Abstract: - The paper presents a software tool used for generating holes in AutoCAD environment. The application allows the user to generate different types of holes, on the parallelepipedic solids, starting from different references as edges, vertex, known points on the solids, or different types of entities which on the solids are placed. The user has the option to modify the hole position on the solid, and constraining it to the hole references. The software tool also has the possibility to multiply the generated holes in a controlled way, using four methods: rectangular, polar, honeycomb and wireframe.

Key-Words: - CAD, Intelligent object, AutoCAD solid, Mechanical part, Hole

1 Introduction
Generating holes depends on multiple facts, such as: references, the geometry of the surface on which the hole is generated, the hole type (threaded or unthreaded), if it is a standard hole or not etc.

It is wellknown that AutoCAD is Computer Aided Design software for 2D and 3D design of solids [5], which has the possibility to create industrial products because of great a variety of commands, which in AutoCAD are implemented. In opposite, AutoCAD environment does not contain specialized tools for holes modelling and design [2], which for being obtained is necessary to be used an important number of commands. The hole design in AutoCAD depends on the user skills and knowledge. Because of these was necessary to be developed a software tool specialized in hole design. The application is able to eliminate the all AutoCAD inconvenience in this direction, and which is able to simplify this step from constructive design of industrial products [1].

2 Presentation of HOLE_MOD tool
The software tool presented in this paper, named HOLE_MOD (HOLE MODelling), has a modular and compact structure, which presents a friendly interface in the dialog between the user and computer. It has been developed using VisualLISP/AutoLISP programming language [4] and OpenDCL dialog control language [6].

2.1 HOLE_MOD block scheme
In Figure 1 the block scheme of HOLE_MOD flow is presented.

In the upper block the input date are inserted, starting from data with basic meaning (the optimum coordinate system, the surface on which the hole is created), and continuing with particular data: reference type (based on the data which the user knows about the hole), hole type (threaded / unthreaded), hole geometry and finally the geometrical parameters.

These data are processed by HOLE_MOD tool and correlated with the selected hole type. The user can choose between 27 constructive hole types: 9 types of gole (figure 2 1-9), every type with 3 alternatives: unthreaded, full threaded, partial threaded (figure 2 a-c).

The data are processed by algorithms custom made for each hole type, as well as function of the selected alternative. As output data the correspondent hole is generated, with the inserted options.
The data which from the upper block are outputted, can be the output data of the HOLE_MOD, or can be the input data for another block, which for the fist block is subdued. With this second block, the user has the option to multiply in a controlled way the generated hole.

Fig. 2 Constructive hole types

2.2 HOLE_MOD tool description

It is well-known that a hole, for being functional, is necessary his coordinates to be known. These are being established with references, from which the hole are being constrained on the solid.

Starting from this reason, the application allows the user to move the Coordinate System in each vertex the user want to. For do this, is necessary just to press the UCS button from figure 3 (top-right on the HOLE form), and select an edge which will be coincident with OX axis. The origin will be the closest endpoint of the selected edge. Than, the user must select the references from which wants to report the hole position. For this, the user can choose between four alternatives.

Alternative 1 - References are a face of the solid and two edges, from which the hole is dimensioned. In this case the FACE button is pressed to allow the user to select the desired face on the solid. Then, the REFERENCE 1 and REFERENCE 2 buttons must be pressed, for enabling the user to select the edges from which the hole will be dimensioned (see figure 3). The result will be materialized in a hole on the solid, which will be normal on the selected face. The hole will be dimensioned according to the selected references. The dimensions will appear in the editing field correspondent to REFERENCE 1, and REFERENCE 2. After this step, the user can modify bought dimensions, changing the values in the editing fields. The dimensions will be updated by pressing the correspondent button on the right of the editing fields.

Alternative 2 - Reference is a revolution entity which on the solid is placed (see figure 4).

Fig. 3 Dialog box corresponding with LINEAR alternative

Fig. 4 Dialog box for CONCENTRIC WITH... alternative
In this case CONCENTRIC WITH… button is pressed, and after that the revolution entity must be selected. In the end, the hole will take the references of the revolution entity previously selected. The dimensions will be displayed in the correspondent editing fields.

Alternative 3 - Reference is a point with known coordinates, in current UCS (figure 5).

The point coordinates must be inserted in the correspondent entity fields, and then the POINT button is pressed. In the end, the hole is generated with the center in the correspondent point, dimensioned. As references will be the current UCS axis.

Alternative 4 – Reference is a point with unknown coordinates in the current UCS, but which can be inserted, and from which are known the coordinates to another point (see figure 6).

In this case FROM… button must be pressed, and then the user will select the point with unknown coordinates. After that, the user must write the relative coordinates at the inserted point, in the editing fields. By pressing WITH… button; the hole will be generated, dimensioned, with current UCS axis as references. The OK button appears in every dialog box and by pressing it the dialog box is closed after the hole is made.

The software tool enables the user to multiply in a controled way these holes, after four methods:

- **Rectangular** method, which has as input data the columns and rows number, the distances between these, and the angle at which the user wants to made the multiplication (figure 7);

- **Polar** method, which has as input data the coordinates of the point from which the multiplication is made, and after case, the number of holes and the angle at which they are displayed, the number of holes and the angle between they and, the angle at which the holes are displayed and the angle between they (see figure 8);
• *Honeycomb* method, which has as input data the number of rows and columns, and the distance between these (figure 9);
• *Wireframe* method, which has as input data a polyline on which the holes are generated and after case, the distance between holes, or the number of holes (figure 10).

![Fig. 9 Dialog box for HONEYCOMB method](image)

![Fig. 10 Dialog box for WIREFRAME method](image)

The command is launched by pressing the OK button from the right-bottom corner of the dialog box of every tab.

3 Case study
To illustrate the working manner of the application, a plate solid was chosen. On this plate different types of holes has been modelled, according to the figure 11. Figure 11 shows the plate before and after the holes generation.

In figure 12 the plate drawing is shown, with 6 classes of holes.

![Fig. 11 The plate before and after the holes generation](image)

![Fig. 12 Drawing plate with 6 classes of holes](image)

The hole types were been modelled in numerical order, with the following options:
• The first hole type was made with TO POINT option; first the UCS was moved in one of the plate vertexes. After that the gravity center dimensions were inserted, by having the dimensions on the drawing plate;
• The second hole type was made by CONCENTRIC WITH... option, this being concentric with the previous;
• The third holes type was made by LINEAR option, having as references the solid edges. After the first hole was made, the RECTANGULAR multiplying method
was used. The result was the four holes of the third type;
- The fourth holes type was made by FROM POINT option; in this case the known point was the plate center. After the first point was made, the POLAR method was used to multiply the last hole;
- The fifth holes type was made by TO POINT option, the multiply method being HONEYCOMB. As result were made four holes;
- The sixth holes type was made by FROM POINT option, and the multiplying method used was WIREFRAME. After the first hole was made, a profile was drawn, on which the multiplication was made. The first hole was in one of the profile endpoint.

4 Conclusions
HOLE_MOD software tool (HOLE MODelling) is very easy to use, developed in accordance with AutoCAD philosophy, has a structured form, and a great number of facilities for holes modelling. All these characteristics made HOLE_MOD an efficient software tool for holes modelling, for all the AutoCAD users.

5 Future works
In the next stage the software tool will be developed starting from the data obtained in the modelling holes process, a database will be generated which will contains information with associative data of the modelled holes [1]. After informatics methods that will be applied on the obtained data, technological information will be generated and associated to the initial holes, using the intelligent object concepts [3].

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References: