FURNIT - User Guide

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1. FURNIT

FURNIT is a program destined to furniture design, operating under AutoCAD at least 2013 .

You can project bodies composed from rectangular plates or plates defined by POLYLINE or CIRCLE.

Materials may be attached on plates. They may be graphically represented as real ones. For graphical representation, use the AutoCAD command "SHAdemode Realistic". You can then turn around furniture, using the AutoCAD command "3DORBIT". To see normally the furniture (by lines), use the AutoCAD command "SHAdemode 2Dwireframe". You can also use the AutoCAD RENDER command, to achieve realistic images of high quality level.

Technology represented by holes, grooves or edgings. can be applied.

On a plate or between 2 plates assemblings can be applied (hinges, pegs, handles, bars, etc.).

Furniture bodies can be composed of several plates and assemblings.

The plates, the holes, the grooves, the edgings, the assemblings, the bodies and the materials can be modified interactively and archived in order to be reused.

Execution drawings for plates, and the lists of materials, technology and costs are automatically generated.

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2. Steps

Type FURNIT to show the toolbars. As an effect, the chamber definition command is generated.

1) Define PLATES first:

- rectangular

- defined by edges (POLYLINE or CIRCLE)

- multiple plates along a POLYLINE, collocated in angle on the edging.

2) If necessary, EDIT the plates and put them suitably.

3) If you want to re-use them, put the plates in the LIBRARY (plates archive).

4) At your choice, you can see the plates from the current session with the function LIST PLATES.

5) At your choice, you can define the EDGINGS (CANTS) or generate HOLES or GROOVES at one plate.

6) At your choice, you can define or locate ASSEMBLINGS (to connect 2 plates, hinges, pegs, handles, bars, etc).

7) Define BODIES formed of more plates and possibly assemblings, using the function DEFINE BODY.

8) If applicable, EDIT the bodies and locate them in an appropriate manner.

9) If you want to re-use them, insert the bodies in the LIBRARY (bodies archive).

10) Optionally, you can see the bodies from the current session using the function LIST BODIES.

11) Draw and generate the lists of amounts with the function DRAWINGS, LISTS.

12) Go back to the status before drawing using the function ERASING DRAWINGS.

In the EDIT functions, when the introduction of a point or a length is required, the program goes automatically in the mode OSNAP ENDpoint. When you have to select an edge of a plate, the edge should be uncut (on all the gauge corresponding size and without fragments with grooves or holes). If not complete one new point of it is required.

If you have to select a hole, get a position on one of its circles!

Some menu functions make repeatedly the same operation. Leave these functions by pressing ENTER again.

The measuring units are: mm (lengths), sq.mm (areas), sexadecimal degrees (angles), kg (mass) and kg/cu.m (density). Area/1.e6 is actually sq.m. Money is that you like ($, EUR, etc.).

If you chose to make operations (UNION or SUBTRACT) between one plate or assembling FURNIT and AutoCAD bodies, the first selected body should be the FURNIT body or assembling!

The UNDO command in AutoCAD is only valid for the commands COPYING, ERASURE, MOVEMENT and ROTATION plates or bodies (not for holes or grooves)!

The modifications made in the materials libraries, grooves, holes, edgings or assemblings cannot have any effect on the existing plates, the bodies, or the assemblings! These shall only have effect on those defined after! The exception is the price of materials. If you change the prices of materials in the library of materials, a new DRAWINGS, LISTS command will take into account the changed values.

The drawn sizes of the plates do not contain the thickness of the edgings!

If you have difficulties in selecting a plate because its edges are covered by the edges of another plate, use the AutoCAD command DRAWORDER, FRONT or BACK!

3. Plates

3.1 Material

This helps to set the current material for the plates. The plates further defined shall be made of the current material. You can set the material by choosing it from a library of materials. Select the desired material and press OK.

You can add new material to existing ones (not in the shareware version). When you will enter the "SHAdemode Realistic" AutoCAD command, the material will be represented according to the "Render material" field from library. For a material name, you can put any value from "Render material"! You can also use the AutoCAD RENDER command, to achieve realistic images of high quality level.

