Furniture design system
3D-CONSTRUCTOR 6.0

User Guide

Contents

Chapter 1. Generalities

1.1. Summary

3D-Constructor 6.0 is a furniture design system. This is an **AutoCAD** application. It works with **AutoCad** 2010, 2011, 2012, 2013, 2015

3D-Constructor in a few words:
- Furniture 3D-model can include parts, bench tops, profiles, hardware and fitting (legs, handles and fasteners)
- possibility to design different types of furniture and interior (kitchens, wardrobes,...)
- 3D work (3D-model can be created and seen from different views)
- 3D-models catalog creation and edition
- texture catalog creation and edition
- photorealistic visualization
- generating drawing of objects according to 3D-model
- generating assembly drawings according to 3D-model
- generating manufacturing reports (cutlist, elements list, material consumption)
- generating calculations
- integration with Cut Optimizer
- integration with CNC-machine

1.2. System requirements

System requirements are defined by AutoCAD.

1.3. Program interface

Program interface includes (see fig. *1.1*):
- drawing area
- menu bar
- quick access toolbars
- command line window
- status bar;
- model / layout bar
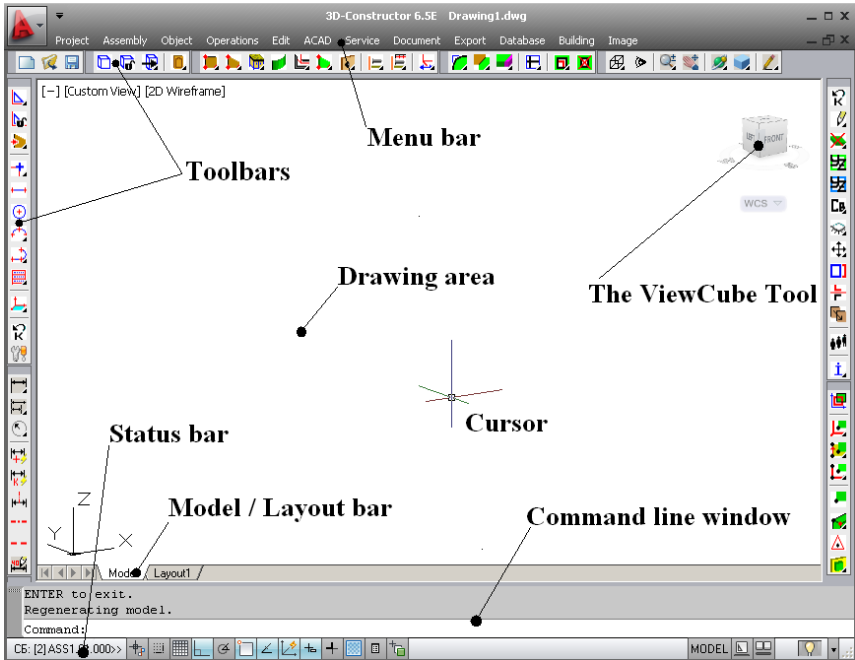
Fig. 1.1   Application window

1.3.1. Menu bar

Menu bar includes full list of command which can be used in the program.

Project
Includes commands to open, save, print, close projects.

Assembly
Includes commands to create and modify assemblies.

Object
Has commands to create parts, bench tops, profiles, hardware and fitting.

Operations
Use this menu to edit objects (edit shape, process a surface, edging, edge processing and other)

Edit

Use this menu to modify object properties, allocation, visibility etc.

Acad

These are commands to create and edit AutoCAD objects

Service

Includes commands for object's information and program settings.

Document

Includes commands to create and modify drawings

Export

Use this menu to create manufacturing reports, calculations, prepare data for optimization software and offload G-codes for CNC-machines.

Database

You can create and modify base of materials, fittings, hardware, catalog of textures, 3D-models and stamps.

Building

With this tool you can create room or house (build a walls, set door and windows)

Image

You can use this menu to set view, control visual style and create photo-realistic visualization.

1.3.2. Toolbars

Use buttons on toolbars to start commands, display flyout toolbars, and display tooltips

Fig. 1.2. Toolbars example "Object"

When you place cursor over a toolbar button, the tooltip displays the name of the button. Buttons with a small black triangle in the lower-right corner are dropdown toolbars that contain related commands. With the cursor over the icon, hold down the left button on your mouse until the flyout toolbar displays.

1.3.3. The Command Line Window

You can see commands, options, messages and prompts in a dockable and resizable window called command window.
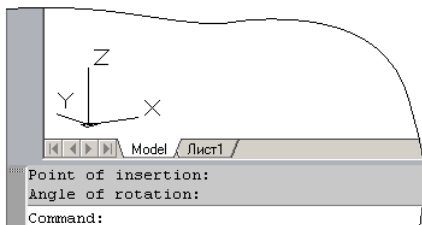
*Fig. 1.3   Command Line Window (sub image of the
Application Window)*

Bottom line of the Command Line is current in a system displays active query. It may be a message or a prompt. In the generally the query command line has the form:

Option1 #R/ *option2 / option3 #W / action:

The command options are separated by a slash "/". Capital letter in each option is key letter to select option. To select and activate an option, you can:

- Enter key letter (R– for option 1, W– for option 3). Key latter is marked with "#".
- Press "**Enter**" on the keyboard to activate the default option. The default option is marked with "*" at the beginning of the option. To change the default option press "**Tab**".
- Press "**Space**" to open options in the shortcut menu. Select one of them by the mouse.

Note. If you press a key letter to select 3D-Constructor options, register and current keyboard layout is not important. (The system allows you to use 3D-Constructor and AutoCad commands. If you work with AutoCad commands, it is necessary to observe the keyboard layout).

Example for query in the command line:
Distance <250> #H / *Centring #W / 2 / 3 / 4 / N / Next #L / Another 0 point:

You may:
- Specify another base point (by mouse)
- Activate one of the options, for example "*Centring*" (press "Enter" or latter "W" on the keyboard)
- Open shortcut menu and select option (press "Space" on the keyboard) see. fig. 1.4.
- Press latter "R" to redraw display
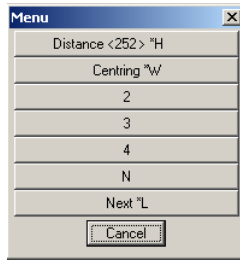
Fig. 1.4.   Example of shortcut menu

Note! The "R" (redraw) option isn't visible in the Command Line but it's available for any 3D-Constructor command. It is a standard feature for any 3D-Constructor command if command requests a point selection or object selection on the screen. It's comfortable to use it after pan and zoom transparently during the command.

1.3.4. Status bar

If there is current assembly in the project its mark and code is displayed in the status bar, see fig. 1.1, page 10.

1.3.5. Mouse right-click as Enter

You need press "**Enter**" in many of the 3D-Constructor commands. In such cases it is convenient, instead of pressing "**Enter"** on the keyboard to use the mouse right-click.

Users who don't work earlier in AutoCAD, it is recommended you complete customization, the right mouse button as follows:
- activate **Project >> ACAD >> ACAD options**,
- activate "User Preferences" tab,
- deactivate "Shortcut menus in drawing area".
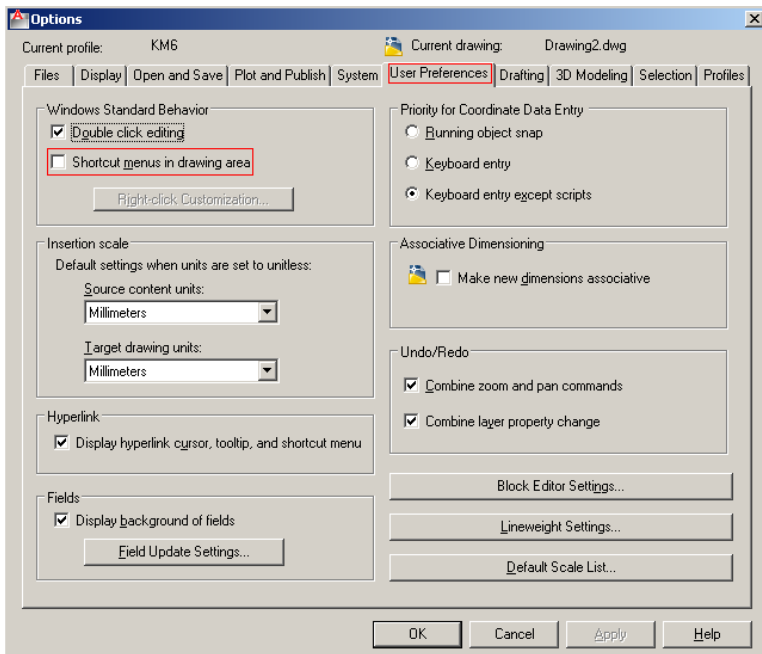
Fig. 1.5   Настройка правой клавиши мышки

Note. "Shortcut menus in drawing area" is deactivated automatically when 3D-Constructor first starts. It may be changed by user later.

Note (for AutoCAD users). Options of the 3D-Constructor command are not printed in the shortcut menu at any state of setting mentioned above. The options are printed in a modal window in the center of the screen by pressing the "Space".

1.4. Coordinate system

Coordinate system (hereinafter called CS) characterized by:
- the point of origin (0,0,0),
- directions of the axes X, Y, Z.

There are next coordinate systems in the 3D-Constructor:
- ACAD coordinate system,
- object coordinate system,
- reference coordinate system,
- operation coordinate system.

ACAD CS is set by user in the project. Most often you used world coordinate system to move or copy objects. The ACAD CS is always shown in the lower left corner of the screen.

Every 3D-Constructor object has its own coordinate system. It is specified in the build phase of the object. The object coordinate system remains unchangeable at all time of object existence. Its icon is shown in fig. 1.6. The X axis has at the end one arrow, the Y axis – two arrows, the Z axis (may be absent) - three arrows.

Fig. 1.6. Object coordinate system

When you reference to objects it is created reference coordinate system. The reference CS is defined by type of reference and it sets position of the object relative to another object.

When you do operation on object it is created operation CS. It defines an operation position relative to base point of the operation.

You may set color and length CS by **Service >> Settings >> General settings**, area "***Dialogue elements colours***".

1.5. Units of Measurement

You create your 3D-model at actual size.

Note. All values in the dialog window are entered in millimeters

Note. The current units precision is defined by "Units" command (enter "units" in Command Line and press **Enter**). Don't change unit's type in the "Units" dialog window. More information, see in the AutoCAD user guide.

1.6. Project structure

All items of the 3D-model are a project. The project consists from assemblies and objects. An assembly may include other assemblies and objects. A top assembly in the project is a product. Example of products is a table, a cabinet. The project structure scheme is presented in fig. 1.7.
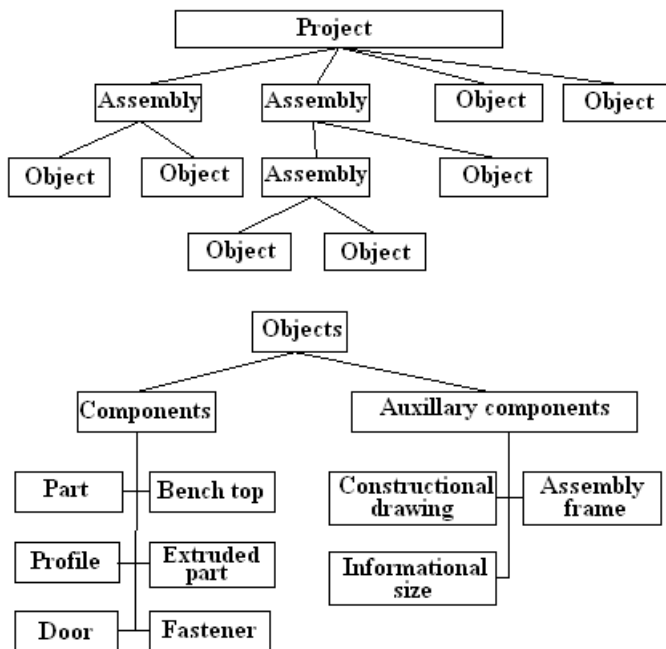
Fig. 1.7. Project structure scheme

1.7. Objects brief information

There are some types of object in the 3D-Constructor. They have different functionality, installation methods and methods of quotation.

Sheet Part

This is a flat solid limited by contour. Its contour may consist of line segments and arcs. Its thickness and texture is defined by sheet material. 3D-Constructor generates drawing of parts, includes them in the task for Cut Optimizer. The cost of part is calculated on the price per square meter of the sheet material (you set price per square meter in the database).

Extruded Part

This is a part with a variable thickness and any texture. It doesn't include in the task for Cut Optimizer. You can use it for any goals.

Bench Top

This is an object created by extruding shape (taken from database) along linear path. The cost of bench top is calculated on the price per running meter.

Bent Sheet

This object is created by extruding along the path of a rectangular section. The path should be smooth. Constructively bent sheet is glued of two or more parts. Every its part is automatically included in the task for Cut Optimizer. The cost of bent sheet is calculated on the price per square meter of the sheet materials.

Profile

This is an object created by extruding shape (taken from database) along curve. The cost of profile is calculated on the price per running meter or on the price per standard liner piece.

Block

These are the furniture accessories (dish racks, baskets and other).

Furniture fittings

These are the handles, legs, hooks and other. Unlike block, furniture fitting is slave object (it will be automatically deleted if its base object is absent).

Fastener

These are the screws, minifix, connectors, hinges, drawer slides and other.

Constructional Drawing

These are auxiliary constructions, which help you to do the 3D-model (to do parts, profiles, bench tops, bent sheets). They are not included in the Quotation. They are not had drawing and price.

Assembly frame

This is one of type auxiliary construction.

Informational size

These are informational dimensions which work in 3D-model. They display distance between two points or between point and surface. Their value is automatically updated when you redraw objects.

Note. In the next sections 3D-Constructor objects are marked with a capital letter, for example Part (this is sheet or extruded part) Assembly, Fastener.

1.8. About parameters and variables

3D-Constructor allows you to build parametric 3D-model. Any object in the project may have parameters and variables. The object parameter may set one of the object dimensions or set an operation parameter. Parameters are denoted by Latin letters and numerals. Some parameters have standard names. For

example: L - length, W - width, H - height. When you are working, the basic parameters of the objects are being automatically generated.

Let's see fig. 1.8. The part has a rectangular contour, one side of which is cutout. "L" and "W" are part parameters, "L1" and "W1" are cutout parameters. "L2" is a parameter referencing operation to a corner of the part.
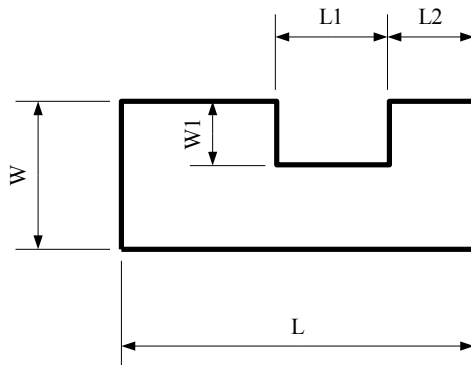
Fig. 1.8. Пример параметрической детали

To define the same parameter value for different operations or different objects, you may use variables. For example, the variable may be created in the part and it value may set two corner rounding. In this case, by changing variable, you will set rounding parameter.

1.9. About object properties

All objects and assemblies in the project have properties. The properties are generated automatically and may be edited by user. To edit them use command **Editor >> Properties**

Different types of objects have different properties but some properties are the same for any objects. For example: Code, Name, Class (assembly, part, standard product, set, no class), price (it may be calculated or attributed to object).

More details see in section 5.6, page. 217.

1.10. About operations

Operation is an action on an object. This is the main tool to edit the shape of object, to make auxiliary points in the object, to create joint. When you set operation parameters, it's possible and comfortable to use object and assembly variables. When you execute an operation, it's important to specify correctly object points and edges, see in section 4.1, page 154.

More details about operations see in Chapter 4.

1.11. About object reference

Object reference defines the relative location of objects. The object which you bind is called editable. The object which you bind to is called the base object. It may be one, two or three base objects for one editable object (it's defined by reference type). When you define reference, you should specify vertexes, edges and surfaces of objects.

If the base object updates its location or dimensions, the editable object will be automatically rebuilt according to reference.

You may reference any 3D-Constructor object except building elements (walls, doors, windows, floor …)

More details about reference see in chapter 6.1, page. 259.

1.12. Construction order of the objects, redrawing objects

In general, you create objects by specifying their dimensions and location per previously created objects. If you need reference object (editable object) to another object which is created after editable object, it isn't possible (the program blocks your action and send message "Wrong construction order"). In this case you may correct construction order and repeat your action. To edit the construction order, use command **Edit >> Construction order of the main objects**, more details see in chapter 5.21 page. 256.

Attention! The editable object must be in the construction order after base object.

If you change parameters, variable, location for any object to update all referenced object it is necessary to redraw them. To redraw objects, use one of the command from menu **Edit >> Redraw**, more details see in section 5.3, page 212.

1.13. Specifying object

You draw objects by specifying
- points,
- edges,
- surfaces,
- objects.

Note. It is difficult to select objects that are close together. In this case, you cycle through the objects. When you specify object (see fig. 1.9 a)) one of them will be highlighted (see fig. 1.9 b)). Press any keyboard key or do left-click by mouse to select another object (see fig. 1.9 c)). When the required object is highlighted, press **Enter** (do right-click) to confirm it.
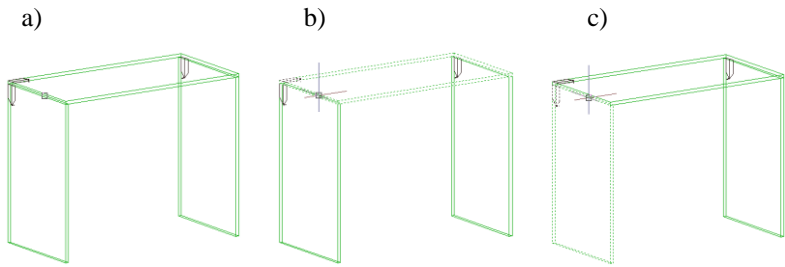
Fig. 1.9 Selecting objects that are close together

1.13.1. Point specifying

Usually points, what you should specify, are part vertexes or assembly vertexes. Remember, two geometrically identical points from different objects are different points. To specify the required point, specify an edge near the point.

Note. If you specify an auxiliary line, which AutoCAD builds for cylindrical and spherical surfaces, you don't select a 3D-Constructor object.

Note. If you can't specify the required object point, specify any another point and activate "Redefine object point" option. When all points of object are highlighted, do left-click near required point to select it.

1.13.2. Edge specifying

To specify an object edge, place the cursor under the edge and do mouse left-click. When the cursor is in position to select an object, it looks as square pick-box, see fig. 1.9 a).

1.13.3. Surface specifying

To specify an object surface, specify one of edges belong to surface.

Note. If you specify assembly edge (assembly edge belongs to adjacent surface) you cycle through the object surface.

1.13.4. Selecting object from "List of object".

There are a lot of commands which have "List" option, for example **Edit >> Properties**

*Product *B / *List *C / Select object*:

The "*List*" opens the list of objects, see fig. 1.10. This list consists of selected object and assemblies which directly or indirectly includes selected object.

For example, if you specify a box part, and the box is included in the cabinet, you will see three objects:

- selected object (BOX.00.003, in the bottom of the list)
- assembly-box (BOX.00.000)
- assembly-cabinet (ASS2.00.000)
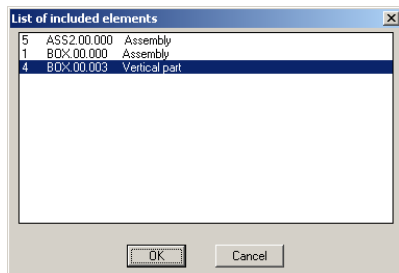
Fig. 1.10.   Selecting object from "List of included element"

1.13.5. Select Multiple Objects

For some 3D-Constructor commands at the Select Objects prompt, you can select many objects at the same time.

Specify opposite corners to define a rectangular area. The objects inside the area change color and become transparent. The direction that you drag your cursor from the first point to the opposite corner determines which objects are selected.

Window selection, see fig. 1.11 a).
Drag your cursor from left to right to select only objects that are entirely enclosed by the rectangular area.

Crossing selection, see fig. 1.11 b).
Drag your cursor from right to left to select objects that the rectangular window encloses or crosses.

a)   b)
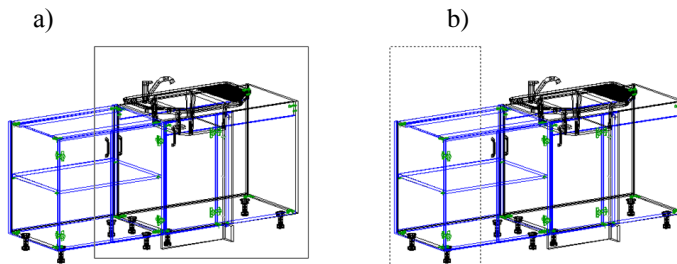
Fig. 1.11   Select Multiple Objects

Chapter 2. Assembly

2.1. Generalities

Assembly is an object that performs two basic functions:
- Logically combines some objects into a single group;
- Simplifies the making of objects which is built by the points and edges of the frame assembly.

Note. It is not necessarily to combine objects in the Assembly. But remember you may add to a database one object or group of objects combined in the Assembly.

The Assembly may include objects and other Assemblies. The Assembly nesting level isn't limited. No object can be included at the same time in more than one Assembly.

When you create a new Assembly its variables are automatically generated. The variables may be used to set the size of Assembly objects. The quantity of variables by default depends on the Assembly form. Variable named "H" always sets the height of the Assembly and isn't displayed on the dimensional Assembly slide (to set "H" default value, execute **Service >> Settings >> Assembly >> setting "Default height"**).

2.1.1. The frame and its subsidiary surface

Any Assembly has a frame. There are one or more points in the frame. It is easy to create objects using Assembly points. Subsidiary points and surface may be added by user.

When you edit an Assembly all the objects referenced with it are automatically rebuilt. The Assembly frame is defined by user at the time of Assembly insertion in the project. The frame usually (but not always) defines size of Assembly.

There are some ways to edit an Assembly.
- To add subsidiary surface, execute:
  Operations >> Assembly wire frame operations >> Sub surface
- To add points in the frame, execute:
  Operations >> Assembly wire frame operations >> any menu item
- To edit variables, execute:
  Edit >> Variables
- To edit position of the subsidiary surface, execute:
  Operations >> Parameters
- To extend of frame or move subsidiary surface, execute:
  Edit >> Stretch
- To make frame visible or invisible, execute:
  Edit >>Visibility >> On, Edit >>Visibility >> Off

Changing the Assembly frame does not always lead to an automatic re-drawing all connected to the frame objects. To redraw the objects included in the Assembly, execute:

Edit >> Redraw >> Assembly
Edit >> Redraw >> Product

A detailed description of the above mentioned commands, see in appropriate sections of this manual.

2.1.2. Logical materials

Each Assembly can have logical materials. The logical material is a special type of the variable. Each material is characterized by a name, a description and a material.

Name
Is a variable of the assembly. It is convenient to do it short, for example: M1, M2... The value of the variable is a thickness of the sheet materials associated with it.

Description
Defines meaning of the logical material. For example shelves, back.

Material
Is one of the sheet materials from the base.

One of the logical materials may be a current. In this case all parts added to the Assembly will have current logical material. Exceptions are objects created by command **Object >> Rectangular part >> Back**. The back always has material named "MZ". If "MZ" is absent in the Assembly, back will have the first item from the list of sheet materials.

The logical materials give you next possibilities:
- Synchronize logical materials with the same name in different models (when you insert 3D-model in the project all parts automatically redefine "M1", "M2" materials according to sets of logical materials in the project).
- Quickly change the part materials without preliminary selection of parts (the parts have already selected by logical material).
- Use reference to the thickness of materials in formulas (you will not have to edit a material thickness in the formulas if material is changed in 3D-model)

More details see section 6.2.4.2, page 289.

2.2. Create Assembly

To create a new Assembly
1. Click **Assembly >> New Assemblly**

2. Select Assembly form from the window, see fig. *puc. 2.1*
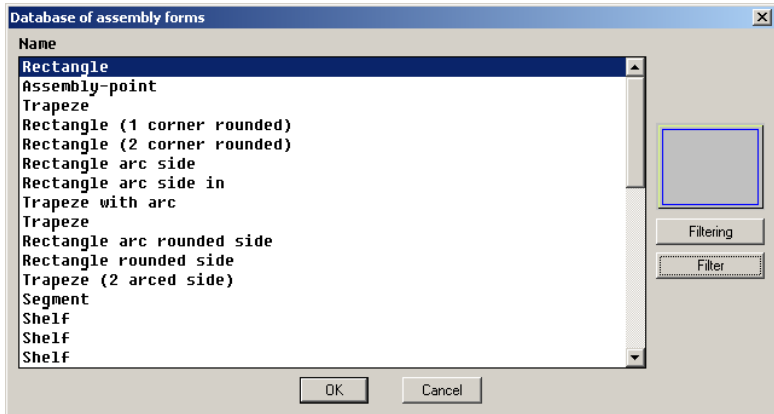
Puc. 2.1. Database of assembly forms

3. The following prompts are displayed:
   Select objects to be included in new assembly: Select objects or click
Enter if there aren't objects which must be included.

4. Edit variables in the next window, see fig. *2.2*.
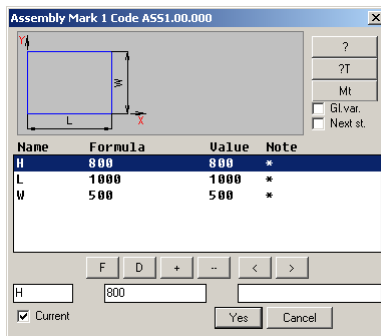
Fig. 2.2. Assembly mark

Note. The operation in the dialog window is considered in section 6.2.2, page 283.

Attention! Setting "**Current**", see fig. *2.2,* sets the new assembly as current after insertion it in the project.

2.3. Current Assembly. Include or exclude objects.

One of the Assemblies in the project can be current. If there is a current Assembly in the project, all newly created objects are included in it automatically. All objects of the current assembly are painted in the same color. There are two colors for objects in the projects. One of them is color for objects included in the current Assembly. Other is for objects out of current Assembly.

To set color of objects, click **Service >> Settings >> General settings >> area "Objects colours".**

You can set current Assembly by dialog window (see fig. *2.2)* or command **Assembly >> Current assembly.**

The following prompts are displayed:

Select new current assembly / *no current assembly: Specify an Assembly or click **Enter** to set mode "no current assembly"

Note. The locked Assembly and the duplicated copy of an Assembly can't be set as current Assembly, more information see in section 2.4, page 26 and section 2.5, page 27.

There are two commands to control an Assembly composition:
Assembly >> Include in Assembly
Assembly >> Exclude from Assembly

Include in Assembly
Includes selected object in the Assembly. One object may be included only in one Assembly.

Exclude from Assembly
Excludes selected object from the Assembly

2.4. Duplicated copy of Assembly

This is special type of Assembly. Any changing in the original will be automatically done in the duplicated copy of Assembly (after its redrawing). You don't have an access to the duplicate objects. It is possible to use only the frame points.

Note. Duplicated copy of Assembly can't be saved to the database; you should save the original Assembly.
Note. You can't create a duplicated copy of Assembly if Assembly has included a duplicate yet.

To create a duplicate, click
Assembly >> Duplicated copy of Assembly

To explode duplicate, click
Assembly >> Explode Duplicated copy of Assembly

2.5. Locked assembly

The program works with the locked Assembly as with the block. The user doesn't have access to its objects. It is possible to use only its frame points.

To lock the Assembly, there are two ways:
- click **Assembly>> Lock Assembly**
- insert a locked Assembly from database (setting "Close after insert" must be on in the database)

Attention! If the Assembly includes other assembly, informational size or it has external references (it references to objects that are not included into the Assembly) it can't be locked.

To unlock the Assembly, click
Assembly>> Unlock Assembly

Note. There is another type of locked Assembly– Tamburat, more details see in section 3.7.8, page 117.

2.6. Insertion model in the project

The model is a single object or an Assembly. When you insert the model in the project you define its position and its coordinate system.

To insert the model in the user coordinate system, click
Assembly >> Typical model in UCS

To insert the model in the local coordinate system, click
Assembly >> Typical model in LCS

Note. Information about UCS and LCS see in section 1.4, page 14.

Above mentioned commands activate one of the next insert methods:
by points *N / by points (face) *K / by parts *C / *without reference *G:

Note. The insertion method by default is defined by the user in the database for each model.

2.6.1. *Method "without reference"*

Inserts the model without reference. The model size is taken from database.

2.6.2. *Method* **"by parts"**

Inserts the model into a rectangular opening. You need to define four or three parts limiting the opening. The model sizes are automatically adapted according to the opening. The model is automatically referenced.

List of prompts:
Base vertical part: Select the first vertical plate limiting the opening.
Flat part 2 (parallel to 1-st): Select the second vertical plate.
Part 3 (perpendicular to edge of 1-st part): Select the third plate.
Flat part 4 (perpendicular to 1-st part): Select the fourth plate or click **Enter** to insert model by selecting three parts.

Attention! If you insert the model by selecting three parts it is necessary to do right selection of the first part, see fig. *2.3* a). The point of selection have to be far from the part 3. In any case, see fig. *2.3* b), you will get a notice "No distance between parts" and command will be cancelled.

Note. The first selected edge defines the front face of the model.

a)   b)
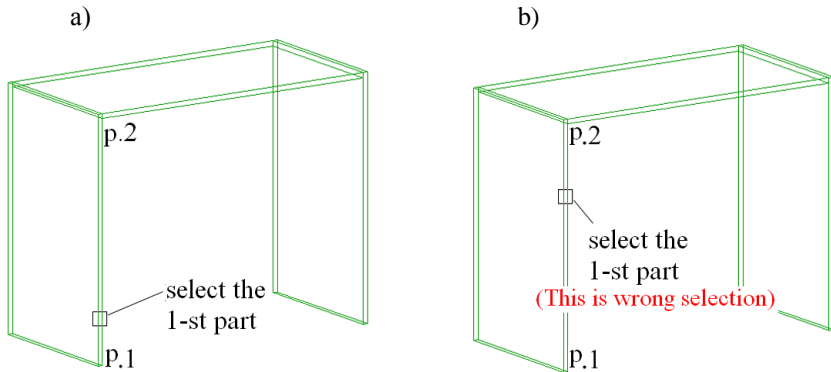
*Fig. 2.3   How to select the first part to insert model
by selecting three parts*

Attention! If the model has left and right codes in the database, the selection order of the first and the second parts will define the model coordinate system (the model CS will be right or left).

Attention! If there is back in the rectangular opening it sets the model depth (the third dimension– variable "W"). In another case the value of "W" is taken from database.

2.6.3. *Method* **"by Points"**

Inserts the model into a rectangular opening by specifying four points. The model size is automatically adapted according to opening and it is automat-

ically referenced. The specified points define model size and model reference. The first specified point sets base of model (point "0", fig. 2.4). The second point sets X-axis and model width (point "1", fig. 2.4). The third point sets Y-axis and model depth (point "2", fig. 2.4). The fourth point sets Z-axis and model height (point "4", fig. 2.4).

Note. The base point sets the model front face. This method may create right or left model.
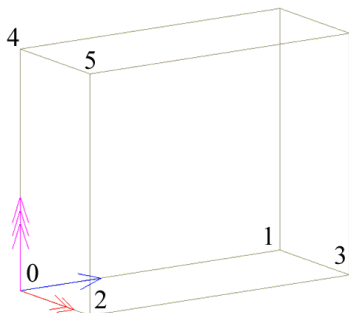
Fig. 2.4   Frame of model

2.6.4. *Method* "*by points (face)*"

This method is similar to the "*by points*" but the first point sets model back.

2.6.5. *Method* "**As product**"

Four mentioned above methods insert model in the current assembly if it is in the project. "As a product" always inserts model in the top level of the project. After inserting model in the project it is automatically executed **Edit >> Magnet**. The magnet allows you to combine furniture. When magnet is turned on, click **Space** button to rotate model around its Z-axis by 90 degrees.

This method allows you to combine furniture set quickly and easily.

To use this method, it is necessary to set insert method "As products" in the database.

Chapter 3. Objects

To set default object properties and default parameters, use **Service >> Settings**.

3.1. Parts

This is a flat solid limited by contour. Its contour may consist of straight line segments and arc segments. You can take typical contour from the database or do it yourself. The contour may be edited with operations (rounding, chamfer, different cuts…)

3.1.1. Overall dimensions of object

Any Parts can have overall dimensions of object (hereinafter box). The box is invisible frame of Part. The Part contour may be situated inside or outside the box. The box points help you to reference object comfortable, see fig. 3.1. You see a shelf which has front and side gaps. The shelf has a box. Its contour is situated inside the box. B1, B2… B8 are the box points. You can use any of them to reference shelf to the vertical Part quickly and comfortable. (The box points coincide with edges of vertical Parts. The gaps are defined with special variables).
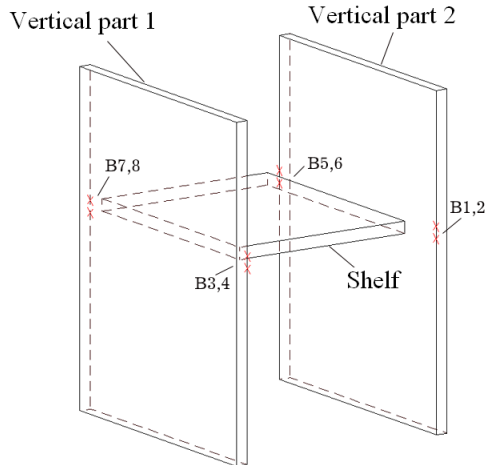
Fig. 3.1   Example of the Part with box
(B1,…B8 are box points)

If you activate box for Part, some special variables will be automatically created. The box variables are:

Lb – box length
Wb – box width
SX1 – the first gap along X axis (from CS origin)

SX2 – the second gap along X axis

SY1 – the first gap along Y axis (from CS origin)

SY2 - the second gap along Y

It's convenient to control the box variable by command **Operations >> Parameters of 1-st**, more details see in section 4.2 page. 156.

a)   b)
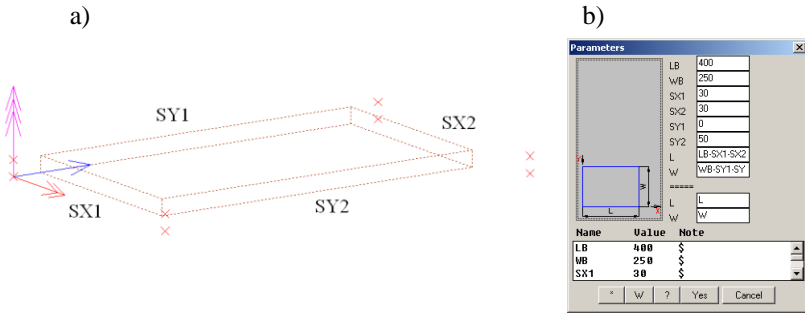
Fig. 3.2 a) Shelf ad its gaps

b) Parameters of the first shelf element

If you activate "***Overal dim. of object***", see fig. 3.3, and use special command to create a shelf, **Object >> Rectangular part >> Shelf**, the box will be automatically created and shelf will be reference by box point.
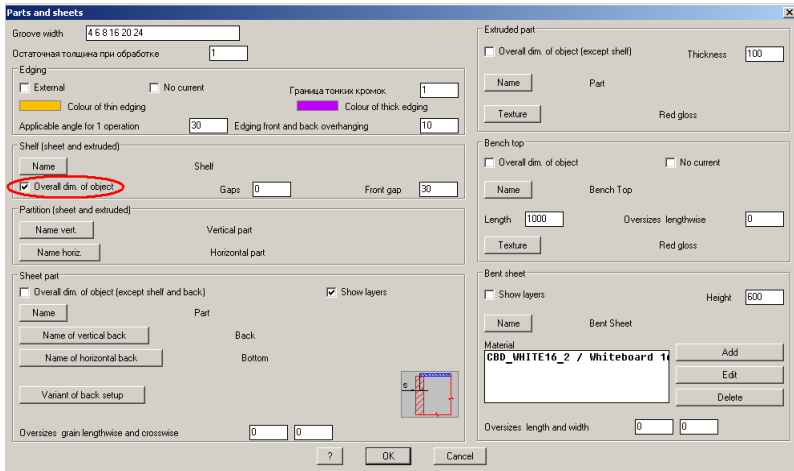
*Fig. 3.3   "**Overall dim. of object**" setting for shelf*
(Service >> Settings >> Parts and Sheets)

If you clear "***Overall dim. of object***" and create a shelf, the box will be absent and shelf will be reference to vertical part by contour point. The reference parameters will have values of gaps.

You can activate box for existing Part by command **Operations >> For part >> Add box.**

Note. There are some special objects (back, door and drawer front) which always have a box. For other objects you can activate box by default in the "Parts and sheets" dialog window, see fig. 3.3.

3.1.2. Sheet Part

There are two groups of commands in the "Object" menu to create a Sheet Part. These are "Rectangular part" and "Part".

More often it is necessary to create the rectangular Parts so commands for the rectangular Part creating are extracted in an auxiliary menu item. Commands from "Rectangular part" item jump through the Part contour selection and the Part parameters editing.

To adjust the sheet materials database, use **Database >> Materials >> Sheet materials**. You can adjust next material properties: code, name, thickness, texture and other.

3.1.2.1. Generalities

To create Part, you define its contour and extrude it according to the sheet material thickness. The Sheet Part thickness, texture, price are defined by database.

When you create a new Part in the Assembly and Assembly has a current logical material, the Part will be created of the Assembly current logical material. In another case it will be created of the first item from the sheet material database.

Note. More details about Assembly logical material see in section 6.2.2.5 page 285.

The Sheet Part may have more than one layer. In this case every layer is defined with one of the item from the sheet material database. The Part thickness is determined by summing up of the layer thicknesses. The program understands it, see fig. *3.4*, as whole Part (it has one drawing, all layer have one edging, you set oversize grain for all layers). In the quotation documents you will see Part and every layer.
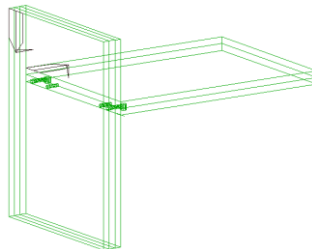
Fig. 3.4   Example of multi-layer parts

Note. To define the glue thickness, you can increase the multi-layer part thickness, more details see in section 5.6.8, page 224.

3.1.2.2. Sheet Part dimensions

There are four dimension types for Sheet Part:
overall by CS
cutting by CS
nominal by CS
cutting by texture

Nominal dimensions by CS

Every Part in the project has its own coordinate system (hereinafter CS), which is determined at the stage of the Part construction. The Part edges may coincide or not coincide with direction of the CS axis. If the Part is already created, you can't edit its CS. The Part and its nominal dimensions by CS are presented in the figure *3.5*.
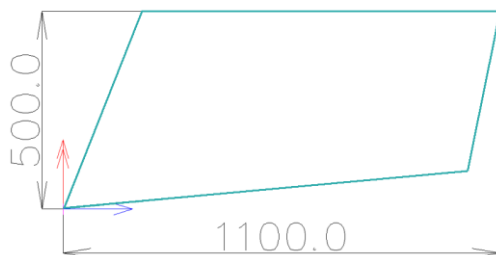
Fig. 3.5   Part and its nominal dimensions by CS

The nominal dimensions by CS are the rectangle circumscribed about the Part contour which edges coincide with direction of the CS axis.

Overall dimensions by CS

This is a sum of the Part dimensions by CS and the external edging thickness. (The edging may be external or internal)

Cutting dimensions by CS

This is a difference between the Part dimensions by CS and the internal edging thickness.

Cutting dimensions by texture

These dimensions define workpiece dimensions. The workpiece must be cut according to texture direction to do the Part.

Cutting dimensions by texture are a rectangle circumscribed about the Part contour one edge of which coincides with texture direction plus oversize grain, see fig. 3.6.

Attention! To calculate workpiece dimensions, the rectangle is circumscribed about cutting contour of the Part and then oversize grains are added to it.
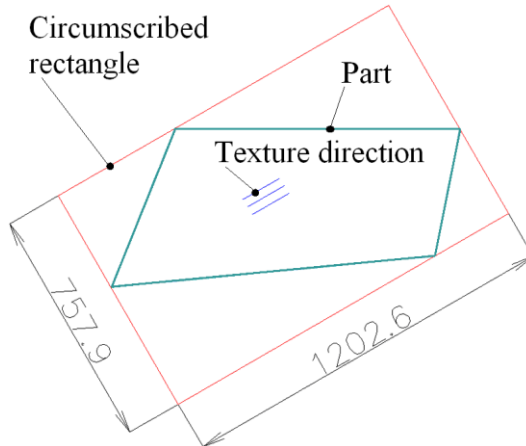
Fig. 3.6   Part and its cutting dimensions according to texture

3.1.2.3. Create a horizontal or vertical Sheet Part

Access methods
Menu: **Object >> Part >> Vertical**
Menu: **Object >> Part >> Horizontal**
Menu: **Object >> Rectangular part >> Vertical**
Menu: **Object >> Rectangular part >> Horizontal**
Toolbar: **Object**
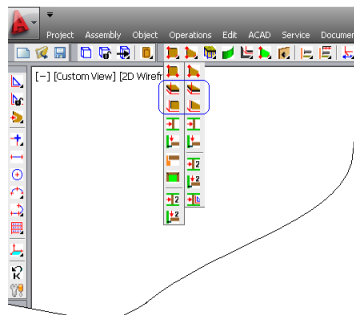
*Fig. 3.7   Toolbars to create horizontal and vertical
           sheet parts*

Summary

These commands create a Part which has one of the typical contours. The contour of the created Part can be edited by operations. The commands from "Rectangular part" menu jump through the contour selection and parameters editing.

List of Actions and Prompts

Activate **Object >> Part >> Horizontal**, Select the Part contour, see fig. *3.8.*
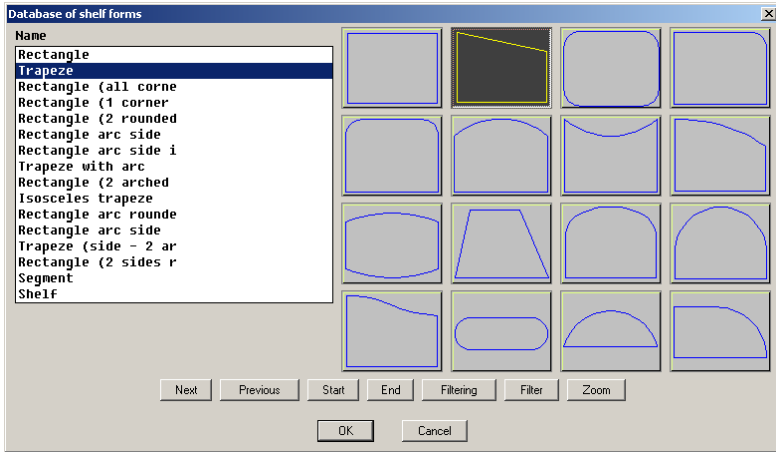
Fig. 3.8. Database of base forms

Enter the Part parameters and press "Yes" in the dialog window, see fig. 3.9. If you insert Part into an Assembly, you will see window presented in fig. 3.9 b). You can use the Assembly variables in the Part parameters.

Note. More details about working in the "Parameters" window see in section 4.1, page. 154.
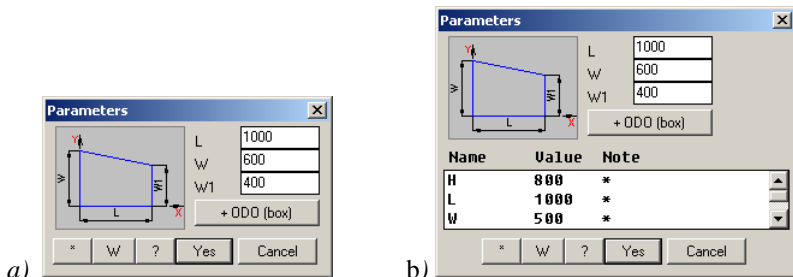
Fig. 3.9. Entering Part parameters

Note. You can enter formulas in the parameters. The mathematical operations and functions, which may be used, see in section 6.2.5.2, page 293.

Attention! Press "ODO (box)" to make a box for the Part. If you press it, the auxiliary variables will be added in the "Parameters" window. More details about box see in section 3.1.1, page 30.

The following prompts are displayed:
Insert point: Specify a point on the screen by mouse left-click
Rotation angle: Determine an angle by mouse or enter the angle value in the Command Line.

3.1.2.4. Create Part by 3 points

Access methods
Menu: **Object >> Part >> By 3 points**
Menu: **Object >> Rectangular part >> By 3 points**
Toolbar: **Object**

Summary
These commands create a Part by specifying three points. The specifying points usually are vertexes of the existing objects. The first specified point is the base point of the Part, the second point sets a texture (grain) direction and its size along X-axis, the third point sets its size along Y-axis. The commands automatically create two variables into the Part in compliance with specified points and set them in the Part parameters. Therefore, if you change position one of the three points, the part will be automatically recreated in compliance with new parameters. More details about variables see in section 6.2, page 281.

List of Actions and Prompts
Execute **Object >> Part >> By 3 point**, select a part contour, see fig. *3.8.*
ODO (box) + / Base point: Specify the base Part point.

Note. To specify point, you can move a cursor to the edge near the point. When the cursor is in position to select an object, it looks as square pick-box, see figure below.
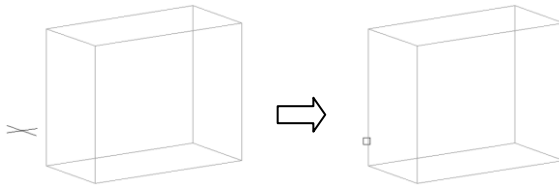
*ODO (box) + / *change point #N / Point of axis X*: Specify the second Part point to assign a texture direction and one of the Part sizes.

ODO (box)

Determines what point will be taken. It may be object point or box point (detail about box see in section 3.1.1, page 30). To specify point of an object, must be displayed "*ODO (box) +*". To specify box point of an object, must be displayed "*ODO (box) -*".

Note. If "*ODO (box) -*" is displayed and the specified object hasn't box, it will be taken the object point.

Change point

Permits you to reselect the point specified before. The new point has to belong to the same object. If you need select point for another object, cancel command and execute it again.
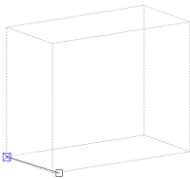
*ODO (box) + / *change point #N / Point of axis Y*: Specify the third part point to assign its second sizes.
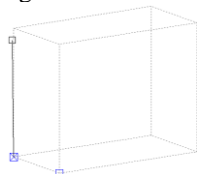
*change point #N / *Apply selected #G*: Confirm the third specified point.
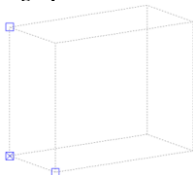
Edit or confirm parameters in the dialog window, see below.

Note, "L" parameter defines Part size along X-axis (the X-axis has one arrow at the end). "W" parameter defines part size along Y-axis (the Y-axis has two arrows at the end). You can see the Part variables in the top variable list (see "Parameters" window). "X" and "Y" (see figure below) were created by specifying three points. You can see the Assembly variables in the lower variable list. Any variables shown in the "Parameters" window may be included in the Part parameters.
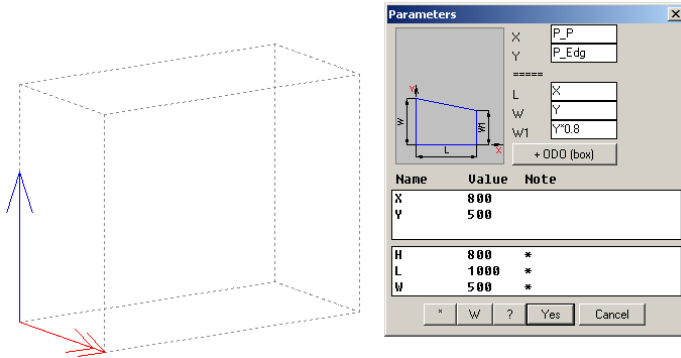
To change direction - any key / ENTER - apply: Specify a direction to extrude Part (the Part is extruded in compliance with thickness of the sheet material)
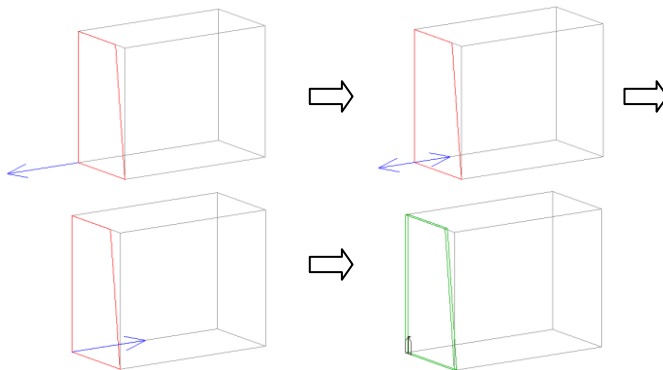
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3.1.2.5. Create Part by drawn contour
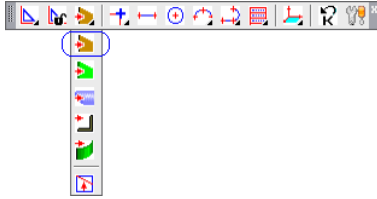
Summary
You can use Constructional Drawing or AutoCAD drawing to make the Part contour yourself. To create Part by drawn contour, execute above mentioned **command**.
More details about Constructional Drawing and creating object by Constructional Drawing see in chapter "Constructional Drawing".
More details about AutoCAD drawing see in AutoCAD user guide.

