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ASPECTS OF OBTAINING ORTHOGONAL PROJECTIONS AND INTELLIGENT INDICATORS BY INVENTOR 2016

Abstract: *The paper aims to preset how to obtain the usual orthogonal projection of view type, sections, details, and auxiliary projections, screenings breaks starting from the 3D model of the designed part and using the special tools offered by Inventor Professional 2016. Also it will be highlighted how to obtain intelligent indicators which completes part of the necessary information automatically. The paper is thus a landmark in the work of execution drawings particularly useful to those users who employ Inventor 2016 software.*

Key words: *Design, CAD, modelling, place views, inteligent border.*

1. THEORETICAL BACKGROUND

Projections arrangement is made according to the standards in force in each country or a group of countries which have uniform standards at a historical moment in their evolution. For this reason all design softwares [1] can work implicitly in both European system of projection and in the US one, but allow in their own settings a number of changes to the parameters in such a way that the graphic tool may adapt to any particular standard. In our country of orthogonal projections they are arranged according to the European system of projection (ISO), but Inventor [2] is installed in several deafult standards of obtaining projections such as: GHOST, ANSI, DIN, ISO, BSI, GB, each them in two variants of file extensions namely DWG and IDW.

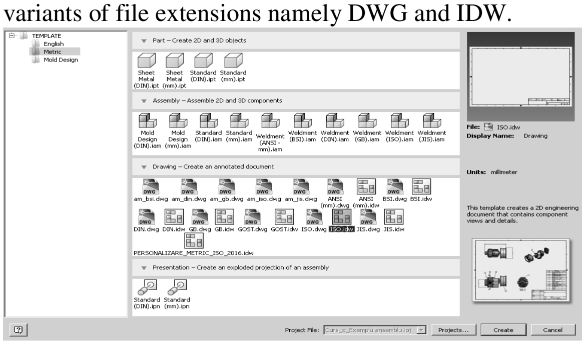

Fig. 1 Preinstalled Inventor Templates

As illustrated in Figure 1, there are 4 (four) categories of templates, namely: for 3D model corresponding to a single piece, for 3D assembly, for the execution or overall drawings [3] and last but not least for the overview exploded. In turn each of these can be customized according to the needs of the designer or company that uses them. Regarding the templates for obtaining drawings, the modifications permitted are those of the drawing format definitions, the arrangement of the frame and the way of defining and board positioning of the indicator. The indicator can be a normal one where the designer manually writes all the requested information or may be under the intelligent variant, in which the indicator automatically takes some information from the physical, mechanical, etc. properties as defined by the designer in the three-dimensional modeling stage. Making a drawing involves the following steps:

- defining a new template or choosing one from the existing ones;
- automatic setting the required size for drawing;
- automatic generation of necessary views and sections and choosing their layout;
- adding axonometric projections in one of 3 different view variants: only with visible edges, with visible edges and also covered or in Shade variant;
- plotting the axes of symmetry;
- sizing the execution or overall drawing;
- adding annotations on the drawing of the type roughness, size tolerances, tolerances of form or position, reference databases, technical notes, etc.

It should be noted that the file type *dwg* or *IDW* is permanently linked to the 3D model or 3D assembly. The link between the two files is not bi-univocal, but any change in the 3D file automatically lead to an upgrade in 2D file with a new opening thereof.

2. TYPES OF PROJECTIONS OBTAINED WITH INVENTOR 2016

The generating projection commands that Inventor may generate are grouped into *Place Views* tab of the corresponding Ribbon in 4 panels denoted as follows: Create, Modify, Sketch, and Sheet. The main commands are: Base, Projected, Auxiliary, Section, Detail, Overlaz, Draft, Break, Break Out, Slice, Crop, and Start Sketch.

2.1 „Create" Panel

Always begin with a projection type Base which will open the window in Figure 2 below. It should be emphasized that open window allows the user choose the 3D file to whom to draw the main projection, choosing the style of representation (Hiden Line, Hiden Line Removed, Shaded), establishing the name of the label to be attached to the projection, setting its visibility and choosing the representation scale. The window presents in turn three tabs: one that relates to 3D component to whom the projection is made, one called "Model State" and the last called "Display Options" which is especially useful in deciding how to represent the standardized threads and fictitious edges in the contact zones.

