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COMPARATIVE ASPECTS OF PROFESSIONAL INVENTOR 2018 VS. 2020 ON OBTAINING 2D DOCUMENTATION

Abstract: *This paper presents a brief description of the novelty elements that come with Inventor Professional2020 compared to Inventor 2018 applicable in mechanical design. It should be mentioned that, some of the most common commands for 3D modeling and obtaining 2D documentation were chosen, in which the similarities and differences were studied, by applying them to the most representative pieces in [1], the specialized documentation. Works published by colleagues from the department [2], [3], [4], [5] were selected for the study. The choice of the two versions was based on the current situation of the existence of educational licenses for both versions in the Faculty of Engineering and Agronomy in Braila. The second part of the paper presents some aspects on how to achieve 2D documentation using Inventor Professional 2020.*

Key words: *Design, CAD, modelling, place views, 2D documentation*

1. GENERAL ON 2D DOCUMENTATION

The evolution of the computer- assisted methods of obtaining 2D documentation has been growing faster as the hardware component has also undergone continuous and upward development. If, in the early years of the mechanical design, when assisted by computer only meant drawing line by line and element by element followed by the completion of each projection by attaching the corresponding quotas, at this moment computer-assisted design involves several modules such as CAD, CAE, CAM, PDM, PLM etc. each having a distinct tradition in its evolution.

If we refer only to CAD, nowadays there are two well-developed forms of assisted design, namely the parameterized and the synchronous one.

The first is the most widespread in the world and has been taken over and developed by all software companies, while the second [6], [7] is implemented exclusively by SIEMENS and Dassault Systems.

Regardless of the use of one of the two forms of 3D modelling, obtaining 2D documentation is similar. Namely there are files with distinct extension containing orthogonal projections, sections, details starting from the three-dimensional model achieved by one of the above mentioned methods.

The quotation is made subsequently with specific tools that allow the updating of the numerical values with each dimensional modification of the geometric model.

The current general situation implies the existence of two files, one for the 3D model and the other with 2D information.

There is only one exception, namely the SIEMENS NX software, where all information can be found in a single file, including those of CAE or CAM type.

The emergence and development of computer-aided design and manufacturing (CAD / CAM) originates from the introduction of automatic systems to monitor and control the production processes. In our country the current level reached especially by the big private companies is CAD / CAE / CAM, in some situations reaching even PDM, PLM.

2. SIMILARITIES AND DIFFERENCES BETWEEN THE INVENTOR'S 2020 AND 2018 VERSIONS

As we have already become accustomed to, every March of the year the version for the following year appears. This mode of occurrence has already become a tradition in the software design world, even though the actual moment in the month may be different from one company to another.

2.1. Similarities and differences in terms of system requirements

From Table 1 it is very clear that Inventor Professional 2020 has significantly higher system requirements than the 2018 version. This fact draws the user's attention to the purchase of a new PC or the need to make at least one PC up-grade PC in terms of RAM and video card.

Table 1

Similarities and differences

	2018	2020
Operating System	▪ Windows 7 SP1 64-bit, ▪ Windows 8 / 8.1 64-bit, ▪ Windows 10 64-bit	• Windows 7 SP1 64-bit with SP1 • Windows 10 64-bit 64-bit
CPU	**Minimum:** Intel Pentium or AMD based	**Minimum:** 2.5 GHz or greater
Memory	**Minimum:** 2GB	**Minimum:** 8 GB RAM for less than 500-part assemblies
Disk Space	**Minimum:** 40GB free space	**Minimum:** Installer plus full installation: 40 GB
Graphics	**Minimum:** 256MB	**Minimum:** 1 GB GPU with 29 GB/S Bandwidth and DirectX 11 compliant
.NET Framework	.NET Framework Version 4.5	.NET Framework Version 4.7 or later.