3.1.2.6. Create Part as frame filler

Summary
This command creates a Part by specifying Profiles and elements of Constructional Drawing. Use it, to create filler of sliding doors or framed door.
The first and second specified objects determine plan of the new Part. Their coordinate system (thereinafter CS) have to belong to one plane. The CS other objects have to be parallel to CS of the first and second objects (They will be projected in the plan of the new object). The paths and elements of the specified object have to determine a closed contour of the new Part.
For example see fig. 3.10

List of Prompts
Material #V / Specify the border fill: Specify the first border, see fig. 3.10, object "1".

Material
Specifies a material of the new Part.

Specify the border fill: Specify next border, see fig. 3.10, object "2".
Specify the border fill: Specify next border, see fig. 3.10, object "3".
Specify the border fill: Specify next border, see fig. 3.10, object "4".
Specify the border fill: Press **Enter** to complete the border selection.

To change direction - any key / ENTER - apply: Specify a direction to extrude the Part and press **Enter**.
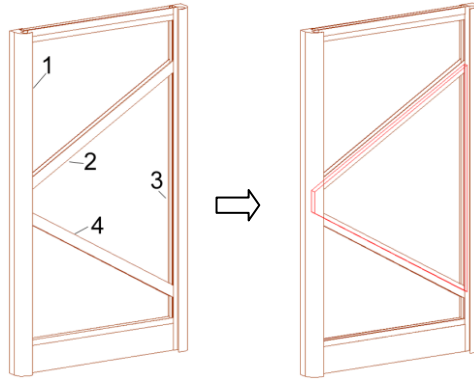
Fig. 3.10   Create part as frame filler.
The Part contour is determined by profile.
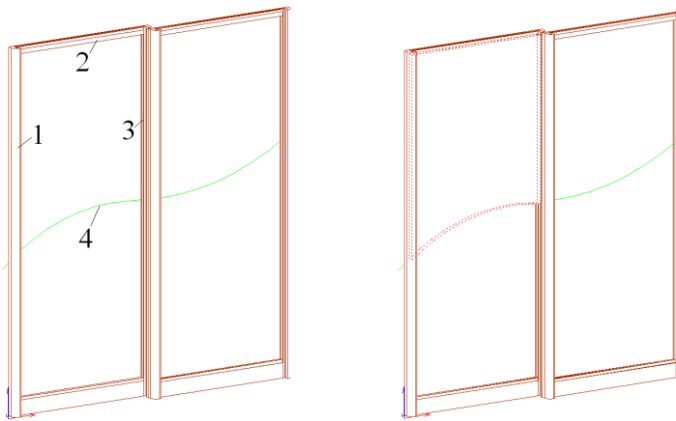
Рис. 3.11   *Create Part as frame filler.*
The Part contour is determined by profile (object 1, 2, 3)
and elements of Constructional Drawing (object 4).

Note. 4 is the path defined in the Constructional Drawing

Note. It leaves gaps between the specified objects and the edges of the new Part in compliance with settings in the Profile database. The gaps are created by "offset" operation.

To edit gaps, use **command Operations >> Parameters**. When you specify filler, every object will be highlighted by different colour, see fig. 3.12 a). The arrows will show offset direction for every Part edge. The

number in the 3D-model will show the parameter number in the dialog window, see fig. 3.12 a). For example, "-@32!SM2Y":

"-" –sets minus offset direction

"@32!" –sets link to the object which has mark "32"

"SM2Y" –sets gap (this is system variable; it is defined in the Profile database).

The moving along Z-axis is automatically included in the Part reference, see fig. 3.12 b). The moving is defined by "SM2Z" system variable (it is defined in the Profile database).
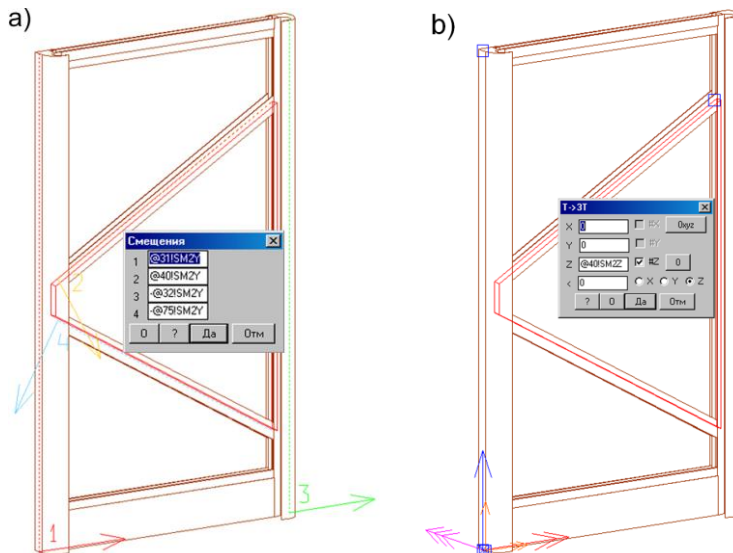
Fig. 3.12 Editing offset parameters and Part reference.

Attention! There are three offset pair in the profile database. It is important to use them correctly. More details see in section 3.3.1.2, page 56.

3.1.2.7. Create rectangular Parts along the path

<u>Access method</u>
Menu: **Object >> Rectangular part >> Parts along the path**

<u>Summary</u>
This command creates rectangular Part along straight lines of specified path. The Parts are created rounded on 90 degree to the plane of the path. The path may be defined with object edges or Constructional Drawing. The path mustn't have brake points.

List of Prompts (the path is defined with a Constructional Drawing)
Select base path: Specify a path, see fig. 3.13
Quadrant (any key) / ENTER - apply: Specify a Part orientation and press **Enter**. Enter in the dialog window the Part high "W" and Part offset "Sm".

Note. This command always creates Parts with box, more details about box see in section 3.1.1 page 30.
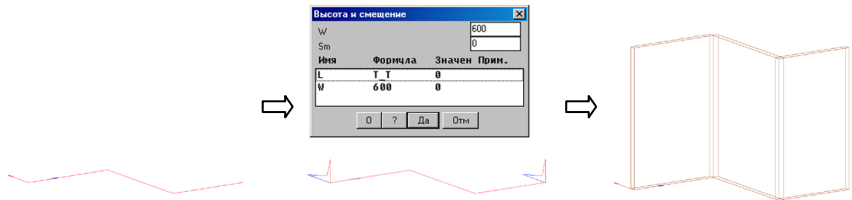
Fig. 3.13   Creating Parts along the path
(The path is defined with Constructional Drawing)

List of Prompts (the path is defined with a Part edges)
Select base path: Specify the first edge
Perimeter #G / Side #C / Next edge: Specify the second edge
Next edge: Specify the third edge
Next edge: Press **Enter** to complete the path selection
Quadrant (any key) / ENTER - apply: Specify a Part orientation and press **Enter**. Enter the part high "W" and part offset "Sm" in the dialog window.

Perimeter
Specifies all part edges

Side
Change the base side. It may be the upper or lower part side.
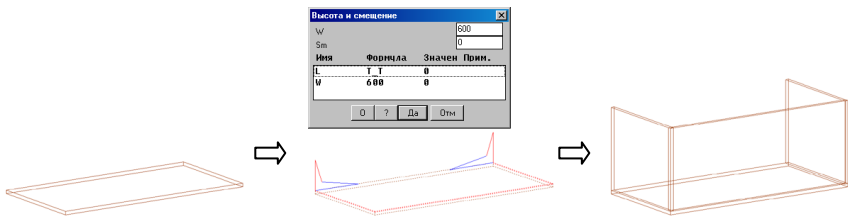
Fig. 3.14   Creating parts along the path
(The path is defined with object edges)

Note. The specified edges have to belong to one object.

3.1.2.8. Take AutoCAD solid

Summary
This command finds AutoCAD solids in the project and converts them into 3D-Conrtructor objects.
If AutoCAD solid is found and its thickness is equal to a thickness of one of sheet materials in a database, the solid will be converted into the Sheet Part. If the solid thickness is not equal to a thickness of one of sheet materials, the solid will be converted into the Extruded Part.

Note. All converted objects take the colour of 3D-Constructor object.

To find and convert AutoCAD solid, it must be done two conditions:
- The solid contour must include only lines and arcs.
- The upper and lower solid plate must be equal.

In other words the solid contour may include rounding, chamfer, cut on side but hasn't to include groove, pocket or fastener holes.

3.1.3. Extruded Part

The program doesn't consider an Extruded Part manufacturing technology. Their mission is a project decorating. The Extruded Parts corresponds to the Sheet Parts, but the Extruded Parts have a variable thickness, an attributed texture and an attributed price. The example of the Extruded Part is a MDF door.

The group of commands to create an Extruded Part corresponds to the commands to create Sheet Part. To access them, use

menu: **Object>> Extruded part**
toolbar: **Object**
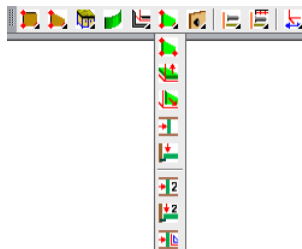
Note. The thickness and texture extruded part by default is determined with settings in the window "Parts and sheets", in the area "Extruded part".

To edit thickness of existing Part, use commands

Edit >> Variables
Operation >> Parameter of 1-st

To set price of existing Part, use command **Edit >> Properties**, field
"**Price**"

3.1.4. Bench Top

The Bench Top has a typical cross-section determined in the database
and a variable length. To set the Bench Top cross-section database, use **Data-
base >> Materials >> Bench Tops database**. You can set cross-section pa-
rameters and material properties (code, name, texture, price for running meter)
in the database.

Note. The face edge of a Bench Top in the drawing is marked with con-
ditional line.

Note. To edit the width of the existing Bench Top, use **Opera-
tion >> For part >> Change bench top width**, more details see in section
4.5.8, page 201.

The commands to create Bench Top are considered below.
The settings for Bench Top by default are determined with the command
Service >> Settings >> Parts and Sheets, area "**Bench top**". If the checkbox
"**No current**" is filled in, you will select a Bench Top cross-section every time
when you will create a Bench Top.

3.1.4.1. Horizontal

Access method
Menu: **Object >> Bench top >> Horizontal**
Toolbar: **Object**
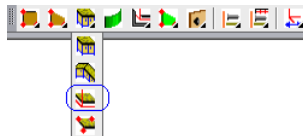
List of Prompts
*Section <BT2> #C / *Next #L / select bench top*: Confirm the cross-
section by default or specify a cross-section to create new Bench Top. The code
cross-section by default is in the Command Line, press **Enter** to confirm it.
You can specify another cross-section by mouse left-click on the existing
Bench Top in the project or by option "Section".
First point of distance <dialogue>: Specify the first point on the screen
to determine Bench Top length or press **Enter** to enter the object length in the
dialog window.
Second point: Specify the second point on the screen to determine the
Bench Top length
Insert point: Specify a point on the screen by mouse left-click
Rotation angle: Determine the angle by mouse or enter its value in the
Command Line.

Section
Opens "Bench Top database" to select cross-section

3.1.4.2. By 3 points

<u>Access method</u>
Menu: **Object >> Bench top >> By 3 points**
Toolbar: **Object**
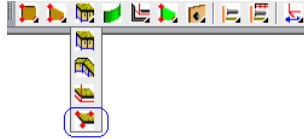
This command corresponds to the command considered in section 3.1.2.4, page. 36.

Note. It is comfortable to have a box for Bench Top, see section 3.1.1, page 30. It gives possibility to do right and left side oversize for existing Bench Top by command **Edit >> Stretch**. To set the box by default, use **Service >> Settings >> Parts and Sheets**, area "**Bench top**", setting "**Overall dim. of object**"

3.1.4.3. Bench top on products

<u>Access method</u>
Menu: **Object >> Bench top >> Bench top on products**
Toolbar: **Object**
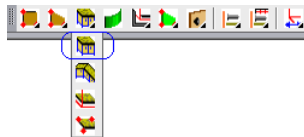
<u>Summary</u>
This is a special command to create a Bench Top for several products installed in a line. You can specify products, walls and line segments created by AutoCAD command to determine the Bench Top position and dimension. The command working is considered in fig. 3.15.

<u>List of Prompts</u>
*Section <BT2> #C / Select product to set up Bench Top: Specify the base product to set up the Bench Top (see fig. *3.15* a), page 47, selection "1"). If you set up a Bench Top by product and walls, the specified product will determine the height of the Bench Top installation. If you set up a Bench Top by two extreme products, the first specified product will determine the height of the Bench Top installation and its border.

Note. The Bench Top is created in compliance with upper plane of the specified Assembly.

Section

Shows you code of current cross-section and permits you to select another cross-section from the database.

Select product to setup Bench Top or wall (to setup against) <ENTER-next>: Specify the second product to determine the Bench Top length or specify the wall to determine one of the Bench Top border (see fig. *3.15* a), page 47, selection "2")

Note. Instead the wall you can specify line segment created by AtoCAD

Select wall (to setup against) <ENTER-next>: Specify next Bench Top border (see fig. *3.15* a), page 47, selection "3") or press **Enter** to work without border.

Select wall (to setup against) <ENTER-next>: Specify next Bench Top border (see fig. *3.15* a), page 47, selection "4") or press **Enter** to work without border.

In the next step you will see the Bench Top preview and dialog window to specify gaps, see fig. *3.15* b), page 47. In the base point of the Bench Top is drawn object coordinate system. The base point corresponds to field "Gap (edge 1)".

The created Bench Top is presented in fig. 3.15 c), page 47.

Note. You can specify Bench Top borders to create the Bench Top in any order.

Attention. When you create the Bench Top by products it is important where the front surface of the first specified product is. Its front surface determines the face edge of the Bench Top. To see the back of product, execute **Service >> Object CS** and specify the product frame. You will see the object coordinate system in the base point of the frame. The base point corresponds to the product back. If you have a product face on the frame back the commands "**Bench top on products**" and "**Group of bench tops**" will work wrong.

There is example in fig. *3.16* page 48 where AutoCAD line segment is specified as border.

Note. The examples considered in this section are displayed in orthographic view. You can create a Bench Top in the 3D-view or in the orthographic view.

a)
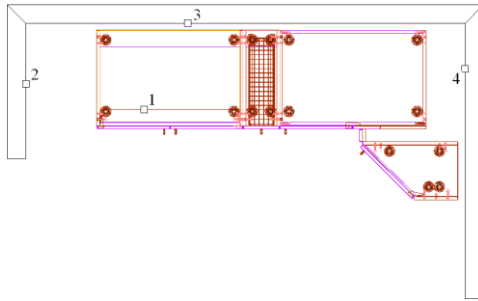
b)
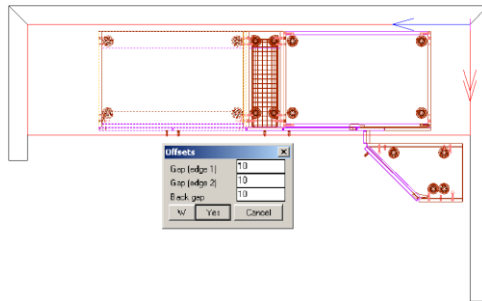
c)
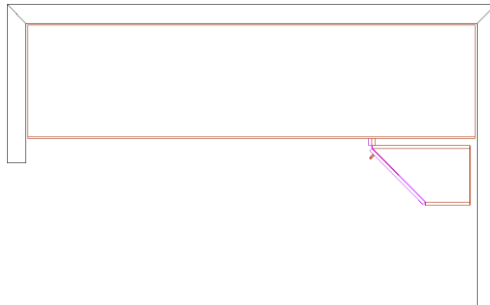
Fig. 3.15   a) Object selection to create the Bench Top.

b) Bench Top preview.

c) Construction result.

a)   b)
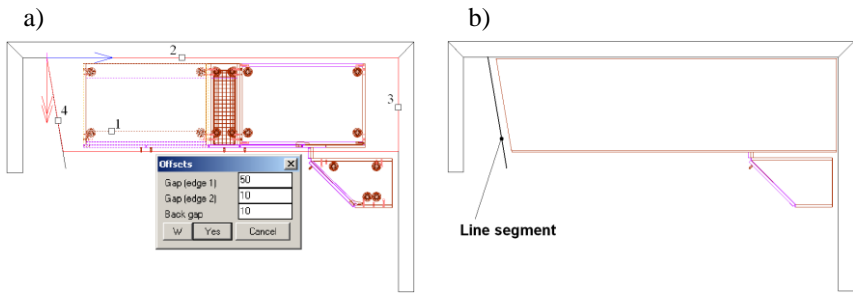
*Fig. 3.16   a) Object selection to create the Bench Pop and the
              Bench Top preview.*

b) Construction result.

In the figure *3.17*, page 49 is displayed example for construction the se-
cond piece of the Bench Top. It is created by specifying the product, the wall
and the first piece of the Bench Top. The order of the object selection is marked
in figure by numbers.

List of Prompts
*Section <BT2> #C / Select product to set up Bench Top: Specify the
corner product, see fig. *3.17* a), page 49, selection "1". It determines the height
of the Bench Top installation and its face edge.
*Select product to setup Bench Top or wall (to setup against) <ENTER-
next>*: Specify edge of the existing Bench Top, see fig. *3.17* a), page 49, selec-
tion "2".
Select wall (to setup against) <ENTER-next>: Specify the wall, see
fig. *3.17* a), page 49, selection "3".
Select wall (to setup against) <ENTER-next>: Press **Enter** to complete
construction.

Enter gaps in the dialog window. You should enter nil in the field "Gap
(edge 2)" to do the close objects.

a)   b)
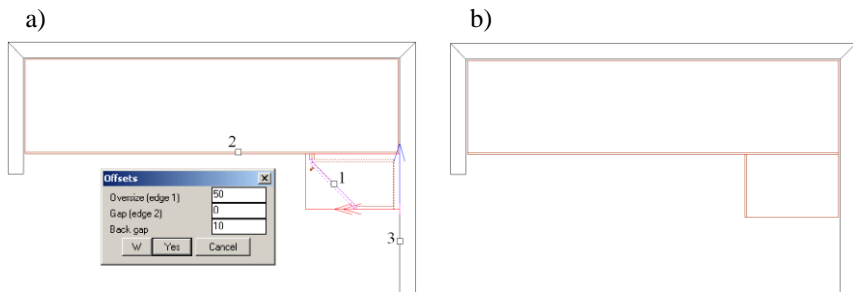
Fig. 3.17   a) Object selection to create the Bench Top and the
                 Bench Top preview.

             b) Construction result.

Note. To cut the Bench Top corner in compliance with product face, it is comfortable to use operation "Cut by surface" (**Operation >> For part >> Edit shape (contour) >> Cut by surface**) see fig. *3.18*.

List of Prompts
Specify point on remaining Part piece (for Bent part – the cut end): Specify the bench top, see fig. *3.18*, selection "1".

*by 3 points / redefine point #N / Cutting surface: Specify the door, see fig. *3.18*, selection "2"

Confirm object side.
To change direction - any key / ENTER - apply: Specify the cutting direction as an example and enter the gap in the dialog window. You should enter minus to do oversize.

a)   b)
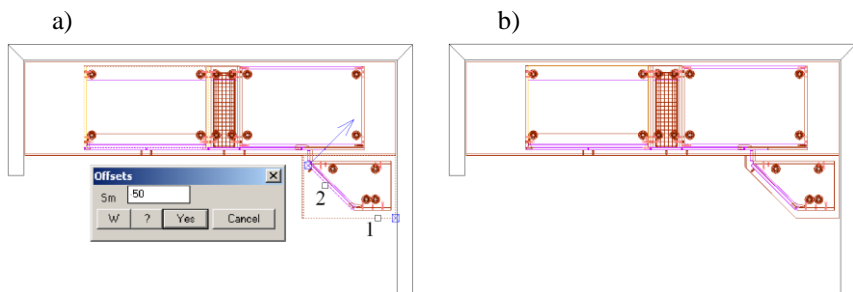
Fig. 3.18   a) Object selection to cut the Bench Top corner.

             b) Result of cutting.

Note. When you install the Bench Top using the "**Bench top on products**" and "**Group of bench tops**", the gap settings by default are set with a command **Service >> Settings >> Parameters of bench top setup**.

3.1.4.4. Group of bench tops

Access method
Menu: **Object >> Bench top >> Group of bench tops**
Toolbar: **Object**

There is a special command to set up a group of bench tops. The basic principles of this command corresponds to those discussed in section 3.1.4.3, page 45. This command feature is an automatic executing of the Bench Tops joint. The joint can be executed by one of the following methods: "by line", "bench top join", "cutting".

List of Prompts
Section <BT2> #C / select product to setup first Bench Top: Specify the first product, see fig. *3.19* a), page 51, selection "1". It determines the height of the Bench Top installation and its face edge.
Select wall (to setup against) <ENTER-next>: Specify the bounding wall of the first Bench Top piece, see fig. *3.19* a), page 51, selection "2"
Select wall (to setup against) <ENTER-next>: Specify the next bounding wall of the first Bench Top piece, see fig. *3.19* a), page 51, selection "3"
Select product to setup second Bench Top: Specify the second product, see fig. *3.19* a), page 51, selection "4". It determines the direction for the second Bench Top piece.
Select wall (to setup against) <ENTER-next>: Specify the bounding wall of the second Bench Top piece, see fig. *3.19* a), page 51, selection "5"
Select wall (to setup against) <ENTER-next>: Press **Enter** to skip this request.
In the next step you will see bench top preview and dialog window to specify gaps, see fig. *3.19* b), page 51. To revers joint, set "**Reverse join**" (the preview will not be redraw, but construction will be reversed). Enter gaps and oversize, then press "Yes" to complete construction. The result is shown in fig. *3.19* c) page. 51.

Note. You can change joint type in the dialog window (see fig. *3.19* b), page 51) but the preview will not be redrawn into the current command (it will be taken into account in next group of bench tops installation).

a)

b)

Offsets

Gap (edge of bench t.1)   10
Back gap Bench Top 1   10
Oversize (edge of bench t.2)   50
Back gap bench t.2   10
Join type
○ By line
○ Bench top profile
◉ Cutting
☑ Reverse join
    W   Yes   Cancel

c)
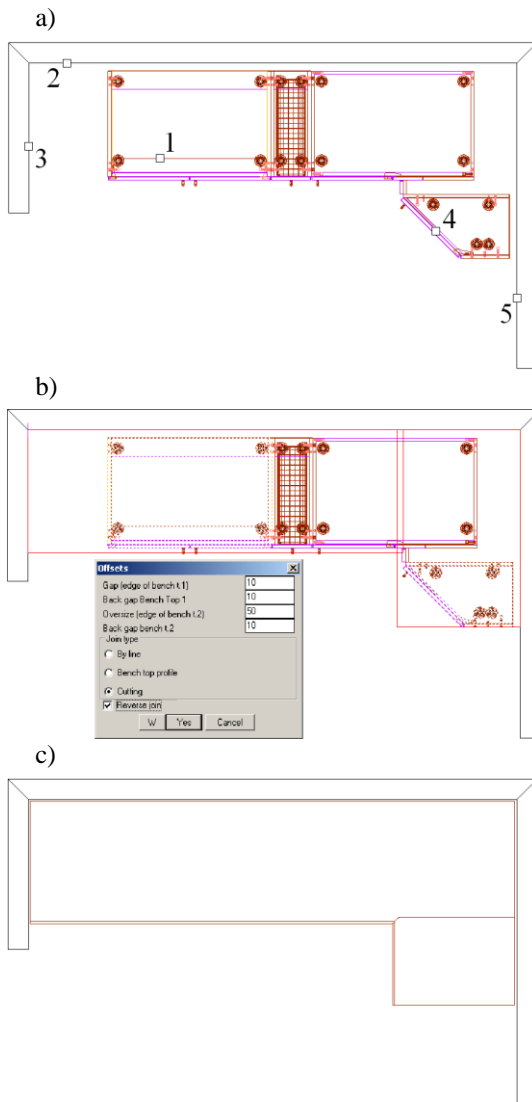
Fig. 3.19   a) Object selection.

b) Bench Top preview.

c) Construction result.

The different types of the bench top joint are presented in fig. 3.20.

a)   b)   c)
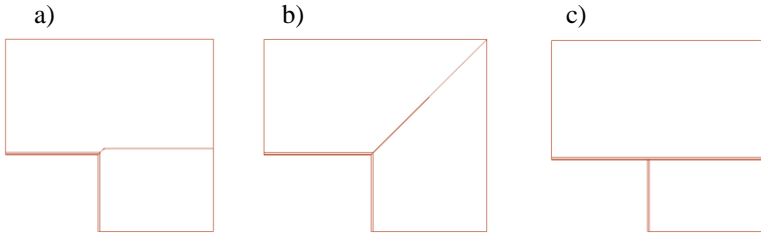
Fig. 3.20   Bench Top joint types

        a) Cutting.

        b) By line.

        c) By Bench Top profile.

To set the joint type by default, use **Service >> Settings >> Parameters of bench top setup**

To set the gap settings by default and joint type by default, use **Service >> Settings >> Parameters of bench top setup**.

To set the cutting parameters by default, use **Service >> Settings >> Parameters of bench top joint**.

3.2. Bent sheet

3.2.1. Generalities

Usually, this is multi-layer Sheet Part its surface is bent along soft path. Its surface sweeping is a rectangular. You can't do any operation to edit a Bent Sheet contour. Every Bent Sheet layer corresponds to one item in the sheet material database. The Bent Sheet thickness is determined by sum of its layers thickness plus auxiliary thickness (if auxiliary thickness is assigned by user, more details see in section 5.6.8, page 224). The Bent Sheet length is determined with its longer layer. The Bent Sheet price may be calculated (as sum of the layers price) or attributed.

Note. The height of a Bent Sheet is always determined with variable "H". This variable is automatically generated when you create a Bent Sheet.

The settings of Bent Sheet by default is determined with command **Service >> Settings >> Parts and Sheets**, area "**Bent sheet**".

3.2.2. Create Bent Sheet

3.2.2.1. Vertical

Access method
Menu: **Object >> Bent sheet >> Vertical**
Toolbar: **Object**

List of Actions and Prompts
Select path to create Bent Sheet, see fig. 3.21
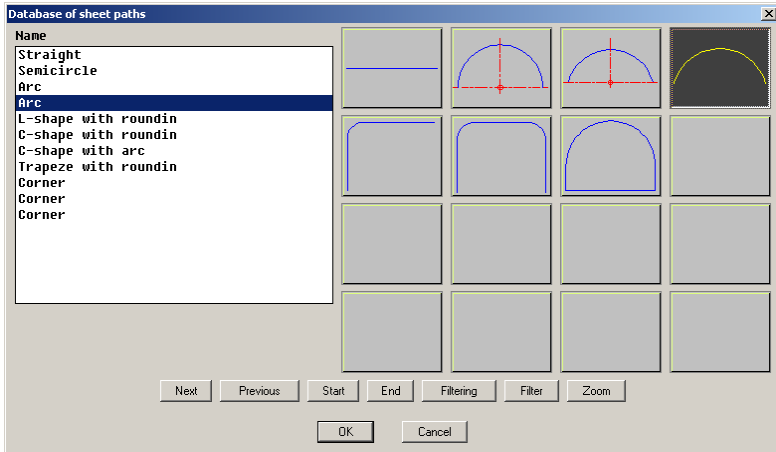
Fig. 3.21   Database of path

Enter the path parameters (in this case, the "L" and "W") and the variable "H" ("H" sets the object height) see fig. 3.22.
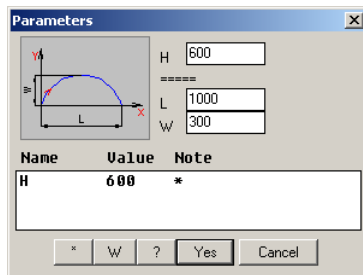
Fig. 3.22   Bent Sheet parameters

Insert point: Specify a point on the screen by mouse left-click

Rotation angle: Determine an angle value by mouse or enter value in the Command Line.

3.2.2.2. By drawn path

<u>Access methods</u>
Menu: **Object >> Bent sheet >> By drawn path**
Toolbar: **Constructions**
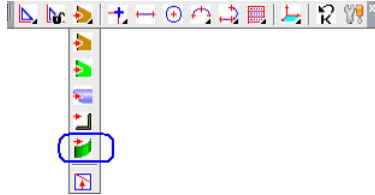
<u>Summary</u>
You can use a Constructional Drawing to make the Bent Sheet path .

Note. The path assigned by Constructional Drawing has to be soft and hasn't to include break points. Otherwise, you can't create the Bent Sheet.

3.2.2.3. By selection…

<u>Access methods</u>
Menu: **Object >> Bent sheet >> By selection…**

<u>Summary</u>
This command creates a Bent Sheet. The Bent Sheet path can be assigned with

- edge of Part
- bent sheet or profile
- Constructional Drawing
- AutoCAD drawing

Note. All path segments can be taken from one object (one Part, one Profile)

The command examined by creating object in compliance with edge of Part, see fig. *3.23*, page 55

<u>List of Prompts</u>
Selection: Specify an edge of the base object

All #D / Edge #H / *List of edges #W / Segment #E: Clarify selection (press **Enter** to take the sequence of edges)

List of edges

Determines the sequence of edges as the path. You can press **Enter** to activate it.

Segment

Determines the path from the base point to one of the points specified by user. The base point is highlighted by a square pick-box, see fig. 3.23 b). All points what you can take as the path end will be highlighted if you activate this option.

Edge

Determines only one specified edge as the path.

Select extruding direction.

To change direction - any key / ENTER - apply: Clarify *extruding direction, see* fig. *3.23* b).

Determine "H" variable and "Sm" offset parameter in the window, see fig. *3.23* c). The plus offset value corresponds to the arrow direction.

The result is presented in fig. 3.23 d).
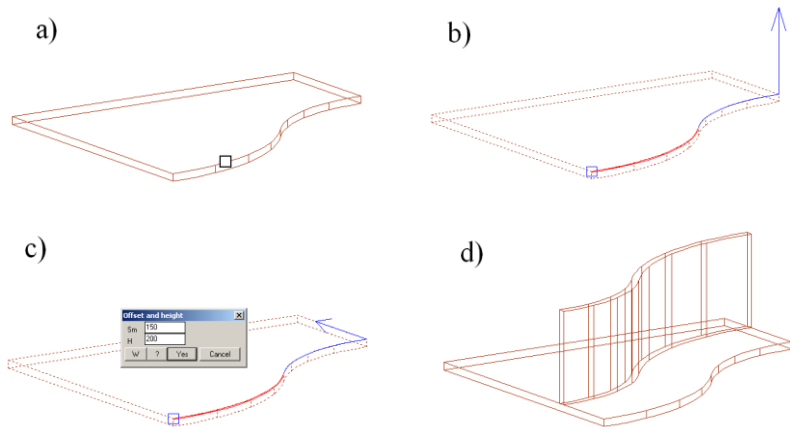
Fig. 3.23   Bent Sheet "By selection…"

The created Bent Sheet will be automatically referenced to base object.

Note. The path assigned by selection has to be soft and hasn't to include break points. Otherwise, it will be not possible to create an object.

To edit offset value, when object has created, use **Operations >> Parametrs**

To edit height of object, use **Edit >> Variables**

To edit length of object in the start or end path point, use **Operations >> Bent sheet operations >> Change length**

3.3. Profile

3.3.1. Generalities

To create a Profile, you have to determine its cross-section and extrude cross-section along the Profile path.

3.3.1.1. Profile cross-section

The Profile cross-section may be parametrical or drawn by user. The drawn cross-section hasn't parameters in the Profile database. The Profile cross-section has base point and four overall points. When you create a Profile, the base point always coincides with Profile path. When the Profile has created, you can reference it using one of overall points. The base point of a Profile cross-section is always in the origin of the cross-section coordinate system, see fig. 3.25 page 58. You can move the cross-section in the database relative its base point. To do this, use "Dy", "Dz" parameters, see fig. 3.25. Note, you will not see the cross-section moving in the slide, but it will be done in the 3D-model.

3.3.1.2. Offset for fillers SM1, SM2, SM3

Three pair of offsets may be set in the profile database relative base point: SM1Y and SM1Z, SM2Y and SM2Z, SM3Y and SM3Z, see fig. 3.25. Above mentioned offset names are variables embedded in a Profile. They will be automatically taken to create object or execute operation in specialized command:

Object >> Part >> Frame filler
Operations >> Profile operations >> Cut by profile

More details about these command see in section 3.1.2.6 page 39 and section 4.6.5, page 203.

The SM1Y, SM1Z, SM2Y, SM2Z, SM3Y, SM3Z allows program to create objects by profile and automatically immerse them in the profile. You can determine the offset conditions by command **Service >> Settings >> Profiles**, see fig *3.24*
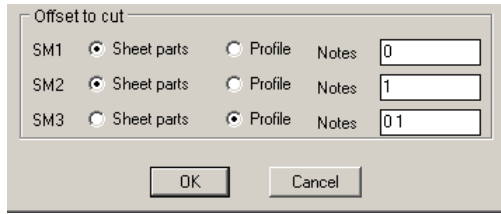
Fig. 3.24   The piece of dialog window "Profile settings"

Every item in the sheet material database and profile database has a "**Note**" field. If this field is empty, it is automatically assigned to note "0".

Let's consider next example.

There are chipboard and some mirror materials in the database. You marked all the mirror materials by note "1"; all chipboard materials leaved empty. The checkboxes (SM1, SM2, SM3) have state displayed in fig. 3.24 page 57. If you create frame filler by command **Object >> Part >> Frame filler**, for the mirror materials will be taken "SM2" and for the chipboard materials will be taken "SM1". An offset value specified in SM1 is set so that it differs from SM2 on the thickness of the seal, which is placed for a mirror. Thereby, if you create frame filler from different sheet materials in the same frame, the filler will have different sizes.

Attention! The program will choose an offset (SM1, SM2, SM3) if you create object by special command. (If you change material for existing object, it will not be done).

3.3.1.3. Profile path

To create Profile, you can take parametrical path from the database or create path yourself. The path shouldn't have break points, but it may have vertexes. It may be closed or opened.

This path includes break point   This path includes vertexes.
(In other word, these are two paths)   (It is not soft)

This path is opened   This path is closed

If the Profile path has vertexes, the Profile will consist of some segments. This will one object in the 3D-model (its code, for example, ASS.00.001) but it will be automatically split into segments in the Quotation

(the code of segments will ASS.00.001_1, ASS.00.001_2… ASS.00.001_N). Two or more segments, which are included in one profile and have equal lengths, are different for program (You will see two objects in the Quotation). If you would like to sum those segments, you should create them as different objects and join them by operations.

To edit the database of profile paths, use **Database >> Shape >> Database of profile paths**.
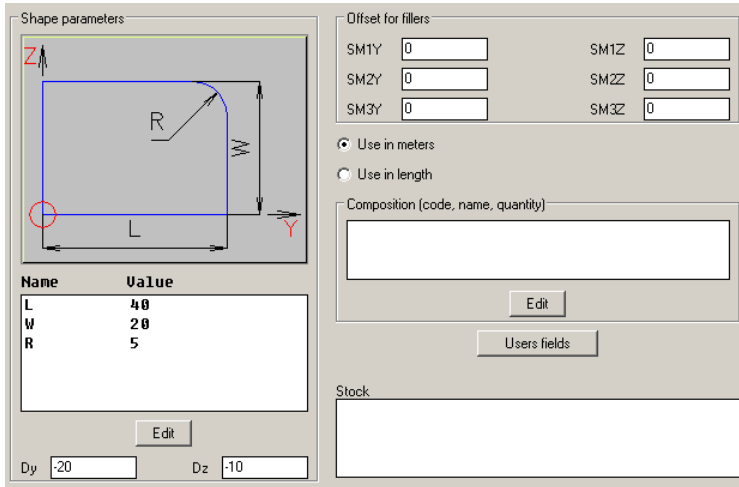
Fig. 3.25   The piece of the "Profile database" dialog window

(You see the Profile cross-section and its coordinate system in the slide.)

To edit the Profile database (Profile cross-section) use **Database >> Materials >> Profile database**.

To change the cross-section of an existing Profile, use **Editor >> Properties**.

To edit the path parameters (if you took parametrical path) use **Operations >> Parameters**

To determine the profile settings by default, use **Service >> Settings >> Profile**.

3.3.1.4. Profile cross-section created by user

To create a Profile cross–section and save it in the database, you should:

• Draw the cross-section. You can draw the cross-section using AutoCAD means (menu ACAD >>Drawing) or Constructional Drawing means (menu Operations >> Constructional Drawing operations). When you draw the cross-section, use scale setting 1:1.

Note. The drawn cross-section must be closed.

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- Combine all segments to have one polyline (It is comfortable to use **ACAD >> Edit >> Join in polyline**).

- Save the cross-section into the Profile database by command **Database >> Save user profile**.

3.3.2. Create Profile

The Profile settings by default are determined with command **Service >> Settings >> Profile**. Note, if the checkbox "**No current**" is filled in, you will select a Profile cross-section in every time to create an object.

3.3.2.1. Horizontal

Access method
Menu: **Object >> Profile >> Horizontal**
Toolbar: **Object**

List of Actions and Prompts
Section <BRAUN_JOIN> *C / *Next *L / select profile: Confirm the cross-section by default or specify a cross-section to create a new object. The cross-section code by default is in the Command Line, press **Enter** to confirm it. You can specify another cross-section by mouse left-click on an existing Profile in the project.

Select a parametrical path from the dialog window. The window corresponds to the window in fig. 3.21, page 53. Enter the path parameters and press "**Yes**".
Insert point: Specify a point on the screen by mouse left-click
Rotation angle: Determine the angle by mouse or enter value in the Command Line.

Section
Opens "Profiles database" to select a cross-section.

Next
Confirms the Profile cross-section by default.

3.3.2.2. By 3 points

Access method
Menu: **Object >> Profile >> Horizontal**
Toolbar: **Object**
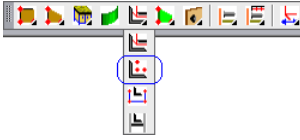
<u>List of Actions and Prompts</u>

*Path *G / *Section <BRAUN_JOIN> #C / Select profile #E / Base point:* Specify a base point of path

 *change point *N / Point of axis X: Specify X point of path
 *change point *N / Point of axis Y: Specify Y point of path
 change point *N / *Apply selected *G: Apply selected Y point by pressing **Enter**.

To change direction - any key / ENTER – apply: Apply direction to create the object.

Path

Selects a parametrical path from the Profile path Database. The straight line segment is a path by default.

Section

Displays the current Profile cross-section code. The last taken cross-section is a cross-section by default. You can specify a Profile cross-section from profile database if the option is activated

Select profile

Selects Profile cross-section. You can specify any existing profile in the project to take its cross-section to create a new object.

Change point
Redefines last specified point.

Puc. 3.26   Selection profile path from database.

The CS origin, X-axis, Y-axis are displayed in the zoomed slide. They have standard positions for each path.

a)   b)   c)

X

0

Y
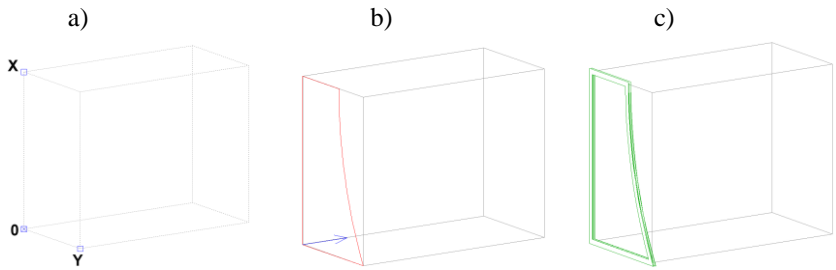
Puc. 3.27 a) Points specified as three base points to deter-
mine Profile path.

b) Preview for the Profile and direction to extrude
the object.

c) Construction result

Note. This command creates two variables by specifying points and set them to the path parameters. More details about variables see in section 6.2, page 281.

3.3.2.3. Profile by points of path

Access method
Menu: **Object >> Profile >> Profile by points of path**
Toolbar: **Object**

Summary
The Profile path consists of straight line segments. The first, the second and the third points of path determine the object plane they can't belong to one line. The other path points are projected onto a profile plane. The path points are specified by existing objects.

Note. If the Profile has created, you can edit the offset distance by **Operations >> Parameters**, see fig. 3.28 d).

List of Prompts
*Section <BRAUN_JOIN> *C / Select profile *E / first point of path: Specify first point of path.

redefine point *N / Cancel *J / next point of path: Specify next point of path.

……………………………………………………………..

redefine point *N / Cancel *J / next point of path: Press **Enter** to complete the path points specifying.

Mirror *P / Quadrant (any key) / ENTER - apply: Specify a cross-section orientation.

Section
Displays the current profile cross-section code. The last taken cross-section is a cross-section by default. You can specify a profile cross-section from the profile database if the option is activated

Select profile
Selects a profile cross-section. You can specify any existing profile in the project to select its cross-section to create a new object.

Redefine point
Redefines the last specified point.

Mirror and Quadrant
Changes the cross-section orientation concerning the profile path.

A construction example is presented in fig. 3.28.
The fig. *3.28* a) shows specified points and their sequence
The fig. *3.28* b) shows the profile cross-section orientation on the pre-view.
The fig. *3.28* c) shows the construction result.
The fig. *3.28* d) shows the **Operations >> Parameters** (You can set off-set distance of existing profile concerning profile path).
The fig. *3.28* e) shows the offset result.

a)

b)

c)

d)

e)
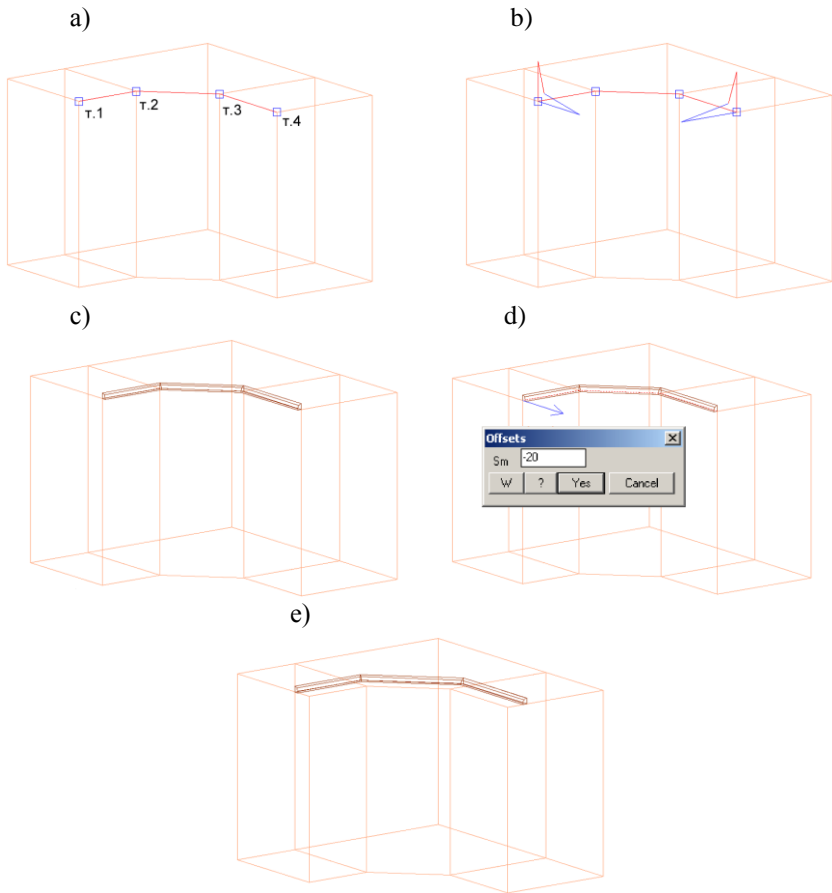
Fig. 3.28 An example of the profile construction

3.3.2.4. Between profiles

<u>Access method</u>
Menu: **Object >> Profile >> Between profiles**
Toolbar: **Object**

<u>Summary</u>
The command is comfortable to create auxiliary profiles between two existing Profiles. The path of creating Profile is always a straight line segment. After creating a new Profile it is automatically trimmed by base Profiles. If the

origins of the base profile coordinate systems belong to one plane, the new profile is trimmed with the operation "**Joint with a wall**" in another case it is trimmed with the operation "**Cut by profile path**". More details about operations see in section 4.6, page 201.

a)   b)
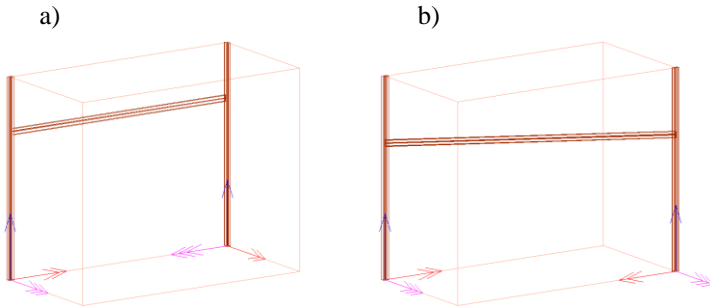
*Fig. 3.29   a)   The origins of the base profile coordinate systems
belong to one plane*

*b)   The origins of the base profile coordinate systems
belong to different planes*

Attention! The specified paths of the base Profiles should be parallel. The base Profile path may include arcs, but it should be specified a straight segment to successfully create a new object.

List of Prompts
*Section <BRAUN_JOIN> *C / Select profile *E / Select straight part of profile: Specify the first base Profile. Note, you set the base point of the new Profile through this selection.
Select straight part of profile: Specify the second base Profile.
Distance <335> *H / *Next *L / Another 0 point: Confirm or specify the base point of the new Profile. The base point is displayed on the screen by square pick-box. To confirm it, press **Enter**.
Mirror *P / Quadrant (any key) / ENTER - apply: Specify a cross-section orientation.
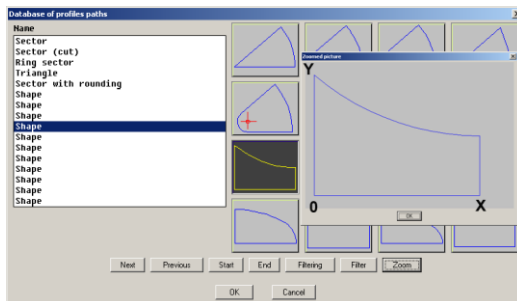
Section
Displays the current profile cross-section code. The last taken cross-section is a cross-section by default. You can specify a profile cross-section from the profile database if the option is activated

Select profile
Selects a profile cross-section. You can specify any existing profile in the project to select its cross-section to create a new object.

Distance

Sets the distance in millimeters from the base point to the Profile path. The current value is displayed in Command Line.

Mirror and Quadrant
Changes the cross-section orientation concerning the Profile path.

a)   b)   c)
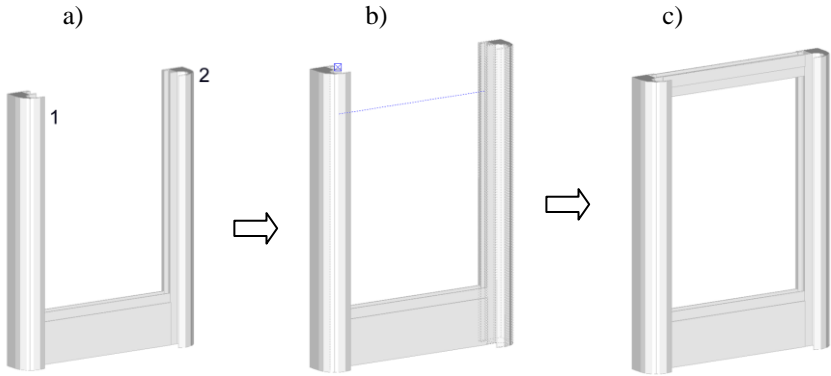
Fig. 3.30   a)   The firs and the second base profiles

b)   Base point and Profile preview

c)   Construction result

3.3.2.5. Between profiles by drawn path

Access method
Menu: **Object >> Profile >> Between profiles by drawn path**

Summary
The coordinate system (hereinafter CS) of a new profile is determined by two base Profile. If the base Profile path includes arcs, you should specify straight segment to successfully create a new object.
The plane of a new Profile has to be parallel to plan of the Constructional Drawing. The path in the Constructional Drawing has to cross the paths of the base Profiles, see fig. 3.31 a), and it shouldn't have break points.

List of Prompts
*Section <BRAUN_JOIN> *C / Select profile *E / Specify straight segment of Profile*: Specify the first base profile, see fig. 3.31 a), selection "1".
Specify straight segment of Profile: Specify the second base Profile, see fig. 3.31 a), selection "2"
Specify path by Constructional Drawings: Specify the path determined by Constructional Drawing, see fig. 3.31 a) and b).

To create an additional profile for the second door, you should repeat command and specify the profiles "3", "4" and path in the Constructional Drawing. The construction result is presented in fig 3.31 c).