The list of the materials from "Render material" is defined in the file "c:\furnit\ MATERIAL\ furnit_materials.dwg", that is automatically loaded in every DWG, when you load the FURNIT program. You can modify or add new materials in this DWG, using AutoCAD MATERIALS command. After changing this file, save it in the same place.

To see normally the furniture (by lines), use the "SHAdemode 2Dwireframe" AutoCAD command.

To load an image in the "Preview" area from the library, you must do the following:

- either you have a material image file (BMP, JPG, etc.) or you generate it using AutoCAD RENDER command and then you save it

- load this file with the AutoCAD IMAGE command, and then right mouse button and "Attach image"

- make a zoom of the image on the full screen, press MATERIAL (or MODIFICATION OF THE MATERIAL) in FURNIT, press SELECT and select the image.

The image of the "Preview" may not perfectly resemble that of AutoCAD. It is because AutoCAD uses only 256 colors in the dialogue area.

3.2 Color

This helps to set the current color for the plates. The plates further defined will have the current color. You can select it interactively.

If you want a "True color", you can use AutoCAD commands to define the color (for example, the PROPERTIES toolbar, then click SELECT COLOR and TRUE COLOR).

3.3 Rectangular plate by origin

This allows to generate a rectangular plate and indicate the position of its origin. You must indicate the type of the plate too: "Lateral" (|| with XOZ, L(length) after OZ, l (width) after OX and g (thickness) after YO), "Front-rear" (|| with YOZ, L-OY, l- OZ, g-XO) or "Horizontal" (|| with XOY, L-OY, l-OX, g-OZ). You must indicate then

the thickness, the width, the length, the name of the plate, and the position of its origin in space ("destination point").

The color and the material of the plate are set with the COLOR and MATERIAL commands.

3.4 Rectangular relative plate

It allows generating a rectangular plate by selecting 2 edges opposite to the side of an existing plate. A perpendicular plate will be generated on the defined side of the 2 edges, oriented from the first selected edge to the second; by the subsequent dialog and following the vectors appearing on the screen, you can indicate values for the sizes of the plate; negative values change the position of a plate in the opposite direction of the corresponding vector. You can also put plates in relation with the parallel sides with the parallelepiped of gauge of the plates resulted from a POLYLINE.

The color and the material of the plate are set by the COLOR and MATERIAL commands.

3.5 Plate defined by outlines

This helps to generating a plate starting from an exterior outline and, probably outlines defining cuts. The outlines should be defined previously by CIRCLE or closed POLYLINE entities, in the same plan, parallel with the plan XOY. The AutoCAD PLINETYPE variable should be set to 0 before describing POLYLINE entities.

You further have to indicate the thickness of the plate, the type: "Lateral" (|| with XOZ), "Front-rear" (|| with YOZ) or "horizontal" (|| with XOY); and the position of the origin in space ("destination point").

The color and the material of the plate are set with the COLOR and MATERIAL commands.

3.6 Multiple plates

This is generating more plates along a POLYLINE, joined in angle on the edging. POLYLINE should be in the plan XOY and contain straight lines! The AutoCAD PLINETYPE variable should be set on 0 before describing POLYLINE entities.

A POLYLINE should be selected and then indicate by a point the part where the program shall make the offset. You have also to introduce the thickness (between the POLYLINE and offset), the length of the plates (perpendicular on the plan XOY of the POLYLINE) and the generic name of the plates. At the generic name add for each plate the number of the corresponding segment of the POLYLINE (1, 2, 3,...).

Finally, the ensemble of plates can be displaced by indicating a destination point. The basic point is the first point of the POLYLINE. The color and the material of the plates are set by the COLOR and MATERIAL commands.

3.7 Library of plates

This helps to introduce, select or extract from the library of the plates.

Use the mouse to select an article of the displayed list.

With "New" you can define a new article name.

With "Edit" you can select an attribute (head of column) of an article.

With "Value" you can modify the selected attribute. Some attributes cannot be modified!

With "Select" you can select one or more AutoCAD entities associated to the current article. In order to take the maximum benefit from the "PREVIEW" area when you select an article from the library, modify the AutoCAD window to be the size of the gauge rectangle of the entities that you intend to select! At the library of bodies the selection of more bodies that can define a PIECE OF FURNITURE is possible.

With "Place" you can extract an article from the library, and you should indicate also its position (the insertion point) and the rotation angle.

