc. but underlying that they do not allow the recovery of the history of the 3D feature generation operations. Since 2016, Autodesk has introduced AnyCAD technology that states that it allows any native file to be opened in other software, while keeping direct connection with the source, meaning that any changes to the source file are automatically passed to the Inventor "*destination"*. Starting with Inventor Professional 2018, the 3D PDF export option has been introduced, which allows to generate a PDF file that can be opened with the current version of Adobe Reader Acrobat, but containing the 3D file from which it has all started. When launching the

"3D PDF" command in figure 3, the interface shown in figure 10 will open.

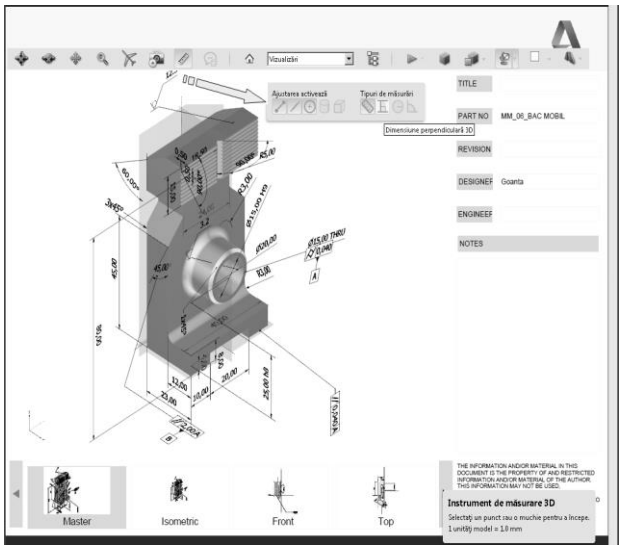

Fig. 11 3D PDF Files

The novelty elements partially seen in figure 11 brought about by this new PDF file are: 3D view of the piece, three-dimensional rotation, movement, magnification, orthogonal or isometric viewing tools, various rendering modes of the 3D model, various sources of light, multiple background colours, and last but not least, what is the most important, the possibility of creating sections directly in PDF and measuring distance between points.

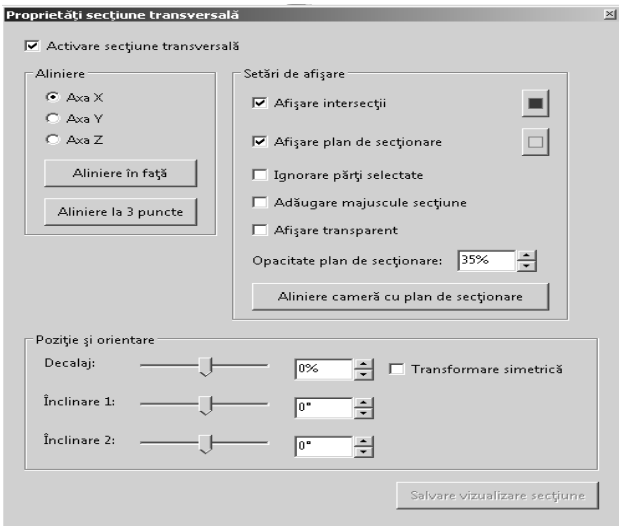

Fig. 12 Window with section plan settings

Figure 12 shows the window where the positioning of the section plane applied directly to the 3D PDF file is made.

4. CONCLUSION AND ACTUAL TRENDS

Virtually all of these 3D viewing and measurement tools included into the 3D file provide a new insight into the communication between the designer and the performer. Additional comments and annotations can be added to the PDF file by each of the participants in the remote dialog. The ability to perform high-resolution zooms combined with the rotation of the three-dimensional object removes any geometry lack of clarity at detail level. The most impressive tool of the moment is the 3D PDF export of the assemblies made in Inventor 2018 as shown in figure 13.

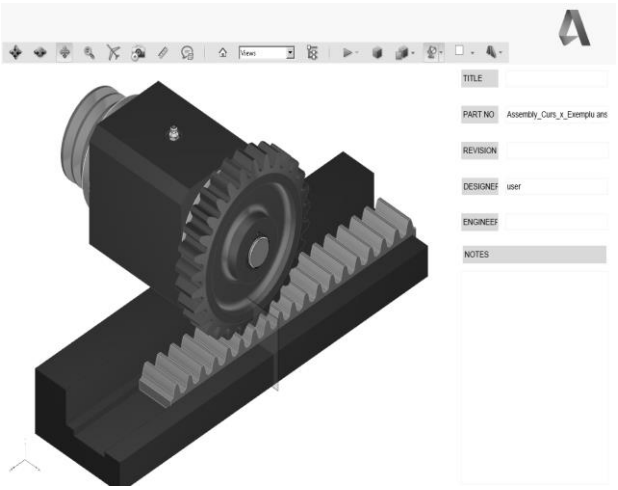

Fig. 13 3D PDF assembly type file

IT Evolution is so fast that in a very near future, design software performance at one time will allow open access to all native software tools by any other top class competitor software.

5. REFERENCES

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