Note. Description of additional options see in section 3.3.2.4, page 63.
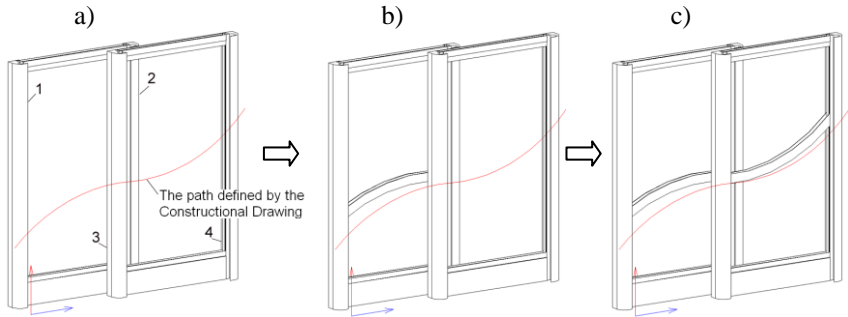
Fig. 3.31   Create profile between two profiles by drawn path.

Note. More details about Constructional Drawing see in chapter "Constructional Drawing".

3.3.2.6. By drawn path

Access method
Menu: **Object >> Profile >> By drawn path**
Toolbar: **Construction**
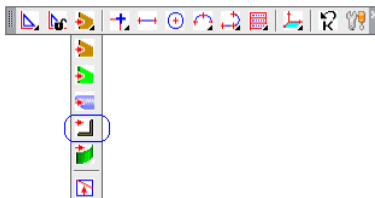
Summary
To create a Profile, you should draw a path by Constructional Drawing in advance and execute this command.

List of Prompts
*Section <BRAUN_JOIN> *C / *Next *L / select profile*: Specify a Profile to define cross-section of a new Profile or press **Enter** to confirm cross-section by default.

*Mirror *P / Quadrant (any key) / ENTER - apply*: Specify a cross-section orientation.
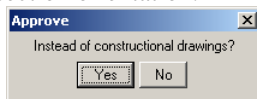
Click "No" if you will create other objects by the Constructional Drawing. Click "Yes" if the Constructional Drawing may be closed.

Note. Description of additional options see in section 3.3.2.4, page 63.

More details about Constructional Drawing see in chapter "Constructional Drawing".

3.3.2.7. By selection

<u>Access method</u>
Menu: **Object >> Profile >> By selection**

<u>Summary</u>
This command corresponds to command considered in section 3.2.2.3, page. 54.

3.4. Body of Revolution

The Body of Revolution is an object created by sweeping a cross-section around the cross-section X-axis. The cross-section is taken from the database of base forms.

There are two commands to create a Body of Revolution.

Object >> Body of revolution >> Horizontal axis –When you create an object, the Body of Revolution X-axis is parallel to XY plane of the World Coordinate System.

Object >> Body of revolution >> Vertical axis –When you create an object, the Body of Revolution X-axis is parallel to Z axis of the World Coordinate System.

<u>List of Actions and Prompts.</u>
- Activate command.
- Specify a cross-section from the database to create object, see fig. *3.32*.
- Enter in the dialog window the cross-section parameters and sweeping parameters, see fig. *3.33*:

ANG– sweeping angle

DX, DY– offset of the cross-section base point about the axis of revolution.

Angle– it determines the inclination angle of the cross-section X-axis relative to the rotation axis.
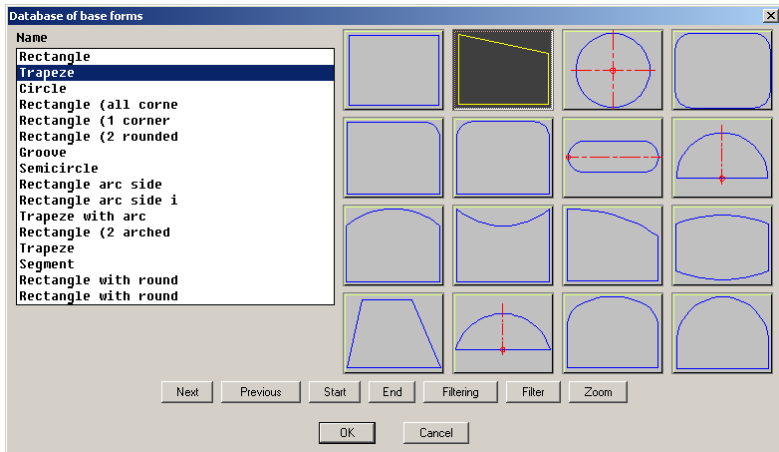
Fig. 3.32   The database of base forms

Note. The cross-section X-axis is always along horizontal edge of the slide.
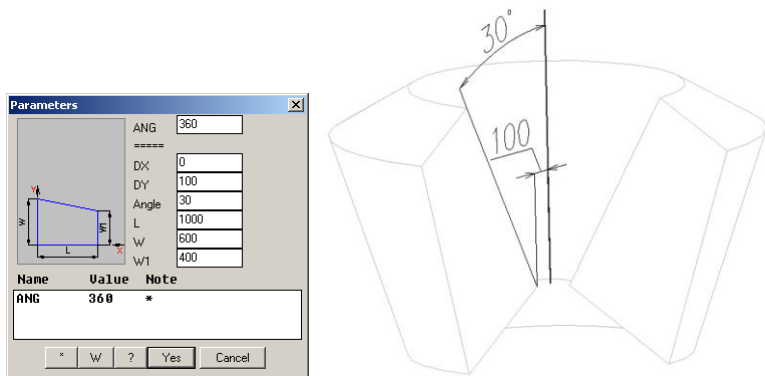
Fig. 3.33   Object creating with a vertical axis.

3.5. Blocks

You can create individual drawing files for use them as blocks. The examples of blocks are a dish rack, a basket, a wardrobe lift and other.

Every block has an insertion base point (it is determined when you make the block). When you insert the block into the drawing, the base point is used as the insertion point.

Every block has some parameters. The parameters quantity is determined by database. (The lags have "H" parameter; the blocks with overall dimensions have "H", "L", "W" parameters). The block parameters define its

base points in the 3D-Constructor. The block base points may coincide with overall block points or not.

Some databases of blocks ("Block with overall dimensions", "Miscellaneous hardware", "Legs (set up on surface)") include variables of block. If you set variable of block in its parameter, you will take block scaling by variable in the project.

3.5.1. Block insertion methods

Access method
Menu:
Object >> Block >> Blocks with overall dim.
Object >> Block >> From miscellaneous Database
Object >> Block >> Sink

Object >> Furniture fittings (handles, legs, miscellaneous) >> Handle 1
Object >> Furniture fittings (handles, legs, miscellaneous) >> Handle 2
Object >> Furniture fittings (handles, legs, miscellaneous) >> Leg on surface
Object >> Furniture fittings (handles, legs, miscellaneous) >> Leg on edge
Object >> Furniture fittings (handles, legs, miscellaneous) >> From miscellaneous Database

Toolbars: Objects
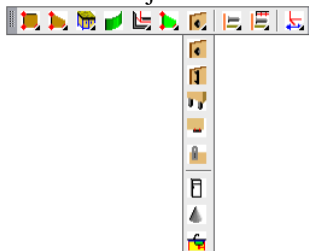
List of Prompts.
*Current <HANDLE1_1> *N / Select *D / Select example*: Specify the block to insert it in the project.

Current
Displays the current block code. The last successfully inserted block will set as the block by default. To confirm the block by default, press **Enter**.

Select
Selects a block from the block database, see fig. 3.34.

Select example
Allows you specify the existing block in the project to insert the same

on door #A / on surf. to direction #Y / on surf. to CS #C / without reference #G / on Bent sheet - point of bent sheet: Specify the method to insert the block.

Note. There are some methods to insert any block in the project. The available insertion methods of the specified block will be displayed in the Command Line when you activate the insertion command. The last successfully completed insertion method will set as the method by default.

Every insertion method is considered below.

Note. When the block is inserted, you can edit the block position by commands:

Edit >> Move Object
Edit >> Object reference >> Block position
Edit >> Object reference >> Reference parameters
And other commands from group "Object reference"
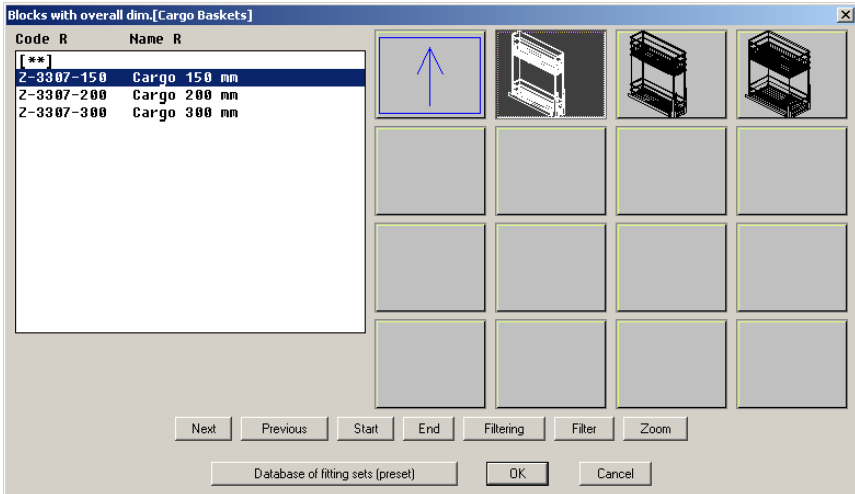
Fig.   3.34   Block selection from a database

Note! When you open a database to select a block it is available a button "Database of fitting sets (preset)". By pressing this button you will insert a group of blocks, which has conditions to insert, more details see in section 3.5.1.8, page 76.

3.5.1.1. Without reference

Summary
Executes a block insertion in the specified point on the screen

Insert point: Specify an insertion point by mouse left-click on the screen.

Rotation angle: Specify a rotation angle by mouse or enter a value in the Command Line.

Note. If you specify the rotation angle by mouse it is comfortable to set "Ortho Mode" by pressing "F8" on the keyboard. In "Ortho Mode" you may easy rotate the block around insertion point with step 90 degree.

When block has been inserted, you may move or reference it by one of the reference command, see section 6.1, page 259.

3.5.1.2. By points

Summary

Inserts and references a block by four points.

List of Prompts.

By Points - base point: Specify a block base point, see fig. *3.35* b), point "1".

*change point *N / Point of axis X: Specify a block X-direction, see fig. *3.35* b), point "2".

*change point *N / Point of axis Y: Specify a block Y-direction, see fig. *3.35* b), point "3".

*change point *N / Point of axis Z: Specify a block Z-direction, see fig. *3.35* b), point "4".

change point *N / *Apply selected *G: Complete command by pressing **Enter**.

Change point
Redefines the last specified point.
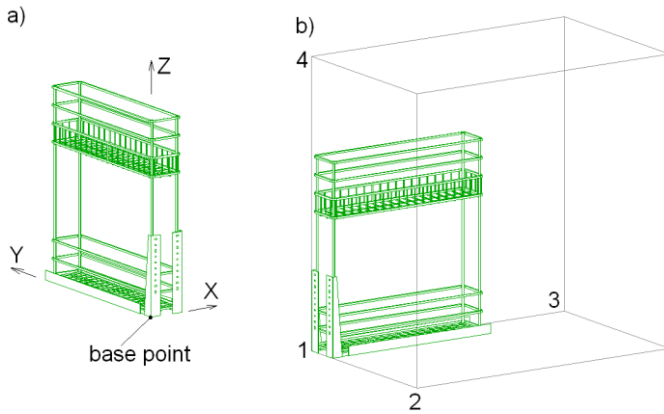
Fig. 3.35 a) Block and its coordinate system

b) Block inserted into the Assembly by four points.

3.5.1.3. On surface to CS

Summary

Inserts a block and references it on a Part surface. The specified Part surface and the block XY-surface are combined.

List of Prompts.

On surface to CS - point of surface: Specify a Part, see fig. *3.37* a).

Note. The block will be referenced to a Part point which is the nearest point to a specified point, see fig. *3.37* b) and c).

Confirm object side.

To change direction - any key / ENTER - apply: Define a Part base surface to insert the block. The base surface is highlighted with an arrow, see fig. *3.37* b). To change the base surface on opposite it, press any key. To confirm the base surface, press **Enter**.

Enter in the dialog window see fig. *3.37* c) reference parameters.

Note. You will see the dialog window, see fig. *3.37* c) if **Service >> Settings >> Blocks, Fittings, Hardware**, checkbox "**Use dialogue to setup**" is filled in. If it is empty, you will see the request presented below.

Insert point: Specify an insertion point by the mouse right-click or enter block coordinates in the command line.
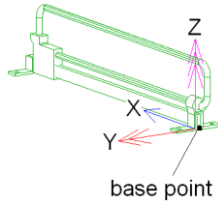
Fig. 3.36   Block and its coordinate system

a)   b)   c)   d)
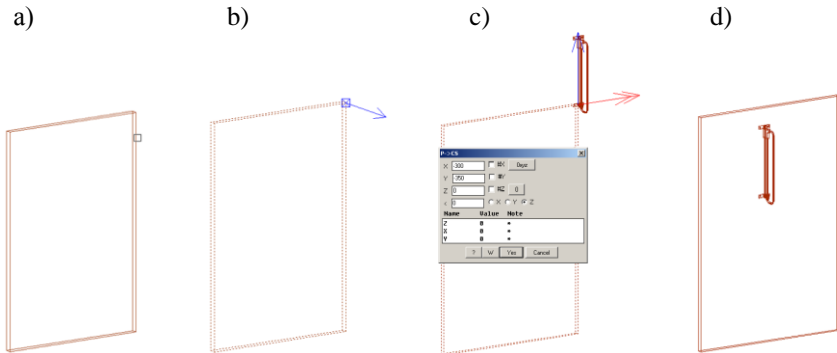
Fig. 3.37   Block installation by "On surface to CS" method.

3.5.1.4. On surface to direction

<u>Summary</u>

Inserts a block and references it on a Part surface. The specified Part surface and the block XY-surface are combined. The block X-axis is oriented along a specified Part edge. (Compare the fig. *3.37*, page 73 and fig. *3.38*, page 74). The block and its coordinate system are displayed in fig. *3.36*, page. 73.

<u>List of Prompts.</u>

On surf. to direction: Define a Part edge, see fig. *3.38* a).

Note. The specified edge determines a reference direction of the block.

Confirm selection (ENTER - confirm):

Confirm object side. To change direction - any key / ENTER - apply: Define a base Part surface to insert the block. The base surface is highlighted with an arrow, see fig. *3.38* b).

*Left-right *K / Rotate *G / *Next *L*: Define a block orientation and press **Enter** to continue. Enter reference parameters in the dialog window, see *3.38* c).

Left-rigth
Flips the block about Y-axis. It creates the symmetrical mirror block.

Rotate
Rotate the block about Z-axis on 90 degree.

Note. You will see the dialog window, see fig. *3.38* c) if **Service >> Settings >> Blocks, Fittings, Hardware**, checkbox "**Use dialogue to setup**" is filled in. Otherwise, you will be prompted insertion point, see section 3.5.1.3, page 72.

a)   б)   с)   д)
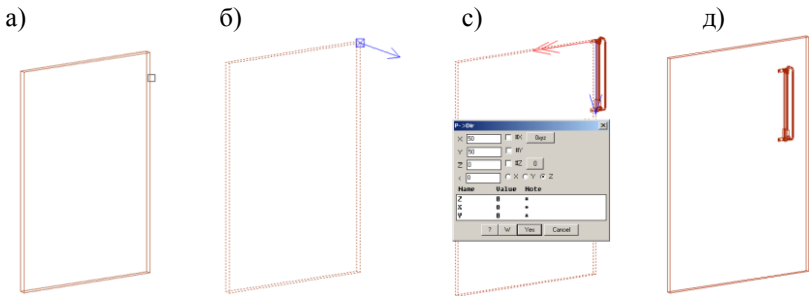
Fig. 3.38   Block installation by "On surface to direction" method.

3.5.1.5. On Bent Sheet

Summary
Inserts a block and references it on a Bent Sheet. The block is referenced along the Bent Sheet path.

List of Prompts.
On Bent sheet - point of bent sheet: Specify a Bent Sheet point, see fig. *3.39* a).
Note! The specified point defines the base Bent Sheet surface. (The block will be positioned relative to the base point so that it does not cross the Bent Sheet).

*change point #N / *Apply selected #G*: Confirm the base point by pressing **Enter** or redefine it, see fig. *3.39* b).

Enter in the dialog window block reference parameters, see fig. *3.39* c).
Note. The "X" parameter defines a block shift along the Bent Sheet path.

a)   b)   c)   d)
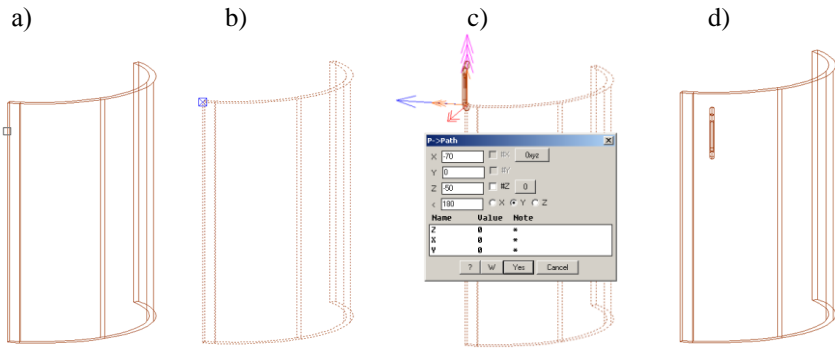
Fig. 3.39   Block installation by "On Bent Sheet" method.

3.5.1.6. On door

Summary

Inserts a block and references it in the center of a door. The door specified surface and the block XY-plane are combined. The door has to have the box to complete insertion by this method. More details about object box see in section 3.1.1, page 30.

List of Prompts.

On Door - select door: Specify a door, see fig. *3.40*.

Note. The block (the handle in fig. *3.40*) will be oriented along a specified edge.

Complete the block insertion by pressing "Yes" in the dialog window.

Note. You can disable an alignment along one of the door dimensions. To do it, replace "#" on a value in the "X" or "Y" parameter.
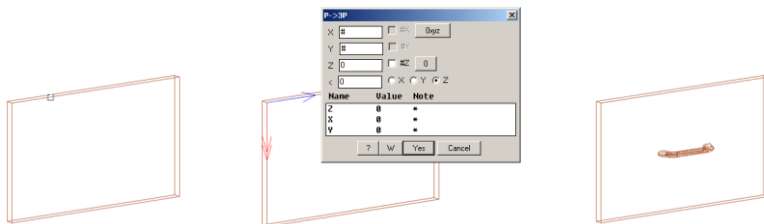
Fig. 3.40   Handle installation on the door

3.5.1.7. In volume

Summary

Inserts and references a block in a space defined by four or three specified Parts.

List of Prompts.

in volume - base part: Specify a base Part, see fig. *3.41*, selection 1.

Flat part 2 (parallel to 1-st): Specify the second Part, see fig. *3.41*, selection 2.

Part 3 (perpendicular to edge of 1-st part): Specify the third Part (it have to be rotated on 90 degree relative to the first Part), see fig. *3.41*, selection 3.

Flat part 4 (perpendicular to 1-st part): Specify the fourth Part, see fig. *3.41*, selection 4. You may press **Enter** to insert the block by three specified Part.

Note. To insert a block by specifying three Parts, you have to specify the first Part so, that the specified point is beyond the edge mid-point concerning the third Part, see fig. *3.41* a).

Enter reference parameters in the dialog window and press "Yes" to complete the command, see fig. *3.41* b).

Note. If the space bordered by Parts has the back Part, you will be able to center block concerning space depth. To do it, set "#" in the "Y" parameter and "#Y" checkbox, see fig. *3.41* b).

a)   b)   c)
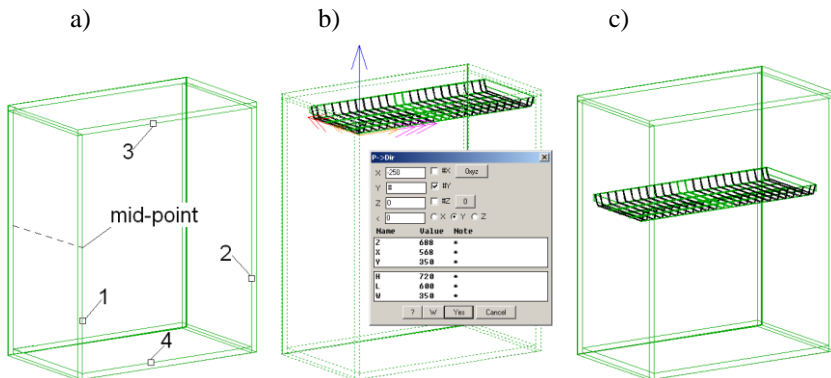
Fig. 3.41   Dish rack installation in the volume

3.5.1.8. Inserting blocks by condition

There is a way to insert the block or furniture by condition. It means that some blocks from the same base may be combined in one group. Each block in such group has its insertion condition. When you insert the block group in the

project it will be inserted only one block which will have the completed insertion condition.

Note, you should combine in a group single-type blocks which differ in one size and one condition to insert. For example, it may be cargo baskets with width 300 mm, 200 mm and 150 mm. Each basket has the insertion condition. The distance between Parts side has to be less than 300, 200, 150 mm, see fig. 3.42 ("X" is a block variable defined as the distance between the cabinet sides)

To combine some block in a group, use **Operations >> Block and furniture fittings operations >> Operations for size settings**. The operation has to be used for an existing block in the project. The block hasn't to have external links.

Follow next steps to combine some blocks in a group:
- create a new project
- insert a block in the project
- combine some block by mentioned above operation. Use one of the block variables to determine the block insertion conditions.
- save the blocks group in a database

To save the created blocks group in the database, use command **Database >> Save block or fitting in preset database**.

To insert a blocks group in the project from the database, press button "**Database of fitting sets (preset)**", see fig. 3.34, page 70
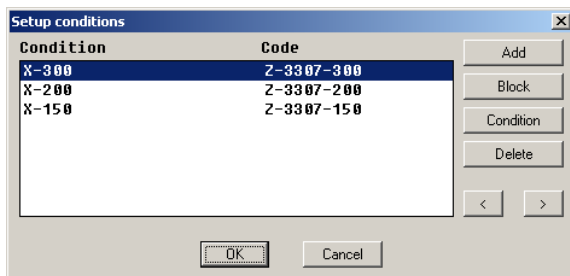
Fig. 3.42   Block insertion conditions

Add
Adds a new block in a block group from the database

Block
Replaces the selected block with another block from a database

Condition
Defines a block insertion condition. The condition is a mathematical expression, which may include numbers, block variables, assembly variables (if

block is included to the assembly). The condition is satisfied if the entered expression is greater than or equal to zero.

Note. If the blocks group includes some blocks and their insertion conditions are satisfied, it will be inserted the first block from the block list which will have satisfied condition, see fig. 3.42. The block list is checked from top to bottom.

Note. The insertion conditions of the blocks group are checked at the time of group insertion in the project. (When you save block group in the database or edit it in the database, the condition correctness will not be checked. In another word, you may imagine the insertion condition which will never be satisfied.)

Delete
Deletes highlighted item from the block group, see fig. 3.42.

To edit the blocks group in the database, open the database "Fitting sets pre-set" (command **Database >> Fittings, Hardware, Blocks**, select base "**Fitting sets pre-set**").

Note. When you insert a blocks group in the project, all blocks will be embedded in the project (You will see one of them in the Quotation but you will see all in the "Control material" dialog window. To open "Control material" window, execute **Edit >> Replace material**).

3.5.2. Blocks with overall dimensions

Access method
Menu:
Object >> Block >> Blocks with overall dim.

Toolbars: Objects
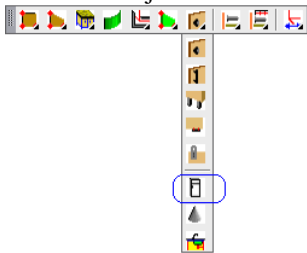
3.5.2.1. Block parameters

Every block with overall dimensions has three parameters in the database, see fig. *3.45 b),* page 82:
"H" – block height
"L" – block width
"W" – block depth

These parameters define eight block points (anchor block points) relative to insertion point (see fig. *3.45 a), points 1, 2...8*).

Note. "H", "L", "W" parameters define the overall dimensions parallelepiped of the block (in other words the parallelepiped circumscribed about the block)

You can set zero in the block parameters. In this case some points (corresponded to zero parameters) will be absent. For example, if the block presented in fig. 3.45, page 82 has H=0, the points 5, 6, 7, 8 will be absent.

Attention! The block parameter have to correspond to the product of the real block size and scale coefficient

For example, **if**:
- cargo sizes (width x depth x height) see fig. *3.45, page. 82,* are 240 x 480 x 530,
- scale coefficients are SX=1, SY=1, SZ=1,
- base points of the cargo have to coincide with its overall dimensions,

  then you should set L=240, W=480, H=530.

If, under the same other conditions, SX=1.25, you should set L=300 (L=240*1.25=300)

It is conveniently for some blocks to set basic (anchor) points so that they don't coincide with its overall points. For example, it is convenient to set fridge basic points according to its body size, see fig. 3.43 (the handle is not included in the parameters). The fridge will be attached to other objects according to body (not according to the handle).
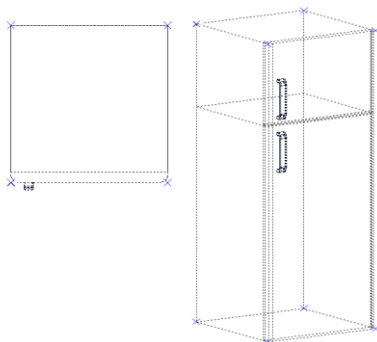
Fig. 3.43 The fridge block and its base point

3.5.2.2. Block variable

The block with overall dimensions has three variables X, Y, Z in the database. It is convenient to use block variable when block size may be changed in the project. For example, a wardrobe lift may be inserted in a wardrobe aperture width 450-600 mm.

When you insert a block in a rectangular opening ("By points", see section 3.5.1.2, page 71 or "In volume", see section 3.5.1.7, page 76):

- "X" block variable is determined by opening size along the block X-axis (it convenient to "L" block parameter)
- "Y" block variable is determined by opening size along the block Y-axis (it convenient to "W" block parameter)
- "Z" block variable is determined by opening size along the block Z-axis (it convenient to "H" block parameter)

In the case of wardrobe lift you should set "X" variable in "L" parameter, see fig. *3.44* and set X=600 mm (it is the product of the real block size and SX coefficient).

Note. You can set parameter diapason by expression in the field "Value". For example, it can be Lisp-expression which includes X variable: [IF (AND (>= X 450) (<= X 600)) X 600]). More details about variables see in section 6.2.5, page 291
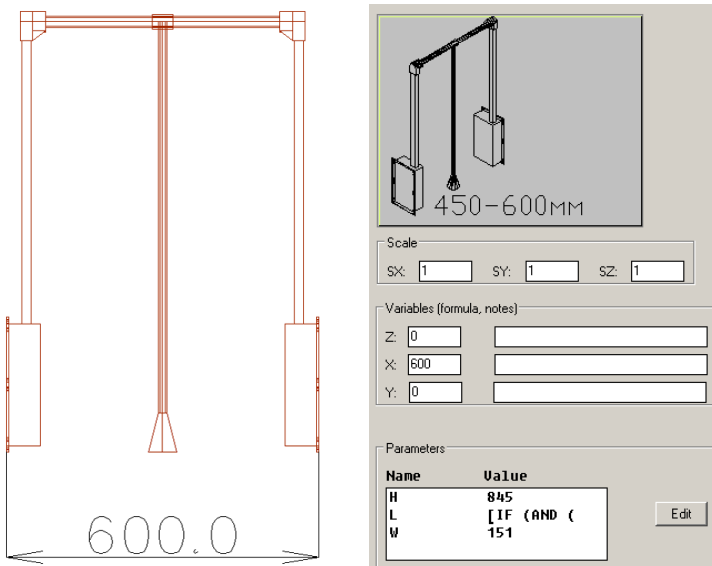
Fig. 3.44   The wardrobe lift block (front view) and the dialog window fragment of the block database

3.5.2.3. Block processing

The block with overall dimensions can drill the Parts which coincide with the block surfaces. The block surfaces are defined by its base points. This object can have six surfaces. They are named in database as:

X – It corresponds to surface "3487", see fig. 3.45, page 82

-X – It corresponds to surface "1265",

Y – It corresponds to surface "2376",

-Y – It corresponds to surface "1485",

Z – It corresponds to surface "5678",

-Z – It corresponds to surface "1234",

The coordinate system origin of the every surface is defined by the block insertion point or its projection in the surface. You can determine processing in any three surfaces, see fig. *3.45 b*).

Attention! You have to do some conditions that the block drills a Part:
- the Part surface and block surface (which has processing) have to be coincided;
- the Part has to be situated outside the block box defined by H, L, W (it hasn't to cross block);
- holes determined on the block have to be projected into the Part contour;
- a mid-point of the block surface has to be projected into the Part contour, for example, see fig. *3.45 a*) point "O".

There is example in figure 3.45. The block processing is defined in direction "-Y". It corresponds to "1485" surface. To have holes in the Part coincided with "1485" surface, the point "O" has to be projected into the Part contour.

Attention! The block processing defined on the one block surface can be projected into one Part. It will be the first Part found by program which satisfies the mentioned above conditions.

Note. If the block parameter is expressed through a block variable, see section 3.5.2.2, page 80, you may use the variable to define a hole coordinates. For example, to have two holes allocated along the block width into "1584" surface, set the first hole coordinates (50, 25) and the second hole coordinates (X-50, 25).

b)
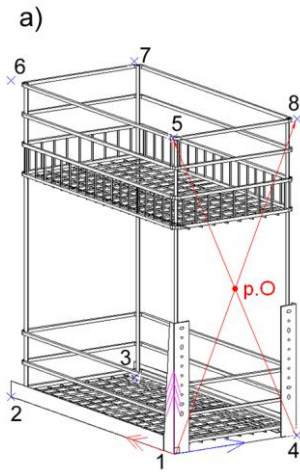
Fig. 3.45 a) Cargo block.

б) Dialog widow of the database.

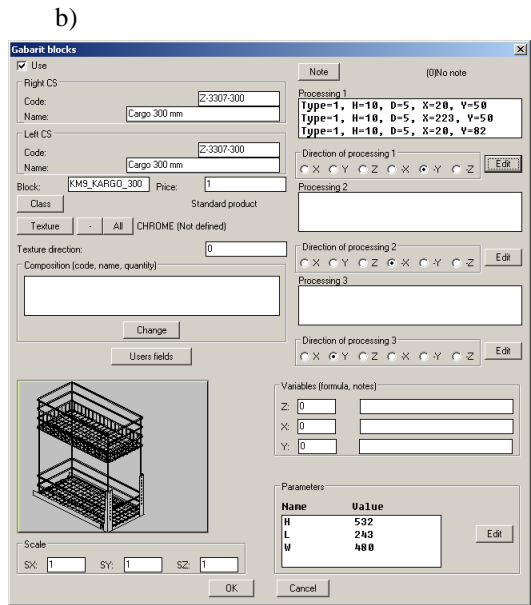The coordinate system (in hereafter CS) for each block surface, where you set the block processing, is presented in fig. 3.46.

a)   b)   c)   d)
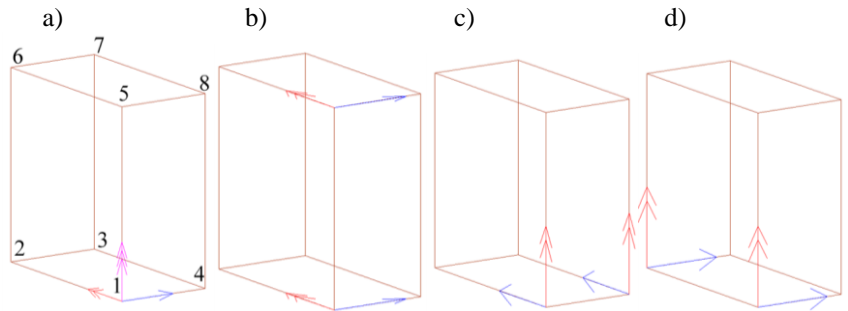
Fig. 3.46   a) Block with overall dimensions and its CS
b) Block CS of the surface "1234" and "5678"
c) Block CS of the surface "1265" and "4378"
d) Block CS of the surface "1584" and "2673"

3.5.3. Blocks from miscellaneous database

Object >> Block >> from miscellaneous Database

Toolbars: Objects
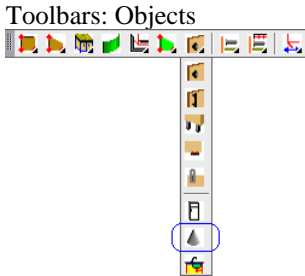
Note. The block from the miscellaneous database is identical to the furniture fittings from the miscellaneous database. But blocks are included into "Construction order of the main object"; fittings are included into "Construction order of fitting and hardware". In other words there are main objects and auxiliary objects. The fittings are auxiliary objects. If it is rationality to include the block in the block database or in the fitting database, you should decide yourself.

3.5.3.1. Block parameters

Every block from the miscellaneous database has five parameters in the database, see fig. 3.47 and fig. 3.48.

"H"   – block height
"L"   – block width
"W"   – block depth

Note. "H", "L", "W" parameters determine the overall dimensions parallelepiped of the block (in other words the parallelepiped circumscribed about the block)

"DX" – determines an offset along X-axis for the parallelepiped circumscribed about the block relative to the block insertion point
"DY" – determines an offset along Y-axis for the parallelepiped circumscribed about the block relative to the block insertion point

Note. If "DX" and "DY" are positive, the block insertion point is inside the parallelepiped circumscribed about the block.

These five parameters define from one to ten block points relative to insertion point see fig. 3.47 and fig. 3.48. The entity of points is determined by user for every block separately.

Note. If block will have an overall dimension points ("H", "L", "W" nonzero), the command **Edit >> Magnet** works correctly.

Note. To understand this topic perfect, see section 3.5.2.1, page 78.

H=60, L=42, W=200, Dx=21, Dy=15
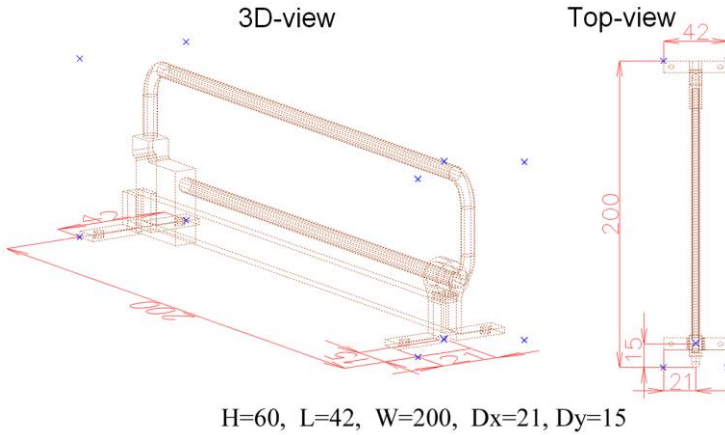
Fig. 3.47   Block which has ten points

H=60, L=0, W=200, Dx=0, Dy=15
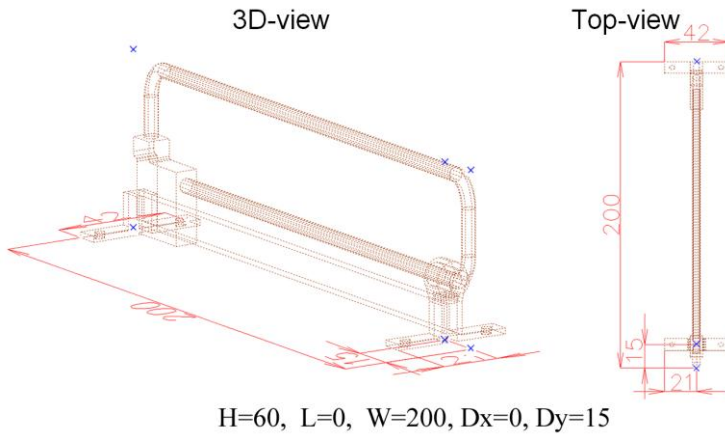
Fig. 3.48   Block which has six points

3.5.3.2. Block processing
The block from the miscellaneous database can drill the holes in the Part which coincide with the block XY–surface. More details see in section 3.5.2.3, page 81.

3.5.4. Sink

<u>Access method</u>
Menu:
Object >> Block >> Sink

Toolbars: Objects
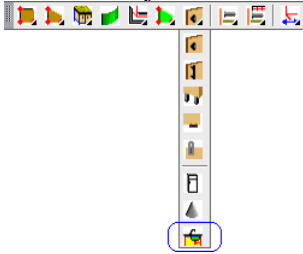
<u>Summary</u>
The block of the kitchen sink can be created and saved in the sink database. The kitchen sink has two parameters "L" (sink length) and "W" (sink width). These parameters determine XY-block surface and four block points (anchor points). More details about parameters see in section 3.5.2.1, page 78.
A feature of the sink database is a possibility to save sink block and its cutting contour. In other words you can make opening in the Part according to the sink form.
Example of object from the sink database is a kitchen sink, a gas cooker, and different trays.
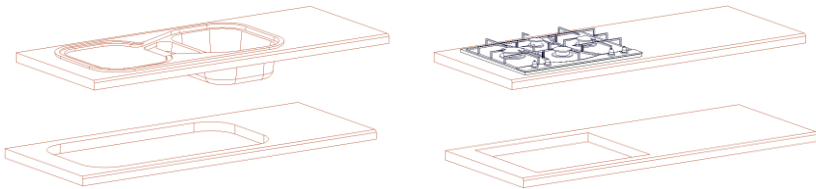
Fig. 3.49   Blocks from the sink database

To cut Part according to sink, use the command **Operations >> For part >> Edit shape (contour) >> Subtract part contour**

Note. There are some conditions to cut successfully:
- The editable Part has to be situated in the construction order higher than the Sink (the Part have to be created first)
- The editable Part surface and sink surface have to be parallel.

Note. The sink cut is parametrical. If you move the sink, the sink cut will be automatically recreated according to new position. If you replace the sink, the sink cut will be automatically recreated.

Note. If you replace the sink to another sink which hasn't a cut contour in the database, you should delete the existing cut from the Part. In any case,

when you try to redraw the Part with the cut, you will get the message "Impossible to build an element".

3.6. Furniture fitting and hardware

There are two groups of objects in the program. These are main objects and auxiliary objects. The program forbids deleting main object if there is another main object referenced to it. The program allows deleting main object if there is another auxiliary object referenced to it. The furniture fittings, hardware and fasteners are auxiliary objects.

Note. The furniture fittings and hardware are identical to blocks, but they are auxiliary objects according to construction order.

The special features of furniture fittings and hardware
- You may delete the object which a fitting or hardware is referenced to
- The fitting or hardware which lost base object will be automatically deleted
- The fitting or hardware can be referenced to another fitting or hardware
- The main object (Part, Block, Bench Top…) can't be referenced to a fitting or hardware

Note. The automatic deleting of the fitting and hardware is allowed with the checkbox "**Delete on error**", **Service >> Settings >> Blocks, Fittings, Hardware**, the area "**Hardware**", checkbox "**Delete on error**"

The insertion methods of the fitting and hardware are considered in section 3.5.1, page 69.

Every fitting or hardware can have processing which is defined in the database. You get holes in the Part drawing if some conditions are satisfied:
- The Part surface has to coincide with the fitting or hardware XY-surface
- The fitting or hardware has to be outside the Part body
- The fitting or hardware insertion point has to belong to the Part surface

Attention. There is an addition condition for handles and legs. The handle or leg has to be referenced to the Part.

3.6.1. Handles

There are two handle databases:
- Handles 1 (handles with one mounting point)
- Handles 2 (handles with two mounting points)

Any handle has one base surface (XY-surface).

Handle 1
Handle 1 has two base points (anchor points). The one of them is the insertion point and another is determined with "H" parameter. The inserted Handle on the Part can drill the Part (the hole will be automatically generated in the Part drawing). The hole type and parameters are determined by user in the database.

Handle 2
Handle 2 has six base points. "1" is the insertion point, see fig. *3.50*. "2", "3", "4" are determined with "H" and "L" parameters. "5" and "6" are mid-points. It may be comfortable to use mid-point to center the handle. The inserted Handle 2 on the Part can drill two holes in the Part. The holes positions are determined with "1" and "2" point.
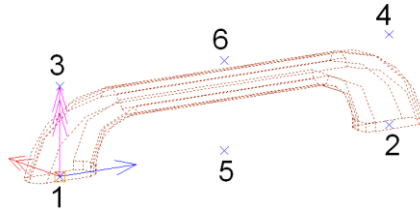
Fig. 3.50 Example of the Handle 2

Note. The item from the Handle 1 database may drill a hole or rout a hole. The item from the Handle 2 database may drill two holes. If a handle installation requires another processing type, for example a pocket, you should include it in the "Miscellaneous hardware" database.

Access method
Menu:
Object >> Furniture fittings (handles, legs, miscellaneous) >> Handle 1
Object >> Furniture fittings (handles, legs, miscellaneous) >> Handle 2

Toolbars: Objects
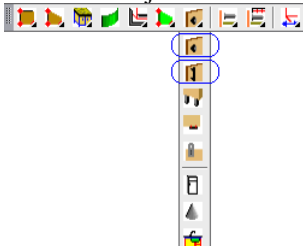
3.6.2. Legs

All legs according to their function are divided into two databases:
- Legs (setup on surface)
- Legs (setup on edge)

Any handle has one base surface (XY-surface) and two base points. The points are determined with "H" parameter in the database.

Note. There is "Z" variable in the "Legs (setup on surface)" database. "Z" variable give you a possibility to create an adjustable leg. More details see in section 3.5.2.2, page 80.

Access method
Menu:
Object >> Furniture fittings (handles, legs, miscellaneous) >> Leg on surface
Object >> Furniture fittings (handles, legs, miscellaneous) >> Leg on edge

Toolbars: Objects
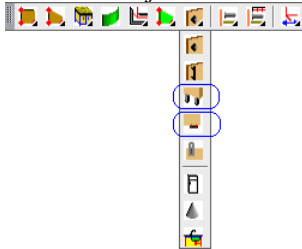
3.6.3. Fitting from miscellaneous database

The description of a fitting from miscellaneous database is identical to the block from miscellaneous database (but fitting is auxiliary object), see section 3.5.3, page 83 and section 3.6, page 86.

Access method
Menu:
Object >> Furniture fittings (handles, legs, miscellaneous) >> From miscellaneous Database

Toolbars: Objects
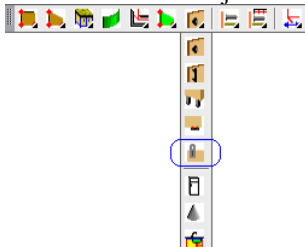
3.7. Applied objects

The applied objects are objects which are a special case one of the mentioned above objects (through some properties, through a creation method). There are next applied objects:

- Shelf,
- Divider,
- Back or Bottom Part,
- Door,
- Drawer
- Tamburat

3.7.1. Divider

The Divider is a horizontal or vertical Part (It may be Sheet Part or Extruded Part). When you create a Divider, you should specify two parallel Parts. Two specified parallel Parts and the Back determine the Divider size. The Divider is jointed to specified Part without gaps. (The gap is a break between the divider edge and the Part surface).

To determine a special name for horizontal and vertical Divider, execute **Service >> Settings >> Parts and Sheets**, area "**Partition**", button "**Name vert.**" and button "**Name horiz.**"

3.7.1.1. Divider

Access methods
Menu: **Object >> Part >> Divider**
        Object >> Rectangular Part >> Divider
        Object >> Extruded Part >> Divider

Toolbar: **Object**

Part	Rectangular Part	Extruded Part
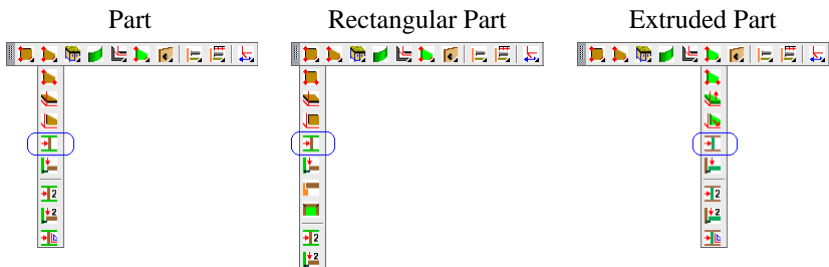
Summary

Creates

- one Divider moved from base point according to specified distance;
- some Dividers, which are placed at equal intervals along the specified distance.

Below is an example of one Part creation.

List of Actions and Prompts
Execute **Object >> Part >> Divider**
Select the Part contour from Database, see fig. 3.8, page 35.

Note. Use **Object >> Rectangular Part >> Divider** to skip the form selection.

ODO (box) + / first part (wall): Specify the first Part, see fig. 3.51 a), page 92.
Note. You specify the Divider face edge by specifying the first Part. Specify the face edge of the first part to create the Divider correctly, see fig. 3.51 a), page 92. (In any case the Divider face will be in the back of the wardrobe)
Note. If the first and the second Parts are different in width, the width of the first specified Part will determine the Divider width (The Divider length is determined with distance between the first and the second Parts).
Note. If the first Part is vertical, the base point by default will be bottom point of the specified edge. If the first Part is horizontal, the base point by default will be determined as the nearest point to a specifying point.

ODO (box) + / second part (wall): Specify the second Part, see fig. 3.51 a) page 92.
Note. You may specify a Part, a Frame edge of an Assembly, an edge of a Constructional Drawing as the second Part.

*distance <186.79918> #H / centering #W / 2 / 3 / 4 / N / next #L / Another 0 point: Determine another base point or confirm the highlighted base point by mouse right click.

Enter distance between the Divider base point and the Divider, see fig. 3.51 b) page 92.

Enter Divider parameters in the dialog window, see fig. 3.51 c), page 92. You skip this step if you will execute **Object >> Rectangular Part >> Divider**.

ODO (box)

Specifies what point will be taken. It may be object point or box point (more details about box see in section 3.1.1, page 30). To specify point of object, must be displayed "*ODO (box) +*". To specify box point of object, must be displayed "*ODO (box) -*".

Note. If "*ODO (box) -*" is displayed and the specified object hasn't box, it will be taken the object point.

Distance

Displays current distance between the base point and the Divider. Activate option to determine the distance in the dialog window.

Centering

Places a Divider at equal intervals along the line. The line is an imaginary line determined by two specified points on the screen.

2, 3, 4, N

Creates two, three, four and N Dividers and places Dividers at equal intervals along the line. The line is an imaginary line determined by two specified points on the screen.

Next

Creates the Divider in a point specified by mouse on the first Part.