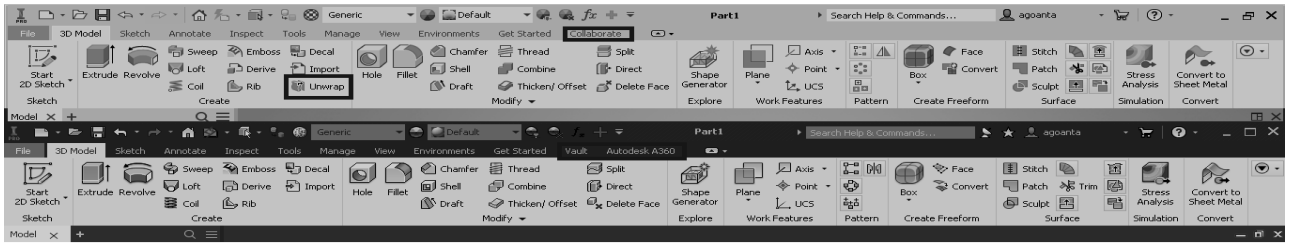

Fig. 1 Up/down start interface – Inventor 2020/Inventor 2018

2.2. Similarities and differences in terms of starting interface

As shown in Figure 1 the only differences at the start interface of both versions related to "Part" files are: "Unwrap" command and "Collaborate" tag. From the comparison of the other labels and panels we will find few new tools added to the 2020 version, namely "Export to DWF", "Shared Views" and "Migrate Settings". These will be the subject of a future work.

2.3. Similarities and differences in terms of basic commands „Extrude" and „Revolve"

If a quick analysis is made of the two start windows for the "Extrude" command, the first one for the 2018 version and the second one for the 2020 version of Inventor Professional, we find that both variants have the same facilities except that the window related to the 2020 version is optimized as to the number of clicks performed. In other words, the window from 2018 version has two tabs, one "Shape" and another "More", which can be found in a single version in 2020.

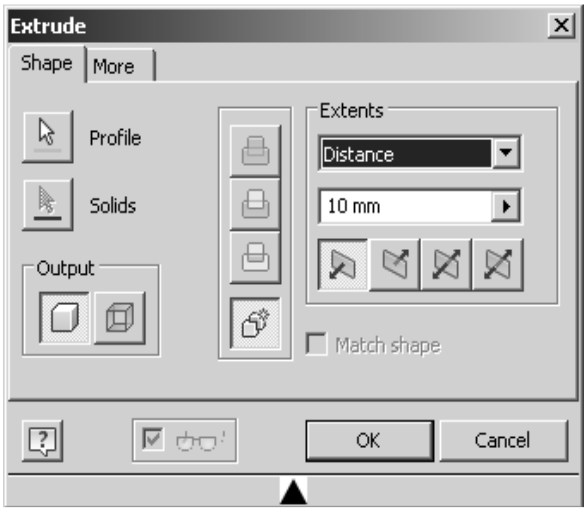

Fig. 2 „Extrude" windows of Inventor 2018

If a quick analysis is made of the two start windows for the "Extrude" command, the first one for the 2018 version (Figure 2) and the second one for the 2020 version (Figure 3) of Inventor Professional, we find that both variants have the same facilities except that the window related to the 2020 version is optimized as to the number of clicks performed.

In other words, the window from 2018 version has two tabs, one "Shape" and another "More", which can be found in a single version in 2020.

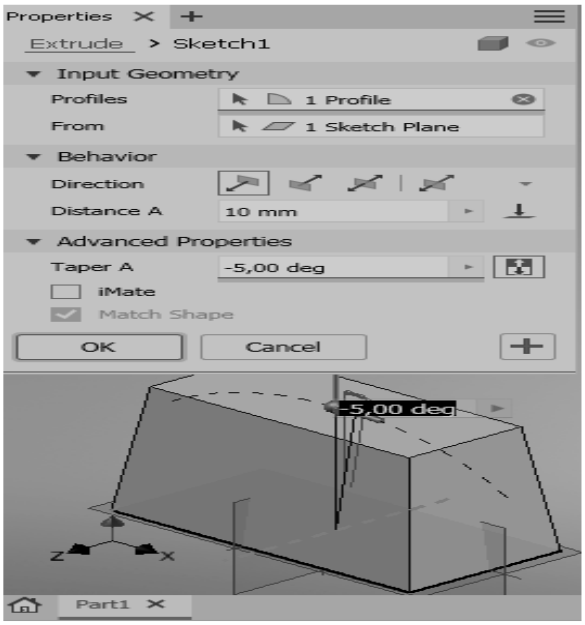

Fig. 3 „Extrude" windows of Inventor 2020

If we extend the analysis to the "Revolve" command, the previous conclusions are still valid. It should also be added that in the case of the selection between the solid and the generated surface, the 2018 version (Figure 4) seems more intuitive compared to 2020 one (Figure 5).