With "Delete" you can delete an article.

With "Sort by" you can arrange in ascending order the displayed list of articles, depending on the selected attribute.

With "Type" the articles that are the selected TYPE are brought at the beginning of the list. These kinds of articles remain sorted by the setting in "Sort by". The TYPE is the last attribute of each article and helps for grouping the articles on types.

With "Print" the list of articles is printed in a file.

Leave the library with OK.

3.8 List plates

It allows the signal on the display of an existing plate in the current session. You can chose between total signal (by typing R) or individual (with a stop between 2 signals). If you type S you leave the command. The signal is made by drawing intermittent lines.

There are also temporarily drawn 2 vectors indicating the thickness and the width. Their vector product is the length.

4. Edit

4.1 Modification of the material

Helps for the modification of the material of a plate. Select first a plate. You can set the material by choosing it from a library of materials. Select the desired material and press OK.

You can add new material to existing ones (not in the shareware version). When you will enter the "SHAdemode Realistic" AutoCAD command, the material will be represented according to the "Render material" field from library. For a material name, you can put any value from "Render material"! You can also use the AutoCAD RENDER command, to achieve realistic images of high quality level.

The list of the materials from "Render material" is defined in the file "c:\furnit\ MATERIAL\ furnit_materials.dwg", that is automatically loaded in every DWG, when you load the FURNIT program. You can modify or add new materials in this DWG, using AutoCAD MATERIALS command. After changing this file, save it in the same place.

To see normally the furniture (by lines), use the "SHAdemode 2Dwireframe" AutoCAD command.

To load an image in the "Preview" area from the library, you must do the following:

- either you have a material image file (BMP, JPG, etc.) or you generate it using AutoCAD RENDER command and then you save it

- load this file with the AutoCAD IMAGE command, and then right mouse button and "Attach image"

- make a zoom of the image on the full screen, press MATERIAL (or MODIFICATION OF THE MATERIAL) in FURNIT, press SELECT and select the image.

The image of the "Preview" may not perfectly resemble that of AutoCAD. It is because AutoCAD uses only 256 colors in the dialogue area.

4.2 Color modification

Helps for the modification of the color of plate. Select first a plate. You can choose then its color, in an interactive manner.

If you want a "True color", you can use AutoCAD commands to define or modify the color (for example, select entities, use the PROPERTIES toolbar, then click SELECT COLOR and TRUE COLOR).

The indicated color becomes the current color.

4.3 Size modification

This is to modify the sizes of a plate, groove or hole.

PLATES

Select first the edge of a plate. You can modify the size || with that edge. The program draws a vector along the plate and displays its length. A new length must be introduced next. If the value is positive, the position of the vector’s vertex is modified. If the value is negative, the vector’s vertex remains unchanged, yet the position of the origin is modified. At the plates resulted from POLYLINE only the thickness can be modified.

GROOVES

After the selection, the system of axes XiYi of the groove will be automatically drawn (see Grooves). Indicate then: width, depth, length, angle, and price per meter of the groove. The implicit displayed values are for the selected groove.

HOLES

After selecting a structure of holes the system of axes XiYi will be automatically drawn (see Holes). Indicate then: diameter, depth, steps, angles and price of the holes. The implicit displayed values are of the selected structure of holes.

If you have to select a hole, get a position on one of its circles!

4.4 Modification of plate or assembling name

This allows the modification of the name of a plate or assembling that you select.

The plates receive names when defined, copied, or mirrored. The name is used when the plates are introduced in the library or the name is introduced at drawing (Drawings,lists).

If you selected an assembling, its price can be modified too.

The assemblings receive names when introduced in the library of assemblings, the name being used in the amounts lists.

4.5 Copy

Selecting PLATE or BODY

It helps to generate a copy in another position of an existing plate (or body). Select first the edge of a plate. The program will draw a vector along the edge. If you introduce a value, a copy displaced on the direction of the vector shall be displayed equal to the value. If you introduce a negative value de displacement shall be made in the vector's opposite direction. If, instead of a value, you select the option "Certain", a displaced copy shall be generated in a certain direction. For the displacement, a basic point shall be indicated on the plate and a destination point.

Selection of GROOVE

It allows the copying of a groove from a plate on another plate.