Another 0 point

Permits to determine another Divider base point. The current base point is highlighted on the screen.

a) Part 1 (the face edge)
Back
the specifying point
Part 2

b) base point

Distance			
Distance			200
Name	Value	Note	
X	484		
Y	500		
H	800	*	
L	1000	*	
W	500	*	

W ? Yes Cancel

c) 3D-view

d)

Parameters			
X		P_Self	
Y		P_Edg	
=====			
L		X	
W		Y:100	
	+ ODO (box)		
Name	Value	Note	
X	484		
Y	500		
H	800	*	
L	1000	*	
W	500	*	

* W ? Yes Cancel

Top view

200

100
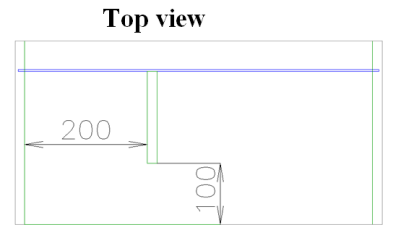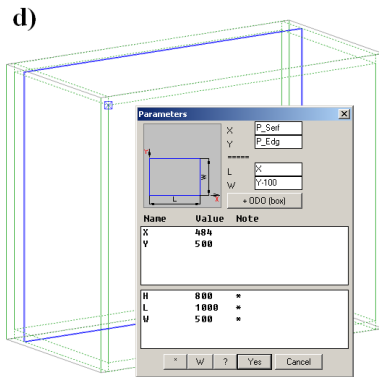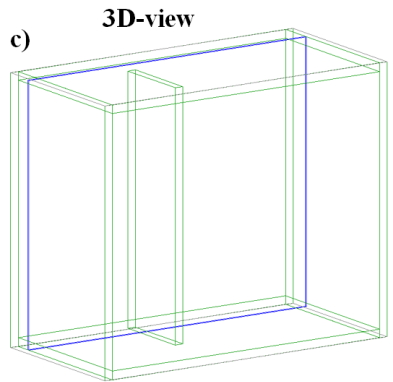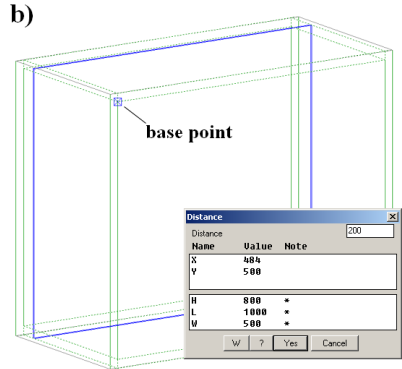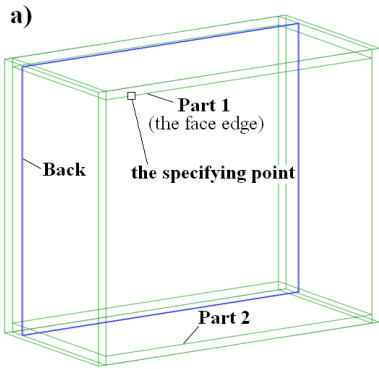
Fig. 3.51 Example of the Divider creating

3.7.1.2. Divider (variant 2)

Menu: **Object >> Part >> Divider (variant 2)**
            Object >> Rectangular Part >> Divider (variant 2)
            Object >> Extruded Part >> Divider (variant 2)

Toolbar: **Object**

Part	Rectangular Part	Extruded Part
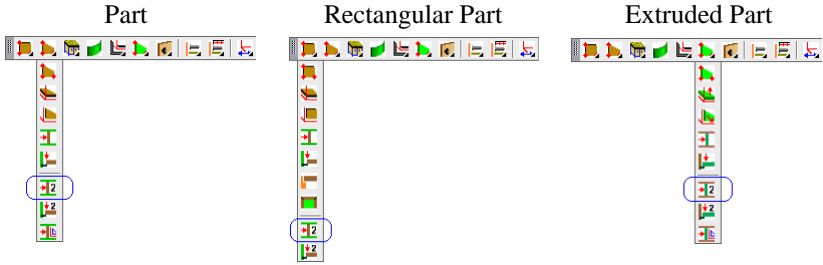
Summary

These commands are identical to the commands considered in section 3.7.1.1, page 89. But these commands calculate the Part parameters according to the specified edge of the first Part, see fig. *3.52*
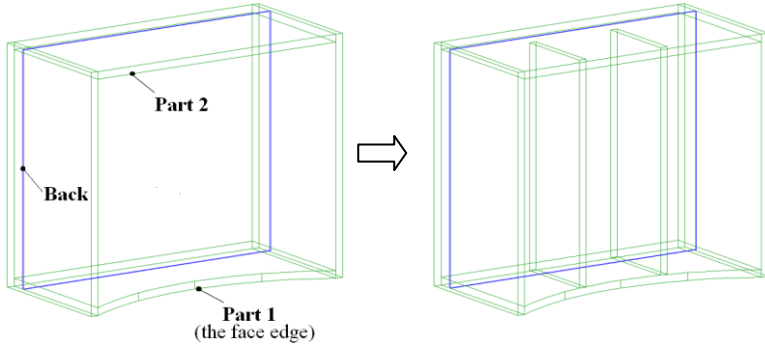
Fig. 3.52   Example of the Divider (variant 2) creating

Note. When you specify the Part 1, it should be specified the arc edge of the Part 1 to create Dividers according to the arc edge.

Attention

The Divider (variant 2) parameters calculated according to the edge can't be edited as usually. (Usually, you can edit the Part parameters in the "Parameters" dialog window by **Operations >> Parameters**)

To edit Divider (variant 2) parameters in the "Parameter" dialog window:

- type space character in the parameter string and then enter new value or expression;
- type an expression which looks as: "initial value", " arithmetical action", "auxiliary value"
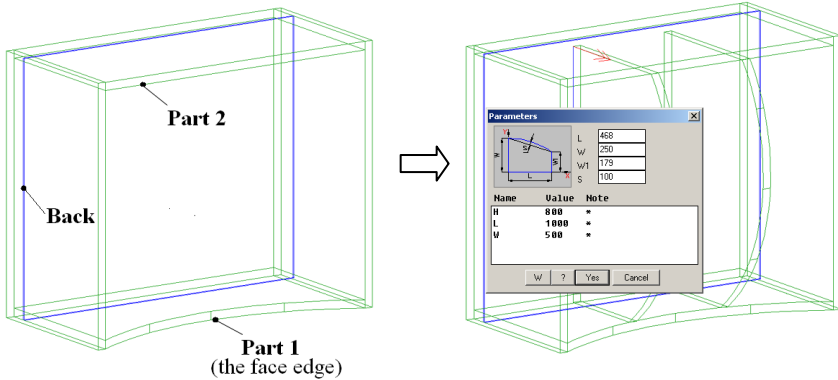
Fig. 3.53   Example of the Divider (variant 2) creating

There are two Parts in fig. *3.53*. Their parameters "L", "W", "W1", "S" were calculated according to installation conditions. The sequence of the Part specifying is free for this Divider form.

Note. To increase calculated "S" value from "100" to "150", type " 150" ("space" and "150") in the "Parameter" window (it is presented in fig. *3.53*). To decrease "W" and "W1" on 30, type in the "Parameter" window "250-30" and "179-30" (250 and 179 are calculated value).
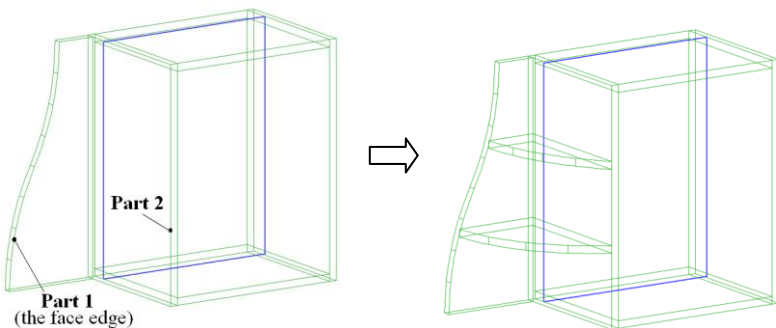
Fig. 3.54   Example of the Divider (variant 2) creating

The list of prompts to create Divider (variant 2) corresponds to it considered in section 3.7.1.1, page 89.

3.7.2. Shelf

Summary

The Shelf corresponds to the Divider but it has in advance determined front and side gaps. To specify Shelf gaps, execute **Service >> Settings >> Parts and Sheets**, determine the "**Gaps**" field (side gaps) and the "**Front gap**" field in the "**Shelf (sheet and executed)**" area. In the "**Shelf (sheet and extruded)**" area you can also determine the Shelf name by default and the state of the Shelf box by default (checkbox "Overall dim. of object"). More details about "box" see in section 3.1.1, page 30.

Access methods
Menu: **Object >> Part >> Shelf**
        Object >> Rectangular Part >> Shelf
        Object >> Extruded Part >> Shelf
Toolbar: **Object**

Part	Rectangular Part	Extruded Part
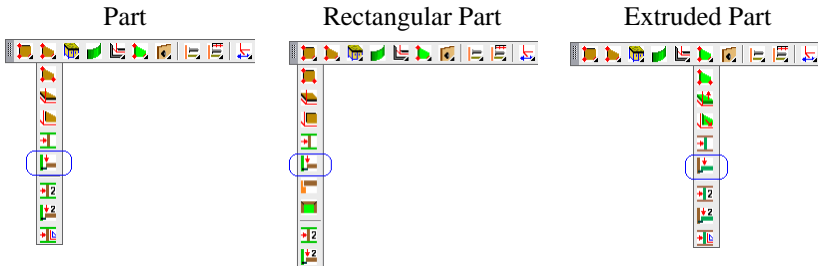
Menu: **Object >> Part >> Shelf (variant 2)**
        Object >> Rectangular Part >> Shelf (variant 2)
        Object >> Extruded Part >> Shelf (variant 2)

Toolbar: **Object**

Part	Rectangular Part	Extruded Part
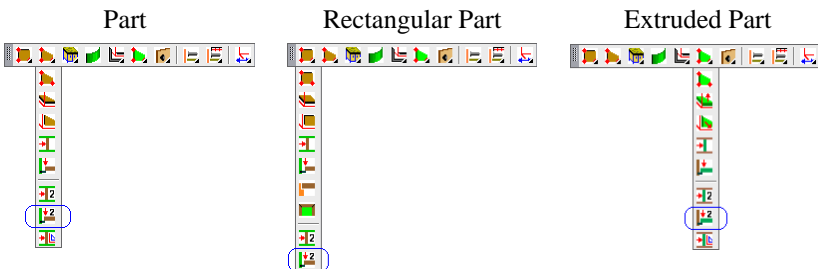
The list of prompts to create Shelves corresponds to it considered in section 3.7.1.1, page 89.

3.7.3. Divider (construction)

Part   Extruded Part
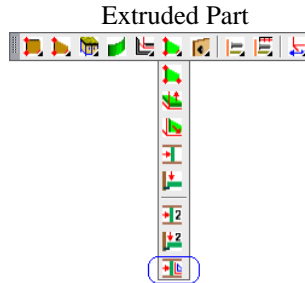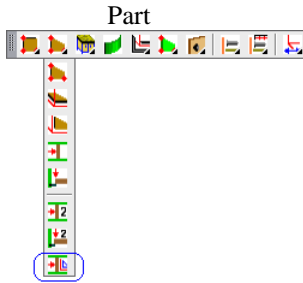
Summary

The Divider (construction) is created with Constructional Drawing. To create the Divider, you should specify two Parts or two Parts and a Back. The back may be found automatically or specified by user (if it wasn't found automatically).

To successfully create the divider, the basic Parts have to satisfy only one condition: their surfaces have to cross the Divider surface (they may be given a tilt, rotated, but they have to cross the Divider, see fig. 3.55)

Note. The Divider (construction) is created perpendicular (rotated on 90 degree) to the plane of the first specified Part.

To create the Divider (construction), the command creates automatically:

- Constructional Drawing
- a joint of Constructional Drawing and base Parts
- an auxiliary straight line edges to close a contour of the Divider (construction)

More details about the Constructional Drawing see in chapter "Constructional Drowing".

The list of prompts to create a Divider (construction) corresponds to it considered in section 3.7.1.1, page 89. The construction example is presented in fig. 3.55
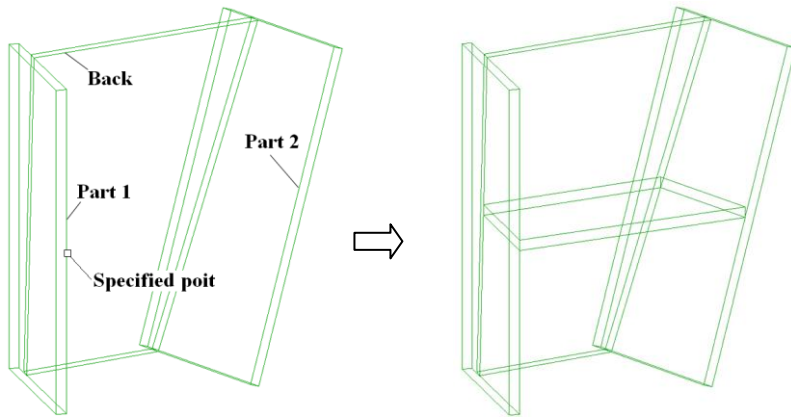
Fig. 3.55   Example of the Divider (construction) creating

3.7.4. Back

Access methods
Menu: **Object >> Rectangular Part >> Back**

Toolbar: **Object**
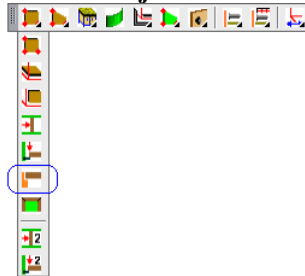
Summary
There are some typical variants to mount Back. To select a variant by default, execute **Service >> Setting >> Parts and Sheets**, press button "**Variant on back setup**" (see "Sheet part" area). Here you can determine also the Back name by default. To determine the variant parameters by default, execute **Service >> Settings >> Back settings (variant)**.

Attention! If there is "MZ" logical material in the Assembly where is created a Back, the Back material will be determined with "MZ". If there is not "MZ" logical material in the assembly or a Back is created outside of the assembly, the Back material will be determined with the first item from the sheet material database.

<u>List of prompts:</u>

Base part: Specify the first Part, see fig. 3.56, the specifying "1"

Note. The specified Part edge determines the Back place.

Part 2 (parallel to 1-st): Specify the second Part, see fig. 3.56, the specifying "2"

Part 3 (perpendicular to edge of 1-st part): Specify the third Part, see fig. 3.56, the specifying "3"

Part 4 (perpendicular to 1-st part): Specify the fourth Part, see fig. 3.56, the specifying "4", or press **Enter** to create the Back by thee Parts.

Enter the Back parameters in the dialog window, see fig. 3.56. Press "Ok" to complete the command.

Note. To change the Back variant, press slide by the left mouse click.

Note. Activate the "up to 0.5" checkbox to create the Back to the middle of the Part thickness, see fig. 3.56, Part 2. The Part numbers in the 3D-model and in the dialog window are displayed.
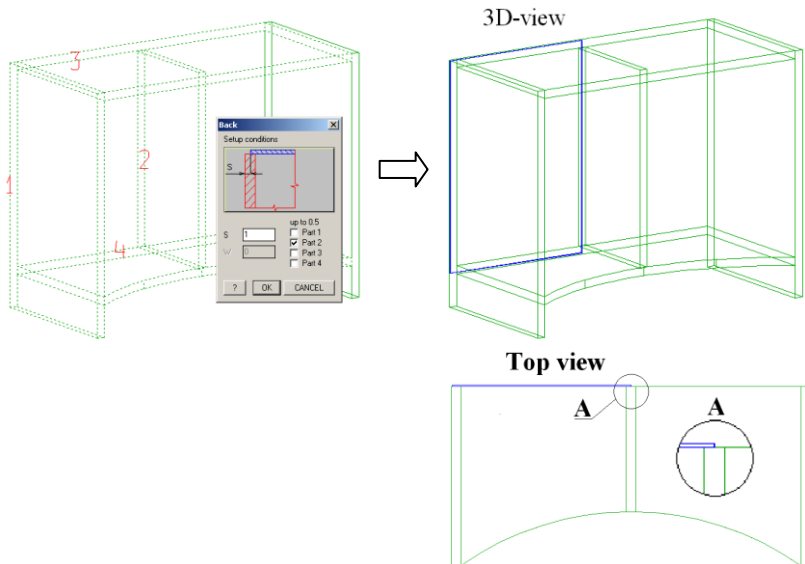
Fig. 3.56. Example of the Back creating

Note. If you create the Back by three Parts, the first specified Part will determine the Back length, see fig. 3.57. The point specified on the first Part has to be behind the middle of an edge relative to the third Part. The distance between the base point of the first Part and the plane of the third Part determines the Back height. The base point of the first Part is the nearest Part point to the specified point, see fig. 3.57
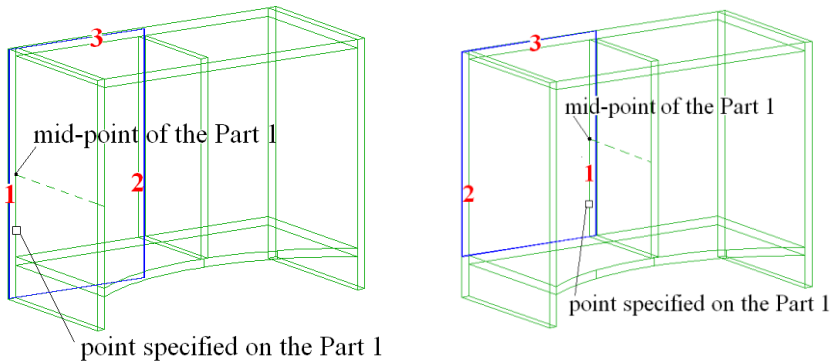
Fig. 3.57 Example of the Back creating by three Parts

(Part 1 and 2 are specified in a different order in the right and left figure)

3.7.5. Toe Kick

<u>Access methods</u>
Menu: **Object >> Rectangular Part >> Toe kick**

Toolbar: **Object**
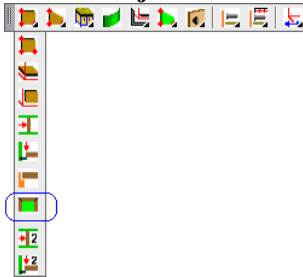
<u>Summary</u>
Creates some straight line segment of a Toe Kick. This command creates the Toe Kick from the chipboard plate.

Generalities:
- The command creates one or more Parts (chipboard plates)
- You draw a Toe Kick by specifying products points or Assembly points. (The assembly points are the points of objects which belong to one assembly).
- The command creates a new Assembly-point and automatically includes all Toe Kick Parts in the Assembly.

- Assembly for a Toe Kick is created at the most top level of the project structure (except for a case when you activate "in Assembly" options). A class of Toe Kick Assembly is a "Set".

Note. In case of "Set" class, the Toe Kick is built on several products at the same time.

- The height of the Toe Kick Parts is equal. Also the Parts have equal oversize for the cut-list and offset from the path line.
- The joint of the Toe Kick Parts is created by "Angular corner joint" operation.
- To edit the all Toe Kick Parts, use variables of the Toe Kick Assembly (the Assembly in which Parts are included). You can use any means for editing each Part separately.

List of prompts

in assembly #C / object point #N / intersection #G / Insert point: Specify the first Toe Tick point, see fig *3.58* a), point "1"

Note. The first point is a base Toe Kick point (The Toe Kick assembly is created in the base point).

Note. If you specify an object point it will be automatically taken the nearest frame point (the point of the Assembly frame).

object point #N / intersection #G / Insert point: Specify next Toe Kick point, see fig *3.58* a), point "2"

cancel #J / object point #N / intersection #G / Insert point: Specify next Toe Kick point, see fig *3.58* a), point "3"

cancel #J / object point #N / intersection #G / Insert point: Specify next Toe Kick point, see fig *3.58* a), point "4"

Enter "height" and "offset" in the dialog window, see fig *3.58* a), and click "Yes". The result is presented in fig. *3.58* b).

In assembly

Includes the Toe Kick into a specified product. The product is determined by specifying the first Toe Kick point.

Object point

Takes points of the specified object as a path point (the Assembly frame points are taken by default).

Intersection

Takes a nonexistent point received by intersection of two objects edges as a path point.

Cancel

Cancels the last specified point.

a)

b)
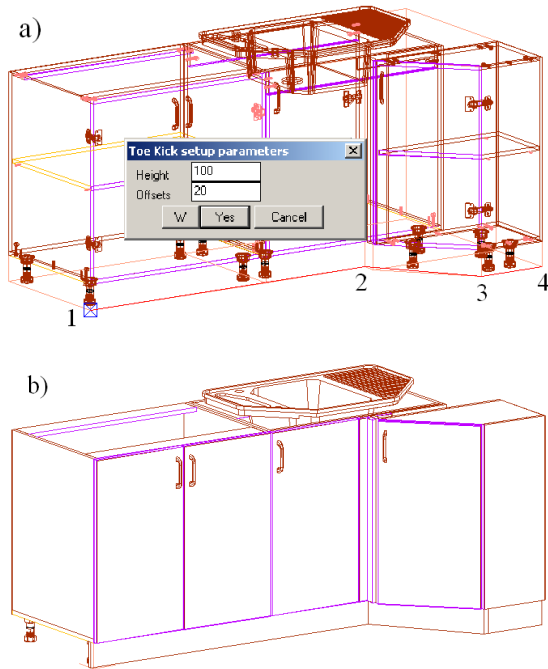
Fig. 3.58   a) Specifying the path points

b) Construction result

Note. When you specify the second and next path points they are pro-jected in the plane determined by the base path point.

Note. To edit Parts order in a Toe Kick joint, use **Operations >> For part >> Joints cuts grooves >> Convert joint oblique angle**, more details about this operation see in section 4.5.7.3, page 196.

To determine the Toe Kick parameters by default, use **Service >> Settings >> Toe kick settings**.

3.7.6. Doors

There are three types of Doors in the program:
- Part Door
- Extruded Door
- Framed Door

Part Door

It is equal to a Part. Its thickness, price and material determined by chip-board material.

Extruded Door

It is equal to an Extruded Part. Its thickness is free (it is determined by user through the variable "H"). Its colour and price determined by user in the door database.

Framed Door

This is an Assembly which may include Parts, Profiles, Fittings, Fasten-ers and other. Its price can be determined in the Door database or calculated according to price its components. (If the field "Price"=0 in the Door database, the Door price will be calculated according to price its components. If the field "Price"=some value in the Door database, the Door price is calculated accord-ing to price per 1 sq. m. determined in the database). The Frame Door splits into components in the Quotation.

Note. It is comfortable to lock Framed Door in the project (there is "Lock after insert" checkbox in the Door database for Framed Door). The locked Door is interpreted by program as one integer object (the user can speci-fy only frame points and edge).

Note. Any type of doors has overall dimensions of object (box). The door gaps are determined with the box variable. More details about object box see in section 3.1.1, page 30.

3.7.6.1. Door insertion (from the Door database)

Access methods
Menu: **Assembly >> Door**

Toolbar: **Assembly**
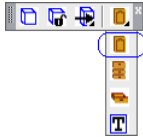
Summary

There are some methods to insert a Door from database. The easiest way to insert a Door is a "No reference" method. If you will use any other methods, you should define a door area. The door area is a rectangle concerning which you determine Door gaps. It is determined by external Part edges if you create an external Door, and it is determined by internal Part edges if you create an internal Door. The door area may be determined by specifying two or three or four Parts. If you create an external Door, you may use "up to 0.5 wall thick-ness" checkbox to determine a Door area concerning half a Part thickness.
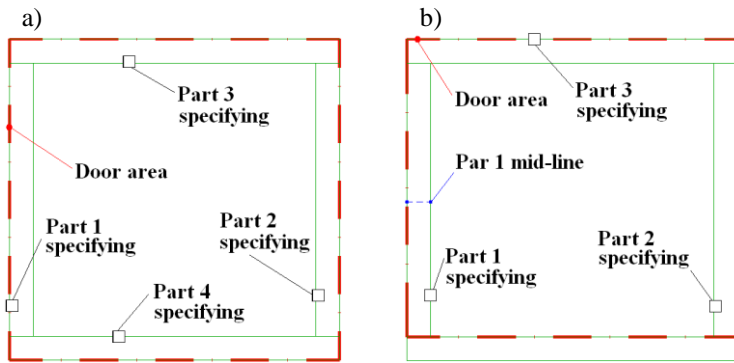
The external Door area is specified with four Parts.

The external Door area is specified with three Parts. The specified point on a Part 1 should be behind the Part 1 mid-point in relation to a Part 3
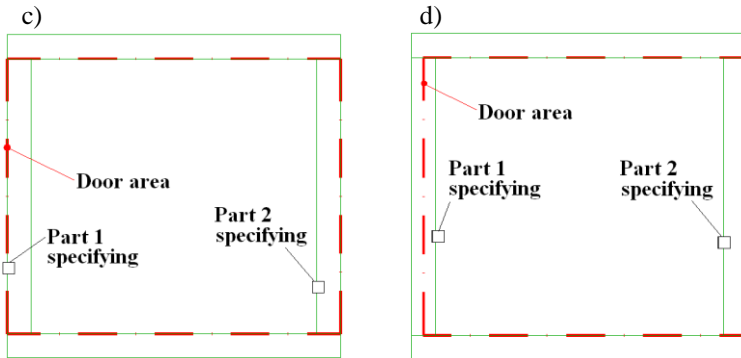
The external Door area is specified with two Parts.

The external Door area is specified with two Parts and "up to 0.5 wall thickness" checkbox for the Part 1

Fig. 3.59 Front view of a cabinet. The Door area determination

Note. When you specify Parts, you may specify internal or external Part edges. Insertion of an external or an internal Door is determined with setup method, see fig. *3.61*. The specifying point place on a Part matters only for a Part 1 when you determine the Door area by three Parts.

Note. When you will create a Door, you may restrict a height of Door area by "Height" field in the dialog window, see fig. *3.61*

Note. The specifying order of the Part 1 and Part 2 determine if it will be created a right or left Door. It is very important when you insert an asymmetrical Door, see fig. *3.60*.

Important. The Door gaps specified in the dialog window, see fig. *3.61* ("S top", "S bottom", "S side") limited Door size relative to Doo area. The

Door gaps are determined by box variables, see section 3.1.1, page 30. (Every door has a box).
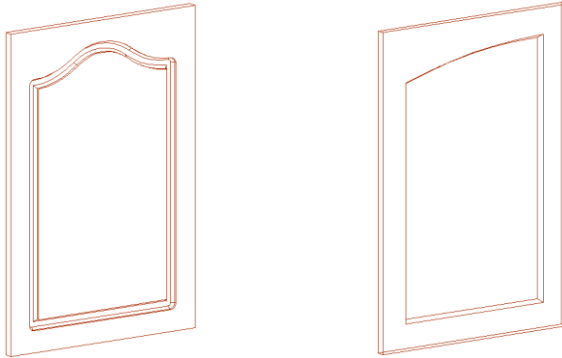
Fig. 3.60   Symmetrical and asymmetrical Door

List of prompts and actions
Execute **Assembly >> Door**
Specify an insertion method and gaps in the dialog window, see fig. *3.61*.
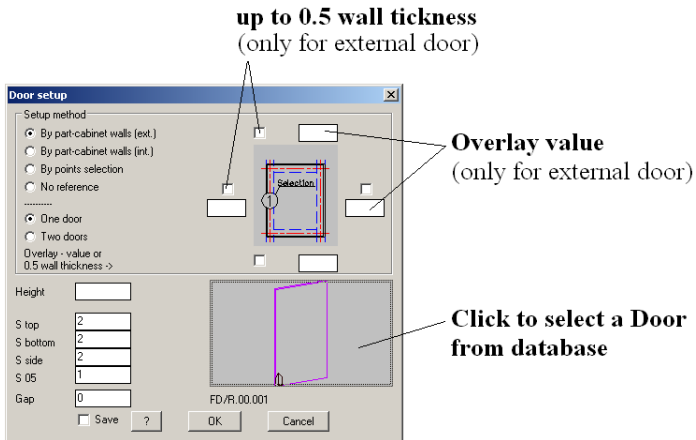
Fig. 3.61.   Door setup dialog window
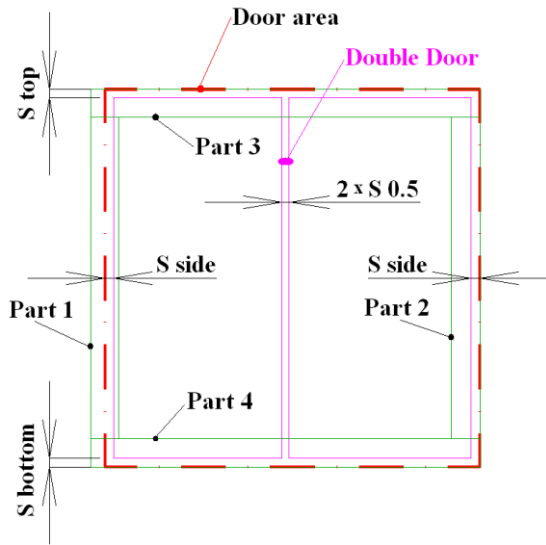
Fig. 3.62   Front view of a cabinet. Door gaps

By part–cabinet walls (ext.)
Creates an external Door by specifying the cabinet Parts
Note. You may specify overlay for an external door, see fig. *3.61*

By part–cabinet walls (int.)
Creates an external Door by specifying the cabinet Part

By points selection
Creates a Door (or two Doors) by specifying three points.

No reference
Inserts a Door from the database without reference.

One Door
If this checkbox will be activated, it creates one door by one of the above mentioned method ("By part–cabinet walls (ext.)", "By part–cabinet walls (int.)", "By points selection").

Two Doors
If this checkbox will be activated, it creates a double door by one of the above mentioned method ("By part–cabinet walls (ext.)", "By part–cabinet walls (int.)", "By points selection").

Up to 0.5 wall thickness

This checkbox restricts door area for an external Door, see fig. *3.59* d).

Note. You may use the "Up to 0.5 wall thickness" checkbox or enter any value in an "overlay value" field, see fig. *3.61*.

Height

The value in this field restricts the Door area height determined by specifying Parts, see fig. *3.59*.

S top

It determines a gap relative to the third specified Part, see fig. 3.61 and fig. 3.62.

S bottom

It determines a gap relative to the fourth specified Part, see fig. 3.61 and fig. 3.62.

S side

It determines gaps relative to the first and second specified Part, see fig. 3.61 and fig. 3.62

S 0.5

It determines a gap between doors of a double door, see fig. 3.61 and fig. 3.62

Gap

It determines a gap between a cabinet and a door. (If you will see a cabinet top view, you see this gap).

Save

Saves gaps value in the dialog window as gaps by default

Note. You can use a formula in the field: "Height", "S top", "S bottom", "S side", "S 05", "Gap".

Relative to a checkbox activated in the dialog window, see fig. 3.61, page 104, the Door will be inserted:
- by Part (external)
- by Part (internal)
- by points (external or internal)
- without reference

<u>Without reference</u>

Summary
This method creates one vertical Door without reference

List of prompts
Insert point: Specify a point on the screen by mouse left-click.
Rotation angle: Specify angle by mouse or enter value in the Command Line

By points (external or internal)
Summary
This method creates one Door or a double Door. The Door coordinate system is determined by three point.

List of prompts
ODO (box) + / Base point: Specify a base Door point, see fig. *3.63*, point "p.1". Note, the base point determine bottom gap and side gap see fig. 3.61, page 104
*ODO (box) + / *change point #N / Point of axis X*: Specify the second point, see fig. *3.63*, point "p.2".
*ODO (box) + / *change point #N / Point of axis Y*: Specify the third point, see fig. *3.63*, point "p.3". Note, the third point determines top gap, see fig. 3.61, page 104
Note. Base point determines the Door origin. The gaps "SX1" and "SY1" are determined by object origin, see fig. *3.63*.
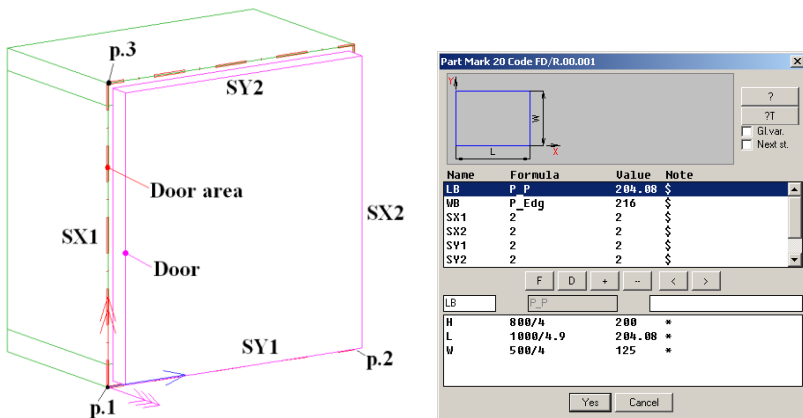
Fig. 3.63   Door and its variable dialog window

By Part (external)
Summary

This method creates one Door or a double Door. The Door coordinate system is determined by specifying Parts. The Parts specifying is considered on page 103.

<u>List of prompts</u>

Base vertical part: Specify a base Part. Note, the base Part determines the Door coordinate system (hereinafter CS), see fig. *3.63* and fig. *3.64*. The Door CS origin is determined by specifying the Part 1. The Door Y-axis is parallel to the edge specified on the Part 1.

Flat part 2: Specify the second Part to determine the Door width.

Part 3 (perpendicular to edge of 1-st part): Specify the third Part or press **Enter** to create Door by two Parts.

Note. You determine the Door Y-axis direction by specifying the Part 3. In other words, you determine where the Door top is by specifying the Part 3.

Note. If you create a Door by specifying three Parts, the specified point on a Part 1 should be behind the Part 1 mid-point in relation to a Part 3, see fig. *3.59* b).

Part 4 (perpendicular to 1-st part): Specify the fourth Part or press **Enter** to create Door by three Parts

Note. You can specify the edge of existing Door instead of Part 4.
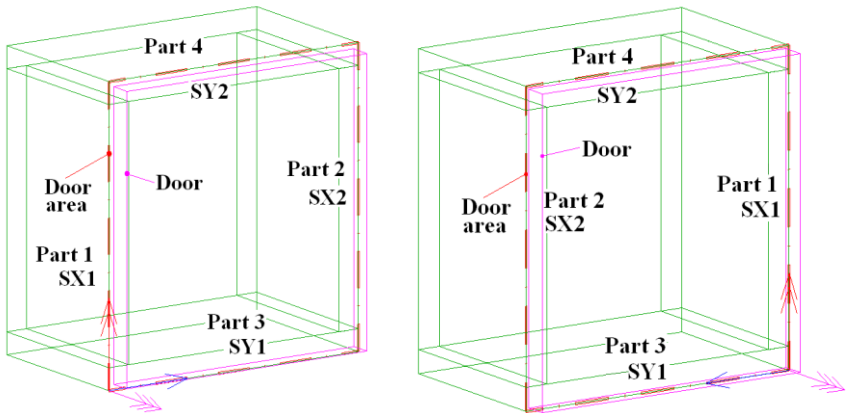
Fig. 3.64   The Parts order specifying and result of the Door insertion

Note. If you create a Door by specifying two Parts, the Part 1determines the Door height and Part 2 determines the Door width.

Note. If you will enter a value in the "Height" field, see fig. 3.61, page 104, the Door height is created from the Part 3.

Important! When you write a Door in the Door database it can has any gaps. But when you insert a Door in the project, the gaps determined in the database is replaced with gaps specified in the "Door setup" window, see

fig. 3.61, page. 104. When the Door has created you can edit its gaps by commands **Operations >> Parameters of 1-st** and **Edit >> Variables**.

By Part (internal)

Summary

This method creates one internal Door or a double internal Door. It works like a "By Part (external)" method.

3.7.6.2. Insertion of Doors group

Access methods
Menu: **Assembly >> Group of Doors-vertically**

Toolbar: **Assembly**
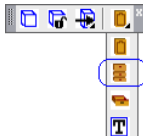
Summary

Creates two or more doors. It is comfortable to use this command to prepare doors to create Drawers by Doors.

List of prompts

The list of prompts is equal to considered in section "By part (external)", page 107. The Door setup dialog window is presented in fig. *3.65*.
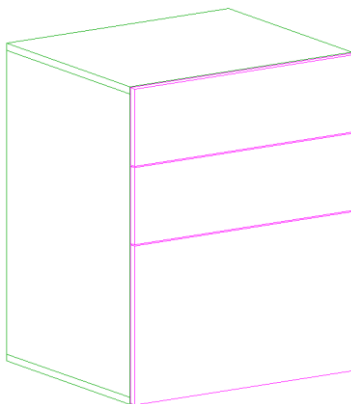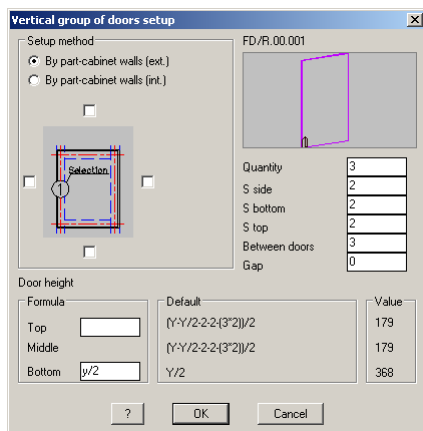
Fig. 3.65   Creation of Doors group

The settings from the "Setup method" area are considered in fig. *3.61*, page *104*.

Click slide by mouse left-click to select a Door from the Door database

Quantity

Determines a quantity doors in the group

S side

Determines gaps relative to the first and second specified Part, see fig. 3.61 and fig. 3.62

S top

Determines a gap relative to the third specified Part, see fig. 3.61 and fig. 3.62.

S bottom

Determines a gap relative to the fourth specified Part, see fig. 3.61 and fig. 3.62.

Between doors

Determines gaps between doors of a group

Gap

It determines a gap between a cabinet and a door. (If you will see a cabinet top view, you see this gap).

Top

Determines a height of the top Door (You determine a Door height, not a Door area height, as it is in the "Door setup" window in the fig. 3.61, page 104).

Note. You can enter value or leave this field empty. If it will be empty, its value is calculated automatically.

Middle

Determines a height of the middle Doors. Note, you can't edit this parameter. It only gives you information.

Bottom

Determines a height of the bottom Door. (You determine a Door height, not a Door area height, as it is in the "Door setup" window in the fig. 3.61, page 104).

Note. You can enter value or leave this field empty. If it will be empty, its value is calculated automatically.

Note. You can use a formula in the "Top" and "Bottom" field. For example see fig. *3.65*, field "Bottom" (the "Y" is a height of Door area, more details see in section 3.7.6.1, page 102).

Important! If the "Top" and "Bottom" fields will be empty, the command creates N equal Doors. If values will be entered into one of them or into both, the height of the middle door (or doors) is calculated automatically. The entered and calculated values of the Door height will be shown in the "Value" area. You can edit "Top", "Bottom", "S bottom", "S top", "Between doors" to take the necessary values of the Door height.

Note. You see real Door heights in the "Value" area of the dialog window presented in fig. *3.65*.

3.7.6.3. Creation of a Door

To create a new Door and write it in the Door database, you should:
- insert the Door template in the project by command Assembly >> Door template, see fig. 3.66;
- if it is a Part Door, edit Door template by operations (more details about operations see in section 4.5, page 158);
- if it is a Framed Door, create Door objects;
- save a new Door in the Door database by command **Database >> Save model** or **Database >> Save model as**
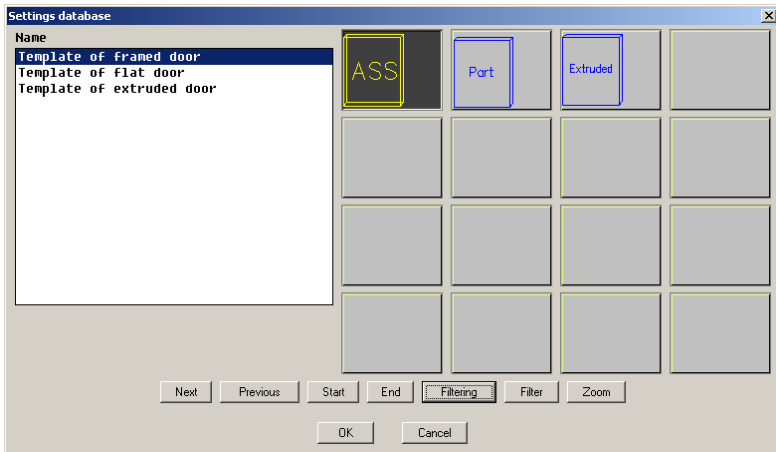

Fig. 3.66. Door template

When a Door template has inserted in the project, you can edit its size and gaps. But it is executed only for convenience of a new Door creation. After saving a new Door in the database and the subsequent insertion in the project, the Door sizes are adapted according to a Door area, see fig. *3.59*, page 103, the

Door gaps are replaced according to gaps in the "Door setup" window, see fig 3.61, page 104 and fig. 3.65, page 109.

3.7.6.4. Special features of Frame Door

When you insert Template of Framed Door in the project, it is automatically determined as current Assembly. It has an "MF" logical material, which defines a Framed Door thickness by default

When you create a Framed Door, you create a usual Assembly which can include different objects (Parts, Profile, Fasteners, furniture Fitting)

It is comfortable to lock Assembly (Framed Door) in the Door database. If you insert a locked Door in the project, you can take only its overall dimensions points (you specify any object included in a Framed Door and automatically take the nearest point of its Assembly Frame). To lock a Frame Door in the database, activate "Lock after insert" checkbox in the database. To edit locked Framed Door in the project, you should unlock it by command **Assembly >> Unlock Assembly**.

Important! When you create a Framed Door the sizes of the Framed Door is determined by its Frame. The Framed Door has box variables (more details see in section 3.1.1, page 30). If you edit box variables, the Framed Door sizes is recalculated (In other words the Frame behavior of a Framed Door similarly to behavior of a Part Door).

Important! Other Assemblies and Informational Sizes haven't to be included in the Framed Door. Otherwise it can't be saved in the Door database.

If a Framed Door is locked in the project, the search functions of holes from Fasteners and furniture fittings process it. If you insert Fasteners or put furniture fittings on the locked Framed Door, they automatically reference to the Frame.

If the Fasteners or furniture fittings are inserted in (put on) an unlocked Framed Door, it is necessary to control an object specifying. In case of Fasteners or furniture installation on a Door as on single object, it is necessary to specify Door Frame edges. In case of Fasteners or furniture installation on Door components, it is necessary to specify edges and points of objects.

When a set of drawings is generated for a product, 3D-constructor generates a plane view of a Framed Door where you will see all operations over a Frame, see fig. 3.67 c). Drawings of objects included in the Framed Door will be generated if the objects require processing.

The Framed Door is not exported in CAM-module (the aggregated Framed Door isn't processed with CNC-machining center).

If there are Sheet Parts in the Framed Door, they will be exported in a Cutlist Data.
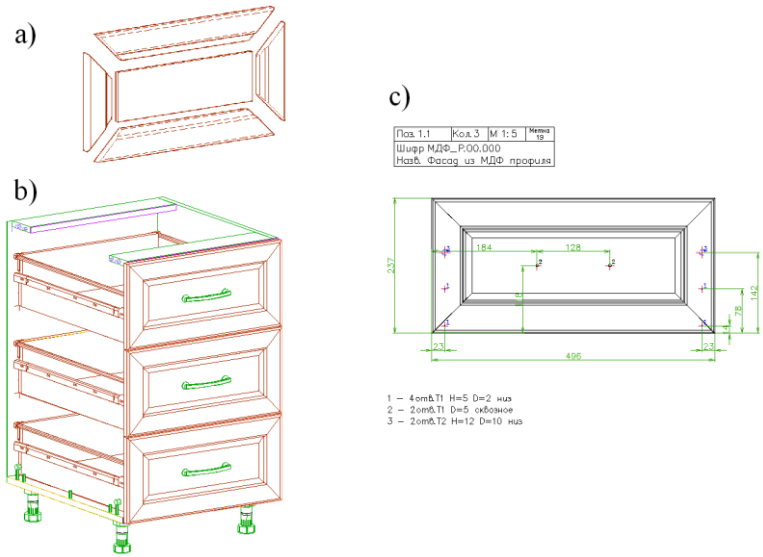
Fig. 3.67   a)   Framed Door

b)   Framed Door in the cabinet
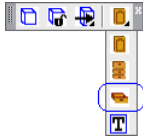
c)   Drawing of Framed Door

3.7.7. Drawers

The Drawer is an Assembly which has special insertion methods and an independent database.

3.7.7.1. Drawer insertion

<u>Access methods</u>
Menu: **Assembly >> Drawer**

Toolbar: **Assembly**

Summary

There are some methods to insert a Drawer from database. The easiest way to insert it is a "No reference" method. If you will use any other methods, you define a Drawer Volume by method. The Drawer Volume is a "box" which limits the Drawer sizes.

List of prompts

current <320M-D-ZRE.00.000> #N / select #D / select drawer: Specify any drawer in the project to insert the same. Press Enter to confirm the drawer by default (its code is in the "Current" option). You can specify a drawer in the database by "Select" option.

Setup method

*by door #A / by door with limitations #J / by points #N / *without reference #G*: Specify an insertion method.

Note. The last executed insertion method is saved as method by default.

Without reference
Summary

This method creates one Drawer without reference

List of prompts

Insert point: Specify a point on the screen by mouse left-click.

Rotation angle: Specify angle by mouse or enter value in the Command Line

By points
Summary

This method creates one drawer. Its sizes and reference are determined by three specified points.

List of prompts

Base point: Specify base point of Drawer. It determines the Drawer face side.

change point #N / Point of axis X: Specify the second point. It determines the Drawer width.

change point #N / Point of axis Y: Specify the third point. It determines the Drawer depth.

change point #N / Point of axis Z: Specify the fourth point. It determines the Drawer height.

By door

Summary

This method creates some Drawers by Doors. The Drawer Volume is calculated by specifying a Door:

- The Drawer Volume width is determined with distance between the left internal cabinet side and the right internal cabinet side. If one of side or both sides are absent, it is determined with distance between the right and left Door edges.
- The Drawer Volume depth is determined with distance between the internal Door side and the internal Back side.

Important! If there is not a Back in the cabinet when you have been inserting the Drawer, the Drawer Volume depth will be determined with one of the cabinet side panel depth. It may be taken left or right panel. If the base side panel is taken incorrectly (right and left panels may be difference in a depth) you will be able to correct the Drawer Volume depth by command **Edit >> Variable**.

- The Drawer Volume height is determined with the Door height.

Important! When you insert a drawer "By door", the Drawer (its frame) will be reference to the bottom Door edge, see fig. 3.68 page 116.

Note. If you use "By door" method the cabinet bottom panel thickness have to be taken into account in the Drawer model.

List of prompts
Select drawer front to setup drawer
Select objects: Specify a Door
Select objects: Specify next Door
Select objects: Press **Enter** to complete Door selection.
Note. You can use "window selection" or "crossing selection" to select some Doors at the same time, see section 1.13.5, page 21.

By door with limitations

Summary

This method creates some Drawers by Doors. The Drawer Volume is calculated by specifying a Door. The width and depth of the Drawer Volume corresponds to the width and depth in the "By door" method. The Drawer Volume height is determined with the Door height but thickness of the bottom and top cabinet panels is taking into account, see fig. 3.69

The "By door with limitation" list of prompts corresponds to the "By door" list of prompts

Note. When you execute "By door" or "By door with limitations" methods it is possible in one command to create some Drawers with identical height. To do it, activate checkbox "For N drawers by fronts H on drawers set as first door", command **Service >> Settings >> Assembly**. If this checkbox is activated the first specified Door will determine the height of all Drawers created

in one command. The correctness of the first Door specifying isn't verified by the program (it is necessary to specify the lowest Door).
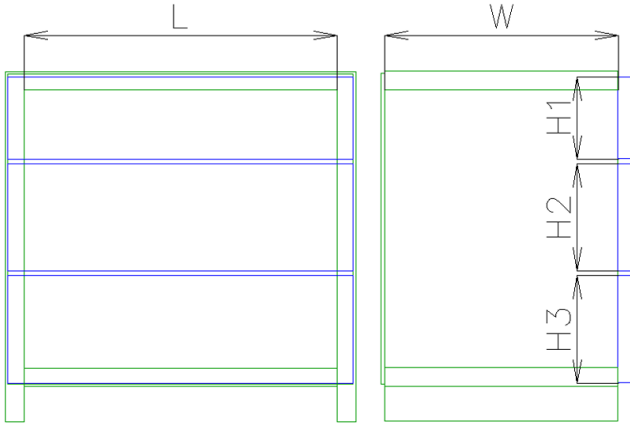

Fig. 3.68 The Drawer Volume according to "By door" method.
L– the Drawer Volume width
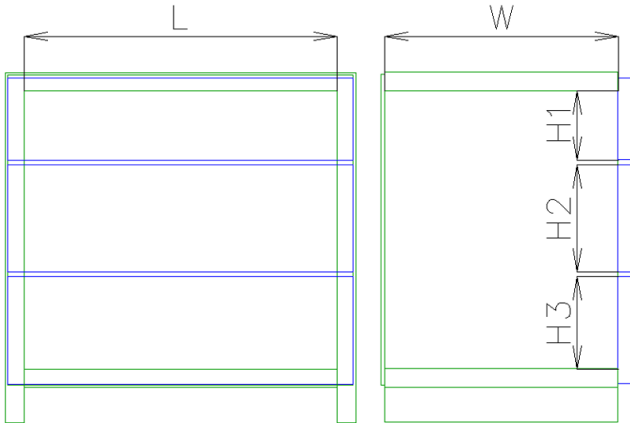*W– the Drawer Volume depth*
H1,H2,H3– the Drawer Volume heights

Fig. 3.69 *The Drawer Volume according to "By door* with limitations*"*
method.
L– the Drawer Volume width
W– the Drawer Volume depth
H1,H2,H3– the Drawer Volume heights

3.7.7.1. Creation of a Drawer

To create a new Drawer and write it in the Drawer database:

- insert the Drawer template in the project by command **Assembly >> Drawer front template**;
- create a Drawer model;
- save a new Door in the Door database by command **Database >> Save model** or **Database >> Save model as**

Note. The Drawer template is a usual Assembly. When you create Drawer model, it is possible to use any objects (Parts, Profile, Fasteners, furniture Fitting, Blocks)

Note. When you create a Drawer model, it is rationality to use fasteners ability to be set in a one object, more details see in section 3.8.13, page 145. This is easiest way to take holes in the Door after Drawer insertion.

Important! Other Assemblies and Informational Sizes haven't to be included in the Drawer. Otherwise it can't be saved in the Door database.

It is comfortable to lock an Assembly (Drawer) in the Drawer database. If you insert a locked Drawer in the project, you can take only its overall dimensions points (you specify any object included in a Drawer and automatically take the nearest point of its Assembly Frame). To lock a Frame Door in the database, activate "Lock after insert" checkbox in the database. To edit the locked Drawer in the project, you should unlock it by command **Assembly >> Unlock Assembly**.

3.7.8. Tamburat-locked Assembly

Tamburat is a furniture panel which consists of two or more layers. The layer can include Part or some Parts. When you design Tamburat with the different operations (process an edge, edging, edit contour, insert Fasteners) the program interprets it as a single object.