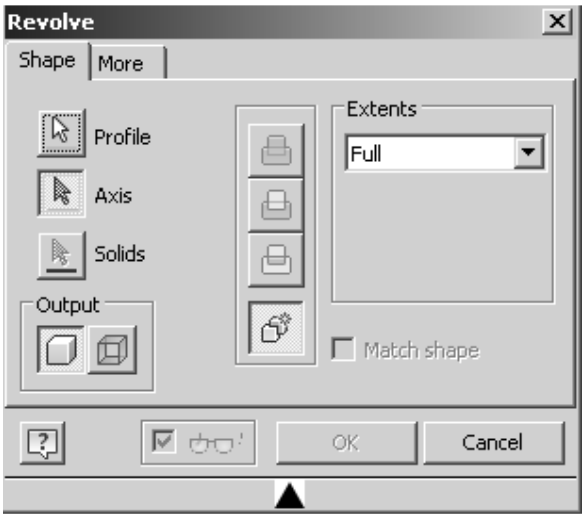

Fig. 4 „Revolve" window of Inventor 2018

3. ASPECTS ON HOW TO ACHIEVE THE 2D DOCUMENTATION

The starting point for obtaining the 2D documentation related to a piece [8] is the completion of the 3D file. To set an example, the file in Figure 6 below was used.

The steps for obtaining the 2D documentation are the following:

- Completing the 3D model.
- Defining a "Template" based on which all 2D drawings will be obtained.
- Creating the 2D file with the "Create New File" command, which at the end, when saved, will have the same name as the 3D file, but with different extension (Figure 7).

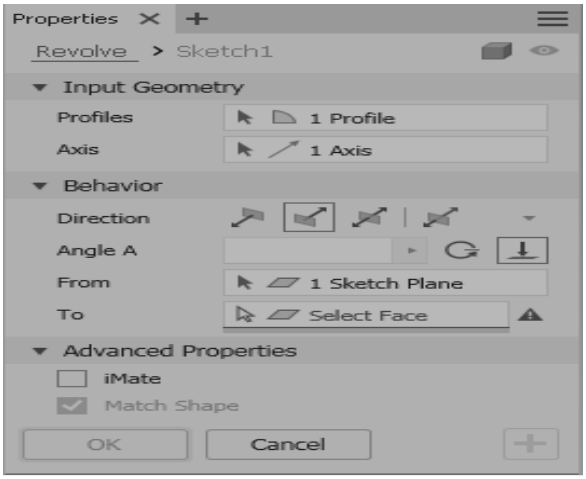

Fig. 5 „Revolve" window of Inventor 2020

- Obtaining the basic "view" type projection with the "Base" command.

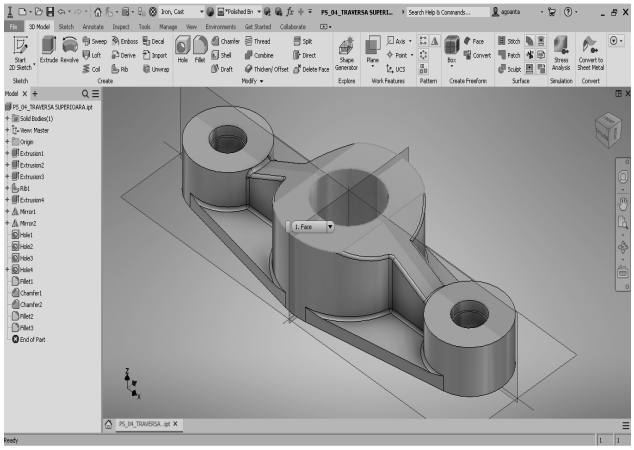

Fig. 6 3D model finalized to obtain the 2D documentation

- Editing the projection of the view type to obtain the fictitious edges or, as needed, the visualization of the covered edges.
- Obtaining the section-type projection with the "Section" command.
- Obtaining the new shadowed area.
- The isometric representation is obtained in the "Shade" variant.