Select a groove first. Indicate then the surface of another plates where you intend to copy the groove: select the first edge and the second one after that. If the first edge does not belong to the parallelepiped of gauge (which is frequent for plates resulted from din POLYLINE) then a second point is requested, the program passing automatically in the mode OSNAP END. In this case, the axis Xi has the origin in the second point and the extremity in the first modified point by OSNAP MIDPOINT (it is in the middle of the selected edge).

The copy is made by the projection on the surface of the new plate, in the direction perpendicular on the surface with the initial groove.

Selection of a HOLE

It allows the copy of one or more structures of holes from a plate to another plate.

Select first one or more structures of holes. Indicate then the surface of another plate where you will copy the structures of holes: select the first edge, then the second. If the first edge does not belong to the parallelepiped of gauge (which is frequently met for plates resulted from POLYLINE) then you will be requested to introduce a second point, the program passing automatically in the OSNAP END mode. In this case the axis Xi has the origin in the second point and the extremity in the first point modified by OSNAP MIDPOINT mode (in the middle of the selected edge).

The copy is made by the projection in the axis direction of a hole on the surface of the new plate.

If you have to select a hole, get a position on one of its circles!

4.6 Erase

It allows the erasure of a plate, of the body, of the groove, of the structure of holes or the selected assembling.

4.7 Move

Selection of PLATE or BODY

It helps for the movement in another position of an existing plate (or body). Select first the edge of a plate. The program shall draw a vector along the edge. If you introduce a value, the movement on the direction of the vector shall be made, equal with the value. If you introduce a negative value, the movement shall be made in the opposite sense of the vector. If, instead of a value, you select the option "Certain", a movement in a certain direction shall be made. For the movement you have to indicate a basic point on the plate and a destination point. If you select for the "Alignement", the movement shall be made with a rotation so that three source points of the plate coincide with three corresponding destination points. In fact, only the source and destination points 1 will be the same. The source and destination points 2 and 3 shall define coincident directions (see the AutoCAD ALIGN command).

If you select "**Incremental**" you can make interactive rotations or movements, with a given increment by typing. First you should introduce the increment (in millimeters for movement and sexadecimal degrees for rotation). For the movement of the plate or the body you shall use the buttons:

- A,a for movement in the sense opposite of the drawn vector

- S,s for movement in the same sense with the vector

- I,i to introduce a new increment

- W,w for ZOOM WINDOW (increasing a detail)

- P,p for ZOOM PREVIOUS (previous image)

- - for ZOOM 0.5X (decreasing twice the image)

- + for ZOOM 2X (increasing twice the image)

- R,r to pass in the rotation mode by typing the buttons A, a, S, s

- M,m to pass in the movement mode by typing the buttons A, a, S, s

- any other button to exit from the incremental mode.

After each A, a, S, s button typing, the total value of the movement shall be displayed.

Selection of GROOVE

It allows the movement of a selected groove.

After the selection, the system of axes XiYi of the groove will be automatically drawn (see Grooves). Then you will indicate: the movement of the groove along the axis Xi and the axis Yi.

Selection of HOLE

This allows the movement of a selected structure of holes.

After selecting a structure of holes the system of axes XiYi of the structure will be automatically drawn (see Holes). Then indicate: the movement of the structure along the axis Xi and the axis Yi.

If you have to select a hole, get a position on one of its circles!

4.8 Rotate

This is destined to rotate a plate or body around a straight line. Select first the edge of a plate. The program shall draw a vector along the edge. If you introduce a value (in sexadecimal degrees), the rotation shall be made around the vector width an angle equal with the value. If you introduce a negative value, the rotation shall be made in the vector's opposite sense. If, instead of a value, you select the option "an Axis", the rotation is made around an axis defined by the user by two points.

If you select "Incremental" you can make interactive rotations or movements, with a given increment.

4.9 Mirror

It is destined to mirror a plate or body about a plan. Select first the edge of a plate.

The program will draw a vector along the edge. If you further select an opposite edge, the plan of mirroring will be already defined. If, instead of the opposite edge, you select for "three points", you should further define

the plan of mirroring by 3 points. Finally, you can also select between erasing or keeping the plate (body) you mirror it on.