See a Tamburat example in fig. 3.70. The Tamburat consists of three layers: top, middle and bottom. The top and bottom layers are the decorate plates. The middle layer is a hoop it adds the thickness. The Tamburat layers are fastened with glue and dowels. When Tamburat is aggregated, some of its edge will be process an edge.

The drawings from 3D-model include:

- drawing of a Tamburat,
- drawing of Parts

Note. The drawing of a Tamburat will be generated if it is processed as single object. The drawing of Parts will be generated if Parts had been processed before Tamburat was created. The process includes holes, routing, process an edge and other

The programs for CNC-machine will be generated by CAM-module for the Tamburat and Parts if they have process.

In the cutlist are included only Parts.

a)   b)   c)
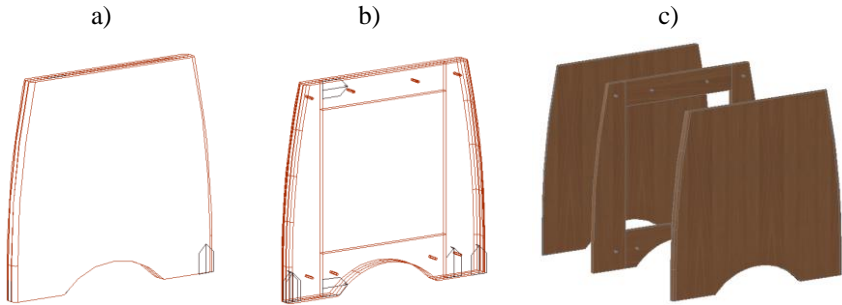
Fig. 3.70 Example of Tamburat
a) aggregated Tamburat, invisible edges are hidden
b) aggregated Tamburat, the edges of the all Tamburat objects are visible
c) Tamburat layers are moved apart

a)   b)

Tamburat contour

Fig. 3.71 *Example of Tamburat details preparation*
a) Tamburat details are shifted (They are shifted to you can see them).
b) Details position when creating the Tamburat
***Note**. The Tamburat contour is shown behind a panel.*

Access method to create Tamburat
Menu: **Assembly >> Tamburat-locked Assembly**

Summary

This command creates a Tamburat Assembly from Parts prepared by user in advance. The frame of Tamburat Assembly is "Point".

List of prompts

Select base part of assembly-tamburat: Specify base Part

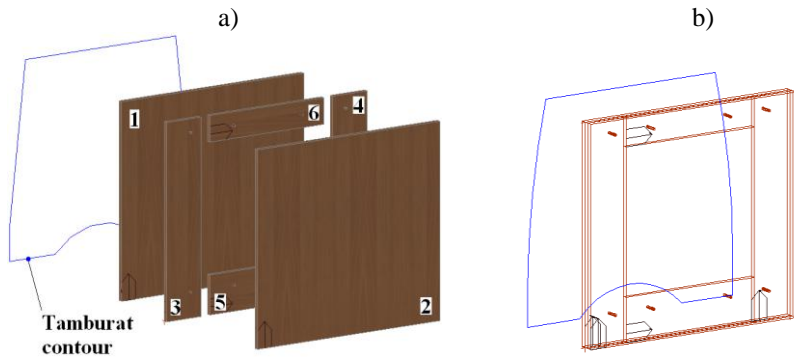**Note**. The base Part is a part to which other Tamburat Parts are referenced. It should be the first in the constructional order. (See fig. 3.71, depending on a construction order it may be Part "1" or Part "2".)

Select part will be processed with base part: Specify other Parts which contour will be processed together with a base Part. See fig. 3.71, Parts "1", "2", "3", "4", "5" have to be processed together to create Tamburat.

Select objects: Press **Enter** to complete selection.

Select objects:

Contour of processed assembly-tamburat <ENTER - contour of base part>: Specify an object determining the Tamburat contour. Press **Enter** to determine the Tamburat contour by base Part. (In an example in fig. 3.71 the contour is determined by 2D polyline)

Note. The Tamburat contour can be determined with:
- Constructional Drawing
- 2D polyline (this is an object created with AutoCAD tools)
- contour of the base Part

Other objects to be included in assembly-tamburat <ENTER - there are no more objects>: Specify any objects which are parts of Tamburat. It can be Parts, Fasteners, Blocks, Fittings. All specified objects will be included in the Assembly of Tamburat. (In an example in fig. 3.71 the auxiliary Tamburat parts are Part "6" and dowels.

Note. All Tamburat parts have to be reference to base Part or each other (All references (variables, objects) have to be closed inside Tamburat Assembly what is creating at this moment)

Select objects: Press **Enter** to complete selection and create a Tamburat Assembly.

The existing Tamburat contour can be edited with operations (Rounding, Chamfer, Cut on a corner, Cut on a side).

Important! The variables of the base Tamburat Part will be automatically copied in the Tamburat Assembly when Tamburat is created with com-

mand "Tamburat-locked Assembly". More details about variables see in section 6.2, page 281.

If according to furniture technology Tamburat is aggregated from rectangular Parts and then its contour is processed, you should prepare rectangular Parts and execute **Assembly >> Tamburat-locked Assembly**. In this case drawing of Parts will have a rectangular contour.

If you include in the Tamburat not rectangular Parts, this means that they have to be processed separately and then will be aggregated in a single object.

In other words, when you prepare Tamburat parts, you determine its manufacturing technology

You can explode Tamburat by command **Assembly >> Unlock Assembly**. In this case will be automatically deleted all operations which have done for it.

Note. If it is necessary to have a thick Part and you needn't date to make this Part (for example you buy it in the shop), you shouldn't use Tamburat. There are other means (multi-layer Part, Extruded Part, additional Part thickness) to decide this matter.

Important! To edit properties of the Tamburat objects, use **Assembly >> Properties of locked object part**. This command corresponds to the **Edit >> Properties**, more details see in section 5.6, page 217.

3.8. Fasteners

A fastener is a hardware device that mechanically joins or affixes two or more objects together. If you set fastener in the 3D-model, program will generate Part drawing which will include holes to insert fasteners. The fastener parameters (hole quantity, hole diameter, hole depth) should be determined in the fastener database in advance.

The fastener in the 3D-model consists of one element and an installation condition. If an installation condition isn't satisfied, the fastener will be automatically deleted from project.

There are some types of fastener in the program (screw, connectors, shelf support, hinges and other). It may be inserted with some methods:

- Shelf edge to Part, see fig. 3.72 a)
- Joint edge to edge, see fig. 3.72 b)
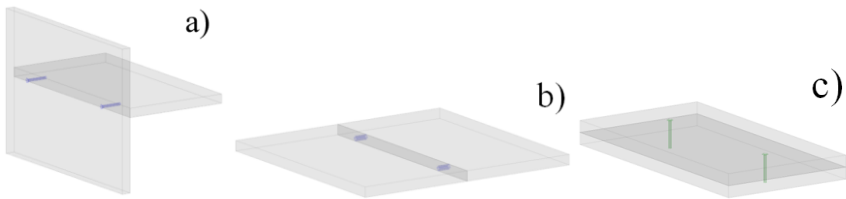- Surface to surface, see fig. 3.72 c)
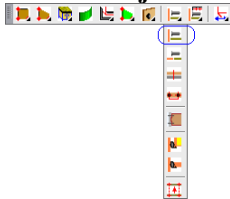
Fig. 3.72   Fasteners insertion methods

Note. When you specify insertion method by command, the program shows you fasteners corresponding to it.

3.8.1. "Shelf edge-Part (dialog)" insertion method

Access methods
Menu: **Object >> Fasteners and fittings >> Shelf edge-part (dialog)**

Toolbar: **Object**

Summary

This command place Fastener group along specified Part edge. The planes of the fastened Part have to be orthogonal (rotated on 90 degree) otherwise the Fastener will be placed in one Part. (Rastexes with angle dowel and hinges are exceptions of this rule).

List of prompts

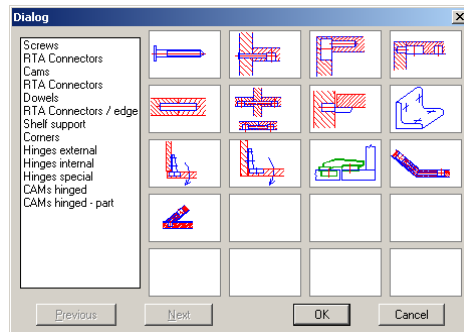Specify the fastener type in the dialog window, see fig. *3.73*. (Let it be RTA Connectors for example).

Fig. 3.73   Fasteners database
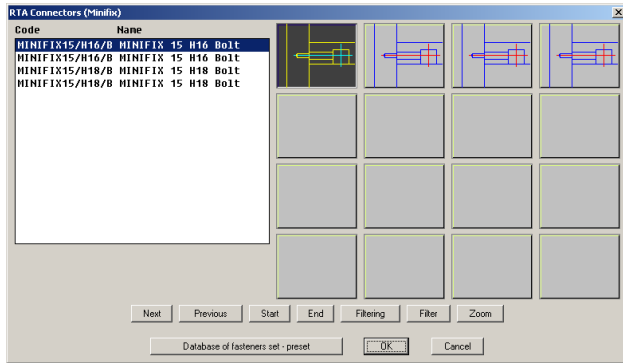
Specify one of the item in the database

Fig. 3.74   RTA connector database

Perpendicular join. Select edge of base part: Specify an edge of the base Part, see fig. 3.75 a) page 123. The Fasteners will be placed along the specified edge.

*next #L / side #C / base point #N / 2-d part: Press Enter to confirm the highlighted second Part or specify the second Part by mouse left-click if it was determined incorrectly, see fig. 3.75 b) page 123.

Enter Fastener parameters in the dialog window, see fig. 3.75 c), page 123. The result of the command is presented in fig. 3.75 d).

Side

Changes the base side of Fastener (The Rastex can be fit in the upper or lower Part surface, see fig. 3.75 b)).

Note. By default the base surface of the horizontal Parts is lower surface. The base surface of the vertical Parts is determined by specifying.

Base point

Specifies the base Fastener point.

Note. The Fastener coordinate system (hereinafter CS) is highlighted in a Fastener base point.

Note. By default the base point is the nearest edge point to the specified point. You can specify any point of objects or a point of an existing Fastener. The specified point will be projected on a Part edge along which the Fastener is placed.
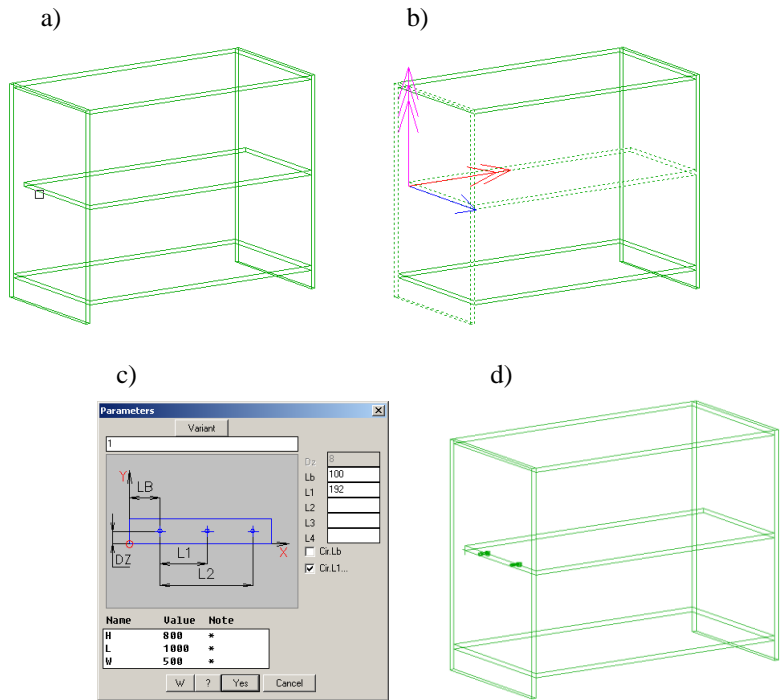
*Fig. 3.75. Insertion fasteners by "**Shelf edge-part (dialog)**"*

To edit parameters of existing Fasteners, use **Operations >> Parameters**

Dz

Determines distance from Part base edge to axis of fastener placing. By default it is determined by database value. You can edit it for some Fasteners (screw and dowel).

Note. If you set "#" symbol in this field the Fastener will be centered concerning Part thickness.

LB

Determines distance from the base point to the first Fastener of the Fastener group, see slide in the fig. 3.75 c).

Note. If you set "#" symbol in this field the Fastener group will be centered concerning base edge

L1

Determines distance from the firs Fastener of the Fastener Group to the second Fastener, see slide in the fig. 3.75 c)..

L2

Determines distance from the firs Fastener of the Fastener Group to the third Fastener, see slide in the fig. 3.75 c).

Ln

Determines distance from the firs Fastener of the Fastener Group to the N-th Fastener.

Note. To increase quantity of the "L" field in the dialog window, see fig. 3.75 c), click "W" button and edit the "List length 3…15" field. If you have to insert more than seven Fasteners, it is convenient to use **Object >> Furniture fitting >> Shelf edge-part (point by point)**

Note. If one of "L1", "L2"…"Ln" parameter is determined so that the Fastener is out of a Part, this Fastener isn't drawn. If all Fasteners of Fastener group is out of a Part, the Fastener group deletes itself.

Cir. Lb, Cir. L1…

If checkbox is activated the entered value in the field "Lb", "L1", "L2"..."Ln" will be rounded according to a step defined in the settings.

To determine rounding step, use **Service >> Setting >> Blocks, Fittings, Hardware**\ field "Rounding (step)"

Installation condition (under "Variant" button)

Determines installation condition of the Fastener group. You can write value or expression in this field. If an installation condition isn't satisfied, the Fastener group will be automatically deleted from 3D-model. The installation condition is satisfied if expression is more or equally to zero.

Variant

Determines auxiliary Fastener groups.

Note. One element of Fastener can include some Fastener groups and their installation conditions. The list of variants is checked from top to down to the first Fastener group with an installation condition more or equally to zero.
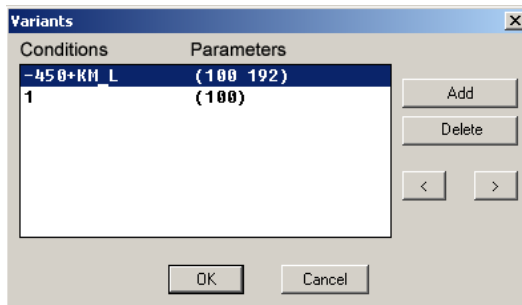

Fig. 3.76 List of variants to insert the Fastener Group

In the example above (fig. 3.76) one Fastener will be insert on a Part with a width up to 450 mm, and two Fasteners will be insert on a Part with a width more than 450 mm.

Note. You can use variables in the Fastener parameters to create a parametrical furniture model. The available variables are present in the "Variable list" in the bottom of the "Parameters" window (fig. 3.75 c) H, L, W variables).

3.8.1.1. KM_L variable

Important! When you enter Fastener parameters (see fig. *3.77*) it is comfortable to use "KM_L" variable.

"KM_L" is a special variable which is automatically created for every Fastener group. It is absent in the "variable list" of the "Parameters" window (see fig. *3.77*) but you can use it in the Fastener parameters.

When you copy Fastener group which parameters are determined through the KM_L variable on other edge, "KM_L" variable will be automatically recalculated. In this way you can quickly copy Fastener group along edges with a different length. More details about Fastener copying see in section 3.8.11, page 144.
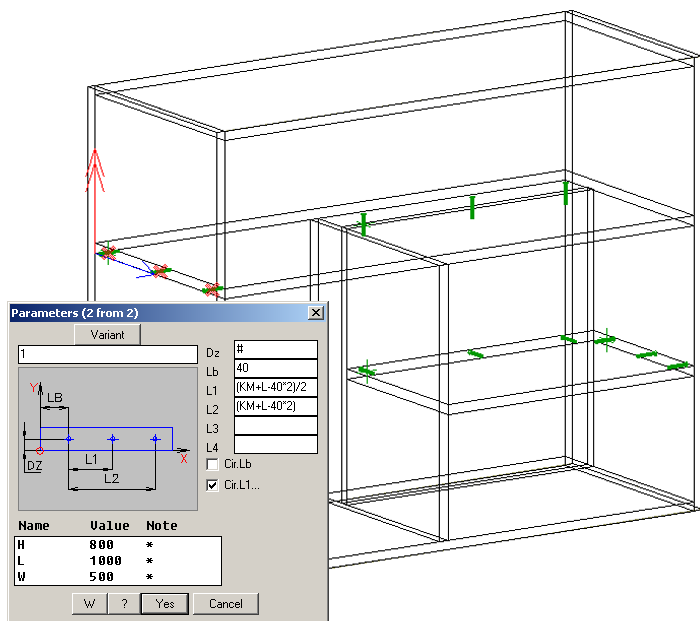

Fig. 3.77. "KM_L" variable in the Fastener parameters

The example (see fig. *3.77*) demonstrates next:
- There is one Fastener group, which includes three screws.
- The screws parameters are defined through "KM_L" variable
- Lb and L2 shifted from the ends of the edge on 40 мм
- L1 is determined as a mid-point between "Lb" and "L2"
- "L1" and "L2" are rounded according rounding step.
- All other Fastener groups in the 3D-model are a copy of it. The Part edges have a different length. The Fastener parameters didn't edit after copying.

Sometimes it is convenient to have "KM_L" as a variable which is determined by some distance defined by user (usually it is defined by edge length on which the Fastener is placed). In this case you can create "KM_L" variable in a Fastener. It overrides the embedded variable.

3.8.1.2. Hinges installation feature

The hinge base Part is a Door. When you insert an external hinge (overlay application) or an internal hinge (inset application) you can specify a front or a rear Door edge. If program finds the second Part and highlights it, the hinge will be automatically inserted into the rear Dore surface.

If you insert a special hinge (from "Hinges special" database), the specified Door edge will determine the hinge base surface of the Door.

Note. The hinge cup is cut in in the Door base surface.

If the second Part to insert a hinge (from any database) isn't automatically found, you can specify it by mouse or insert hinge only on a Door.

3.8.1.3. Shelf support installation feature

The row of holes over and under a shelf support is its a special feature, see fig. *3.78*. The auxiliary holes for the shelf supports are present in the 3D-model and Part drawings.
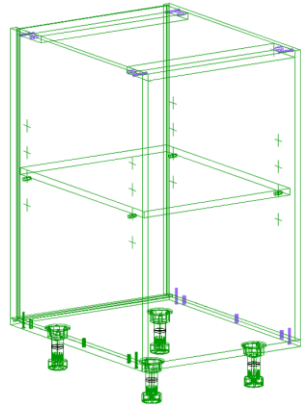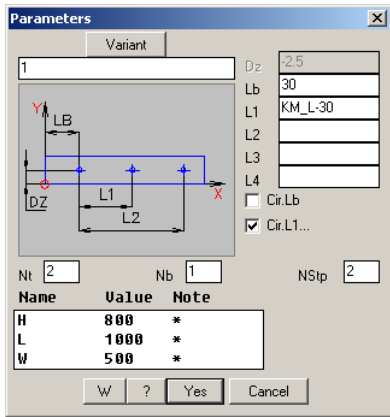
Fig. 3.78   Shelf support parameters

Nt (top)
Determines the auxiliary holes quantity over the shelf support

Nb (bottom)
Determines the auxiliary holes quantity under the shelf support

Nstp (step)
Determines repetition factor
Note. The step is determined by **Service >> Setting >> Blocks, Fittings, Hardware**\ field "Rounding (step)". "Nstp" determines the steps quantity between two holes. For example, if the step is 32 and "Nstp" is two, the distance between holes will be equal to 64 mm.

3.8.2. "Edge to edge (dialog)" insertion method

<u>Access methods</u>
Menu: **Object >> Fasteners and fittings >> Edge to edge (dialog)**

Toolbar: **Object**
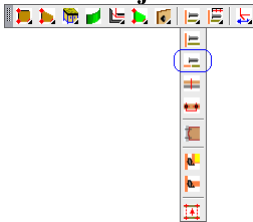
<u>Summary</u>
This command fix two Parts which is jointed edge to edge, see fig. 3.72 б) page 121. The command dialog corresponds to "Shelf edge-Part" dialog, see section 3.8.1 page 121.

3.8.3. "Surface to surface (dialog)" insertion method

This command fixes two or more parallel Parts, see fig. 3.79 a). The fastener can be based on a Part edge, on a Part point and on a two Part points.

Note. You can determine auxiliary points on a Part before insert Fasteners by this command. To determine auxiliary points, use command **Operations >> For part >> Point of contour >> Points XY (Points AR)**, see. fig. 3.79 b).
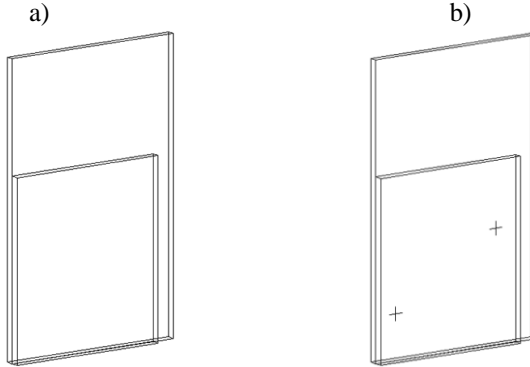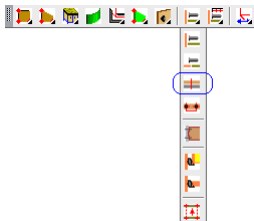

a)   b)

Fig. 3.79. Parallel Parts with the adjacent surfaces.

Access methods

Menu: **Object >> Fasteners and fittings >> Surface to surface (dialog)**

Toolbar: **Object**

3.8.3.1. Fixing two Parts

Activate command; select for example a dowel form database.

*point #N / Base edge: Specify a Part edge, see fig. 3.80.

Note. If you specify one of an adjacent edge, you automatically will cycle through the adjacent Parts. The specified object will be highlighted, press **Enter** to select it or any key (or mouse left-click) to select another adjacent Part

next #L / 2-d part: Confirm the second highlighted Part or specify another Part, see fig. 3.80 b). Enter the Fasteners parameters in the dialog window.

Point
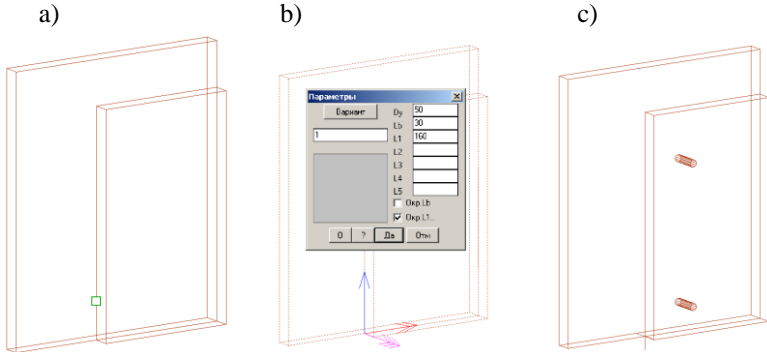Inserts Fasteners by point or two points, see fig. 3.81, page 130.

a)   b)   c)

Fig. 3.80   Fixing Parts by specifying edge

      a)   Edge specifying

      b)   Entering Fasteners parameters

      c)   Result of the command

Note. "Dy" determines a shift of Fastener group along Y-axis, see fig. see fig. 3.80 b). This is a distance between specified edge and Fastener group.

Other Fastener parameters are considered in section 3.8.1, page 121.

The "**Point**" options is considered more details below.
point #N / Base edge: Press **Enter** to activate the "point" option.

Base point: Specify a Part base point, see fig. *3.81* a) specifying "1"

redefine base point #N / *X / -X / Y / -Y / Direction point: Specify the second Part point, see fig. *3.81* a) specifying "2".
Note. If you specify two points, you can use "KM_L" variable. In this case it is defined by distance between two specified points. If you specify one points and determine direction by "**X/ -X / Y / -Y**" option, "KM_L" value is equal to zero.

next #L / 2-d part: Confirm or specify the second fixed Part and enter the Fastener parameters in the dialog window.

Redefine base point

Allows you to redefine previous point.

X

This option determines Fastener group direction along X-axis of the first fixed Part. The fastener will be shifted in positive direction. In this case you haven't to specify the second point. Fastener will be inserted by one point and X-axis direction.

-X / Y / -Y

These options are identical to mentioned above.

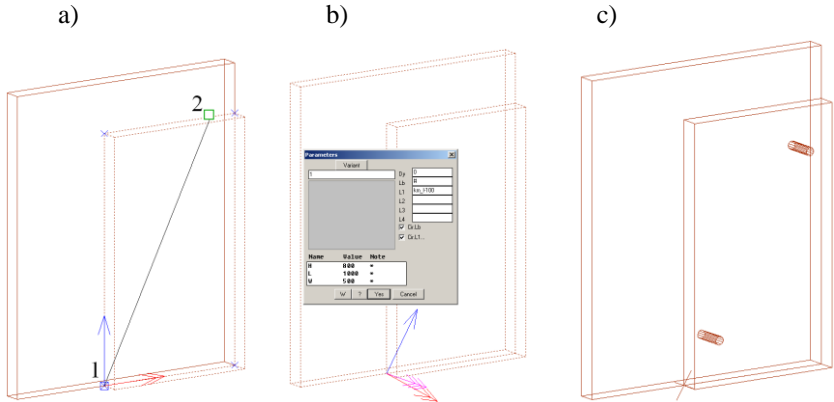a)   b)   c)

Fig. 3.81   Fixing Parts by specifying two points

Important! If you insert dowels or fittings for surface, you have to specify an internal edge of the adjacent Parts. If you insert screws, you have to specify an external edge of the base Part, see fig. *3.82*
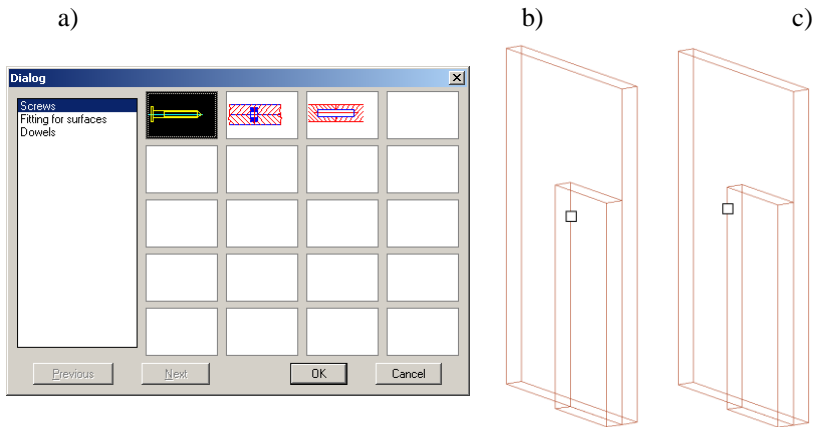
a)   b)   c)

Fig. 3.82 Fixing adjacent Parts

       a) Fastener Database to fix adjacent Parts

       b) Specifying <u>internal</u> edge (or points) of the base Part to insert dowels or fittings for surface

       c) Specifying <u>external</u> edge (or points) of the base Part to insert screws
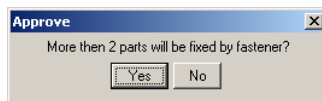
3.8.3.2. Fixing more than two Parts

Activate command; select for example a dowel form database.

*point #N / Base edge: Specify an edge one of external Parts, see fig. *3.83* a), specifying "1".

Note. If you insert dowels, you have to specify an internal edge, see fig. 3.82.

*next #L / 2-d part: Specify the second external Part, see *3.83* b), specifying "2".

Confirm fixing more than two Parts.

Next part: Specify auxiliary Part (middle Part)

Next part: Press **Enter** to complete selection.

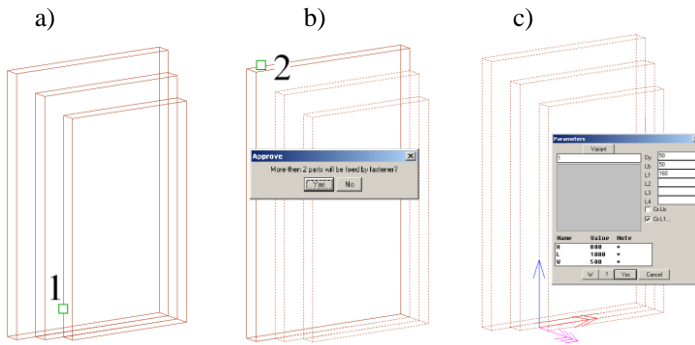Enter Fasten parameters in the dialog window, see fig. *3.83* c).

Fig. 3.83 *Fixing three parallel Parts by dowels*

The fig. 3.84 shows the Part specifying to fix the same three Parts by screws.
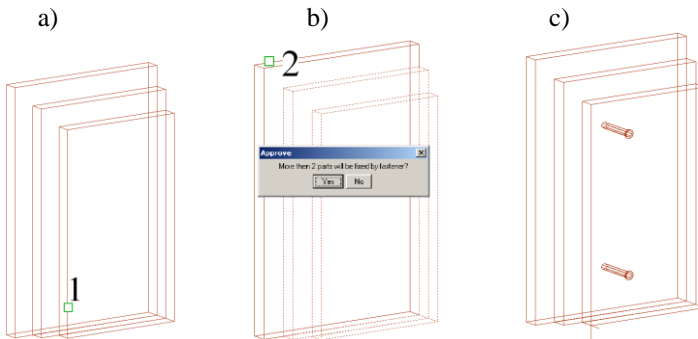
Fig. 3.84 *Fixing three parallel Parts by screw*

Important! When you fix three and more Parts, only you (not a program) control if specified Fastener is suitable (if its length is suitable) to fix three and more details. In the external Part drawings will be done holes correspond to the database settings. In the internal Part drawings will be done reach-through hole.

To successfully fix three and more Parts, the distance between adjacent Part surfaces have to be equal to zero.

3.8.4. Auxiliary Fasteners insertion

Menu: **Object >> Fasteners   and   fittings >> Additional   fasteners (dialog)**
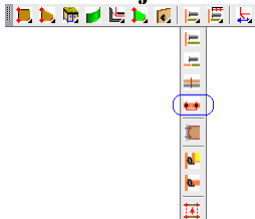
Toolbar: **Object**

Summary

Auxiliary Fasteners are Fasteners based concerning existing Fastener. The main and auxiliary Fasteners form a single group which can be copied and saved in the database. As an auxiliary Fastener can be inserted a screw or a dowel.

List of prompts
Activate command; select a screw or a dowel from database.

Base point: Specify one of an existing Fastener, see fig. 3.85. Enter auxiliary Fastener parameters in the dialog. The dialog window is consider in section 3.8.1, page 121.
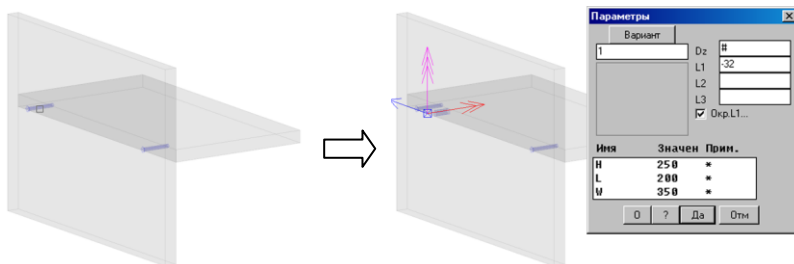
Fig. 3.85   Auxiliary Fasteners insertion

Note. "Dz" parameter of the auxiliary Fastener is automatically taken from the main Fastener. In other words, auxiliary dowels will be aligned in one line concerning minifix.

Note. You can insert one or some auxiliary Fasteners concerning one main Fastener in the Fastener group.

To   edit   auxiliary   Fastener   parameters,   execute   **Operations >> Parameters** and specify an auxiliary Fastener. It comfortable, to use

"list of elements" option, to specify an auxiliary Fastener from a Fastener group.

To delete an auxiliary Fastener, execute **Operations >> Delete,** activate "list of elements" option, specify a Fastener group, specify an auxiliary fastener from the "list of elements".

3.8.5. Fastener insertion by specifying points

There are some commands corresponded to commands considered in sections 3.8.1, 3.8.2, 3.8.3, 3.8.4. The Fastener position in these commands is defined by mouse left-click (not in the dialog window). It is comfortable to use these command, when it is necessary to insert five and more Fasteners. You can specify approximately Fastener positions with these commands and then edit them with **Operations >> Parameters**.

Access methods
Menu:
Object >> Fasteners and fittings >> Shelf edge – Part (point by point)

Object >> Fasteners and fittings >> Edge to edge (point by point)

Object >> Fasteners and fittings >> Surface to surface (point by point)

Object >> Fasteners and fittings >> Additional fasteners (point by point)

3.8.6. Tenon and mortise

Access methods
Menu: **Object >> Fasteners and fittings >> Mortise and tenon (dialog)**

Toolbar: **Object**
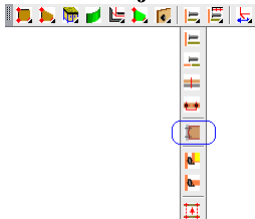
Summary
This command allows you to determine joint of two Parts by tenon and mortise. In addition details are fixed by screw. The tenon is cut in the first specified Part; the mortise is cut in the second Part. Two specified Parts have to be orthogonal (rotated on 90 degree).

<u>List of prompts</u>

Activate command, select a screw type from the database to fix Parts in addition, see fig. *3.86* a), page 136.

Important! It is comfortable to press "Database of fasteners set-preset" to select screw type, screw parameters, tenon and mortise parameters from "preset" database, see fig. *3.86* a), page 136.

Perpendicular join. Select edge of base part: Specify an edge of the base Part, see fig. *3.86* b), page 136.

next #L / side #C / base point #N / 2-d part: Press **Enter** to confirm the second fixed Part, see fig. *3.86* c).

Enter tenon and mortise parameters in the dialog window, see fig. *3.86* d), (Th– tenon thickness, W– tenon length, S– tenon shift concerning Part end, L– tenon length)

Enter screw parameters in the dialog window, see fig. *3.86* e). The screw parameters description see in section 3.8.1, page 121.

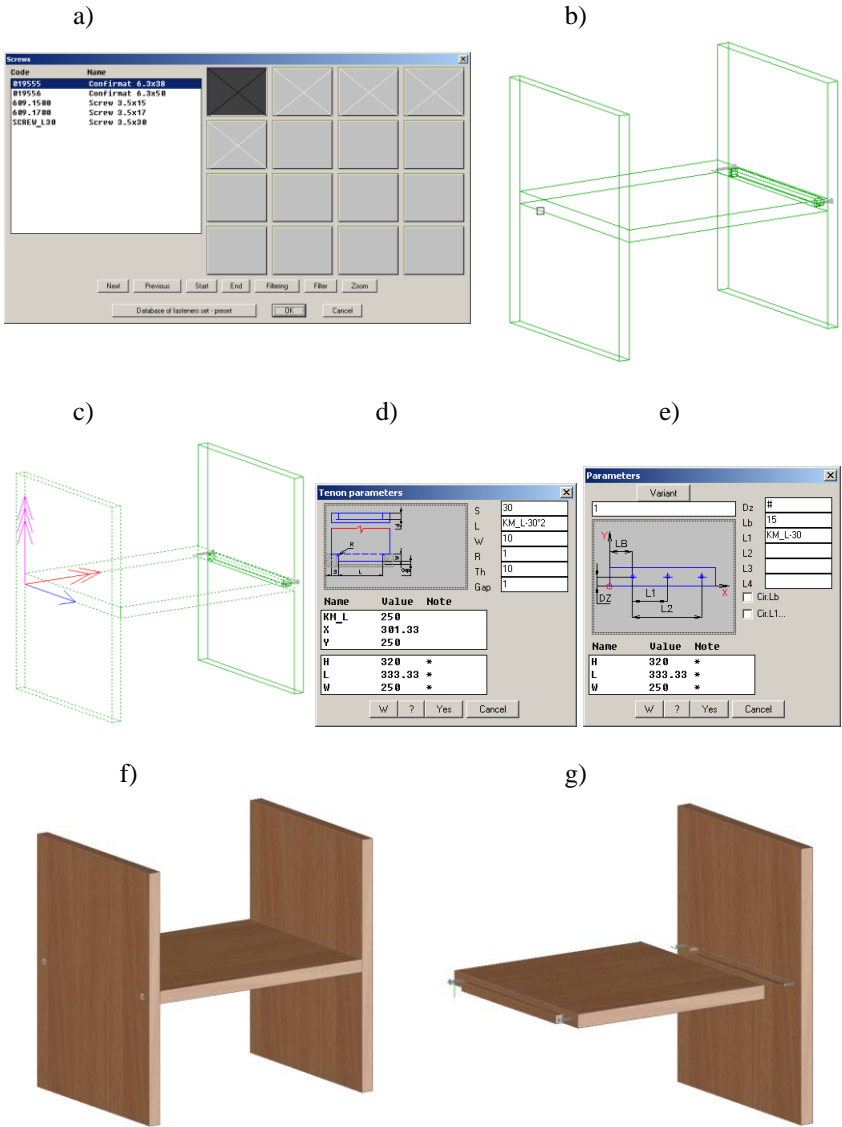You can see result in fig. *3.86* f) and g).

a)   b)

c)   d)   e)

f)   g)

Fig. 3.86   Three Parts fixed by tenon and mortis

3.8.7. Drawer runners

3.8.7.1. Drawer runner insertion on a surface

<u>Access methods</u>
Menu: **Object >> Fasteners and fittings >> Set runner on a surface**

Toolbar: **Object**

<u>Summary</u>

This command inserts one runner or pair of runners. If you insert a pair of runners the base Parts have to be parallel.

Note. The runners can be calculated in the quotation as a pair or by the piece. To set the control, execute **Service >> Settings >> Blocks, Fittings, Hardware,** control "Spesify drawer runners in pair (2 pcs as a set)"

<u>List of prompts</u>

Select edge of base part: Specify an edge of the base Part, see fig. 3.88, "ab" edge. Enter the runner insertion parameters in the dialog window, see fig. 3.87 (offset along X-axis and Y-axis, see fig. 3.88).

*No mate / Select edge to insert a mating runner: Press Enter to complete the command or specify an edge to insert a mating runner. The mating runner will have same insertion parameters as a base runner.
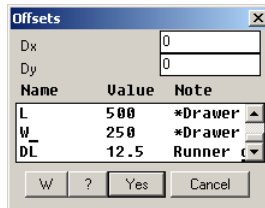

Fig. 3.87 Runner insertion parameters

Important! If a gap between a cabinet side and a drawer side is equal to runner thickness ±0.9 mm, it will be generate holes in the cabinet side drawing. The runner holes and runner thickness are determined in the runner database.

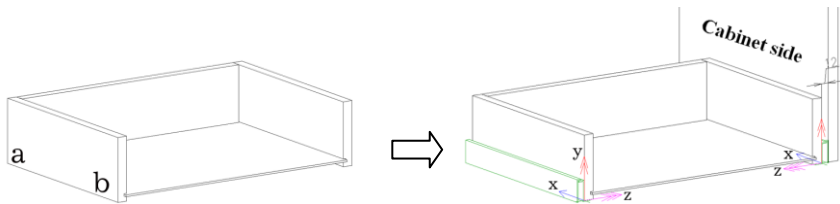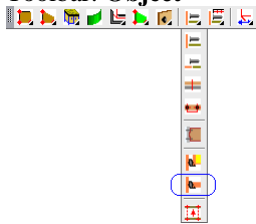*Fig. 3.88   Runner insertion on a surface*

3.8.7.2. Drawer runner insertion on an edge

<u>Access methods</u>
Menu: **Object >> Furniture fittings >> Set runner on an edge**

Toolbar: **Object**

<u>Summary</u>

This command inserts runners to a sliding shelf, see fig. 3.89. The list of prompts corresponds to it considered above, see section 3.8.7.1.
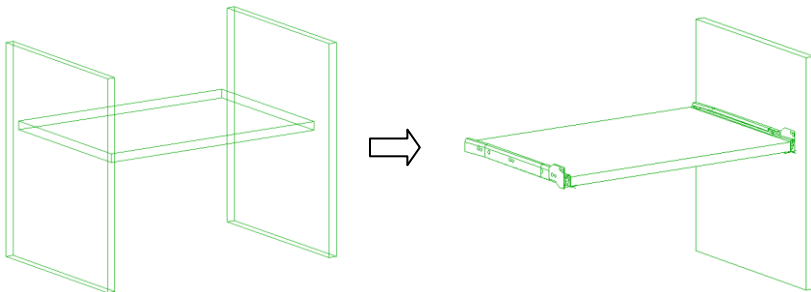

Fig. 3.89   *Runner insertion on an edge*

3.8.8. Fastener insertion on a Profile

3.8.8.1. Fastener insertion on Profile joint - dowels

Menu: **Object >> Fasteners and fittings >> Set fastener on Profile joint - dowels**

Summary
Inserts dowels on a profile bisecting joint, see fig *3.90*, page 139.

List of prompts
Insert point: Specify line of a profile bisecting joint, see fig *3.90*.
side #C / direction #Y / *next #L: Confirm fastener orientation or determine it by "side" and "direction" options. Enter the Fastener parameters in the dialog window.

Side
Changes the basic side of a Fastener, see fig. *3.91* page 140.

Direction
Changes the Fastener direction, see fig. *3.91* page 140.

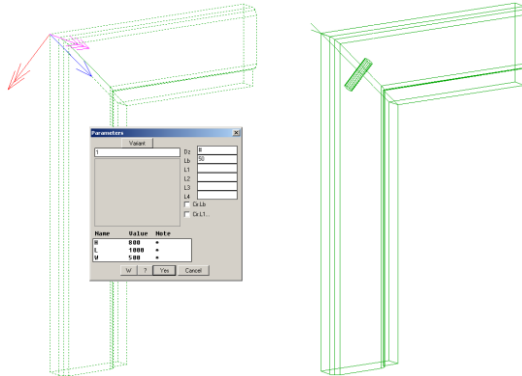*Fig. 3.90   Dowels insertion on the Profile bisecting joint*

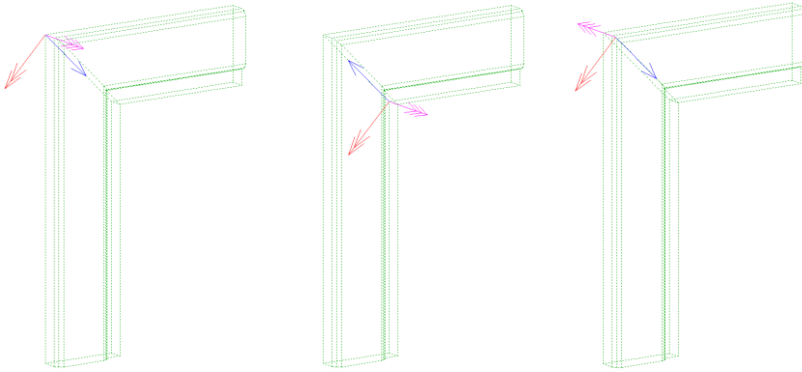a)   b)   c)

Fig. 3.91   a) Fastener orientation determined by specifying

b) "a)" orientation edit by "Direction" option

c) "a)" orientation edit by "Side" option

3.8.8.2. Fastener insertion on Profile joint - cams

<u>Access methods</u>
Menu: **Object >> Furniture fittings >> Set fastener on Profile joint - cams**

<u>Summary</u>
Inserts minifixes on a profile bisecting joint. The list of prompts corresponds to it, considered in section 3.8.8.1, page 139.

3.8.8.3. Fastener insertion "Profile-board (on edge)"

<u>Access methods</u>
Menu: **Object >> Furniture fittings >> Set fastener: Profile- board (on edge)**

<u>Summary</u>
Fixes a Part and a Profile, см. рис. *3.92* стр. 141
Note. XY-planes of the fixed objects have to be parallel, see. fig. *3.92* a).

<u>List of prompts</u>
Profile: Specify a Profile, see. fig. *3.92* b), page 141, specifying "1".
base point #N / 2-d part: Specify a Part, see. fig. *3.92* c), page 141, specifying "2".
*next #L / base point #N / 2-d part: Press **Enter** and enter Fastener parameters (see. fig. *3.92* c)) in the dialog window.

Base point

Determines Fastener base point (if you are not satisfied with base point taken by default)

2-d part
Allows you to reselect fixed Part (if it has been specified by mistake)

a)   b)   c)

d)   e)
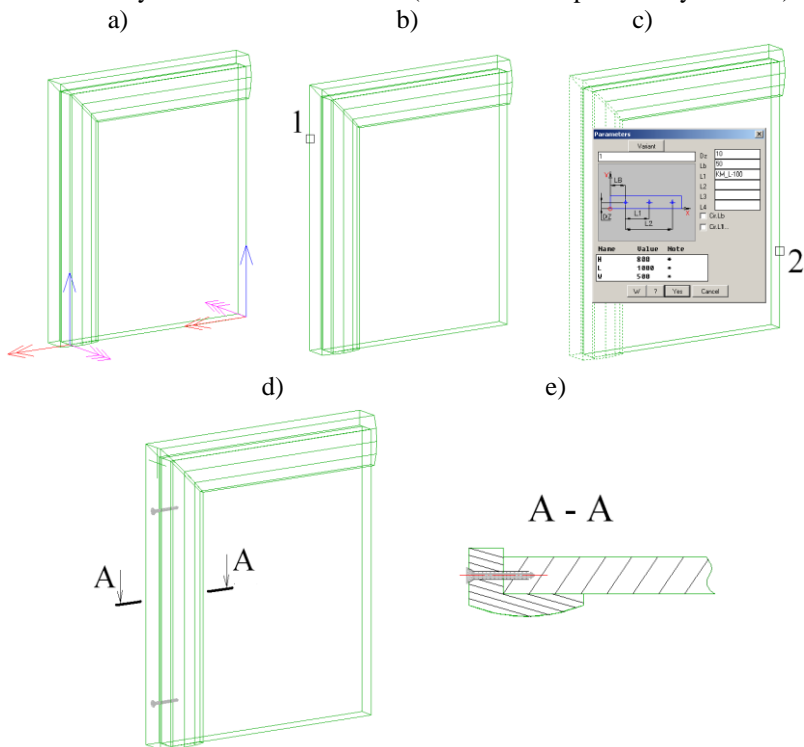
Fig. 3.92   Fastener insertion "Profile-board (on edge)"

3.8.8.4. Fastener insertion "Profile-board (on surface)"

Access methods

Menu: **Object >> Furniture fittings >> Set fastener: Profile- board (on surface)**

Summary

This command corresponds to command mentioned above, but it fixes Profile to a Part plane, see fig. 3.93 a).

Note. XY-planes of the fixed objects have to be parallel, see. fig. *3.92* a).
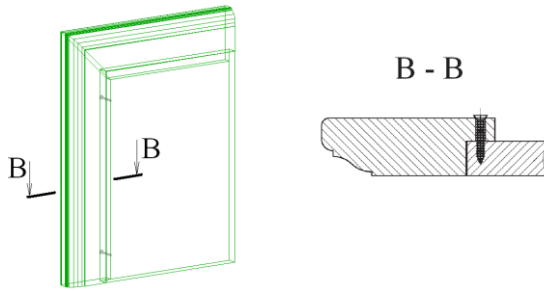
Fig. 3.93   Fastener insertion "Profile-board (on surface)"

3.8.9. Fastener insertion from a "Database of fitting sets-preset"

You can determine Fastener parameters, Fastener variants (see section 3.8.1, page 121) in the project and save adjusted Fastener group in the special database. Then adjusted Fastener group can be used in any project.

To save adjusted Fastener group in the database, execute **Database >> Save fitting set in database of preset fittings**

To insert adjusted Fastener group, execute one of command:

Menu:

      Object >> Fasteners and fittings >> Shelf edge-part (preset)

      Object >> Fasteners and fittings >> Edge to edge (preset)

      Object >> Fasteners and fittings >> Runner on a surface (preset)

      Object >> Fasteners and fittings >> Runner on an edge (preset)

Toolbars:

Else you can press "**Database of fasteners set-preset**" button in the database dialog window (see fig. 3.74, page 122) to insert adjusted Fastener group.
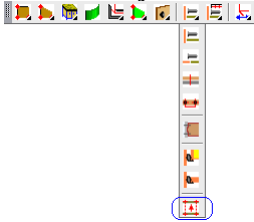
The list of prompts corresponds to one of the mentioned above.

3.8.10. Fastener projection

Toolbar: **Object**

Summary
You can project inserted Fastener onto other Parts to have some Fasteners placed along one line. The Fastener can be projected only onto Parts of one product.
Note. Projected Fastener parameters (see fig. 3.75 c), page 123) can't be edited ("Dz" parameter is an exception for dowels and screws). They are equal to base Fastener parameters. But you can replace projected Fastener (minifix by screw for example), it has own properties (code, position and anoter)

List of prompts
Select projecting fastener: Specify Fastener which is being projected
*replace fasteners #P / Select edge of base part: Specify an edge on which the Fastener is being projected, see fig. 3.94, edge 1
*next #L / side #C / 2-d part: Confirm second Part by pressing **Enter** or specify it.
Select edge of base part: Specify an edge on which the Fastener is being projected, see fig. 3.94, edge 2.
*next #L / side #C / 2-d part: Confirm second Part by pressing **Enter** or specify it.
Select edge of base part: Press **Enter** to complete command
Note. If you forget to press **Enter** to complete command, all preliminary taken Fastener projections will be automatically deleted.