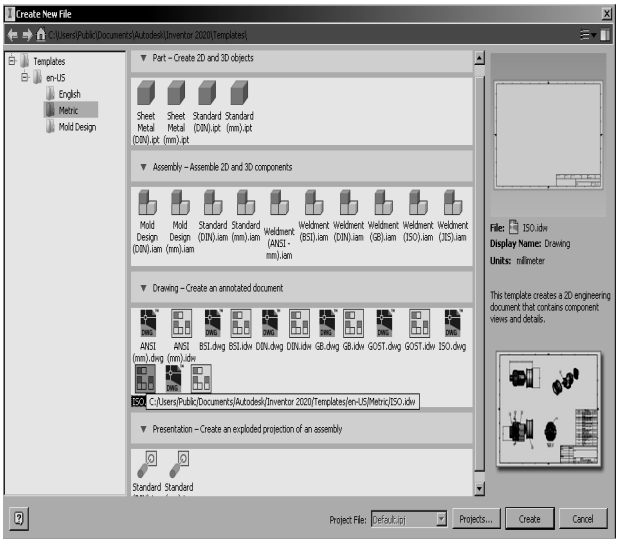

Fig. 7 Window of „Create New File" command.

- Editing the section-type projection by hiding the default section and creating with the command "Start Sketch" a new border for the section. It is especially applied to the representation of longitudinally sectioned stiffening nerves.
- Add all the necessary dimensions and symmetry axes, tolerances, deviations of shape and position, roughness, technical inscriptions.

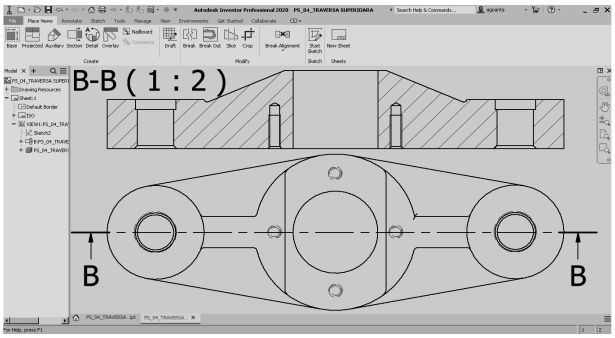

Fig. 8 View and section type projections

In the case of the piece shown in figure 6, we created the ISO.idw type 2D file with the command "Create New File" using as an "ISO template" a default file (Figure 7) and the view projection was obtained - with the "Base" command from the "Created" panel shown in Figure 9.

Figure 8 illustrates the two projections not edited namely, the horizontal projection of view type and the vertical projection of section type.

Figure 10 shows all the vertical and horizontal projections but edited according to ISO technical drawing rules. Additionally it contains an axonometric type shadow representation.

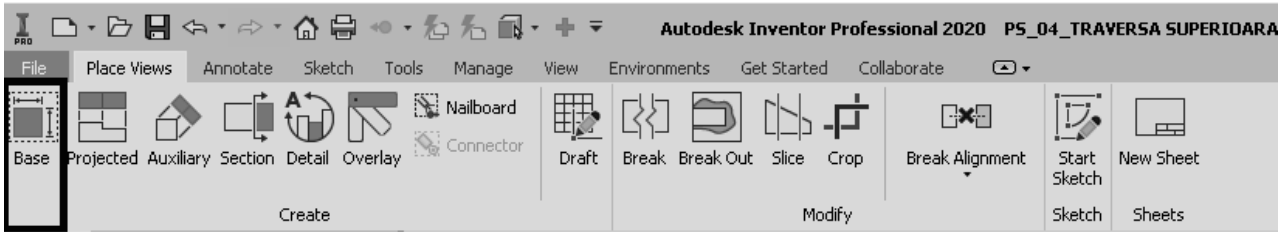

Fig. 9 Positioning „Base" command in the „Create" panel.