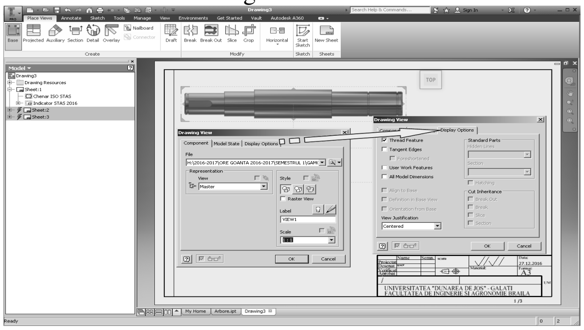

Fig. 2 Obtaining an independent projection of Base type

After obtaining the base projection one can move on to achieve orthogonal or auxiliary projections of view type by launching commands „Projected" or "Auxilary". The application is extremely simple in if with the first command mentioned, after indicating the baseline projection, the site of the "Projected" projection arrangement is selected while in the case of the two commands listed, it will be indicated in order the initial projection followed by an item of line/segment type that underpin the new system of reference by which to generate the new projection. *This second command starts from an existing project of "parent" type in which it is selected a direction given by a line / segment* that should actually be an actual edge or a contour of the "parent" projection, because there is no possibility of creating it artificially as in case of „Section" or "Break Out". It should be noted that there are specific situations when the auxiliary projection in lined with an element of line or segment type is to be a part of the whole piece, when it is necessary the application in particular conditions of the order "Break Out" in order to remove from the projection that portion which should not be visible.

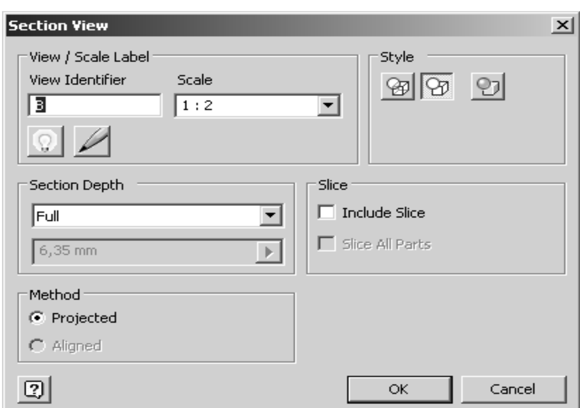

Fig. 3 Window imposing sectioning parameters

„*Section*" command generates sections starting from a basic projection, by drawing the sectioning path. Its application can become somewhat delicate, especially in case the section plan is drawn with some difficulty. In most cases applying the command "Section" assumes indication of the baseline projection, drawing the sectioning path, pressing the right mouse button and selecting „*Continue*", imposing the sectioning parameters presented in Figure 3 and finally indicating the place in which to draw the requested section. There are also particular situations when a sectioning path must be made with some inclination provided with the constraint to pass through a fixed point, as is the case shown in Figure 4.

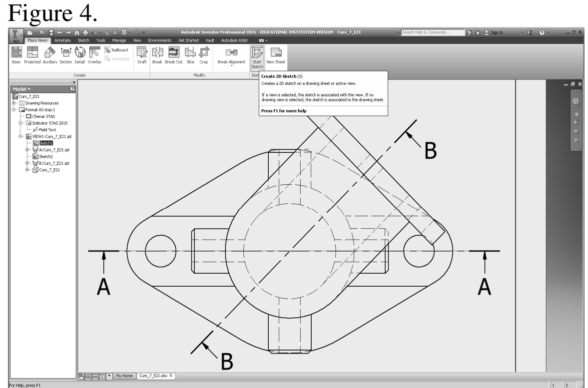

Fig. 4 Sectioning path inclined to 45^0

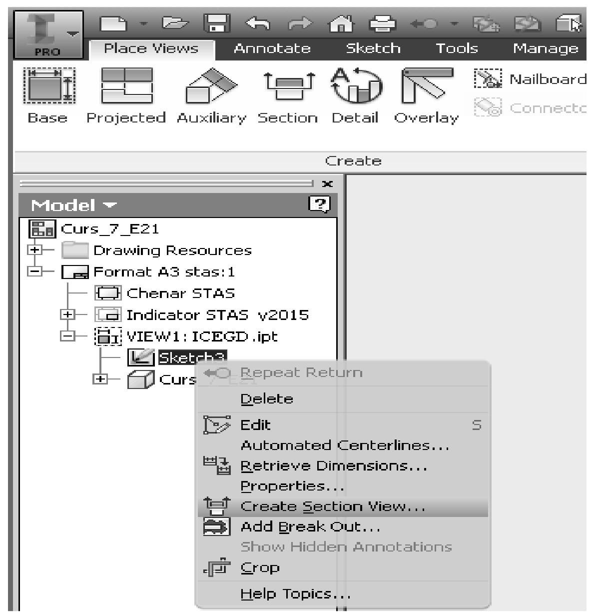

Fig. 5 Turning a sketch into a sectioning path

In this situation the classical sectioning command followed by the drawing of sectioning path can not be applied any longer because the tools provided by Inventor for constraints are not active. Consequently, the designer must define a new sketch projection over which he wants to draw the sectioning path, to impose the necessaery constraints, to complete the sketch and subsequently to declare it to the browser as route of sectioning/splitting, as shown in Figure 5.