Replace Fasteners
Replaces Fastener in another from database. (You can specify a minifix to project and replace it with a self-support, for example).

Side
Changes the base side of Fastener (The Rastex can be fit in the upper or lower Part surface, see fig. 3.75 b)).
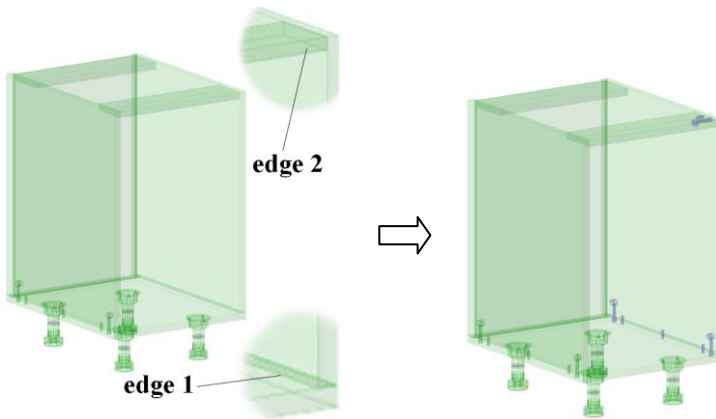
Fig. 3.94   Fastener projection

Note. The Fastener projection which is out of a Part edge is not drawn in 3D-model and not calculated in the quotation, see fig. 3.94, edge 2 (only one minifix and one dowel from Fastener group are present in 3D-model)

3.8.11. Fastener copying

Access methods
Menu: **Edit >> Copy**

Toolbar: **Edit**

Summary
You can copy existing Fastener group it allows you quickly insert Fastener into all joints. The list of prompts corresponds to it considered in section 3.8.1, page 121.

Note. If you not complete copying, all preliminary taken Fastener copies will be automatically deleted.

Внимание! You can't copy fastener group from one product to other products if the Fastener group references to Assembly variables or Part variables. "KM_L" variable is an exception, more details about "KM_L" see in section 3.8.1.1, page 125.

3.8.12. Options for Fastener size settings

Summary
You can determine a condition to insert a Fasteners which has a different size (Recognize difference between "Options for size settings" and "Variants" in the Fastener parameters dialog window). "Options for size settings" allows you to combine some single-type Fasteners in one Fastener. For exampe, you can combine some single-type runners in one "clever" runner. You will have one item to insert in the special database ("Database of fitting sets-preset"). When you insert combined runner in the 3D-model, one of the runners will be inserted concerning a drawer depth.

List of prompts
Execute command and specify a Fastener in the project.

Create a combined Fastener by dialog window, see fig. 3.95. This dialog window corresponds to it considered above, see fig. 3.42, page 77. More details about working with this window see in section 3.5.1.8, page 76.
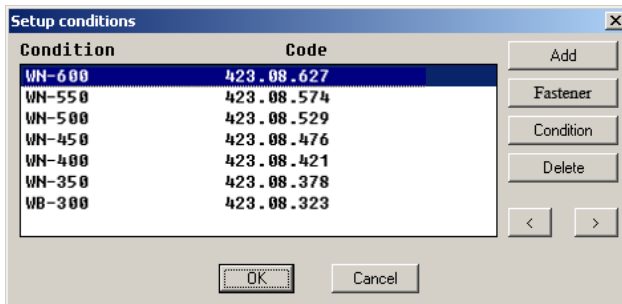

Fig. 3.95 Диалоговое окно "Условие установки крепежа"

Note. It is comfortable to use "KM_L" Fastener variable into condition.

3.8.13. Fastener inserted in one Part

There is capability to insert Fastener in one Part. The Fastener inserted in one Part can automatically find the second Part of the joint. This Fastener property allows you to create a universal typical model.

For example, if you create a drawer model, you can insert drawer runners and Fasteners, see fig. 3.96. When you insert this drawer in the cabinet and generate drawings, the holes in the cabinet side drawings and in the door drawing will be drawn automatically in case of the second Part search condition satisfaction.
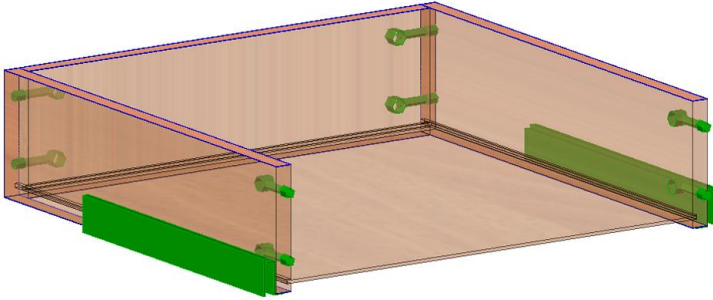
Fig. 3.96   Model which includes Fastener inserted in one Part

To insert Fastener in one Part, use one of above mentioned command. When program will request the second Part, press **Enter**. You will get the message (see below) and Fastener will be inserted in one Part.

Note. This message will be not given if you set drawer runners.

When you generate Part drawings, the automatic search of the second fixed Part will be executed for all Fasteners inserted in one Part.

If you execute **Service >> Information** and specify a Fastener inserted in one Part the automatic search of the second fixed Part will be executed only for this Fastener, you will get an informational message, where the mark of found Part will have "*" sign, see fig. 3.97.
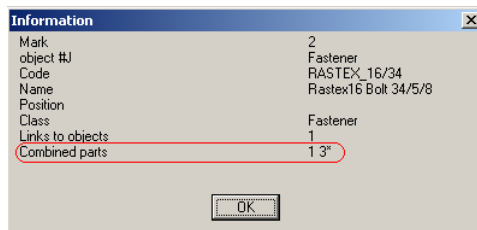

Fig. 3.97.   Fastener information

> *Note.*
> *The mark of Part, on which a Fastener is inserted, is "1".*
> *The found Part mark is "3", it is signed by "*".*

Conditions of automatic search of the 2nd fastened Part:

If the fasteners are inserted in one Part and a gap between two fastened Part (see fig. 3.98) is less than 1 mm, the second fastened Part will be found automatically.

If the drawer runner is inserted in one Part and a gap between two fastened Parts (between a drawer side and a cabinet side) is less than (Tr+1) mm, the second fastened Part will be found automatically. "Tr" is a drawer runner thickness determined in the drawer runner database.

If the shelf-support is inserted in one Part (in a shelf) and a gap between two fastened Part (between the cabinet side and the shelf) is less than 7 mm, the second fastened Part will be found automatically.

Note. The search of the second Part is executed from an Assembly and from a locked Assembly.

Important! If a hinge is inserted on one Part (on a door) the search of the second fastened Part is not executed in any case.

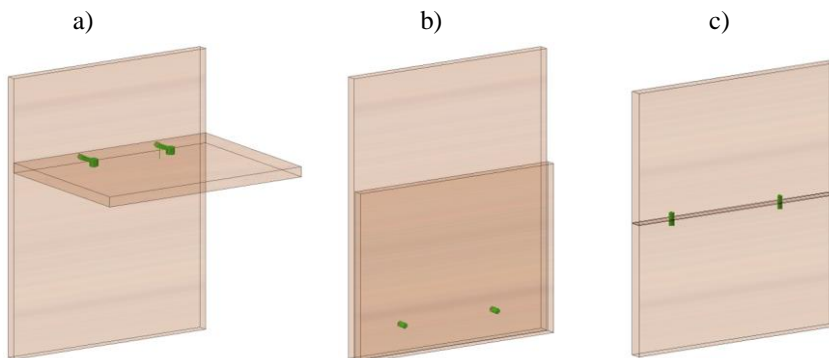a)   b)   c)

Fig. 3.98.   Different cases of the fastened Parts

3.8.14. "Change fitting side" operation

Access methods
Menu: **Operations >> Furniture fittings operations >> Change fitting side**

Summary
This command changes the base side of specified Fastener

List of prompts
Select fastener: Specify a fastener. See the operation example in fig. *3.99*.
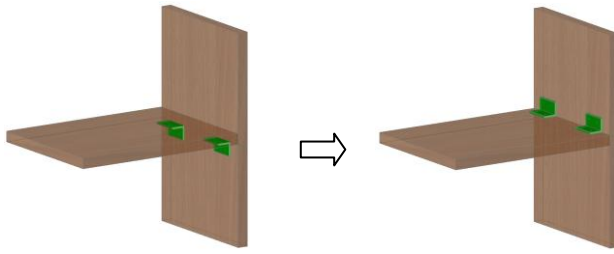
*Fig. 3.99. Result of "*Change fitting side*"*

3.8.15. Fastener properties

Any Fastener in the project has properties. To edit Fastener properties, execute **Edit >> Properties** and specify Fastener.

In detail the "Properties" window is considered in section 5.6, page 217. In this section is considered special Fastener properties, see fig. *3.100*, page 149.

Class

When you insert Fastener in the project it will be given the "Fastener" class. In this case Fastener will give holes in the Part drawings and it will be included in the Quotation. If you change "Fastener" for "No class", the Fastener will give holes in the Part drawings but it will be not included in the Quotation.

Off 1

Determines whether Fastener will give holes in the drawing of the first Part of joint. (The first Part of joint is the base Part of joint, you specified it the first to fix two Part). If this field is determined as "ON", the holes will be given.

Off 2

Determines whether Fastener will give holes in the drawing of the second Part of joint.

Note. If you not determine Fastener holes in the database, the holes will be absent in the Part drawings irrespective of "Off 1" and "Off 2" values.
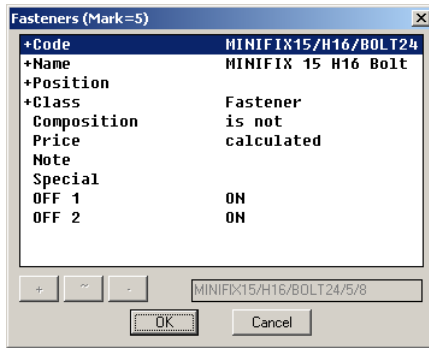
Fig. 3.100 Fastener properties

3.9. Constructional drawing

This object description see in chapter " Constructional drawing".

3.10. Informational size

The Informational Size can be set in the 3D-model. It shows any distance determined by user. It redraws, recalculates, and can be saved in the database together with a 3D-model.

The special feature of the Informational Size is his plane. His plane depends on a view point in the current moment. It will be redrawn when you change the view point therefore its value can be read by user in any time.

Note. If you change the view point, you should redraw Informational Size to read it. To redraw size, execute **Edit >> Redraw >> All sizes**.

There are two types of Informational Size: Orthogonal and Between points.

3.10.1. Informational size between points

Access methods
Menu: **Object >> Informational size >> Between points**
Toolbar: **Object**

Summary
This command creates a size which shows the shortest distance between any two points.

List of prompts
ODO (box) + / Base point: Specify the first Size point

ODO (box) + / *change point #N / Second point: Specify the second Size point

change point #N / *apply selected #G: Press **Enter** to apply specified point

Select point of dimension line: Determine a position of the Size line by specifying point on the screen.

ODO (box)

Specifies what point will be taken. It may be object point or box point (detail about box see in section 3.1.1, page 30). To specify point of object, must be displayed "*ODO (box) +*". To specify box point of object, must be displayed "*ODO (box) -*".

Note. If "*ODO (box) -*" is displayed and object hasn't box, it will be taken the object point.

Change point

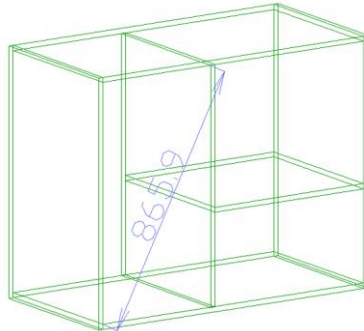Allows you to reselect the last specified point.

Fig. 3.101 Informational Size between two points

3.10.2. Orthogonal informational size

Access methods
Menu: **Object >> Informational size >> Between points**
Toolbar: **Object**

Summary

This command creates a size which shows a distance between projections of two points to any axis of the base object coordinate system.

List of prompts
ODO (box) + / First point: Specify the first Size point

*ODO (box) + / *change point #N / Second point*: Specify the second Size point

*change point #N / *apply selected #G*: Press **Enter** to apply specified point

*assembly #C / 1-st object / base object: Specify a base object

Direction of dimension line *X / Y / Z: Specify a base axis of the base object

Select point of dimension line: Determine a position of the Size line by specifying point on the screen.

ODO (box)

Specifies what point will be taken. It may be object point or box point (detail about box see in section 3.1.1, page 30). To specify point of object, must be displayed "*ODO (box) +*". To specify box point of object, must be displayed "*ODO (box) -*".

Note. If "*ODO (box) -*" is displayed and object hasn't box, it will be taken the object point.

Change point

Allows you to reselect the last specified point.

Assembly

Selects an object Assembly as a base object of the Informational Size.

1-st object

Selects the first specified object (by specifying the first point) as a base object of the Informational Size.
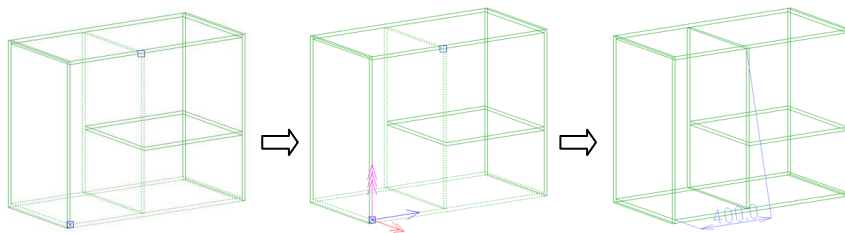

*Fig. 3.102 Orthogonal informational size*

3.10.3. Informational size editing

To edit the Informational Size, use **Edit >> Stretch**, more details about this command see in section 5.15 page 246. This command has some special feature concerning Informational Size. It works differently depending on a specifying point.

3.10.3.1. Size line position editing

To edit the Size line position, execute **Edit >> Stretch**, specify one of the extension Size lines or the Size line, specify a new Size line position.

Note. It is comfortable to use "Ortho mode" to move Size parallel one of the WCS axis. To on/off "Ortho mode" press "F8".
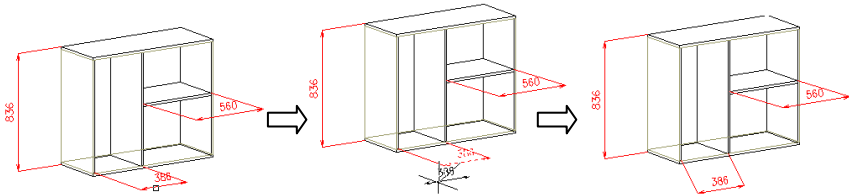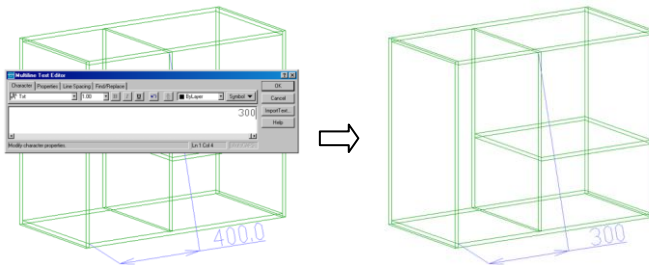

3.10.3.2. Size text position editing

To edit the Size text position, execute **Edit >> Stretch**, specify Size text, specify a new Size text position.
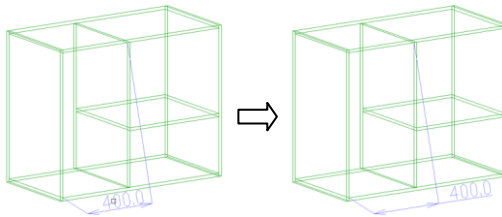
3.10.3.3. Size text editing

To edit Size text, execute **ACAD >> Edit >> Edit text**, specify Size text and edit it in the special dialog window. More details see in AutoCAD user's guide.

Chapter 4. Operations

Operation is an action which creates a new object element, for example a new base point or a cut on a side. There is a cut on a side example in the fig. 4.1. The cut on a side is determined with two parameters, see fig. 4.1 a), and with reference parameters, see fig. 4.1 b). The operation reference determines the operation position concerning operation base point.
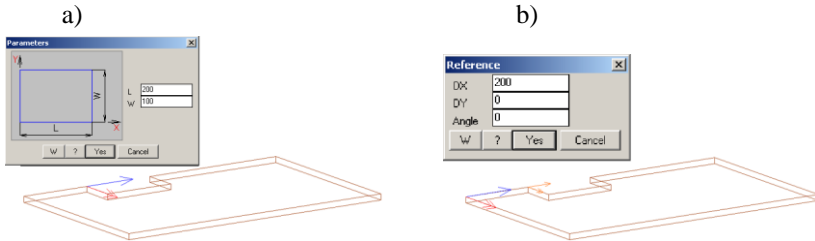
a)   b)

Fig. 4.1 "Cut on side" operation
a) Operation parameters
b) Operation reference parameters

Any operation is determined with:
− Operation coordinate system (hereinafter CS). The operation CS X-axis is defined by specifying Part edge.
− Operation parameters, these are "L" (cut length) and "W" (cut width) in above mentioned example. Note, "L" is determined along X-axis of the operation CS (look at the operation slide and at the Part, fig. 4.1 a)).
− Operation reference CS. The operation reference CS X-axis is defined by specifying Part edge. The point of the operation reference CS origin is the nearest object point to a specifying point.
− Operation reference parameter. These are offsets along X and Y-axis and an operation rotation angle, for example, look at the operation parameters slide, fig. 4.1 b).

All existing operations are in the "Operations" menu. Here you see all operations admissible for each object type (the menu is grouped in compliance with object type). One and the same operation can be admissible for different objects (for an Assembly, for a Part, for a Profile).
If you execute an operation, you can edit it by commands:
Operations >> Parameters
Operations >> Parameters of 1-st
Operations >> Reference

To delete operation, use **Operations >> Delete**

4.1. Operation parameters

List of Prompts
list of elements #C / select element: Specify the editable operation

list of elements

Opens the operation list by specifying any edge of object. Use it to select one of object operation from the list if it is difficult to specify the operation, see fig. 4.3, page 155.

Important! The result of the "parameters" operation depends on specifying point, see fig. 4.2. If you specify the chamfer edge, you will edit the chamfer parameters. If you specify an edge of the Part original contour, you will edit the original contour parameters.
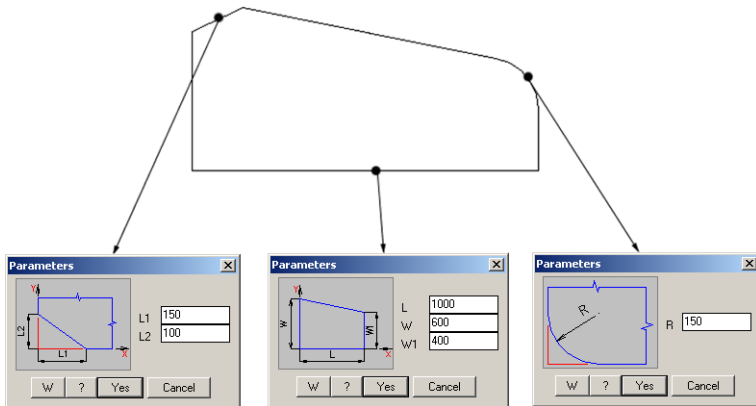

Fig. 4.2.   The operation result depend on specifying point

The trapeze is an original contour. The chamfer and the rounding are two executed operations

Note. The "W" button in the "Parameters" dialog window allows you to set view of dialog window.
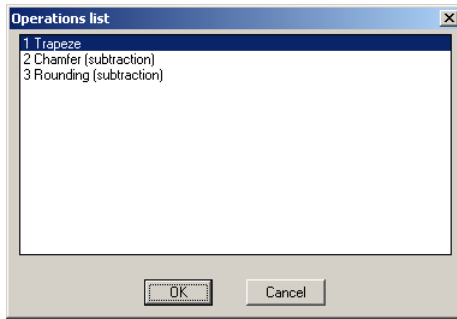
Fig. 4.3 Operation list

If the edit object has variables or it is included in an Assembly, which has variables, you will see the variable list in the "Parameters" dialog window, see fig. 4.4. You can use variables from the variable list to determine the operation parameters. More details about variables see in section 6.2, page 281.

Object variables list

Assembly variables list
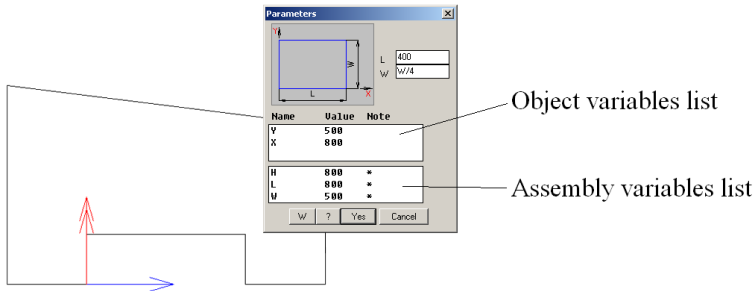
Fig. 4.4 The operation parameters of the Part included in an Assembly. The Part and the Assembly have variables.

Note. To copy a variable name from the variables list, execute mouse left double click. To insert a variable name in the parameter field, press "**Ctrl**"+"**V**".

Note. Press "**?**" button in the "Parameters" dialog window to open Catalogue list. You can use some information from catalogue list to determine object parameters. More details about catalogue list see in section 6.2.2.3, page 284.
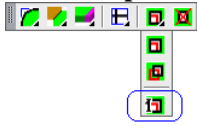
4.2. Parameters of the 1-st element

Access methods
Menu: **Operations >> Parameters of 1-st**
Toolbar: **Operation**

List of Prompts
Select object: Specify an editable object.

Summary
It edits parameters of the first operation and object variables in the same dialog window. The first operation is always the original Part contour. The specifying point doesn't matter.
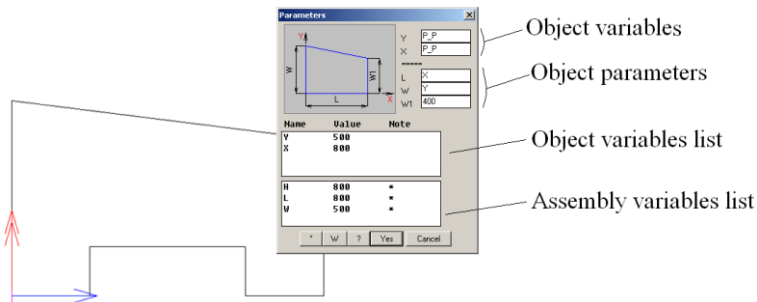

Fig. 4.5   Parameters of the first operation (of the original Part contour)

In above displayed example the first operation is a trapeze contour. It is determined with "L", "W", "W1" parameters. The "L" and "W" parameters are determined with "X" and "Y" Part variables. The "X" and "Y" are distance-variable (their value are determined with some distance in the 3D-model). More details about variables see in section 6.2.5, page 291.

Note. You can determine any object variable or object parameter in this window through some distance in the 3D-model. To execute it, set the cursor in the parameter or variable field and press "*" button.
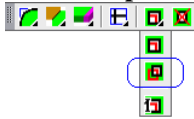
Note. It is comfortable to use this command to edit box variable of a Part, more details about Part box see in section 3.1.1, page 30.

4.3. Reference parameters

Summary

The operation reference parameters set the operation position on the object concerning an operation base point, see fig. 4.1. b), page 153.

Important! Differ an operation reference and object reference. The first determines operation position on the object; the second determines object position in 3D-model.

List of Prompts

The list of prompts corresponds to it considered in section 4.1, page 154.

4.4. Operation deleting

Summary

It deletes specifying operation. You should remember:

- It is impossible to delete the first operation by this command. To delete object (the first operation), execute **Edit >> Delete >> Objects.**
- It is impossible to delete operation if any object or other object operation refers to it (For example, you can use some point of an existing operation as a base point for other operation).

Note. If you need delete an operation on which there are references, you can:

- sequentially delete all operations which are referred on it and then delete it
- disable a model control (**Service >> Settings >> General settings**, checkbox "Model control"), delete operation, redraw 3D-model, enable the model control. In this case you will break your 3D-model (all undefined references will be changed with parameters (the variables will be set with current values and they will turn into parameters))
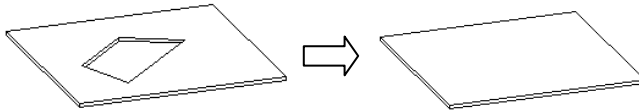
Fig. 4.6.   Operation deleting

4.5. Part operations

4.5.1. Part contour editing

4.5.1.1. Rounding

Access methods
Menu: **Operations >> Part operations >> Edit shape (contour) >>**
Rounding
Toolbar: **Operation**
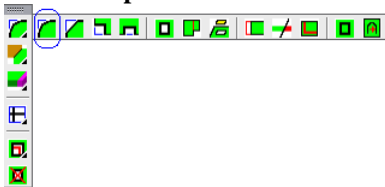
Summary
It rounds two Part edges by entered radius.
Note. The edges length has to be longer than rounding radius.

List of prompts
Select corner to be rounded: Specify an edge near the corner.
Enter the cut parameter in the dialog window and press "Yes".
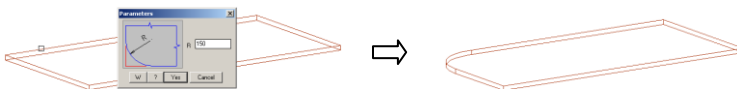
Fig. 4.7   Part edges rounding

4.5.1.2. Chamfer

Menu: **Operations >> Part operations >> Edit shape (con-tour) >> Chamfer**

Toolbar: **Operation**
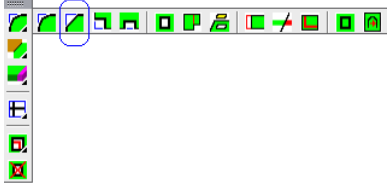
Summary
It creates chamfer of the specified Part edges.
Note. The edges length has to be longer than chamfer length.

List of prompts
Select corner to be cut: Specify an edge near the corner.
Enter the cut parameter in the dialog window and press "Yes".

Fig. 4.8   Part edges chamfer

Note. The specified edge determines the operation X-axis on the Part. The X-axis in the slide is parallel to the bottom slid edge, see fig. *4.8*.

4.5.1.3. Cut on a corner

Menu: **Operations >> Part operations >> Edit shape (con-tour) >> Cut on a corner**

Toolbar: **Operation**

Summary
It cuts the Part corner. The cut form is taken from the cut corner data-base, see fig *4.9*.

<u>List of prompts</u>

Specify the cut form in the database.

Select corner: Specify an edge near the corner. Enter the cut parameter in the dialog window and press "Yes".

Fig. 4.9   Cut corner database

Fig. 4.10   Cut on the Part corner

Note. The specified edge determines the operation X-axis on the Part. The X-axis in the slide is parallel to the bottom slid edge, see fig. 4.10.

Note. The "Rounding" and the "Chamfer" are special cases of "Cut on a corner".

Note. You can set the cut corner database by command **Database >> Shapes >> Database of corner cuts**. You can delete some not used forms or add new forms from the central database, which is edited only by software developer.

4.5.1.4. Cut on a side

<u>Access methods</u>
Menu: **Operations >> Part   operations >> Edit shape (contour) >> Cut on a side**
Toolbar: **Operation**
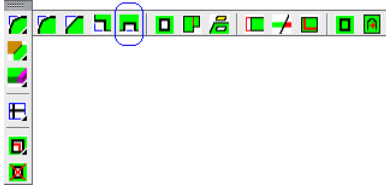
<u>Summary</u>
It cuts the Part side. The cut form is taken from the database of cuts (on side).

<u>List of prompts</u>
Specify the cut form in the database, see fig. 4.11.

Select edge: Specify a Part edge. The specifying point determines operation base point. It will be the nearest Part point to specifying point.

Enter the cut parameter and the cut shift ("DX" parameter) in the dialog window (see fig. 4.12) end press "Yes".

mirror #P / *apply as set #G: Press **Enter** to create the cut as you see in the preview view.

Mirror
Creates a mirrored cut. The cut will be mirrored about midline of the cut, which are parallel to X-axis of the operation coordinate system.

Note. This request will be given if the cut form is asymmetrical concerning operation Y-axis.
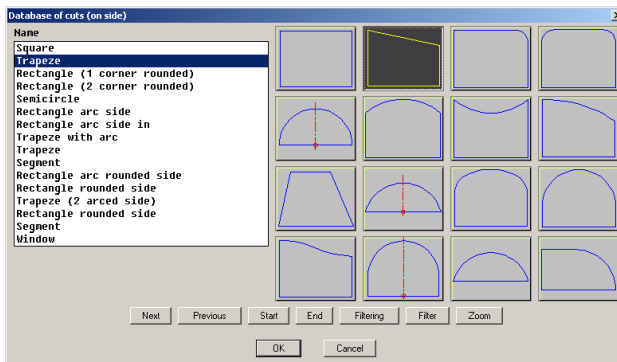
Fig. 4.11   Database of cuts (on side)
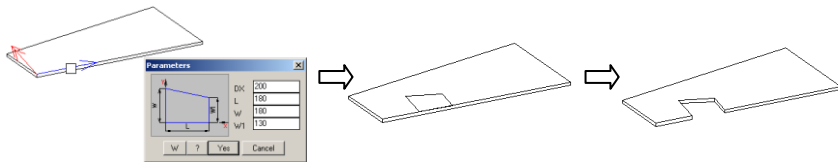
Fig. 4.12   Cut on the Part side
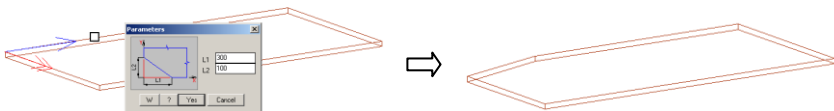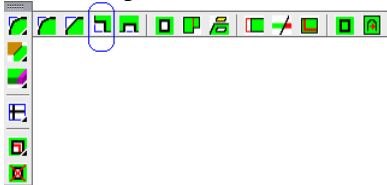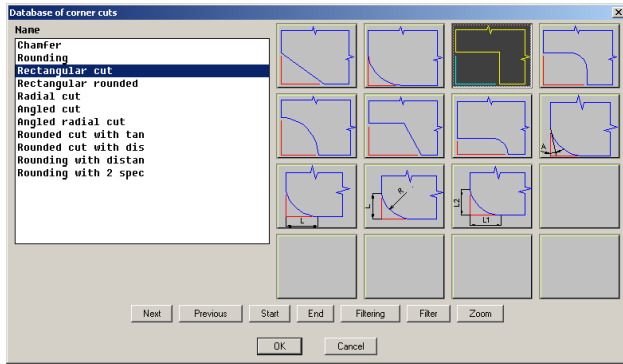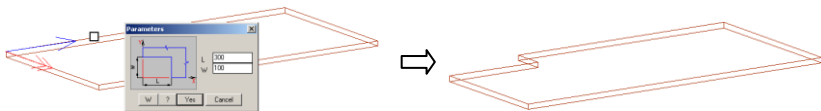
Note. The specified edge determines the operation X-axis on the Part. The X-axis in the slide is parallel to the bottom slid edge, see fig. 4.12.

Note. You can set the cut corner database by command **Database >> Shapes >> Database of cuts (on side)**. You can delete some not used forms or add new forms from the central database, which is edited only by software developer.

4.5.1.5. Opening

Access methods

Menu: **Operations >> Part   operations >> Edit shape (contour) >> Opening**

Toolbar: **Operation**
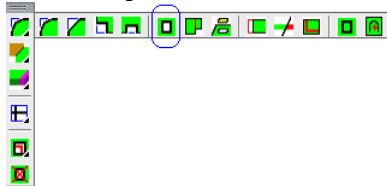
Summary

It creates an opening on the Part surface. The opening form may be taken from the database or created by user.

This command has some operation mode. They are considered below.

Note. You can set the opening database by command **Database >> Shapes >> Database of cuts**. You can delete some not used forms or add new forms from the central database, which is edited only by software developer.

List of prompts (exemple 1)

The operation CS is determined with the Part CS, an opening contour is taken from the "database of cuts")

by contour #R / Base point: Specify an opening base point. It will be the nearest Part point to the specifying point, see fig. 4.14 a).

Specify the opening form in the database, see fig. 4.13.

redefine base point #N / *direction along CS of object #C / 1-st direction point: Press **Enter** to determine the operation coordinate system (hereinafter CS) by Part CS, see fig. 4.14 b).

Enter the opening parameters and the opening position parameters in the dialog window, see fig. 4.14 b).

rotation angle #E / reverse X #X / reverse Y #Y / reference point #N / *apply #G: Press **Enter** to create the opening as you see in the preview view, see fig. 4.14 d).

Fig. 4.13 Database of cuts
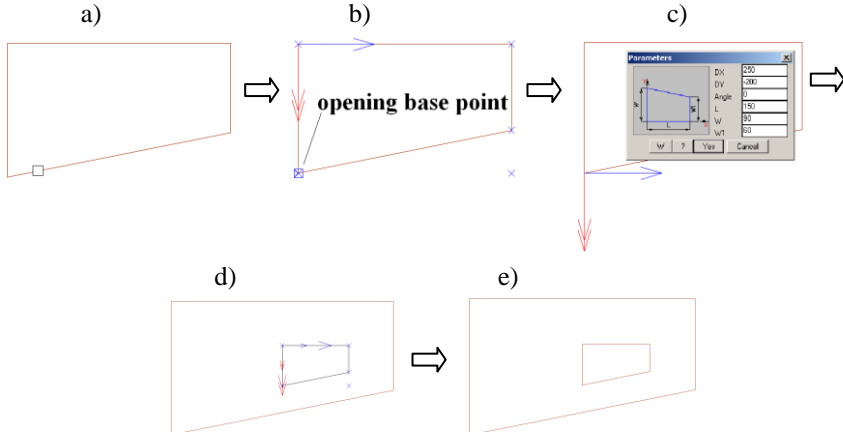
Fig. 4.14 "Opening" operation, example 1

Additional options
By contour
Creates the opening by contour drawn by user. More details see below.

Redefine base point

Allows you to redefine the operation base point. You can specify one of the highlighted points, see fig. 4.14 b).

1-st direction point

Allows you determine X-axis operation CS by specifying two Part points, see fig. 4.15. More details see below.

Rotation angle

Determines the opening rotation angle concerning operation X-axis
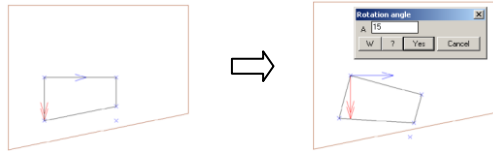
Reverse X

Reverses the operation X-axis
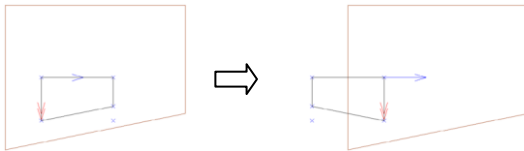
reverse Y

Reverses the operation Y-axis
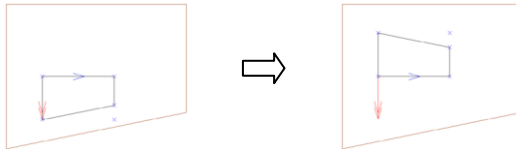
reference point

Redefines the base point in the opening contour
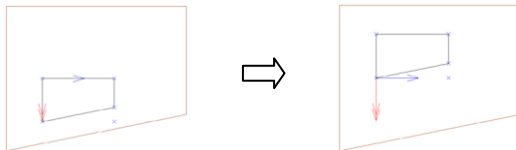
List of prompts (example 2)

The operation CS is determined with two specified points; an opening contour is taken from the "database of cuts".

by contour #R / Base point: Specify an opening base point. It will be the nearest Part point to specifying point, see fig. 4.14 a).
Specify the opening form in the database, see fig. 4.13.

*redefine base point #N / *direction along CS of object #C / 1-st direction point*: Specify the first Part point to determine X-axis of the operation CS, see fig. 4.15 a), point "1"
2-d direction point: Specify the second Part point to determine X-axis of the operation CS, see fig. 4.15 a), point "2".
1-st centering point on Y (ENTER - without centering): Press Enter to activate "without centering" mode for Y-axis.
Enter the opening parameters and the opening position parameters in the dialog window, see fig. 4.15 b).
*rotation angle #E / reverse X #X / reverse Y #Y / reference point #N / *apply #G*: Press **Enter** to create the opening as you see in the preview view, see fig. 4.15 c).
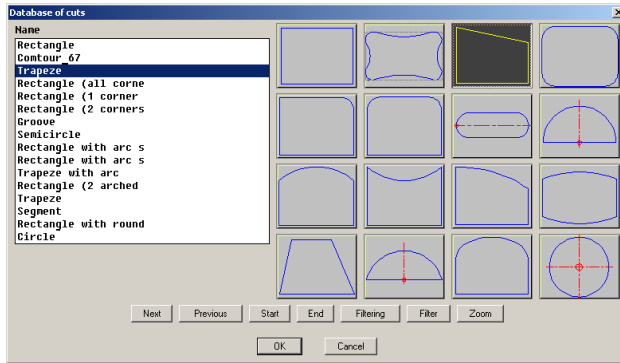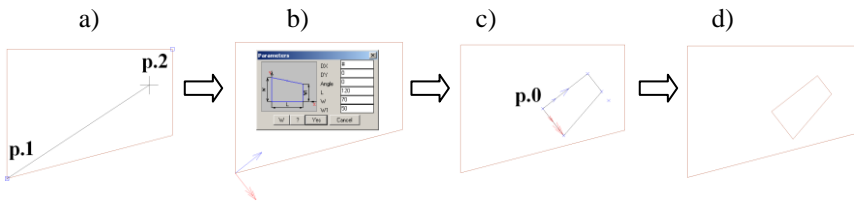

Fig. 4.15 "Opening" operation, example 2.

Note. You can center opening base point (see fig. 4.15 c), point "0") concerning line determined by point "1" and point "2". To do this, enter "#" sign in "DX" parameter. To center the opening base point concerning Y-axis, you should specify additional points (see "*1-st centering point on Y*" request).

The opening contour may be created by user. You can create the opening contour by "Constructional drawing" or AutoCAD primitives (lines and arcs). The opening contour should be prepared before executing the "Opening" operation.

by contour #R / Base point: Press **Enter** to activate "by contour" option.

Part to be edited: Specify a Part which will be edited

Projecting contour: Specify an opening contour.

Note. The opening contour may be determined with another Part. In this case the edited Part and a Part specified as the opening contour should be parallel. The edited Part should to be situated in the "Construction order" after the subtracted Part.

Important! The opening contour hasn't to split Part into two parts. If it happens, the operation will be canceled.
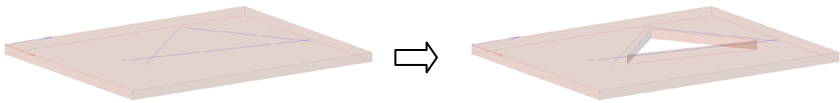
Fig. 4.16 "Opening" operation, example 3. The opening contour is determined with "Constructional drawing"

Note. More details about Constructional Drawing see in chapter "Constructional Drawing".

4.5.1.6. Joint

Access methods

Menu: **Operations >> Part   operations >> Edit shape (contour) >> Join**

Toolbar: **Operation**
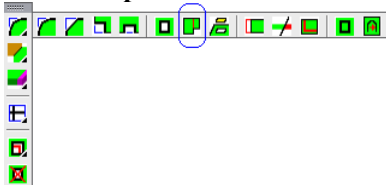
Summary

It combines the edited Part and an additional contour. The additional contour may be taken from the database of cuts or created by user. Also the additional contour may be determined with another Part.

<u>List of prompts</u>

The list of prompts corresponds to it considered in section 4.5.1.5, page 162.

4.5.1.7. Subtract Part contour

<u>Access methods</u>
Menu: **Operations >> Part   operations >> Edit shape (contour) >> Subtract part contour**
Toolbar: **Operation**
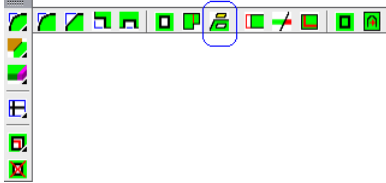
<u>Summary</u>

It subtracts contour one Part from another Part contour. You can specify a block from the "Sink" database as a Part which will be subtracted.

Note. The "Construction order" doesn't matter for this operation.

<u>List of prompts</u>
Part to be edited: Specify a Part which will be edited.
Object to be subtracted: Specify a Part which will be edited
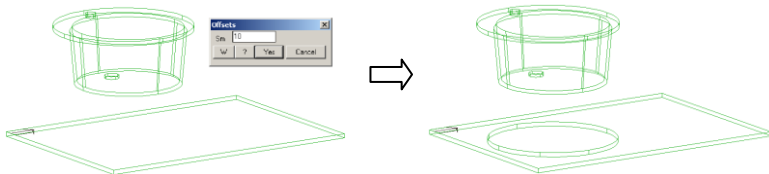
Fig. 4.17   "Subtract Part contour" operation

4.5.1.8. Cut by line

<u>Access methods</u>
Menu: **Operations >> Part   operations >> Edit shape (contour) >> Cut by line**
Toolbar: **Operation**
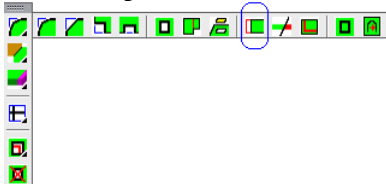
Summary

It cuts a Part. The cutting line may be determined with:
- two Part points
- a Wall (For example, it can be created with a command **Build >> Wall >> Polywall**)
- a Line (It can be drawn with a command **ACAD >> Drawing >> Line**)

List of prompts

by wall or Line #C / First point of cutting line: Specify the first Part point

redefine the first point #C / second point of cutting line: Specify the second Part point

Confirm side to be cut.

To change direction - any key / ENTER - apply: Press **Enter** to cut the highlighted by arrow Part piece

Enter an offset distance in the dialog window, see fig. *4.18*.
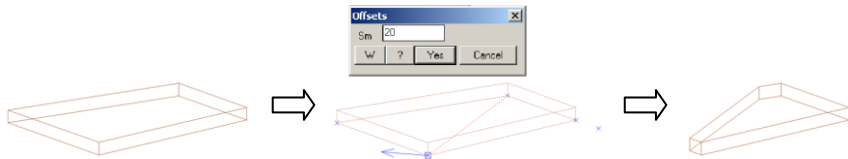
Fig. 4.18.   "Cut by line" operation

4.5.1.9. Cut by surface

Access methods

Menu: **Operations >> Part   operations >> Edit shape (contour) >> Cut by surface**

Toolbar: **Operation**
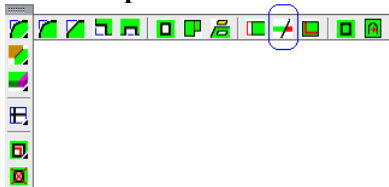
Summary

It cuts a Part by surface. The surface can be determined with three points belong to different objects or with another Part.

Note. The object, which determines cutting surface, should be placed in the "Constructional order" before the edited Part.

Specify a point on the remaining piece of the Part (for Bent Sheet – cut end): Specify a Part point. The point determines the Part piece which will be remained after the operation executing.

*by 3 points / redefine point #N / Cutting surface: Press **Enter** to activate "by 3 ponts" method.

Base point: Specify the first point of the cutting surface.

*change point #N / Point of axis X: Specify the second point

*change point #N / Point of axis Y: Specify the third point

change point #N / *apply selected #G: Press Enter to complete the operation.

Enter an offset value in the dialog window. Look at fig. *4.19* to see the example. The cutting surface (ABC) is determined with three Assembly points.

Redefine point

Redefines a Part piece which will be remained after the operation executing.

Cutting surface

Specifies object which determine a cutting surface. You can specify a Part, an Assembly surface, a Constructional Drawing.

Change point
Redefines the last specified point.
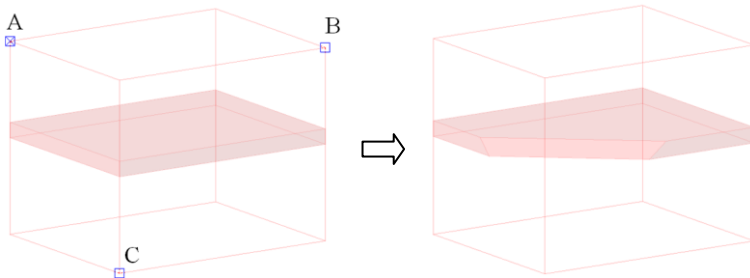
*Fig. 4.19   "Cut by surface" operation. The cutting surface
is determined with three Assembly points*

4.5.1.10. Angular cut of the edge

<u>Access methods</u>

Menu: **Operations >> Part   operations >> Edit shape (contour) >>
Angular cut of the edge**

Summary

It cuts a Part end by surface. The edge of the Part end coincide with a cutting surface.

List of prompts

Cut edge: Specify the edge of the edited Part. Note, the specified edge coincide with cutting edge, see fig. *4.20*, page171.

*side #C / parallel #G / perpendicular #H / angle #E / Point 1 of cutting surface of parallel to one: Specify the first point to determine cutting surface, see fig. *4.20*.

change point #N / Point 2 of cutting surface of parallel to one <ENTER - due to edge and point 1>: Press **Enter** to determine cutting surface by point and cutting edge, see fig. *4.20*.

Note. Object, which determines cutting surface, have to be situated in the Constructional order above the edited Part.

Side

Changes specified edge of the Part end to the opposite edge.

Parallel

The cutting surface is determined with a cutting edge and specified Part. The cutting surface is parallel to specified Part.

Perpendicular

The cutting surface is determined with a cutting edge and specified Part. The cutting surface is perpendicular (round on 90 degree) to specified Part, see fig. 4.22, page 172

Angle

The cutting surface is determined with a cutting edge and the edited Part. The cutting surface coincides with cutting edge and is rotated on some angle concerning the edited Part. The rotation angle is entered in a dialog window.

Change point

Redefines the last specified point.
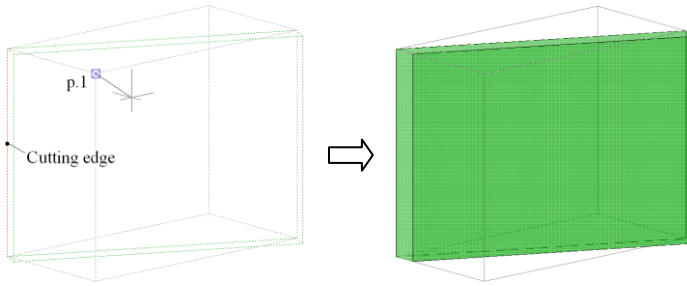
Fig. 4.20   *"Angular cut of the edge" operation, example 1*

Cutting surface is determined with the cutting edge and point 1.
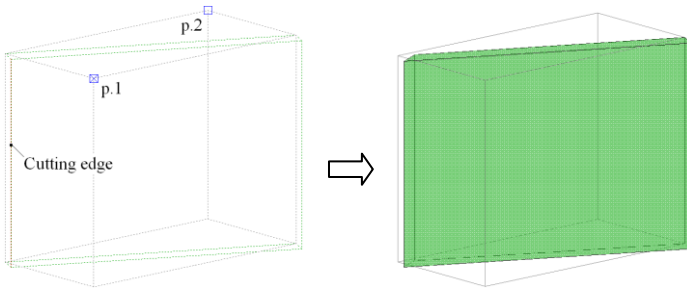
Fig. 4.21   *"Angular cut of the edge" operation, example 2*

Cutting surface is determined with the cutting edge and two points (point 1 and point 2).
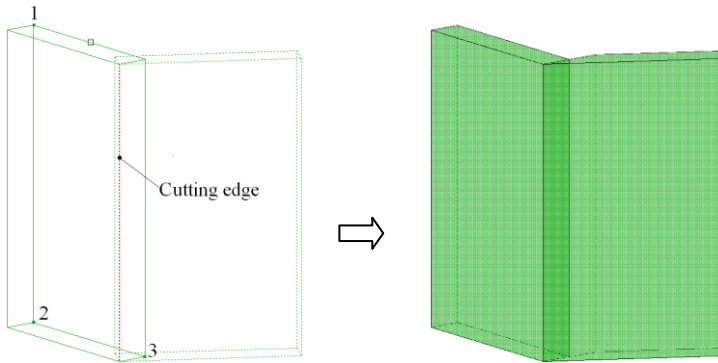
Fig. 4.22. "Angular cut of the edge" operation, example 3

> *Cutting surface is determined with the cutting edge and the surface perpendicular to the specified Part surface (perpendicular to the "123" surface).*

4.5.1.11. Offset edge

Access methods

Menu: **Operations >> Part operations >> Edit shape (contour) >> Offset edge**

Toolbar: **Operation**
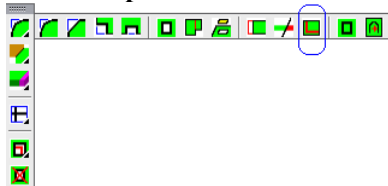
Summary

It replaces specified Part edges by shifted edges.

Note. The base Part points are remained on a Part contour. You can specify them after an offset executing.