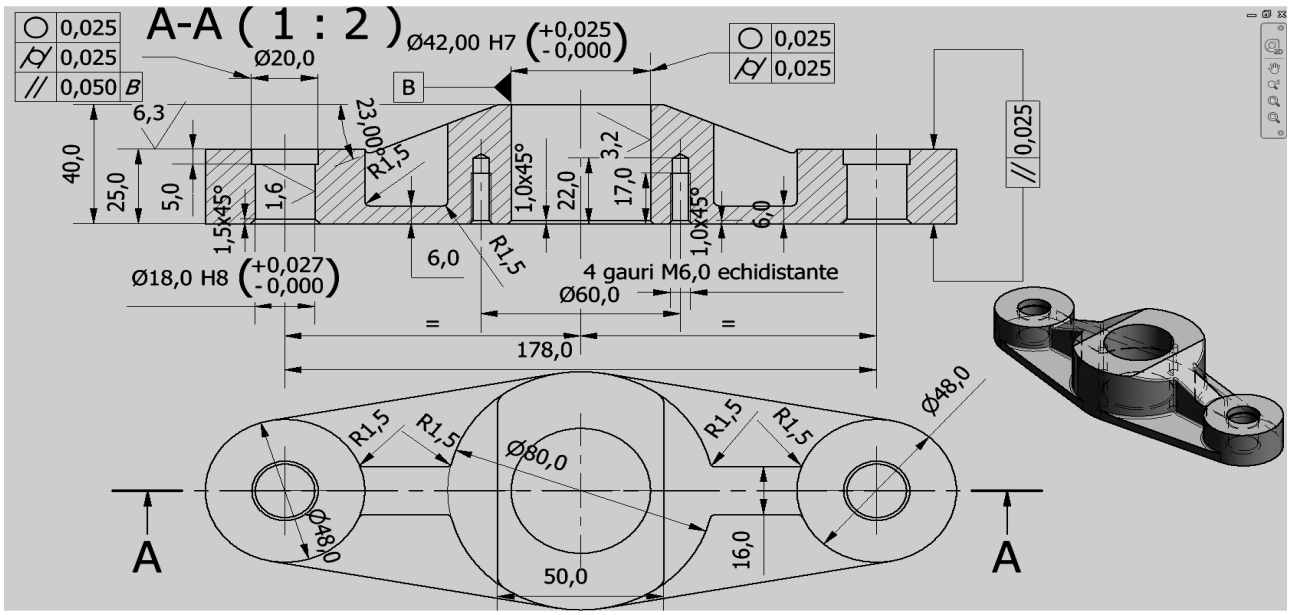

Fig. 12 The 2D drawing associated with the chosen solid.

In the last stage of the above classification is performed with the commands from the label "Annotated" presented in Figure 11. The final result of the edits for the 2D drawing of the solid chosen for this work is presented in Figure 12. 3D models made with other software with earlier versions can be opened by both Autodesk Inventor versions [7], [8].

4. CONCLUSIONS

At the end of the paper I present some conclusions regarding the differences between the two versions of Professional Inventor but also related to the univocal link between the 3D file and the 2D file made in any version of Inventor:

- There are significantly different requirements for the hardware resources necessary to run the two versions, but it should be noted that the tests performed used a laptop PC equipped with i3 2.3 GHz, with SSD, AMD mobility Radeon HD 5000 Series, 4 GB RAM , DirectX 11, 1 GB video.
- There are a few optimizations of the basic commands for performing as few clicks as possible. We can mention the case of the commands "Extrude", "Revolve", "Sweep", "Hole".
- There is a unique determination link between the 3D file and the 2D file, in that the changes at the 3D level automatically involve the updating of the projections and the corresponding quotas in the 2D file.
- In the case of changing the name of the 2D file, the connection with the 3D file must be restored using the "Replace Model Reference" command on the "Manage" tab, the "Modify" panel, which is to be found only in the 2D file.

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