The command „Detail" allows the extraction of details on the scale imposed for the less clear portions of projection obtained. After launching this command, select the projection „parent" which aims to extract the detail, followed by performing settings in the window visualized in Figure 6, indicating a point around which to make the cut, delimitation of the portion to be extracted, followed by an indication on the sheet of paper of the point where it will be placed. It should be noted that if the projection "parent" is of view type then the detail is a projection of view type as well, and if the projection „parent" is section type, then the detail is a section too.

Command „Draft" help create a special projection in the sense that it has nothing to do with the 3D model of the part being designed. Activating it opens the window in Figure 7, where it can be established the name of that projection and scale of representation and subsequently by confirmation „OK" it can be passed directly in the drawing environment.

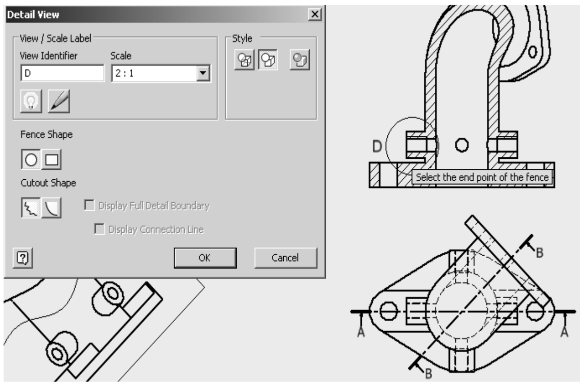

Fig. 6 Setting command „Detail"

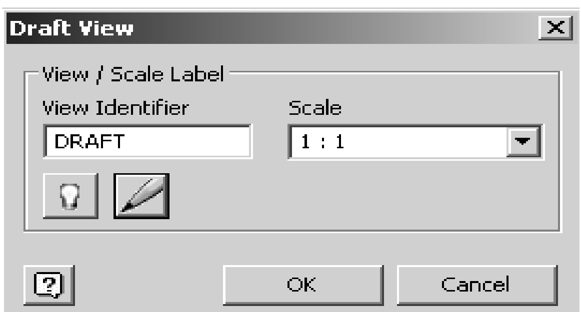

Fig. 7 „Draft" window

2.2 „Modify" Panel

The first instrument on "Modify" panel is the command "Break" which applies mainly to very long pieces to be represented so as to be easier to interpret and also allocating a smaller format but able to allow representation of all details of the item being designed. Next command "Break Out" is designed to perform ruptures, based on sketches made with closed contour by command "Start Sketch" launched before the application of the command. To explain the application, I go back to the individual cases mentioned above of the projections of "Auxiliary" type where a certain portion of the projection obtained should be removed or kept.

Therefore, I have to underline that for applying the command "Break Out" a sketch of type closed contour type is drwan as shown in Figure 8, but the depth required from the selected point is much higher than the depth of the piece, which lead to complete removal of the portion contained within the closed contour.

Command „Slice" has a somewhat narrower scope of application, it can produce multiple and successive slices in a hollow piece. To obtain such slices, the following steps maus be taken : previously drawing section lines on the basic projection, launching the command mentioned, selecting projection "Projected View" that will generate the desired slices (and deleting the initial projection) followed by the indication of only one of the lines set. It should be noted that at the end of this command all the lines being sketched will disappear from the basic projection.

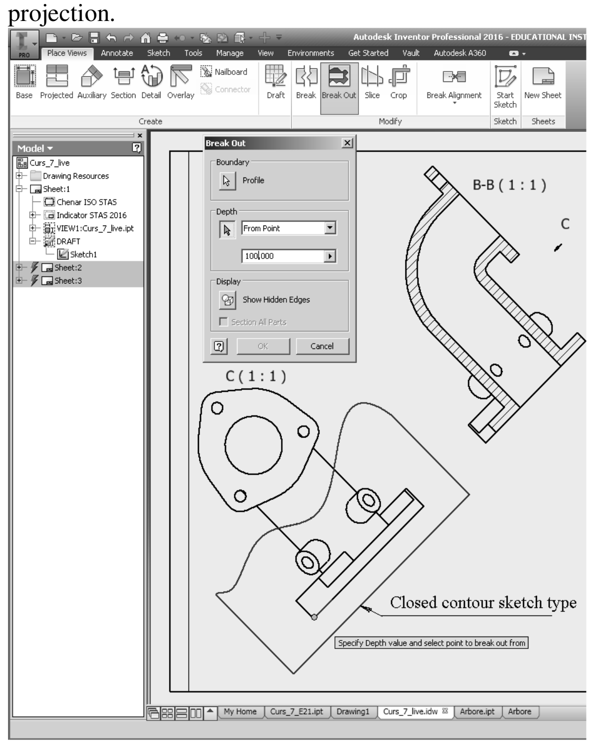

Fig. 8 The particular case of command „Auxiliary"