List of prompts

Select edge for offset: Select a Part edge

perimeter #G / Select edge for offset: Select next Part edge.

perimeter #G / Select edge for offset: Select next Part edge.

…..

perimeter #G / Select edge for offset: Press **Enter** to complete operation.

Enter offset distance in the dialog window. To shift edge according to the arrow in the preview view, enter the positive value. To shift in another side, enter the negative value.

Note. All specified edges have to belong to one Part.

Perimeter

Selects all Part edges on perimeter.

4.5.2. Auxiliary points of the Part contour

4.5.2.1. Points XY

<u>Access methods</u>

Menu: **Operations >> Part   operations >> Point   of   contour >> Points XY**

Toolbar: **Operation**
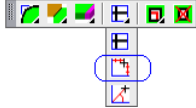
<u>Summary</u>

It creates auxiliary points in Part contour. The new points are created concerning one of the existing Part point. Their positions are determined by user in rectangular coordinate system.

Note. To see all existing Part points, use option "*change point*" in any command where it is provided.

<u>List of prompts</u>

Select base point on object: Specify a base point of the operation

Another base point (ENTER - selected): Specify another base point or press **Enter** to apply specified.

Enter coordinates of an auxiliary point in the dialog window and press "Yes". Enter coordinate of next auxiliary point and press "Yes".

Press "Exit" button to complete operation.
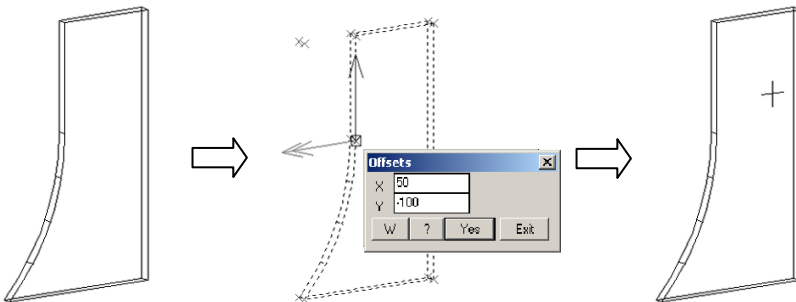
Fig. 4.23.   Creating of auxiliary points by "Points XY" operation

4.5.2.2. Points AL

Toolbar: **Operation**
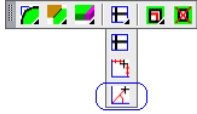
Summary
This operation corresponds to the operation considered in section 4.5.2.1 page 173. But auxiliary points position is determined in polar coordinate system.

4.5.2.3. External points projection

Summary
It creates new points on an edited Part by projecting points of base Part.

List of prompts
Specify a Part surface to edit: Specify one of the Part points to define an editing surface.
Confirm side of object
To change direction - any key / ENTER - apply: Press **Enter** to apply highlighted by arrow surface. Or press any key to select the opposite surface.
Projecting point: Specify a projected point on the base Part
change point #N / next projecting point: Specify next projected point.
change point #N / next projecting point: Press **Enter** to complete command

Note. The base object has to be placed over the edited Part in the Construction order.

Change point
Redefines the last specified point on the base Part.
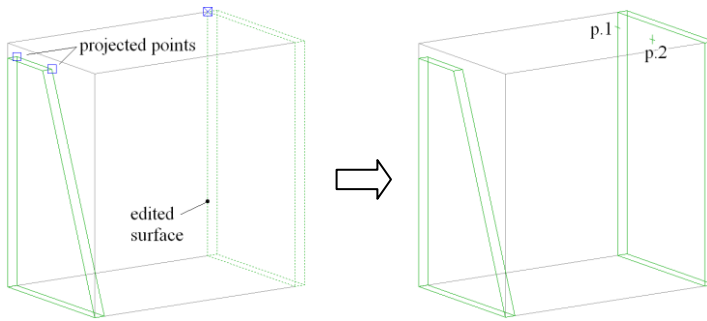
*Fig. 4.24. "External points projection" operation
(p.1 and p.2 is a result of the operation)*

4.5.2.4. Intersection points with external edges

Access methods

Menu: **Operations >> Part operations >> Point of contour >> Intersection points with external edges**

Summary

It creates new points on an edited Part by intersecting specified edge and surface.

List of prompts

Specify a Part surface to edit: Specify one of the Part points to define an editing surface, see fig. 4.25, point "0".

Confirm side of object

To change direction - any key / ENTER - apply: Press **Enter** to apply highlighted by arrow surface. Or press any key to select the opposite surface.

external edge: Specify an edge of the base Part, see fig. 4.25, edge "1"

undo / next external edge: Specify next edge, see fig. 4.25, edge "2"

undo / next external edge: Press **Enter** to compete the command. (The point "1" and the point "2" is a result of the operation, see fig. 4.25)

Note. The external edges may be taken from different object.

Undo

Removes the most recent edges added to the operation.
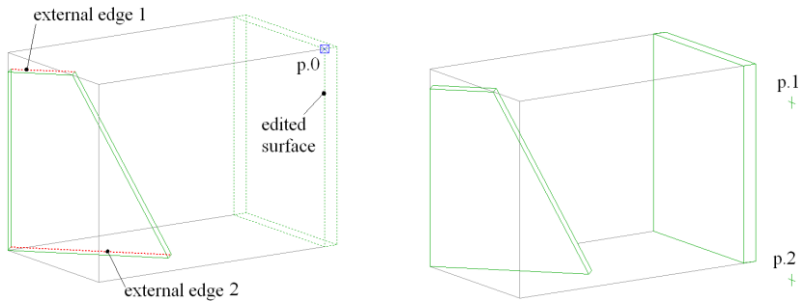
Fig. 4.25. "Intersection points with external edges" operation
(p.1 and p.2 is a result of the operation)

4.5.2.5. Intersection points with external surface

Access methods

Menu: **Operations >> Part operations >> Point of contour >> Intersection points with external surface**

Summary

It creates new points on an edited Part by intersecting specified edge and surface.

List of prompts

Specify a Part surface to edit: Specify a Part to be edited
Confirm side of object
To change direction - any key / ENTER - apply: Define a Part surface to be edited.

External edge: Specify an edge of the base object (the base object has to be situated in the construction order upper than edited object).

cancel #J / next external edge: Specify next edge of the base object
cancel #J / next external edge: Press Enter to complete operation.

There is an example for this operation, see fig. 4.26 page 176. Note, it was created the intersection points for arc.
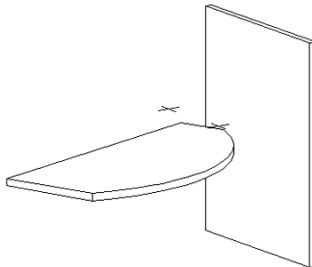
Fig. 4.26. "Intersection points with external surface" operation

4.5.3. Add box

<u>Access methods</u>
Menu: **Operations >> Add box**

<u>Summary</u>
It creates an overall dimensions (hereinafter box) of the Part. More details about Part box see in section 3.1.1, page 30.

<u>List of prompts</u>
reference to contour points #G / Part to be edited: Select Part to add the box.
reference to contour points #G / Part to be edited: Select Next Part.
reference to contour points #G / Part to be edited: Press **Enter** to complete command.

Note. When you add the Part box, the reference of the edited object will be automatically replaced to the box points (The reference of the). If you would like to keep the reference on object points, use "***reference to contour points***" option.

When the operation has been completed, you don't see any changes. But Part variables and Part reference are edited. The box is added.

4.5.4. Process a Part surface

4.5.4.1. Pocket

<u>Access methods</u>
Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Pocket**
Toolbar: **Operation**
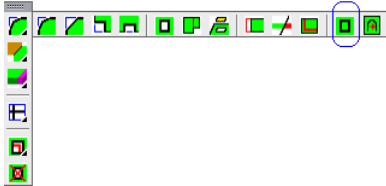
<u>Summary</u>
It creates a pocket on the Part surface. The pocket form may be taken from the database or created by user. The list of prompts corresponds to it considered in section 4.5.1.5 page 162.
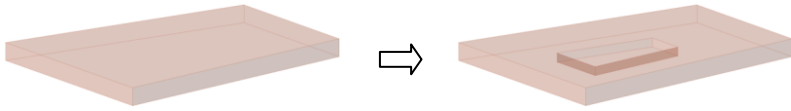
Fig. 4.27   "Pocket" operation

Note. The Part thickness inside the pocket has to be more than a "residual Part thickness" setting (command **Service >> Settings >> Parts and Sheets**, setting "residual Part thickness")

4.5.4.2. Routing

Access methods

Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Routing**

Toolbar: **Operation**
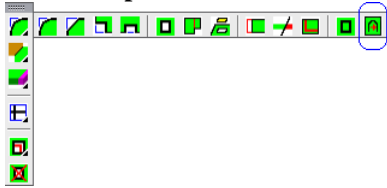
Summary

It creates a routing on the Part surface. The routing section is taken from the "Cutter profile database". The routing path may be taken from the "Database of processing (routing) paths" or created by user.

Note. The Part thickness after routing has to be more than a "residual Part thickness" setting (command **Service >> Settings >> Parts and Sheets**, setting "residual Part thickness").

Note. You can set the "Cutter profile database" by command **Database >> Processing >> Cutter profile database**. You can delete some not used cutter profile or add new cutter profile from the central database, which is edited only by software developer.

Note. You can set the "Database of processing (routing) paths" by command **Database >> Shapes >> Database of processing (routing) paths**. You can delete some not used routing paths or add new routing paths from the central database, which is edited only by software developer.

The list of prompts corresponds to it considered in section 4.5.1.5 page 162.
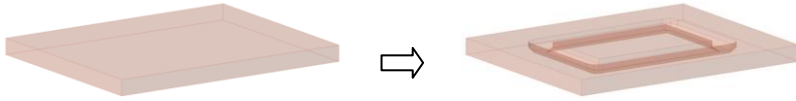
Fig. 4.28   "Routing" operation

4.5.4.3. Change side of routing

<u>Access methods</u>
Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Change side of routing**

<u>Summary</u>
It changes routing surface of Part on opposite Part surface.

<u>List of prompts</u>
Select processing to change side of processing: Specify an edge of the routing path.

Fig. 4.29   " Change side of routing" operation

4.5.4.4. Replace routing

<u>Access methods</u>
Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Replace routing**

<u>Summary</u>
It changes routing section with another routing section from "Cutter profile database".

<u>List of prompts</u>
Select processing to replace: Specify an edge of the routing path.

4.5.4.5. Processing by points XY

<u>Access methods</u>
Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Processing by points XY**

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Summary

It creates processing on the Part surface. The processing may has some standard types.

There is an example below, where you can see the creation of three holes with their coordinates (300; 50), (350; 50), (400; 50).

List of prompts

Select base point on object: Specify the operation base point on a Part, see fig. 4.30.

Another base point (ENTER - use selected): Press **Enter** to apply specified base point or specify another point

Fig. 4.30   The operation base point is specified

Enter in the dialog window (see fig. 4.31 a), page 180) coordinates of the first hole and press "Yes"

Enter in the dialog window (see fig. 4.31 b)) coordinates of the second hole and press "Yes"

Enter in the dialog window (see fig. 4.31 c)) coordinates of the third hole and press "Yes"

Press "Exit" in the dialog window (see fig. 4.31 d)) to finish coordinates entering.

Note. When you press "Yes", the previous point of the hole will be displayed in the 3D-model.

a)   b)   c)   d)
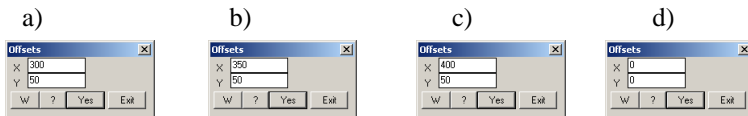
Fig. 4.31   The points coordinates entering

Enter the hole parameters in the next dialog window (see fig. 4.32) and press "Yes" to compete operation. To have a through-hole, enter H=0.

Note. Press "Change type" button (see fig. 4.32) to change the hole type.
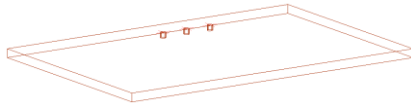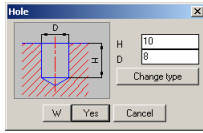
Fig. 4.32   The hole parameters entering

Note. If you create a hole with countersink, this hole type will be displayed in 3D-model simply (without countersink).

Fig. 4.33   Editing the holes parameter

To edit coordinates of existing holes, use **Operations >> Parameters** (it may be comfortable to use "list of elements" in option to specify processing).

To edit processing type, use **Operations >> Part operations >> Process a surface (Pocket, routing)>> Edit processing by points**.

To delete processing, use **Operations >> Delete**. (It deletes a group of holes determined in one operation)

4.5.4.6. Processing by points AL

Access methods

Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Processing by points XY**

Summary

It creates processing on the Part surface. This command corresponds to the command considered in section 4.5.4.5, page 179, but the processing coordinates in this operation are determined in polar coordinate system.

4.5.4.7. Processing by points with step

Access methods

Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Processing by points with step.**

Summary

It creates processing on the Part surface. The processing may has some standard types. The processing (holes) is placed at equal distance along the specified line.

List of prompts

Select base point on object: Specify the operation base point on a Part, see fig. 4.34 a), page 183, point "0".

*along X #X / along Y #Y / redefine base point #N / Select point to specify direction: Specify the second point to define the operation direction or press **Enter** to define the operation direction correspond to the X-axis of the Part CS.

Enter the operation parameter in the dialog window:

"SmX", "SmY" – These parameters shift the first processing point concerning base point (p.0) of the operation.

"Step" – This parameter determine the step between holes.

"N" – This is a quantity of holes in operation.

Specify the type and hole parameters in the next dialog windows. To have a through-hole, enter H=0.

See the operation result in fig. 4.34 b), page 183.

Along X

Sets the operation direction along the X-axis of the Part CS.

Along Y

Sets the operation direction along the Y-axis of the Part CS.

Redefine base point

Redefines the operation base point.

Для редактирования координат, шага и количества обработки следует использовать **Операции >> Параметры**. Удобно воспользоваться опцией "Список элементов" внутри операции "Параметры", см. рис. 4.33 стр. 181.

Для редактирования типа и параметров обработки следует использовать команду **Операции >> Для плоской детали >> Обработка плоскости >> Редактирование обработки по точкам,** см. раздел 4.5.4.9, стр. 184.

Для удаления обработки следует использовать **Операции >> Удаление.** Удаляется вся обработка заданная внутри операции.

a)

p.0

b)
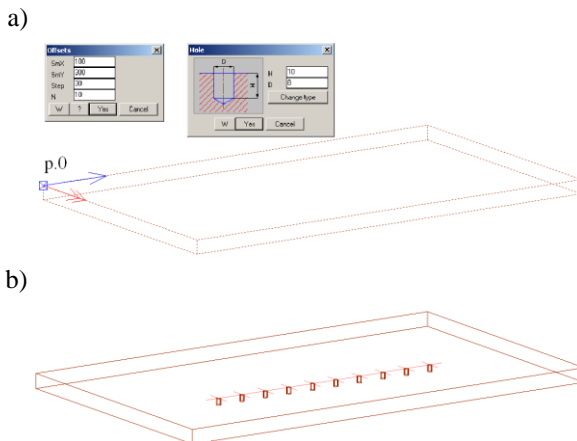
Fig. 4.34   "Processing by points with step" operation

To edit coordinates of existing holes, use **Operations >> Parameters** (it may be comfortable to use "list of elements" in option to specify processing).

To edit processing type, use **Operations >> Part operations >> Process a surface (Pocket, routing)>> Edit processing by points**.

To delete processing, use **Operations >> Delete**. (It deletes a group of holes determined in one operation)
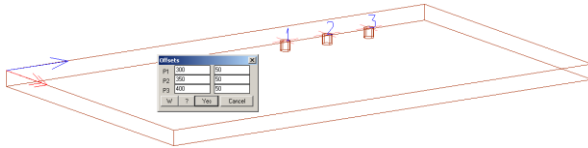
4.5.4.8. Process part along an edge

Access methods

Menu: **Operations >> Part operations >> Process a surface (Pocket, routing)>> Process part along an edge.**

Summary

It creates processing on the Part surface. This operation corresponds to it considered in section 4.5.4.7, page 181. The processing (holes) coordinates in this operation are determined in the dialog window presented in fig. *4.35*, page. 183.
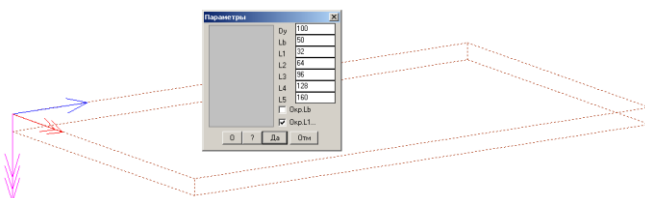
Fig. 4.35   "Process part along an edge" operation

4.5.4.9. Edit processing by points

Summary
It edits parameters and a type of the processing by points and processing along edge. See sections 4.5.4.5 (page 179)–4.5.4.8, (page 183) to take information about processing operations.

4.5.5. Edge processing

4.5.5.1. Edge processing

Toolbar: **Operation**
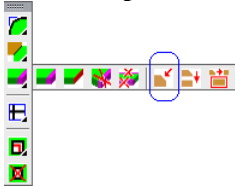
Summary
It mills the end of Part, see fig. *4.37*, page 185. The mill section is taken from the "Database of edge processing".

Note. You can set the "Database of edge processing" by command **Database >> Processing >> Database of edge processing**. You can delete some not used processing section or add new processing section from the central database, which is edited only by software developer.

List of prompts
*select #D / edge #H / List of edges to put edge processing <Rounding>: Specify the edited edge
perimeter #G / edge #H / List of edges to put edge processing <Rounding>: Specify the next edited edge
perimeter #G / edge #H / List of edges to put edge processing <Rounding>: Press **Enter** to complete operation

Note. The specified Part edge will be highlighted, in preview view, see fig. 4.37, page 185. You can specify edges of different Parts in one operation.
Note. If you specify an edge which is included in a smooth connected sequence of edges it will be automatically taken the sequence of segments.

Select

Sets section of edge processing. The section is determined from the "Database of edge processing", see fig. 4.36. Note, the name of current section you see in the command line, for example "Rounding".

Edge

Selects only specified edge even if it is included in a smooth connected sequence of edges to process them.

Perimeter

Selects all edges on Part perimeter to process them.
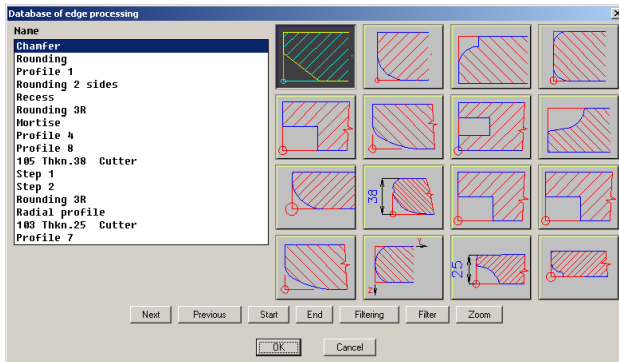
Fig. 4.36 Database of edge processing
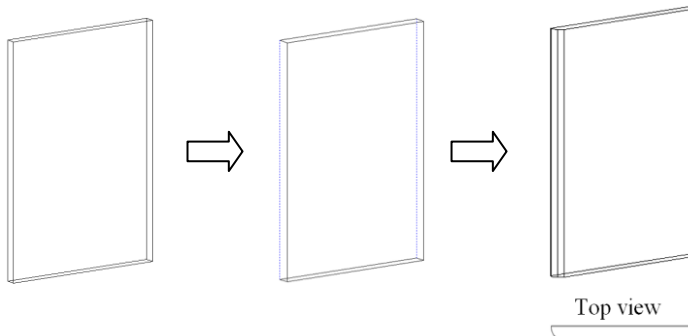
Fig. 4.37. Edge processing

Note. You can edit some section parameters on the Part by command **Operations >> Parameters**

4.5.5.2. Side of processing

Summary
It changes side of processing to opposite Part side.

List of prompts
Select processing to change side of processing: Specify the Part end which edge is processed to change side of processing.
cancel #J / Select processing to change side of processing: Specify the next processed edge.
cancel #J / Select processing to change side of processing: Press **Enter** to complete operation.
Note. You can specify edges of the different Parts in one operation.

Cancel.
Excludes the last specified edge from a set of the edited edges.
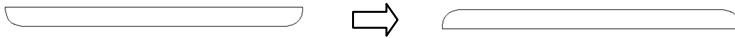
Fig. 4.38. "Side of processing" operation (this is top view)

4.5.5.3. Replace processing

Summary
It replaces processing section with another section from the "Database of edge processing".

List of prompts
Select edge to replace processing: Specify the Part end which edge is processed.
cancel #J / Select edge to replace processing: Specify the next processed edge.
cancel #J / Select edge to replace processing: Press **Enter** to complete operation.

Cancel.
Excludes the last specified edge from a set of the edited edges.

4.5.5.4. Remove processing and remove all processing from a part

<u>Access methods</u>

Menu: **Operations >> Part operations >> Edge processing >> Remove processing**

Operations >> Part operations >> Edge processing >> Remove all processing from the part

Toolbar: **Operation**
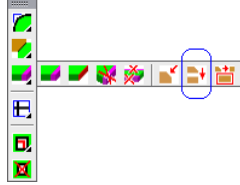
Remove processing   **Remove all processing from the part**

<u>Summary</u>

The first operation (remove processing) deletes a processing from specified edge of the Parts. The second operation deletes all edge processing from the Part by specifying any Part edge.

4.5.6. Edging

The edging operations complete Parts and Bent Sheets with an edgebanding strip. The thickness of an edgebanding strip can be subtracted or added to the Part contour. If it is subtracted, the edgebanding strip is named internal. If it is added, the edgebanding strip is named external. More details see in section 3.1.2.2, page 33.

Note. When program generates Part drawings, the dimensions can be created concerning the Part contour (cutting Part contour) or Part contour plus edgebanding strip (overall dimensions Part contour including edgebanding strip thickness)

The length of the edgebanding strip glued to Part can be presented in the Quotation document. The strip length can include overhang. The edgebanding strip glued to the smooth connected sequence of segments is one piece of strip (the overhang is set to the piece of strip glued by one technological operation). Two connected segments are smooth if their adjacent angle is less than "applicable angle for 1 operation" (**Service >> Setting >> Parts and Sheets**, setting "**applicable angle for 1 operation**")
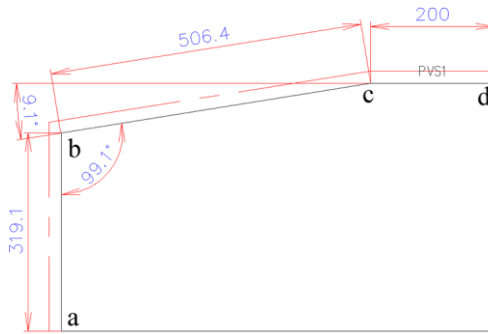
Fig. 4.39 Part drawing with edgebanding strip

See example above. If "applicable angle for 1 operation" is equal to 10 degree, you will get two edgebanding strip pieces "ab" and "bcd" (you will need two technological operations to glue strip).

Conditionally all edgebanding strips are split into thick and thin strip. The thick and thin strips have different colour in the project. This helps you to control the edging visually. The border of thin and thick edgebanding strip is determined by **Service >> Setting >> Parts and Sheets**, area "edging", setting "thin and thick border". If the strip thickness is less than the setting, this is a thin strip.

To determine a database of the edgebanding strips, use **Database >> Materials >> Edging database**.

4.5.6.1. Edging thick

<u>Access methods</u>
Menu: **Operations >> Part operations >> Edging >> Edging thick**
Toolbar: **Operation**
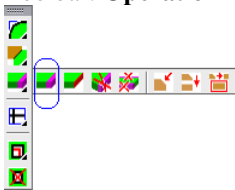
<u>Summary</u>
This command creates thick edgebanding strip along specified Parts edges.

<u>List of prompts</u>
select #D / selection #E / edge #H / external #Y / List of edges for edging <PVC_2>: Specify the Part edges to have edging them.
perimeter #G / edge #H / external #Y / List of edges for edging <PVC_2>: Specify the next edge

perimeter #G / edge #H / external #Y / List of edges for edging
<PVC_2>: Press **Enter** to complete operation

Note. The specified Part edge will be highlighted, in preview view, see fig. *4.40*, page 189. You can specify edges of different Parts in one operation.

Note. If you specify an edge which is included in a smooth connected sequence of edges it will be automatically taken the sequence of edges.

Select
Sets specified section of the edgebanding strip from "Edging database" as current section. The current section code is displayed in the command line, for example "PVC_2".

Selection
Sets the edgebanding strip section by specifying edge with a strip in the 3D-model as current section. The current section code is displayed in the command line, for example "PVC_2".

Edge
Selects only specified edge even if it is included in a smooth connected sequence of edges to glue an edgebanding strip.

Perimeter
Selects all edges on Part perimeter to glue an edgebanding strip.

External
Turns on mode of an external edging. (It is available if an internal edging mode has been already activated)

Internal
Turns on mode of an internal edging. (It is available if an external edging mode has been already activated)

Note. You can set edging mode by default. To do this, use **Service >> Setting >> Parts and Sheets**, area "edging", setting "**External**". If this checkbox is empty, the internal edging mode is activated.
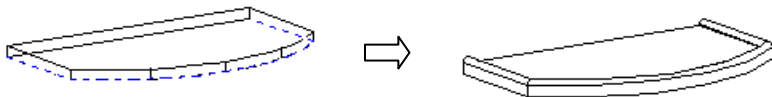
Fig. 4.40.   Part edging

4.5.6.2. Edging thin

Summary
This command creates thin edgebanding strip along specified Parts edges. It is correspond to above mentioned command but when you select a strip from "Edging database", you will see only those strips which will have a thickness equal or less than setting "thin and thick border" (to have more details see section 4.5.6, page 187).

Note. There is an edgebanding strip with thickness equal to zero in the "Edging database". This strip doesn't change Part dimensions, it is not displayed in the 3D-model, but it is calculated in the quotation and is displayed in the Part drawings. To visually control it in the 3D-model, use **Service >> Edging highlighted** or **Edit >> Replace material**.

4.5.6.3. Internal-external

Summary
It changes mode of existing edgeing.

List of prompts
Execute command, select in the dialog window the mode of the operation
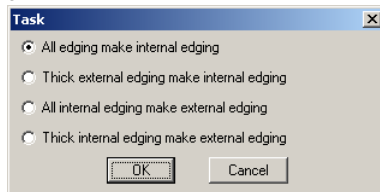
Select objects to replace edging <project>: Select a Part to edit it or Press **Enter** to edit all Parts in the project. (If you press **Enter**, the below displayed prompts will be skipped).
Select objects: Select next Part
Select objects: Press **Enter** to complete the operation

4.5.6.4. Remove list of edging

Menu: **Operations >> Part operations >> Edging >> Remove list of edging**
Toolbar: **Operation**
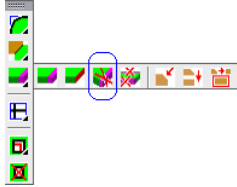
Summary
It deletes an edgebanding strip from the specified Part edges.

List of prompts
Edge to be deleted: Specify an edge with an edgebanding strip.
cancel #J / Edge to be deleted: Specify next edge
cancel #J / Edge to be deleted: Press **Enter** to complete the operation

Cancel.
Excludes the last specified edge from a set of the edited edges.

4.5.6.5. Remove all edging from a part

Access methods
Menu: **Operations >> Part operations >> Edging >> Remove all edging from a part**
Toolbar: **Operation**
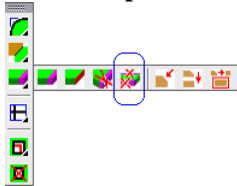
Summary
It deletes all edging from specified Parts.

List of prompts
Select objects to remove edging
Select objects: Specify a Part with an edging
Select objects: Specify next Part

Select objects: Press **Enter** to complete the operation

4.5.6.6. The way to edit and highlight edging

The command **Edit >> Properties** allow you to see and edit edging of the specified Part. You can edit the strip type and the edging mode (internal/external). To edit edging, select some edging string in the "Properties" window and press the "~" button.
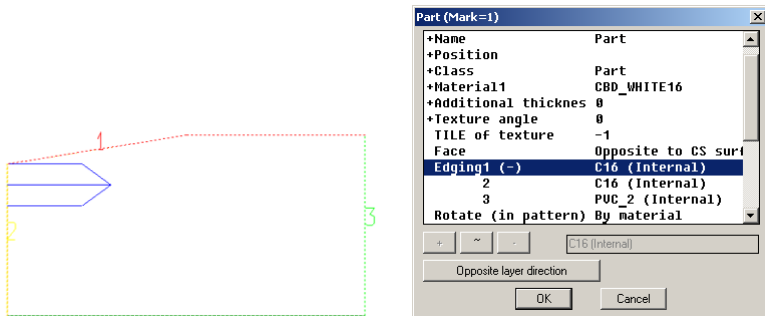
Fig. 4.41.   Edging editing in the "Properties" dialog window

In the Edging area (see fig. *4.41*) you see all Part edging. Edging numbers in the 3D-model correspond to edging numbers in the dialog window. The different edging in the 3D-model is marked by different colour. The edging in the dialog window is split into technological operation (there is three edging operation for example above) so you can see the same strip code in the "Properties" window (for example edging 1 and 2)

Note. If the "Edging 1" in the "Properties" window is marked by "(+)", it means that there are some edging operations for the Part. To open the edging list in the "Properties" window, execute left mouse double click.

To edit an edgebanding strip type for some Parts in the project, use **Edit >> Replace material**. You see all strip types used in the project in this dialog window (see fig. 4.42) and can replace it. To highlight the strip in the 3D-model, specify it in the dialog window (for example "PVC1") and press "Highlight" button.
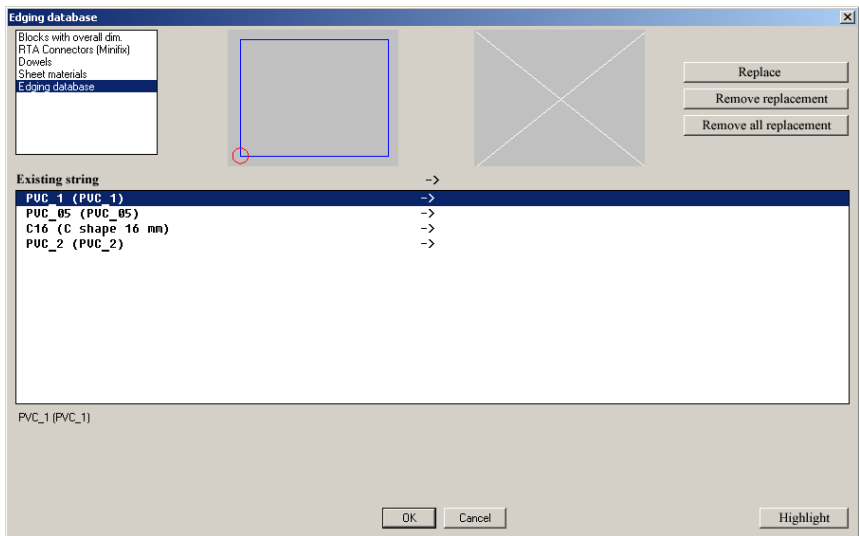
Fig. 4.42 "Replace material" dialog window

To execute a visual control of the edging Part, use **Service >> Edging highlight** (see fig. 4.43). Note, the thick and thin edging are highlighted with different colors. To set colour of the thick and thin edging, use **Service >> Setting >> Parts and Sheets**, area "edging", settings "**Colour of thin edging**" and "**Colour of thick edging**".
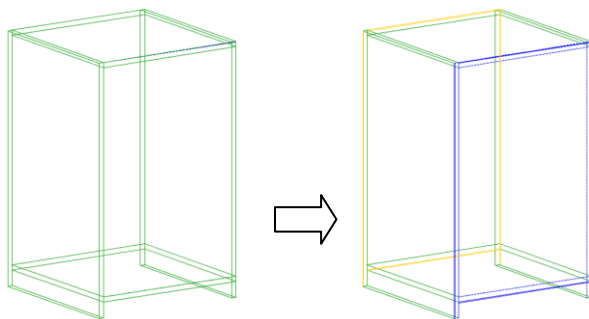
Fig. 4.43 Edging highlighting

More details about these commands see in appropriate sections.

4.5.7. Joints, cuts, grooves…

This operation group creates joint of two Parts. Their feature is an absence of auxiliary points on Parts after the joint execution. Besides this the edited Parts don't depend on the "Construction order" (they can have any "Construction order").

Important! The intersection of the jointed Parts is a necessary condition for the successful operation application. This applies to any operation from this group considered below. Sometimes the Parts can be automatically extended, but in total these are cutting operations.

Note. Don't use these operations as a basic tool to determine the Part sizes. These operations do your 3D-model more "heavy" (If you can create Parts with necessary dimensions (by 3 points, by divider and other) it is more preferable)

4.5.7.1. Bisecting joint

Access methods

Menu: **Operations >> Part   opera-
tions >> Joints cuts groves… >> Bisecting joint**

Toolbar: **Operation**
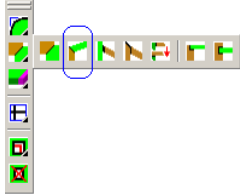
Summary
It creates a bisecting joint of the two Parts or two Profiles

List of prompts
Specify the cut edge of the first object (part or profile): Specify the first Part. Note, the specified point determines the Part end which is cut.
Specify the cut edge of the second object: Specify the second Part. Note, the specified point determines the Part end which is cut.
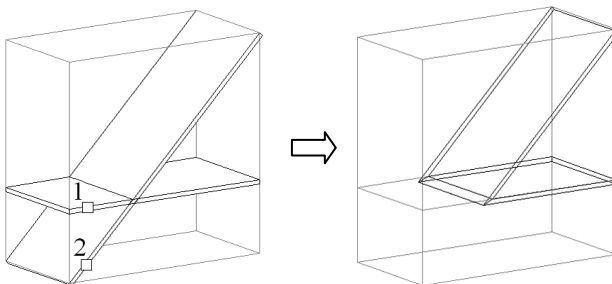
*Fig. 4.44. Bisecting joint of two Parts.
(1 and 2 are specified points)*
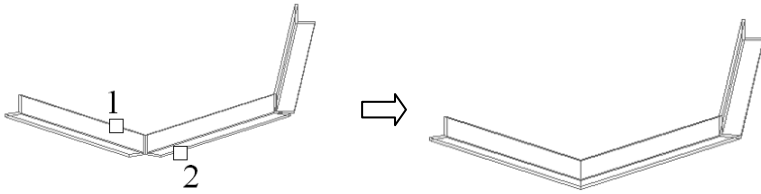
Fig. 4.45. Bisecting joint of two Profiles.
(1 and 2 are specified points)

4.5.7.2. Angular joint

<u>Access methods</u>
Menu: **Operations >> Part   operations >> Joints cuts groves… >> Angular joint**
Toolbar: **Operation**

<u>List of prompts</u>
First part (part to be joined to a wall): Specify the first Part
Second part (the wall): Specify the second Part
Note. The points of specifying don't matter. This operation always cut the shorter ends of the Parts.
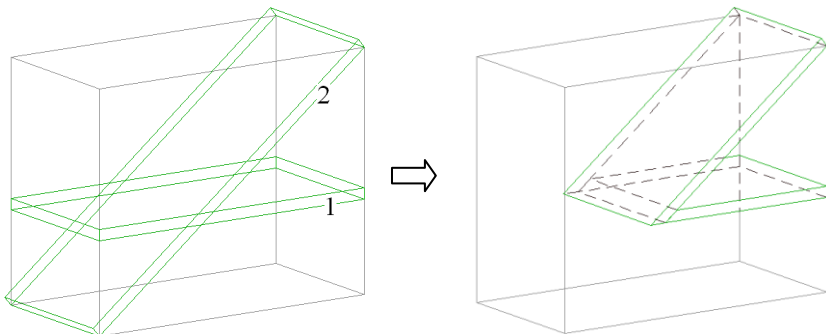
Fig. 4.46.   Angular corner joint example
(1-the first specified Part, 2-the second Part (a wall)).

4.5.7.3. Reverse angular joint

Menu: **Operations >> Part operations >> Joints cuts groves… >> Reverse angular joint**
Toolbar: **Operation**

Summary
It reverses angular joint, see fig. *4.47*.
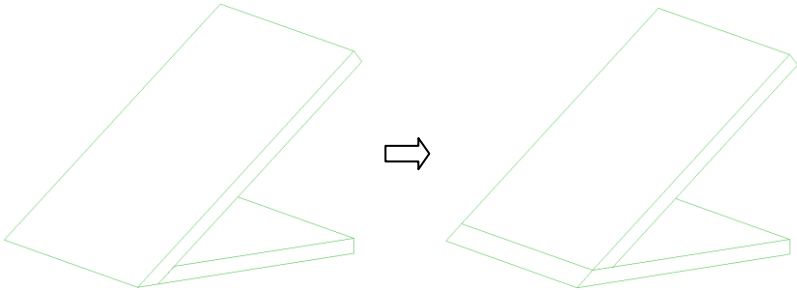
Fig. 4.47 Angular joint reverse

4.5.7.4. Joint with a wall

Menu: **Operations >> Part operations >> Joints cuts groves… >> Joint with a wall**
Toolbar: **Operation**
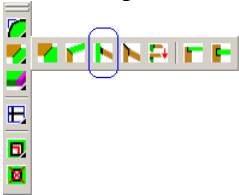
Summary
It cuts an editable object by surface of a base object. You can cut:
- a Part by Part or by Profile or by Bent Part;
- a Profile by Part or by Profile or by Bent Part;
- a Bent Part by Part or by Profile or by Bent Part.

Important! There are some conditions to complete the operation successfully:

- If you cut a Part by Part, the Part surfaces have to be crossed
- If you cut a Profile by Part or a Part by Profile, the Profile XY-plan and the Part XZ-plane have to be parallel
- If you cut a Profile by Profile or Profile by Bent Part, the XY-plane of two objects have to be parallel.
- If you cut a Bent Part by Profile or by Bent Part, the XY-plane of two objects have to be parallel.

Note. If the editable object is situated in the "Construction order" bottom the base object (bottom the wall), you will be able to use the auxiliary object points created with operation to create other objects.

List of prompts

Specify point on the remaining part of the Part (for Bent Sheet and Profile – jointed end): Specify point on the editing object.

Cutting Part (wall): Specify cutting object.
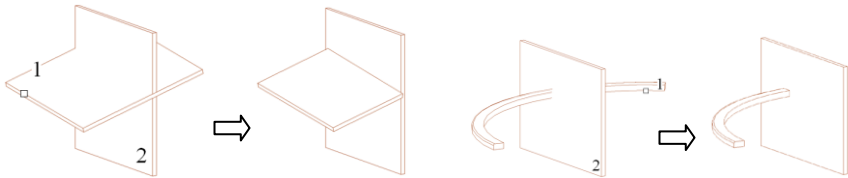
Fig. 4.48. "Joint with a wall" operation

4.5.7.5. Joint in a surface of parts

Access methods

Menu: **Operations >> Part   operations >> Joints cuts groves… >> Joint in a surface of parts**

Toolbar: **Operation**

List of prompts

Specify point on the remaining part of the first Part: Specify the first Part

Specify point on the remaining part of the second Part: Specify the second Part.

Note. The specified points determine the Part end which is desired.

Note. To have success in executing the operation, the Parts have to be parallel and they have to be intersected, see fig. 4.49.
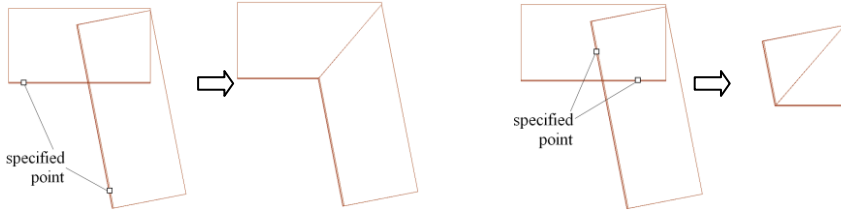
Fig. 4.49 "Joint in a surface of parts" operation

4.5.7.6. Cut by profile

Access methods

Menu: **Operations >> Part   operations >> Joints cuts groves… >> Cut by profile**

Summary

It cuts a Part or a Profile by Profile path projection. (The cutting path is projected onto editable object)

Important! The "offset to cut" is automatically set in the joint parameters. In other word you can have a gap between the path projection and desired piece of the Part, more details see in section 3.2.2.3, page 54.

List of prompts

Specify point on the remaining part of the Part (for Profile - jointed end): Specify an editable object

Cutting profile: Specify the cutting profile.
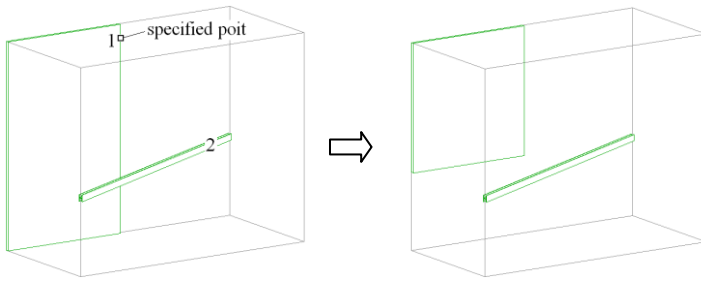
Fig. 4.50 "Cut by profile" operation

4.5.7.7. Groove (quarter)

<u>Access methods</u>
Menu: **Operations >> Part operations >> Joints cuts groves… >>
Groove (quater).**
Toolbar: **Operation**
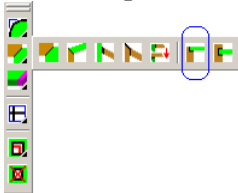
<u>Summary</u>
It creates a through quarter groove in one Part according to another Part.
To have success in the operation, two Parts have to be intersected and their surfaces have to be orthogonal, see fig. 4.51.

<u>List of prompts</u>
Select part to make cut in: Specify an editable Part
Select part to be cut in: Specify the Back Part
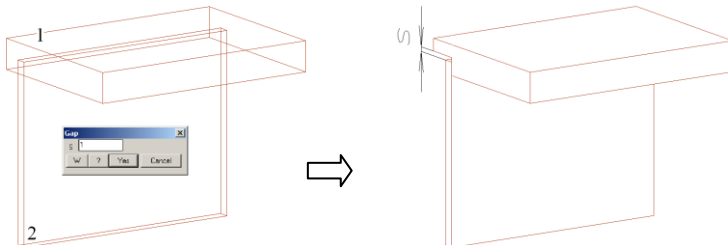
*Fig. 4.51. "Groove (quarter)" operation
(1– editable Part, 2– Back)*

4.5.7.8. Groove

Menu: **Operations >> Part   operations >> Joints cuts groves… >> Groove**
Toolbar: **Operation**
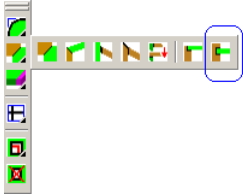
Summary
It creates a through quarter groove in one Part according to another Part. To have success in the operation, two Parts have to be intersected and their surfaces have to be orthogonal, see fig. 4.51.

List of prompts
blind (not through) #Y / Select part to make cut in: Specify an editable Part to make a through groove or press Enter to make a blind groove.
Select part to be cut in: Specify the Back Part
Enter operation parameters in the dialog window, see fig. *4.52*

Top view
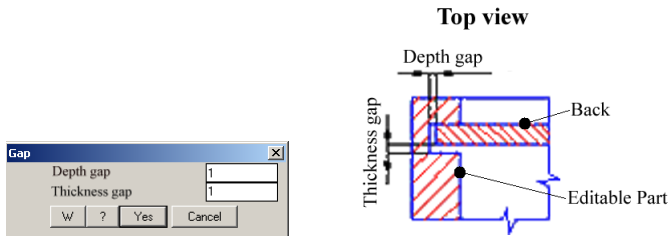
Fig. 4.52.   Groove parameters
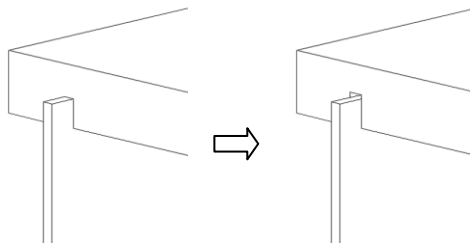
Fig. 4.53.   "Groove" operation

4.5.8. Change bench top width

Summary
It reduces a bench top width concerning the width defined in the database. (The bench top has a standard section)

List of prompts
Bench top to be edited: Specify a Bench Top.

4.6. Profile operations

4.6.1. Change orientation

Summary
It orients the Profile cross section concerning Profile path.

List of prompts
Select profile: Specify an editable Profile
mirror #P / Quadrant (any key) / ENTER - apply: Press any key to change quadrant of the Profile cross section concerning Profile path, see fig. *4.54*.

Mirror
Transforms the Profile cross section by mirroring it across the section bisector of angle, see fig. 4.55.

Note. When you activate command options, you will see a special sign of the Profile cross section in the preview view to understand the operation result.
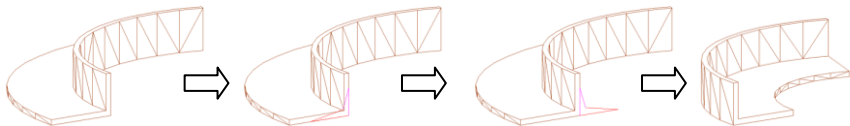
Fig. 4.54   Changing a quadrant of the Profile cross section
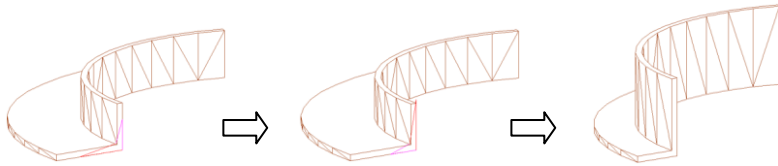
Fig. 4.55   Mirroring Profile cross section

4.6.2. Change length of profile

<u>Access methods</u>
Menu: **Operations >> Profile operations >> Change length of profile**

<u>Summary</u>
It increases or decreases a Profile length along the Profile path. You can edit the Profile length in the start path point or in the end path point.

<u>List of prompts</u>
Select profile: Specify editable Profile.
Enter in the dialog window the value of the start or end increment. If you enter negative value, the Profile length will be shorted.
Note. The start path point is marked with square pick box.
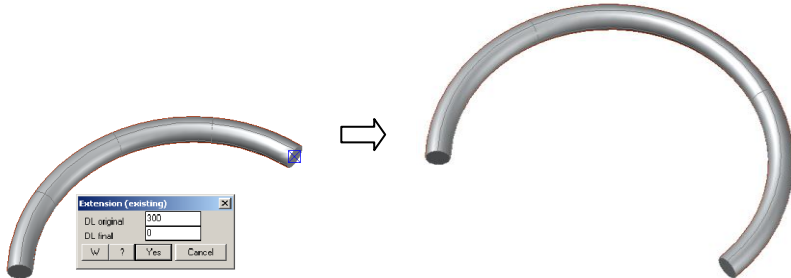
Fig. 4.56   "Change length of profile" operation

4.6.3. Bisecting joint

<u>Access methods</u>
Menu: **Operations >> Profile operations >> Bisecting joint**
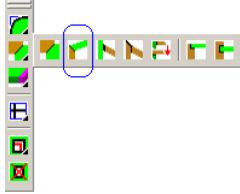
Toolbar: **Operation**

The operation is considered in section 4.5.7.1, page 194.

4.6.4. Joint with a wall

Access methods
Menu: **Operations >> Profile operations >> Joint with a wall**
Toolbar: **Operation**
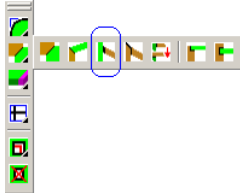
The operation is considered in section 4.5.7.4, page 196.

4.6.5. Cut by profile

Access methods
Menu: **Operations >> Profile operations >> Cut by profile**
The operation is considered in section 4.5.7.6, page 198.

4.6.6. Auxiliary points

Access methods
Menu: **Operations >> Profile operation >> Points >> Points XY**
Menu: **Operations >> Profile operation >> Points >> Points AL**
Toolbar: **Operation**
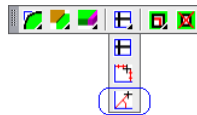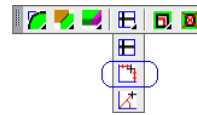
Point XY	**Point AL**

These operations correspond to them considered in section 4.5.2.1, page 173 and section 4.5.2.2, page 174.

4.6.7. Subtract part contour

Menu: **Operations >> Profile operations >> Subtract part contour**
Toolbar: **Operation**
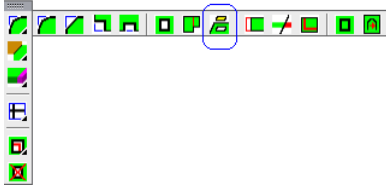
Summary
It creates a circular opening in the Part round a profile, which passes through the Part. The opening radius is a parameter of the operation.
Note. To have operation completed successfully, the Profile path has to be one straight segmen and it has to be perpendicular to the Part surface.