4.10 Explode

Decomposes a body in the plates and possibly its composing assemblings.

4.11 Trim plate

It erases from the exterior of a plate's gauge the possible holes or grooves erroneously drawn and the add-ons of the AutoCAD bodies made with the UNION command!

4.12 Stretch

It lengthens or shortens a plate or a body along an edge.

It is selected first by a point an edge of an individual plate or contained by a body. Automatically and temporarily there shall be drawn the section in the plate or body, perpendicular on the edge in the point indicated at the selection. This section divides the plate or the body in 2 portions. It is also temporarily drawn along the edge a vector. The user shall further indicate what portion of a plate remains fixed (form unchanged), the portion that contains the origin of the vector or the other one. Then indicate the distance of extended (with '-' to shorten).

The plates that are section shall extend suiting the distance, keeping the form of the fixed portion unchanged. The structures of holes (if there are any) are automatically exploded in individual holes that should be moved, if they are in the extended portion. The depth or the diameter of the holes is not affected!

The grooves will be moved too, if they are in the extended portion. If the groove is on one side and the other of the sectioning plan, then, the part of it, which is in the extended part of a plate, will be extended too. In other words, the length and the width of the groove is extended too. The depth of the groove is not affected by the extension!

The edgings are affected by the extension too, the application length being modified too.

In the case of bodies, the plates or the assemblings in the part that remains unfixed, move along the selected edge, with the extension distance. The assemblings and the sectioned plates of the din body which are not parallel with the direction of tension are not tensed! These plates or ASSEMBLINGS shall be signaled by a message and by re-drawing temporarily with an interrupted line.

The plates defined by outlined are transformed by the extension on rectangular! If the extension distance is negative, the plates, the holes or the grooves passing through the tension over the sectioning plan will be eliminated.

5. Technology

5.1 Edgings

It is helpful to attach or modify the edgings on the sides of a plate. Select first one plate. The program shall draw temporarily 2 vectors representing the length and the width. Their product is a vector resulting from the plate. The length origin represents the LEFT, and the extremity the RIGHT of the plate. The origin of the width represents DOWN, and the extremity TOP.

It is also drawn temporarily, in the center of the plate, a scheme of emplacement and the type of the existing edgings.

In the dialog window appearing there will be displayed information about: the material and the thickness of the plate and about edgings (if defined). For each side you can indicate a type of edging, typing EDIT and selecting the edging from the library.

The mode of combination at the corners of the edgings can be selected by typing on the images of the corners of the dialog window.

The drawn sizes of the plates do not include the thickness of the edgings! The existence of edgings at one plate results from the lists of amounts and from the execution drawing of the plate (see Drawings,lists).

5.2 Grooves

It generated a reamed channel on a plate. In order to indicate the surface to ream, select first an edge of the surface. If the edge do not belong to the parallelepiped of gauge (frequent case at plates resulted from POLYLINE) then you will be requested to introduce a second point, the program passing automatically in the OSNAP END mode. In this case the axis Xi is originated in the second point and the extremity in the first point modified by OSNAP MIDPOINT (in the middle of the selected edge).

Further on select the second edge of the surface. The second edge can be in fact any right line of an AutoCAD entity. Thus an initial system of axes was defined (of index i). There is another system of axes of the groove (of index f) having Xf oriented along the groove length.

Then you can select with "Edit" a type of groove from the library of grooves or you can define a new type indicating: the channel width, the depth, length, angle of the axis Xf compared to the axis Xi and the price of manufacturing per linear meter of a groove. The width will be generated along the positive sense of the axis Yf. The width is above the axis Xf. If you choose for "Edit", select the wanted article and exit the dialog window with OK.

Then the position of origin of the axes system XfYf compared to the axes system XiYi shall be indicated, by the distance from the axis Xi and the distance from the axis Yi.

5.3 Holes

This generated a structure of holes on a plate. In order to indicate the surface of hole, select first an edge of the surface. If the edge does not belong to the parallelepiped of gauge (which is frequent for plates resulted from POLYLINE) then you will be required to introduce a second point, the program passing automatically in the OSNAP END mode. In this case the axis Xi is originated in the second point and the extremity in the first point modified by OSNAP MIDPOINT (in the middle of the selected edge).