Command „Crop" is actually a variant of command "Break Out" with the difference that the sketch previously provided will keep the portion delimited from the projection. Also concerning the differences it must be mentioned: command "Crop" can be applied only on projections of "child" type not "parent", it is not allowed to specify the level of the cut depth, it allows the selection of the remarkable points such as MidPoint, Circle etc. and the outline/contour is just circular or rectangular.

Also in the Modify Panel there are to be found the tools to align or to remove the alignment from projections: "Horizontal", "Vertical", "In Position", and "Break Alignment". In the default variant, except for axonometric projections, all other projections are generated by Inventor in variants of vertical, horizontal or inclined alignments. If it is desired to break the alignment vertically or horizontally, apply the command "Break Alignment". This will finally allow translation on paper of the child type projection against that of parent type, but additionally it will add a new name text for the child projection and will mark with an arrow direction the viewing of the parent projection.

2.3 „Sketch" and „Sheets" Panels

These last two panels each contain a single command namely "Sketch" and "New Sheet", respectively. The first allows realization of sketches used to apply commands "Section", "Break Out" or "Slice" and the second leads to the generation of several worksheets. The simplicity of their application requires no further description.

3. INTELLIGENT INDICATORS

Inteligent /smart indicators [4] on the execution or overall drawings are based on their ability to extract and automatically register some information directly from the 3D model such as: the name of the maker of the 3D model, the name of the maker of the 2D drawing, the item name, the material the piece is made from, date when 2D drawing was made, general roughness, the drawing number, the total number of drawings, etc. In general all this information automatically retrieved and those entered manually are contained in a template file in whose Browser there are two categories of elements: "Drawing Resourses" (with subsections Sheet Formats, Borders, Title Blocks, Sketch Symbol) and "Sheet 1, 2, 3, etc. as shown in Figure 9.

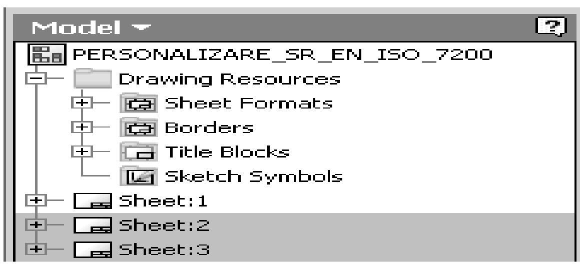

Fig. 9 Browser Template

The steps for creating the template are: frame editing, "Block" indicator editing, defining what information will automatically be retrieved, saving under a personalized name and copying in the template files directory of the Inventor

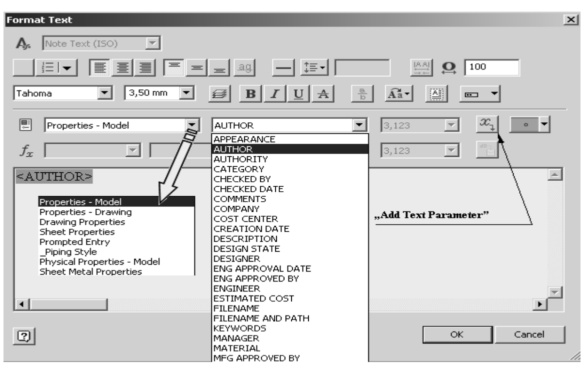

Fig. 10 W*indow for information automatic retrieval*

To reach the window in Figure 10 where properties and information that will be retrieved automatically in indicator are defined, the resource "Title Block" is edited with the right mouse button. As shown in this figure the various information grouped into s groups of properties like : Properties-Model, Properties-Drawing, Drawing Properties, Sheet Properties, Prompted Entry, Physical Properties-Model, etc. can be selected and added to the indicator by "Add Text Parameter" button and imposing the insertion site and the writing size.

4. CONCLUSIONS

The conclusions on the two issues addressed in this paper refer to the following benefits to the tool design activity made available to engineers by Inventor 2016:

- Obtaining rapidly and accurately the main projections of an execution drawing;
- Ability to edit each projection to account for every feature of the existing standards.
- Possibility to obtain extremely fast some very complex axonometric representations;
- Upgrading instantly when opening the IDW file for each change in the IPT 3D file;
- Increasing the responsibility of each participant of the design team through the use of intelligent indicators.

6. REFERENCES

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