Select then the second edge of the surface. The second edge can be in fact any right line of an AutoCAD entity. Thus, an initial system of axes was defined (of index i). The structure of holes has more rows of parallel holes. There is another system of axes of the structure (of index s) having the axe Xs oriented along the first row of holes.

You can select then with "Edit" a type of structure of holes from the library of holes or you can define a new type indicating: the diameter of the hole, the depths, the number of holes (or distance) on Xs, the number of holes (or distance) on Ys, the step (distance between holes) on Xs, the angle of the axis Xs from the axis Xi, the step on Ys, the angle of the axis Ys from Xi and the price of manufacturing one hole.

If you choose for "Edit", get a position on the wanted article and exit the dialog window with OK.

Indicate then the originating position of the system of axes XsYs from the system XiYi by the distance from the axis Xi and the distance from the axis Yi.

Optionally, you can make a single hole and you should introduce the number of holes 1 on Xs and Ys. You do not need to introduce in this case the distances between the holes and the angles of the structure’s axes.

6. Bodies

6.1 Define body

It is used to define by selection of the plates and the assemblings composing a body. The end of selection is made by typing ENTER.

6.2 Library of bodies

This helps to introduce, select or extract from the library of the plates.

Use the mouse to select an article of the displayed list.

With "New" you can define a new article name.

With "Edit" you can select an attribute (head of column) of an article.

With "Value" you can modify the selected attribute. Some attributes cannot be modified!

With "Select" you can select one or more AutoCAD entities associated to the current article. In order to take the maximum benefit from the "PREVIEW" area when you select an article from the library, modify the

AutoCAD window to be the size of the gauge rectangle of the entities that you intend to select! At the library of bodies the selection of more bodies that can define a PIECE OF FURNITURE is possible.

With "Place" you can extract an article from the library, and you should indicate also its position (the insertion point) and the rotation angle.

With "Delete" you can delete an article.

With "Sort by" you can arrange in ascending order the displayed list of articles, depending on the selected attribute.

With "Type" the articles that are the selected TYPE are brought at the beginning of the list. These kinds of articles remain sorted by the setting in "Sort by". The TYPE is the last attribute of each article and helps for grouping the articles on types.

With "Print" the list of articles is printed in a file.

Leave the library with OK.

"The depth" represents the gauge designed on the axis OX, "the height" on OZ and "the length" on OY.

"Price.mat." represents the price of materials of the plates composing the body. The price for holes, grooves, edgings, assemblings, and the total price result from the command Drawings,List.

6.3 Library of libraries

It allows the use of more libraries of bodies. In the first appeared window select or define (by the field "New") a library of bodies. Exit this window with OK. Further on a window specific to the set library will appear. When you define a new library in the first window, this library will have a sole article the first article from the main library of bodies BODY.

6.4 List bodies

It allows the signal on the display of the existing bodies in the current session. You can select between total signal (typing R), and individual (with a pause between 2 signals). Press S to exit the command. The signal is made by drawing intermittent lines.

6.5 Drawings,lists

This generates the execution drawings and the files with lists of the plates and eventually the assemblings of a body. You can choose between 2D or 3D drawing. Introduce first the name of the body. There is an implicit name if the body resulted from placing a body from the library. You will introduce then the sized of a sheet of draw in millimeters. For each plate, in the AutoCAD PSPACE mode will be generate automatically a window for each indicated sized, representing the execution drawing. If the plate results from POLYLINE, another window will be generated (called "p" after the number of the plate) where the outlined are defined. There is another ensemble drawing where each plate is counted automatically in the weight center. The number of the plate appears on the execution drawing too. On the ensemble drawing the assemblings are counted. The number of the assembling is used in the file described below. The color of the dimensioning shall be the current color for the plates.

In the case of holes and grooves, 'a' and the number that follows represent the depth.

In case of 2D drawing 's' indicated that the hole or groove are behind the plate, 'f' indicates the front of the plate, and 'p' and the number that follow indicate the depth of the axis of the hole or groove in the front of the plate (in case of holes or grooves made on edging).

By typing S you can set the following 2D drawing options:

- from the corner; the dimensioning is made from the nearest corner; or else, from the origin of the plate

- the holes depths; write the depth of the holes preceded by 'a'

- the groove depth; write the depth of the grooves, preceded by 'a'

- the holes number; write the number of holes in the structure

- every hole; each hole in the structure shall be quoted separately

- the separate drawing front-rear; for the plates with holes or grooves; 2 drawings will be drawn, on the first the holes and the grooves appearing (to the user) in front are draws, and on the second those that appear behind

- compulsory step structure; it works if "every hole" is set; if the indicated step is equal to the step on X or Y of a structure, the holes in this structure shall not be dimensioned separately.

If a plate has edgings, in the upper right part of its drawing will be drawn the sketch of location of the edgings.

Another file with the name of the body and the LST extension will be generated in the subdirectory C:\FURNIT\TMP. This file contains the lists of assemblings, of the plates, sized, materials, grooves, holes, edgings and prices.

For each plate another file is generated, its name being composed of the name of the body and number of the plate. The extension is LST, and the subdirectory C:\FURNIT\TMP. This file contains geometrical, and technological information, and prices.

6.6 Erasing drawings

Return to the image before the Drawings,lists command.

7. The room

7.1 Help

You can view this help.

7.2 The room

A parallelepiped will be generated, representing the room where the furniture will be located. The length will be introduced (oriented along the axis OX), the width (along the axis OY) and the height of the room (along the axis OZ).

7.3 Delete room

The parallelepiped room will be deleted.

8. Assemblings

8.1 Generating assembling

This transforms an existing plate (with or without holes) in assembling. The assemblings are pieces that, located on 1 plate or 2 neighboring plates can generate automatically holes. The assemblings represent in fact the iron part (hinges, handles, bars, etc.).

The editing commands of the plate are valid for assemblings too. You can generate and edit holes or grooves too!

8.2 Library of assemblings

Using the button SELECT, you can introduce a new assembling in the library. After the selection of an assembling you must indicate by three points the front (the plan) of the assembling. The point 1 defines the origin O, the point 2 the axis OX, and the point 3 the axis OY.

Using the button PLACE, you can place 1 or more assemblings at equal distance along the axis OX, on the front of a plate. You will need to select the front of the plate indicating by a point the first edge. Then you can indicate, possibly the change of the sense of this edge. By this first edge you defined the origin O and the axis OX of the front of the plate. Indicate then the second edge by a second point. By a second point you defined the axis OY of the front of the plate. The assembling shall be placed to coincide with the front of the plate and the system of axes XOY of the assembling to coincide with the system XOY of the front of the plate.

In order to reuse the maximum of the "PREVIEW" zone when you select an article from the library, modify the AutoCAD window to be similar to the rectangle of the gauge of the entities that you will select!

You should indicate by distances the movements of the assembling:

- along the axis OX,

- along the axis OY,

- along the axis OZ.

For any movement you can indicate previously that you want a movement in the "Opposite" sense to the sense of the axis. The "Opposite" option is useful when you indicate a distance by 2 points on the screen.

Introduce then the number of assemblings along the axis OX and the distance between 2 successive assemblings. For a negative value of the distance the placing shall be in the opposite sense of the axis OX.

Optionally, a second front can be indicated from another plate, by 2 of its edges.

The holes of a front of the assembling shall be copied automatically on the placing front of the first plate and on the indicated front of the second plate, if the front of the assembling is parallel with a corresponding front of the plate!

The assembling might not contain holes!

With "New" you can define a new article name.

With "Edit" you can select an attribute (head of column) of an article.

With "Value" you can modify the selected attribute. Some attributes cannot be modified!

With "Delete" you can delete an article.

With "Sort by" you can arrange in ascending order the displayed list of articles, depending on the selected attribute.

With "Type" the articles that are the selected TYPE are brought at the beginning of the list. These kinds of articles remain sorted by the setting in "Sort by". The TYPE is the last attribute of each article and helps for grouping the articles on types.

With "Print" the list of articles is printed in a file.

Leave the library with OK.

8.3 List assemblings

This allows to signal on the display of the existing assemblings in the current session. You can select between the total signal (typing R) or individual (with stop between 2 signals). Type S to exit the command. The signal is made by drawing intermittent lines.

Temporarily are drawn 2 vectors indicating the thickness and the width. Their vectorial product is the length.