List of prompts
Part to be edited: Specify an editable Part.
Object to be subtracted: Specify a Profile, which passes through the Part.
Enter the opening radius in the dialog window, see fig. 4.57.
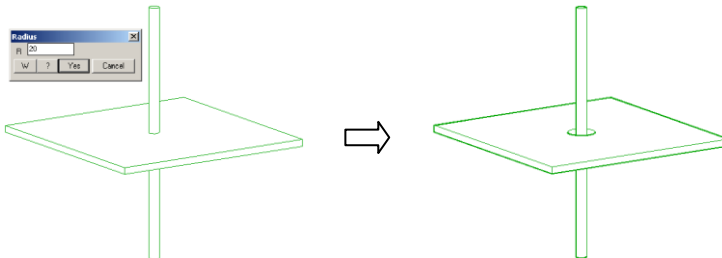
Fig. 4.57 "Subtract part contour" operation

4.7. Bent Sheet operations

4.7.1. Change direction for Bent Sheet thickness

Access methods
Menu: **Operations >> Bent sheet operations >> Change direction for bent sheet thickness**

Summary
It changes direction for Bent Sheet thickness concerning Bent Sheet path

List of prompts
Select bent sheet: Specify an editable Bent Sheet
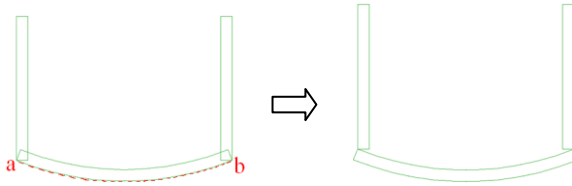
*Fig. 4.58   "Change direction for bent sheet thickness" operation
("ab" is a Bent Sheet path)*

4.7.2. Change length

Access methods
Menu: **Operations >> Bent sheet operations >> Change length**

This operation corresponds to it considered in section 4.6.2, page 202.

4.7.3. Cut by surface

Access methods
Menu: **Operations >> Bent sheet operations >> Change length**
Toolbar: **Operation**
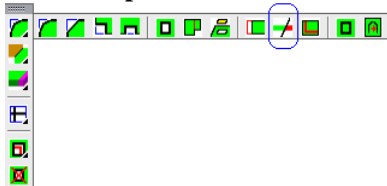
Summary
    It cuts an editable object by surface. This operation is equal to operation
considered in section 4.5.1.9, page 168.

4.7.4. Joint with a wall

Access methods
Menu: **Operations >> Bent sheet operations >> Joint with a wall**

Toolbar: **Operation**

The operation is considered in section 4.5.7.4, page 196.

The Bent Sheet is cut with Part

The Part is cut with Bent Sheet
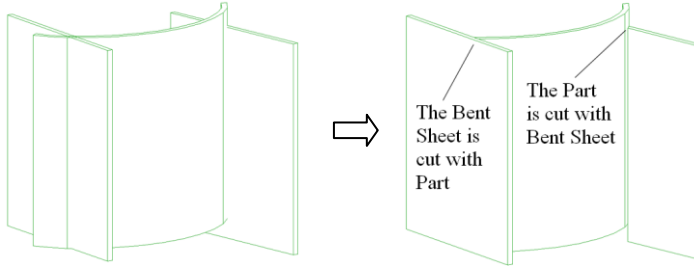
Fig. 4.59   "Joint with a wall" operation

4.7.5. Auxiliary points

Access methods
Menu: **Operations >> Bent sheet operations >> Points >> Points XY**
Menu: **Operations >> Bent sheet operations >> Points >> Points AL**
Toolbar: **Operation**
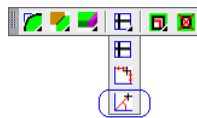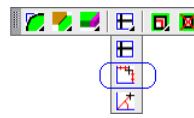
Point XY   **Point AL**

These operations correspond to them considered in section 4.5.2.1, page 173 and section 4.5.2.2, page 174.

4.7.6. Edging

The operation group from menu **Operations >> Bent sheet operations >> Edging** is correspond to the operation group considered in section 4.5.6, page 187.

4.7.7. Edge processing

The operation group from menu **Operations >> Bent sheet operations >> Edge processing** is correspond to the operation group considered in section 4.5.5, page 184.

4.8. Body of revolution operations

4.8.1. Change orientation of a section

Access methods
Menu: **Operations >> Body of Revolution operations >> Change orientation of a section**

List of prompts
Select body of revolution: Specify an editable object.
rotation angle #E / reverse X #X / reverse Y #Y / *apply #G: Select an option to edit the object, for example "rotation angle". Enter the angle value into the dialog window, see fig. 4.60.

Rotation angle
Determines the angle between the revolution body axis and cross section X-axis, see fig. 4.60.

Reverse X
Reverses the cross section X-axis (mirrors X-axis across cross section Y-axis).
Reverse Y
Reverses the cross section Y-axis (mirrors Y-axis across cross section X-axis).

Fig. 4.60   Entering cross section rotation angle for the body of revolution

4.8.2. Auxiliary points

Point XY	Point AL

These operations correspond to them considered in section 4.5.2.1, page 173 and section 4.5.2.2, page 174.

4.9. Block and furniture fitting operation

4.9.1. Options for size settings

This operation is considered in section 3.5.1.8, page 76.

4.9.2. Editing block scale

Summary
This operation edits scale factors of the specified block. The scaled block is saved only in the current project (it isn't saved in the block database). This operation edits only nonstandard blocks (they may have classes "No class" or "Part"; the class is an object properties).

Note. If you set the block class equal to "No class" or "Part" for the block with the class "standard product" the block reference to the database will be destroyed.

List of prompts
Select block to be edited: Specify an editable block, enter new scale factors into the dialog window, see fig. 4.61
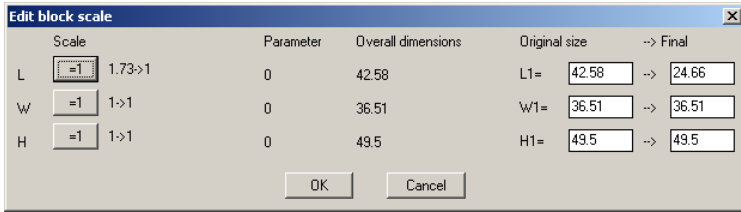
Fig. 4.61 Редактирование масштаба блока

Original and Final size

When you open the dialog window to edit the block scale factors, the original and final sizes will be equal and will display the current block sizes in the 3D-model. In next step you will be able to set scale factors equal to "1" by pressing "=1" buttons or set another value by fitting the quotient of the final and original size. In other words, the new scale factor is obtained by dividing a final size by an original size.

Note. To update the scale factor after original or final size editing, set focus in another field.

Overall dimensions
Displays the current block sizes. The value is update by pressing "Ok".

Parameters
Displays the block parameters

Scale
Displays the current scale factor.

4.10. Fastener and Furniture fittings operations

4.10.1. Change fitting side

Access methods
Menu: **Operations >> Fastener and Furniture fittings operations >> Change fitting side**

The operation is considered in section 3.8.14, page 147.

4.10.2. Options for size settings

Access methods
Menu: **Operations >> Fastener and Furniture fittings operations >> Options for size settings**

The operation is considered in section 3.8.12, page 145.

4.10.3. Revers tenon

Summary
This operation changes a tenon base surface. Note, it works for "tenon and mortise" joint created with command **Object >> Fastener and Furniture fittings >> Mortise and tenon (dialog)**.

List of prompts
Select tenon: Specify the Part tenon
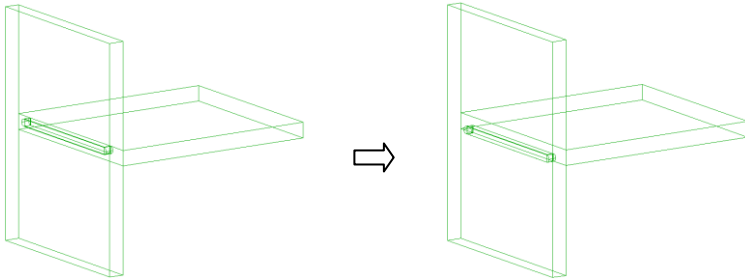
Fig. 4.62   "Revers tenon" operation

4.11. Assembly wire frame operations

4.11.1. Sub surface

Summary
It creates a sub-surface in the Assembly frame. The auxiliary edges and points of the sub-surface may be used to create new objects.
To edit a sub-surface position, use:
Edit >> Stretch
Operations >> Parameters
To delete a sub-surface, use **Operations >> Delete**.

Specify assembly at the dividing point: Specify the Assembly.
*Dialogue / Specify the position of the plane: Determine sub-surface position by mouse or press **Enter** to define sub-surface position into dialog window.
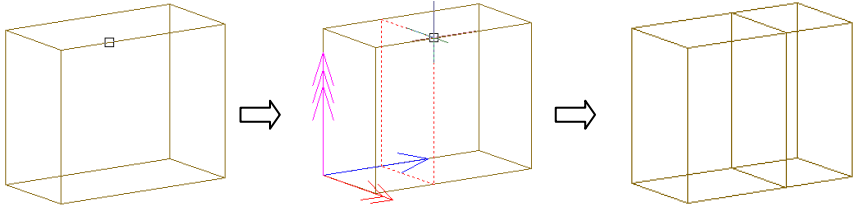
Fig. 4.63. "Sub surface" operation

4.11.2. Auxiliary points

Access methods
Menu: **Operations >> Assembly wire frame operations >> Points XY**
         Operations >> Assembly wire frame operations >> Points AL
         Operations >> Assembly wire frame operations >> External points projection
         Operations >> Assembly wire frame operations >> Intersection points with external edges
         Operations >> Assembly wire frame operations >> Intersection points with external surface

Toolbar: **Operation**

Point XY	**Point AL**
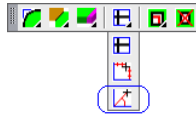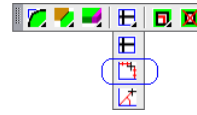
These operations correspond to operations considered in section 4.5.2, page 173.

4.12. Constructional drawing operations

These operations are considered in chapter "Constructional drawing".

Chapter 5. Editing of objects

5.1. Undo

<u>Access methods</u>
Menu: **Edit >> Undo**
Toolbar: **Edit**

<u>Summary</u>
It cancels the last executed command effect.

5.2. Redo

<u>Access methods</u>
Menu: **Edit >> Redo**

<u>Summary</u>
It reverses the last executed "Undo" (see section 5.1) effect.

5.3. Objects redraw

When you change something in the 3D-model (for example object variables, an object reference, an object operation and other) some objects bound up with the edited objects will not redrawn automatically. To redraw some objects selectively, there is a special group of commands.

Redraw all
<u>Access methods</u>
Menu: **Edit >> Redraw >> All**
Toolbar: **Edit**
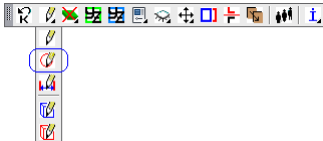
<u>Summary</u>
It redraws all objects in the projects.

Redraw selected object

Menu: **Edit >> Redraw >> Selected object**
Toolbar: **Edit**
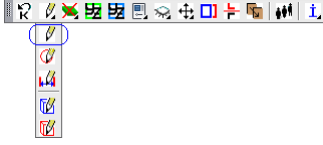
Summary

It redraws only selected objects. To select objects, you can specify them by the mouse left-click or use multiple selection. More details about object selection see in section 1.13, page 19).

Note. If you specify the Assembly frame to redraw it by this command, only Assembly frame will be redrawn (the Assembly objects will not be redrawn).

Redraw Informational Size

Menu: **Edit >> Redraw >> All sizes**
Toolbar: **Edit**
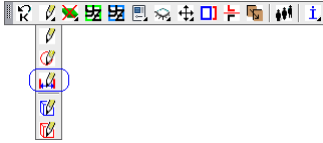
Summary

It redraws all Informational Sizes in the project.

Note. The Informational Sizes are created with program so that the Size text is parallel to the screen plan. If you change the view point, it is comfortable to use this command to redraw all Informational Size.

Redraw selected Assembly

Menu: **Edit >> Redraw >> Assembly**
Toolbar: **Edit**
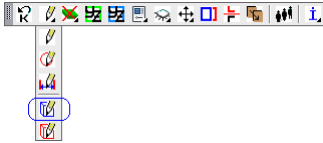
Summary

It redraws the specified Assembly and all object included in the Assembly.

*current assembly / select assembly: Press **Enter** to redraw the current assembly or specify an assembly by mouse left-click. Note, to specify an Assembly, you may specify any object included in the Assembly or Assembly frame.

Redraw selected Product
Access methods
Menu: **Edit >> Redraw >> Product**
Toolbar: **Edit**
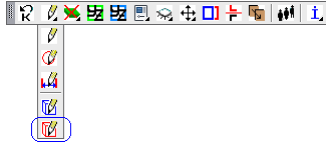
Summary
It redraws the specified product and all object included in the product. (The product is an Assembly located at the top level of the project).

List of prompts
*current assembly / select assembly: Press **Enter** to redraw the current assembly or specify an assembly by mouse left-click. Note, to specify a product, you may specify any object included in the product Assembly or its frame.

Redraw duplicates of Assemblies
Access methods
Menu: **Edit >> Redraw >> All assemblies duplicates**

Summary
It redraws all Assemblies' duplicates in the project.
Note. If you change origin Assembly (for example add a new Parts), you will have to redraw duplicated Assembly to see changes in the 3D-model. Note, to create a duplicated Assembly, you should execute **Assembly >> Duplicated copy of Assembly**.

5.4. Objects deleting

Summary
Attention! When you delete any objects, the program will control its links. If there are some links from other objects to the deleting objects, the deleting operation will be disabled, you will get the message:
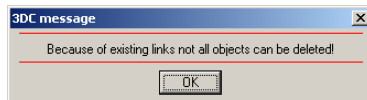
Note. You can delete a linked object, but you should have disabled the "Control" setting (**Service >> Settings >> General Settings** "Control" checkbox). To delete a linked object, you should:

- disable the "Control" checkbox;
- delete objects;
- redraw project (**Edit >> Redraw >> All**);
- enable the "Control" checkbox

Note, you destroy the 3D-model by above mentioned action (the variables and references will be replaced by current numerical values).

Access methods
Menu: **Edit >> Delete >> Objects**
Toolbar: **Edit**
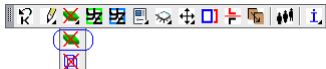
List of prompts
Select objects to be deleted.
Select objects: Specify objects. You can select objects individually (by mouse left-click) or select multiply objects (by specifying a rectangular selection area).
Select objects: Specify objects.
Select objects: Press **Enter** to complete command.

Access methods
Menu: **Edit >> Delete >> Assembly**
Toolbar: **Edit**
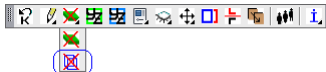
List of prompts
*product #B / *list #C / Select assembly*: Specify an Assembly by mouse left-click. Note, to specify an Assembly, you may specify any object included in the Assembly or its frame.

Product
Specifies an Assembly-product (an Assembly at the top level of project) by specifying its frame or any object included in the Assembly-product.

List
Selects an Assembly from the "List of included elements", see fig. 5.1.
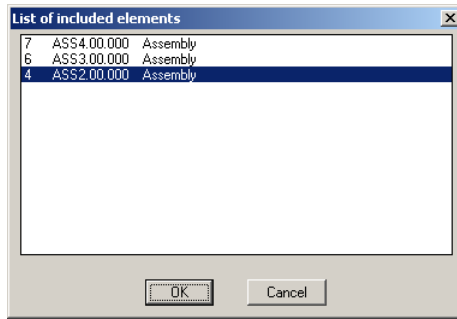
Fig. 5.1 List of included elements

The specified object is included in the ASS2.00.000 Assembly, ASS2.00.000 is included in the ASS3.00.000 Assembly, ASS3.00.000 is included in the ASS4.00.000 Assembly-product (ASS2.00.000, ASS3.00.000, ASS4.00.000 are the codes of the Assembly).

5.5. Editing variables of object

Summary

There are three commands to edit variables of object. These commands edit object variables through the same dialog window, see fig. *6.22*, page 283. See section 6.2.2, page 283 to get the detail information about working into the window.

Access methods (to edit variables of an object)
Menu: **Edit >> Variables**
Toolbar: **Edit**

List of prompts
*product #B / *list #C / Select object*: Specify an Assembly by mouse left-click. Note, to specify an Assembly, you may specify any object included in the Assembly or its frame.

Product
Specifies an Assembly-product by specifying its frame or any object included in the Assembly-product.

List
Selects an Assembly from the "List of included elements", see fig. *5.1*, page 216.

<u>Access method</u> (to edit variables-parameters of an object).

Note. The variable-parameter is a variable which is defined by numerical value (not by formula).

Menu: **Edit >> Variables-Parameters**

<u>List of prompts</u>

The list of prompts corresponds to the considered above list of prompts.

<u>Access methods</u> (to edit variables of Assembly-product).

Menu: **Edit >> Variables**

Toolbar: **Edit**

<u>List of prompts</u>

*object #J / *list #C / Select product*: Specify any object of Assembly-product to determine the product.

Object

Edits variables of the selected object.

List

Edits variables of an Assembly. The Assembly is selected from the "List of included elements", see fig. *5.1*, page 216.

5.6. Properties

These are non-graphical features which have any object of program.

There are next object properties:

- Mark;
- Code;
- Name;
- Position;
- Class;
- Material;
- Section;
- Additional thickness;
- Texture parameters;
- Face;
- Edging
- Cutting parameters;
- Composition;
- Price;
- Oversize;

- Drawing;
- Note;
- Description;
- Special.

The properties of an existing object can be edited by **Edit >> Properties** (this command can be also executed from the "project-tree" dialog window). Some object properties can have values by default, which are determined with object settings (**Service >> Setting**…)

The different types of objects have different properties therefore the "Properties" dialog window will look different for these objects. The example of the "Properties" window is presented in fig. 5.2.
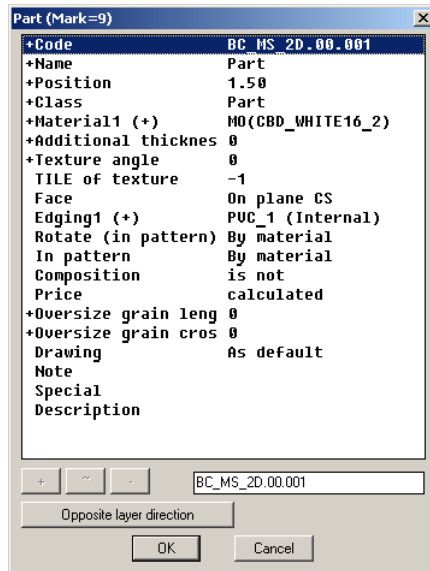
Fig. 5.2. "Properties" dialog window

Important! When you generate quotation documents, generate Part drawings, export Cutlist Data, the program calculates the quantity of identical objects. Two objects (two Parts, two Profiles, two Bent Sheets and other) are identical if they have identical sizes and properties marked by "+" sign in the "Properties" dialog window (code, name, position, class and other).

5.6.1. Mark

The mark is an unique identifier of objects in the project. It is generated by program and can't be edited by user. It is used into the informational and search operations.

5.6.2. Code

There is one code system in the program. The code format has next structure:

<Product>.<Assembly>.<Object>[/<piece of smth.>][suffix]

<**Product**>

It is the first code part, which is determined with product code. The product is an Assembly-product or an object-product (this is an object at the top of project structure). This is a string which can include any latter.

Note. The codes of Assemble-products look as:

<Prefix1>.00.000;

<Prefix2>.00.000;

<Prefix3>.00.000;

…

The code of an object-product looks as:

<Prefix>.00.001;

<Prefix>.00.002;

<Prefix>.00.003;

…

The <Prefix> value by default is determined by **Service >> Settings >> General settings**\ field "**Base code (Prefix of assembly code)**".

<**Assembly**>

It is a numerical Assembly number (which includes the object) in the product. For example ASS.01.000, ASS.02.000…ASS.99.000. If the object are included in an Assembly-product, it will have "*.00.*", for example ASS.00.000.

Note. This code format presupposes that any Assembly can include up to 99 other Assemblies.

Note. If an Assembly has a "Set" class, their objects will take codes according to the higher situated Assembly (according to the project structure).

<**Object**>

It is a numerical object number in the Assembly. For example, the first three Parts in the Assembly which number is five in the product will look as: ASS.05.001, ASS.05.002, and ASS.05.003

Note. This code format presupposes that any Assembly can include up to 999 objects.

Note. If an Assembly has a "Set" class, their objects will take codes according to the higher situated Assembly (according to the project structure). For example, the first three Parts in the Assembly which number is five in the product and which has a "Set" class will look as: ASS.00.001, ASS.00.002, and ASS.00.003

<Piece of smth.>

It can shows in quotation documents:
- a layer of a Bent Sheet
- a segment of a Profile which path has vertexes
- a Part which is attributed in the properties

For example, if two Parts are attributed to a Part which has a code ASS.05.003, their codes in quotation documents will look as ASS.05.003/1, ASS.05.003/2.

<Suffix>

It is a string which may be added by user. For example ASS.05.001TEST.

Principles of codes generation:

1. The function of codes generation assigns new object codes according to above considered code format. And it reassigns codes of existing objects by user require. To reassign codes, you should edit an Assembly code (command **Edit >> Properties**) and enable a checkbox "Reassign codes" in the "Properties" dialog window.

Attention! The Assembly objects will be reassigned if:
- the Assembly code and objects codes are created according above mentioned code format
- the code prefix of objects included in the Assembly corresponds to the code prefix of the Assembly before reassigning.

(If some objects code doesn't correspond to the Assembly code it will not reassign. For example, if the Assembly code is "ASS.00.00" and the Part code is "AS.00.000", the Part code will be not reassigned)

2. You can block the code reassigning for Assembly objects. To do this, enter prefix "*" in the object code. For example, if the Assembly code is "ASS.00.00" and the Part code is "*ASS.00.002", the Part code will not be reassigned.

Note. The "*" is a service sign, it will not be shown in the Part drawing and in the quotation documents.

3. You can block the code reassigning for an Assembly and its objects. To do this, enter prefix "#" in the Assembly code. For example, if the Assembly code is "ASS.00.00", enter "#ASS.00.00"

Note. The "#" is a service sign; it will not be shown in the Assembly drawing and in the quotation documents.

4. You can use "size replacement strings" into an object code. These strings will be replaced by current sizes in the object drawings and quotation documents. The "size replacement string" have to be marked with < > signs in the code. For example, "ASS<L>.00.00",

you will see "ASS600.00.000", where "L" is an Assembly length. There are next "size replacement strings":

L, W, **H** – length, width and height (thickness) of object.

LS, **WS**, **HS** – length, width and height of an Assembly which include the object.

LI, **WI**, **HI** – length, width and height of an Assembly-producte which include the object.

5. It is possible to use "replacement strings" in the object code. The "replacement string" have to be marked with < > signs in the code. The values of the "replacement strings" are determined by user in the "General settings" dialog window (command **Service >> Settings >> General settings**).

   Note. The "replacement strings" will be replaced in the drawings and quotation documents

6. When you insert the Doors, Drawers and Typical models in the project, their codes will able to be reassigned. To reassign models codes, enable checkboxes "After door insert generate new code", "After drawer insert generate new code", "After typical model insert generate new code (except products)" (command **Service >> Settings >> Assembly**). If above mentioned checkboxes are disabled, the models will take codes from the database.

   Attention!

7. The codes of standard objects and standard models and their composition are not reassigned.

5.6.3. Name

This is a string which describes the object meaning, for example "shelf", "side", "top"

You can type object name or select it from a name bank in the "Properties" dialog window.

Note. To fill in the name bank, execute Service >> Setting \ select type of object (for example "Parts and Sheets")\ press "Name" button, see fig. 5.3. The different objects have different name banks.

Note. You can use the "replacement strings" into the name, more details see in section 5.6.2.
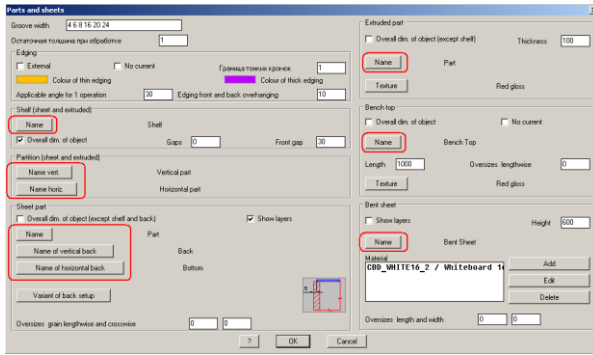
Fig. 5.3   Press "Name" button to edit a name bank

5.6.4. Position

This is a numerical object property. It may be displayed on the object drawing and quotation documents.

To generate the positions automatically, activate the checkbox "Regenerate positions after command execution". In this case before generating drawing, quotation documents and CNC-programs, they will be generated.

To generate the positions manually in any time, execute **Edit >> Assign POSITIONS.**

Note. Recognize the object "position" on the view of assembly drawing and the property of an object "position". The position on the view of assembly drawing can include: object code, object mark, object dimensions and object position. The property "position" is a numerical string.

The following agreements are taken to generate positions of objects:
- The assembly-product position is a number according to the continuous numbering of products in the project. The product numbering is starting with number 1 (1, 2, 3, 4....).
- The position of objects created in the program (Part, Bent Sheet, Bench Top…) consists of two parts. These are number of a product and an object number into the product, for example 1.50, 1.55, 3.52. The object numbering is starting with number which is assigned with the setting "Starting value of objects position in assembly" (**Service >> Setting >> General settings**)
- The standard object positions (fastener, leg, handle…) are assigned according to continuous numbering. The starting number is assigned with the setting "Starting value of standard objects position". This is 200 by default. (**Service >> Setting >> General settings**)

5.6.5. Class

This property defines the quotation of object and the possibilities of its editing. There are next classes in the program:

- assembly,
- part,
- standard product,
- set,
- no class,
- material.

The "**assembly**" class is assigned to Assemblies. Any Assembly may include other Assemblies and Parts; it combines them logically in one furniture product. More details see in chapter "Assembly".

The "**set**" class is a type of the assembly class. If an Assembly has the "set" class, it will not be given out in the quotation. Its objects will be included in the quotation into the upper Assembly according to project structure.

Note. The "**set**" class may be determined for an item of the database (of the hardware, fitting and fastener database).

The "**part**" class is assigned to Parts which will be created according to project.

The "**standard product**" class is assigned to standard object which are used for furniture creating, for example fasteners, handles, kitchen appliances and other. You can't edit their properties and size.

The "**no class**" class may be assigned to any object. These objects are not given in the quotation documents, in the drawings, in the cutlist but they influence drawings of other objects and visual representation of the project (the "no class" fastener gives processing on connected objects)

The "**material**" class is intended for non-graphical materials which may be included into objects composition, for example glue, paint, vinyl film and other.

5.6.6. Material

This property is available only for Sheet Parts and Bent Parts.

If the created Part is included in an Assembly its material can be assigned by:

- one of the sheet materials form database
- a current logical material of the Assembly (more details see in section 6.2.4, page. 289).

If a new Part is not included in an Assembly its material is automatically assigned by one of the sheet materials.

To assign the current logical material of a new Assembly, use **Service >> Setting >> Assembly**, area "**Logical material**"

To assign the current logical material of the specified Assembly, use **Edit >> Variable**, specify an Assembly, press "**Mt**" button in the dialog window.

Note. There is a special command to replace material for some selected objects, more details see in section 5.20, page 255.

5.6.7. Section

This property is available for molded parts (Profile, Bench Top). It defines cross-section of the object. You can replace object cross-section to any other of a database.

5.6.8. Additional thickness

It increases the thickness of Part, for example, to take in account the glue thickness of multilayer Part.

Note. The additional thickness is always added to the amount of thickness of layers of a Part. It proportionally increases the thickness of every Part layer in a 3D-model. You will not see the gap between layers.

5.6.9. Texture

It is available for fittings, hardware, blocks, Extruded Parts, Bench Tops. You can define an object texture in the database or in the project.

a)   b)
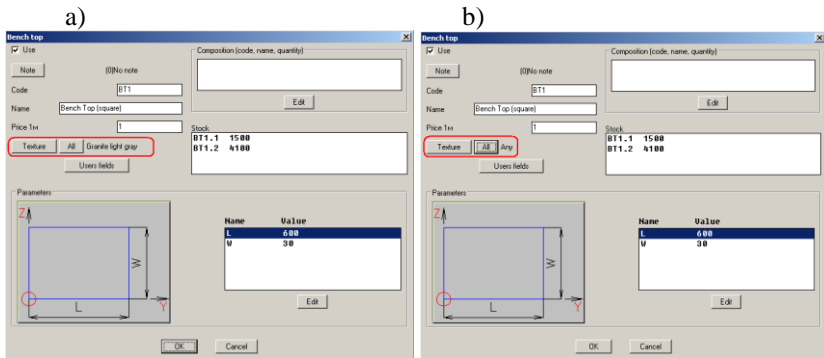
Fig. 5.4   a)   Texture is defined in the database

b)   Texture will defined in the project

Note. To define the object texture by default in the project, use Service >> Setting, select type of object and press "texture" button in the dialog window, see example bellow for Bench Top and Extruded Part.
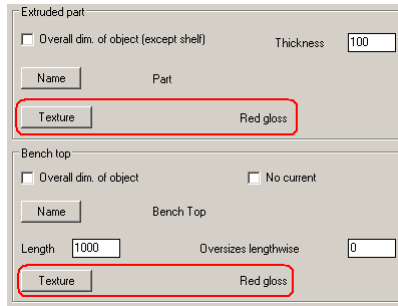
Fig. 5.5   Fragment of dialog window "Parts and sheets" settings

5.6.9.1. Texture angle

It determines the angle of texture rotation. Zero corresponds to the positive direction of the Part X axis. 90 degree corresponds to the direction of the Part Y axis.

You can see the Part texture angle in the 3D-model (by face side designation, see section 5.6.10, page 225) and in the Part drawings.

5.6.9.2. Tile of texture

Determines the scale of the mosaic pattern, which is composed of a bitmap texture. There are three modes to use the tile of texture:

- tile=0. The bitmap is automatically stretched to the object. The scale for X axis and Y axis to stretch the bitmap is calculated separately.
- tile= 1000 (any positive value in millimeters). The bitmap is stretched to assigned value and then is tiled to cover the all object.
- tile= -1 (any negative value in millimeters) The bitmap is tiled to cover the all object. (The bitmap size is defined in the texture directory).

Note. The tile mode by default is determined with setting "Default TILE value" (**Service >> Setting >> General settings**).

5.6.10. Face

It is available only for Parts, defines the face side of the Part.

The face side designation (see fig. *5.7*) displays the face side and texture angle. You may switch off/on it into a 3D-model, determine its dimension and set its colour (**Service >> Settings >> General settings**, area "Face side designation", see fig. *5.6*.

Note. If you switch off the face side designation into a 3D-model, it will appear temporarily on the Part inside the "Properties" command (**Edit >> Properties**)

Note. If the face side designation is switched on into a 3D-model, you may temporarily switch off it with **Edit >> Visibility >> On/Off face block**.

Fig. 5.6   Fragment of "General setting" dialog window
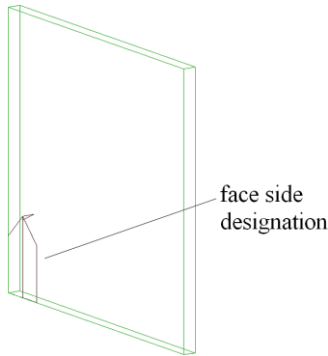
Puc 5.7   Face side designation in a 3D-model

Note. When generating Part drawings, the Part drawing may be ordered face side up or face side down.

5.6.11. Edging

It is available only for Parts, more details see in section 4.5.6.6, page 192.

5.6.12. Cutting properties

The cutting properties are "Rotate (in pattern)" and "In pattern". They are available only for Sheet Parts.

In pattern

Determines whether the Part is offloaded in the cutlist with command **Export >> Export for optimization software**. (The cutlist is prepared for optimization software).

If it has a "Not re-define" value, the Part will not be offloaded in the cutlist.

If it has a "By material" value, whether the Part will be offloaded is determined with a sheet material property "Don't offload to optimization pro-

gram". If the checkbox "Don't offload to optimization program" (see fig. *5.8*, page 227) is filled in in the sheet materials database, the Part will be not offloaded. If it's cleared, the Part will be offloaded.

Rotate (in pattern)

Determines whether the Part is allowed to be rotated by optimization software on the pattern.

If it has an "Allow" value, the Part may be rotated on the pattern.

If it has a "By material" value, whether the Part may be rotated is determined with a sheet material property "Grain". If the checkbox "Crain" (see fig. *5.8*) is filled in, the Part rotation is prohibited. If it's clear, the Part may be rotated.

Fig. 5.8   Sheet material properties in the sheet material database

5.6.13. Composition

Important! There are a composition assigned to an object in the project, and a composition assigned to an item in the database.

5.6.13.1. Composition in the database

If the composition is assigned to an item of the material database, its quantity is assigned:

- for 1 square meter (for sheet material)
- for 1 meter (for edging, profile, bench top)
- for 1 piece (for fasteners, fittings, hardware)

To calculate total quantity of assigned material in the project, the assigned material quantity will be multiplied on material quantity on which it is assigned to in the quotation module.

Note. If you assign a piece element to an item of the material database (for example to profile material), its calculated quantity will be rounded to the nearest integer in the quotation module.

Note. You can't control the composition assigned to the database items, which have class "material", "standard product" and "set", in the project. You will see it only in the quotation module.

Important! If you assign a composition to a database item of, which has class "part" (see fig. 5.9), and insert it in the project, all data (and composition also) from database will be copied in the project and the object will lose database connectivity after insertion. (If you then change database, the changes will be not taken into account in the project).
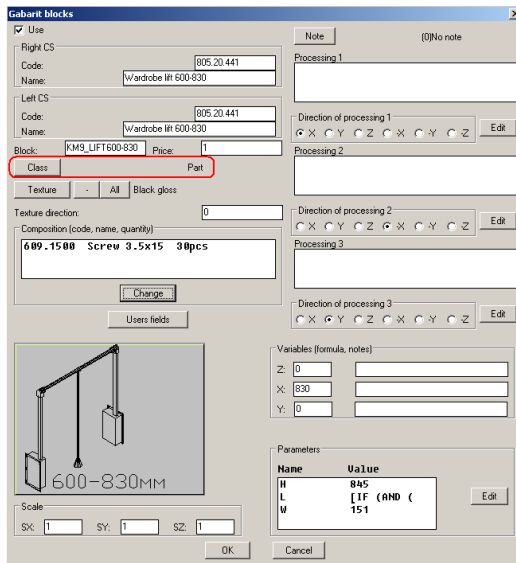
Fig. 5.9   Example of database item with class part

5.6.13.2. Object composition in the project

This is no graphical object, which is offloaded to quotation documents. To assign a composition to an object, use "Properties" dialog window, the "Composition" string.

The composition quantity may depend on properties of object (for example on the Part perimeter) or be set as a number. If it depends on object properties, you may use special variable to assign quantity, see fig. 5.10, string 2 or reference to typical calculation, see string 3. The full list of special variable see in section 6.2.6, page 296.

Note. If you define calculation for material, the field quantity will define material quantity for one calculation unit, for example 0.25kg for 1 square meter, see fig. 5.10, string 3.

Fig. 5.10   Object composition dialog window

Add

Assigns a new string of the composition. The composition may be taken from database shown in fig. *5.11 page 230*.

Edit

Edits the specified string

Delete

Deletes the specified string

Buttons "<", ">"

Define strings order in the dialog window.

Buttons "+"

Defines one of the typical calculations for selected string.

Buttons "-"

Deletes a typical calculations for specified string.

Buttons "?"

Shows the typical calculation and its parameters.

Note. The "Record" column displays "+" if the material is in the database (everything is ok). It will displays "?" symbol if material is deleted in the database. In other words the data in the project outdate, you should correct them.
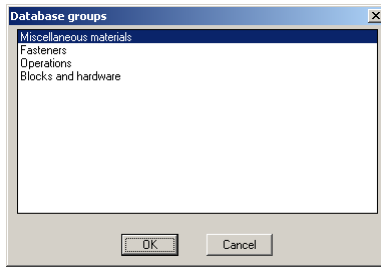
Fig 5.11   Databases to assign the object composition

Note. If you assign paint to a Part, the Part will be painted, when you activate photorealistic mode, according to its colour. If you assign some different paints to a Part, the Part will be painted according to the last paint in the composition list.

Note. If you assign Paint or Materials (activate "Miscellaneous materials") to an object, you may specify material quantity through the "Material usage directory". Press "Usage catalogue" (see fig. *5.12*) to open it.
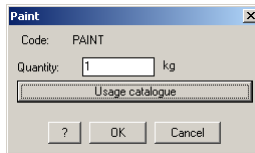
Fig. 5.12   Assigning the paint into the object composition

Note. You can assign an additional composition to an Assembly frame too.

5.6.14. Price

The price of object may be calculated or attributed. In the "Properties" dialog window you see only attributed price. The calculated price of object is shown only in the quotation documents.

Note. If you assign attributed price to an object it suppress the calculated price in the quotation document.

The price of objects is calculated according next principle:
- The price of standard product is always attributed; you set it in the database
- The price of Sheet Part is calculated as multiply of the 1 sq. m. pattern price and Part square. The Part square is a rectangular square circumscribed around the Part one edge of which coincides with its texture grain (direction).
- The price of moulded part (profile, edging, bench top) is calculated as multiply of the 1 m. pattern price and Part length.

5.6.15. Oversize grain

The oversize grain is a technological oversize to produce the object. It differs for different type of objects.

There are an oversize along and an oversize cross texture for Sheet Part. Note they are given to cutting dimensions of the Sheet Part, more details see in section 3.1.2.2, page 33.

The Bent Sheet oversize is given along length and height of the Bent Sheet. The Bent Sheet oversizes are added to it dimensions and then the cutting dimensions of Bent Sheet layers are calculated.

The Profile and Bench Top oversize is always given along object length.

Important! You can assign an oversize as the property of selected object (in the "Properties" window) or as the property of the object type (in the object setting by default, for example by **Service >> Setting >> Profile**, setting "Oversize for length")

The oversize in the "Properties" dialog window suppresses the oversize in the object settings. In other words, if the oversize (two oversizes for Parts) in the "Properties" window is equal to zero, it (they) will be got from object settings.

Note. You can control object dimensions and its oversize by **Service >> Object information**, see fig. 5.13.
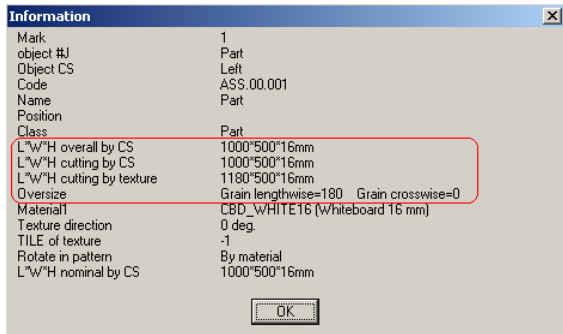
Fig. 5.13   Object dimensions and its oversize

5.6.16. Drawing

This property may have two states "As default" and "Without drawing".

Without drawing
If this state is enabled, the drawing for Part is not generated.

As default

If this state is enabled, the existence or absence of the Part drawing is defined by settings of the "Drawing set". To control "Drawing set", use **Service >> Drawing settings >> Set of drawings**.

Note. If you execute **Document >> Parts drawing >> Parts views** and specify Part, the Part drawing will be generated irrespective of "Drawing" property. This property influences object selectivity only for "Drawing set".

5.6.17. Note

This is a string which length is 120 characters.

5.6.18. Description

This is a text field with unlimited length. The text is typed with Notepad editor. To return to 3D-Constructor after text editing, close Notepad.

Important! This field is saved only in the current project (in the *.dwg file). It doesn't copy, if you copy the object.

5.6.19. Special

This field is used for interfacing 3D-Constructor and external software. It is not used usually.

5.6.20. Off (for fastener)

See section 3.8.15, стр. 148.

5.7. Match properties

Access methods
Menu: **Edit >> Match properties**
Toolbar: **Edit**
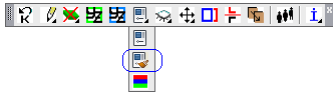
Summary
Apply the properties of a selected object to other objects.

Note. The type of the source object and type of the destination objects have to be equal. If you specify a different type object as a destination object the command will be canceled.

List of prompts
*product #B / *list #C / properties database #< / Select object example*: Specify the object which properties you want to copy, enable checkbox in the dialog window (see fig. *5.14*) to specify the matched properties.

product #B / list #C / Select destination object: Specify the object to which you want to copy the properties of the object-example.

product #B / list #C / Select destination object: Specify next destination object or press Enter to complete the command.

Product

Selects product as a destination object by specifying any object of product.

List

Selects a destination object from object list. The object list is created by specifying object.
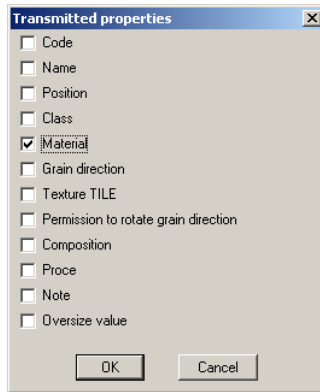
Fig. 5.14   Dialog window to specify object properties to copy

5.8. Match properties (group of object)

Access methods
Menu: **Edit >> Match properties (group of object)**

Summary

It corresponds to command considered above, but it allows you to select a group of destination objects.

List of prompts

Select object example: Specify the object which properties you want to copy, enable checkbox in the dialog window (see fig. *5.14*) to specify the matched properties.

*by project #G / product #B / list #C / Select destination object: Specify the object to which you want to copy the properties of the object-example.

By project

Selects all objects in the project which type is equal to the object-example as destination objects.

Product

Selects all objects included in the product which type is equal to the object-example as destination objects.

List

Selects a destination object from object list. The object list is created by specifying object.

5.9. Colour

Access methods
Menu: **Edit >> Colour**
Toolbar: **Edit**

Summary
Sets colour setting for specified objects.

Note. The object colour by default is set by **Service >> Settings >> General settings**, the area "**Objects colours**". To restore object colour by default set the colour "ByLayer" or "ByBlock".

List of prompts
Select objects to change colour

Select objects: Select the objects to which you want to change colour. Note, you can use "Window selection" or "Crossing selection", more details see in section 1.13.5, page 21.

5.10. Controls the visibility of objects

5.10.1. Visibility Off

<u>Access methods</u>
Menu: **Edit >> Visibility >> Off**
Toolbar: **Edit**

<u>Summary</u>
Hides selected objects.

<u>List of prompts</u>
Select objects to hide <ENTER - dialogue>

Select objects: Select the objects which have to be hidden or press **Enter** to open dialog window, see fig. 5.15, page. 235. Note, to select objects, you can use "Window selection" or "Crossing selection", more details see in section 1.13.5, page 21.
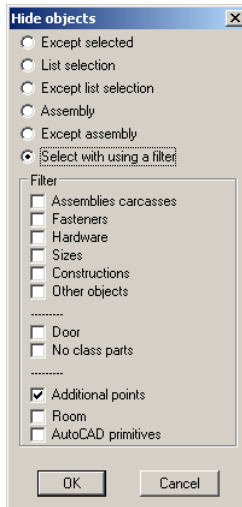
Fig. 5.15 "Hide object" dialog window

5.10.2. Visibility On

<u>Access methods</u>
Menu: **Edit >> Visibility >> On**
Toolbar: **Edit**

<u>Summary</u>
Displays objects previously hidden.

<u>List of prompts</u>
all #D / filter #A / *last set #G: Press Enter to display last hidden objects.

All
Displays all hidden objects in the project

Filter
Displays hidden objects in compliance with a filter settings, see fig. *5.16*.
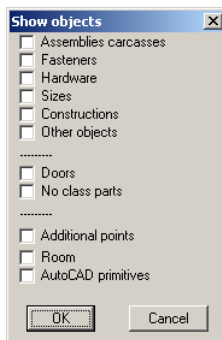
Fig. 5.16   "Show objects" dialog window

5.10.3. Using the "project tree" to control object visibility

You can use "Project tree" to control object visibility. To activate project tree, execute **Service >> Project tree**

To hide any object through the "Project tree", you should select it in the "Project tree" and press "Hide" button ⊠ in the bottom of the tree. Instead of pressing "Hide" button you can execute right mouse click on the object in the "Project tree" and select "Hide" from the pull-down menu.

Note. If you select Assembly in the "Project tree" which is closed in the "Project tree" (see fig. *5.17*, Assembly with mark 36) and press "Hide" button, the Assembly frame and all objects included in it will be hidden. If you select

Assembly in the "Project tree" which is opened in the "Project tree" (see fig. *5.17*, Assembly with mark 19) and press "Hide" button, only the Assembly frame will be hidden.

To display object selected in the "Project tree", utilize "Show" button ![icon] in the bottom of the "Project tree" or "Show" item from the pull-down menu.

To display only Assembly frame, select an opened tree node and press "Show" button. To display the Assembly frame and all included objects, select a closed tree node and press "Show" butten.
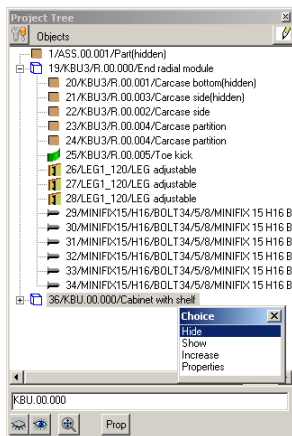
Fig. 5.17   Project tree dialog window

5.10.4. Control visibility of the "Face side"

See section 5.6.10, page 225.

5.11. Command to control objects position

The commands from this group only place objects (not reference them). If you want to fix achieved result in 3D-model, you should use a reference after object placement.

5.11.1. Move

Access methods
Menu: **Edit >> Move**
Toolbar: **Edit**
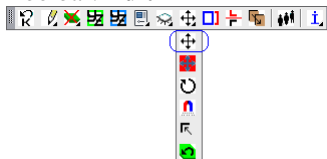
Summary
Moves selected objects.
Note. If you specify an object which has referenced to another object or to which other objects are referenced, all referenced object will be selected to move automatically.

List of prompts
Select objects to be moved
Select objects: Select object to move. Note, to select objects, you can use "Window selection" or "Crossing selection", more details see in section 1.13.5, page 21.
Select objects: Press **Enter** to complete selection.
New point: Specify a new point on the screen. Instead of a new point specifying it is possible to set moving direction by mouse (the "Ortho mode" should be activated by pressing F8 before) enter the moving distance in the command line and press **Enter**.

Note. It is possible to activate "Base point" enquiring for this command. To enable it, use **Service >> Settings >> General settings**, checkbox "**On copy and move do not ask point 1**".

5.11.2. Move object

Access methods
Menu: **Edit >> Move object**
Toolbar: **Edit**
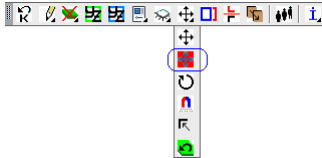
Summary
Moves specified object.
Note. If a moved object is referenced, its reference will be edited by this command. If there are objects referenced to a moved object, their positions will be updated after project redrawing.

List of prompts
*product #B / *list #C / Select object to be moved*: Select object.
New point: Specify new point on the screen. Instead of a new point specifying it is possible to set moving direction by mouse (the "Ortho mode" should be activated by pressing F8 before) enter the moving distance in the command line and press **Enter**.

Product
Selects all objects included in the product to be moved.

List

Selects an object from object list to be moved. The object list is created concerning specified object.

5.11.3. Rotate

<u>Access methods</u>
Menu: **Edit >> Rotate object**
Toolbar: **Edit**
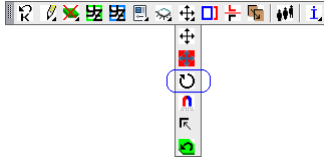
<u>Summary</u>

Rotates specified objects around a base point. The axis of rotation passes through the specified base point and is parallel to the *Z* axis of the current UCS.

Note. If you specify an object which has referenced to another object or to which other objects are referenced, all referenced object will be selected to move automatically.

<u>List of prompts</u>
Select objects to be rotated

Select objects: Select object to move. Note, to select objects, you can use "Window selection" or "Crossing selection", more details see in section 1.13.5, page 21.

Select objects: Press **Enter** to complete selection.

Base point (rotation axis): Specify a point on the screen.

Rotation angle: Determine rotation angle. You can set angle by mouse or enter a value in the command line.

Note. To rotate an object with step 90 degree, it is comfortable to activate "Ortho mode" by pressing F8.

Note. To determine base point it is possible to use Object Snaps (standard AutoCAD feature).

5.11.4. Magnet

<u>Access methods</u>
Menu: **Edit >> Rotate object**
Toolbar: **Edit**
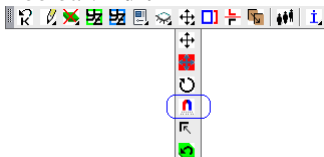
Summary

Coincides two nearest points of editable object and base object. The area of magnate is determined with **Service >>Settings >> General settings**, setting "**Magnet searching area**".

If after objects points coinciding there is some angle between any two axes of objects coordinate system and it is less than "Additional rotation degree", the editable object will be automatically aligned. The align operation coincides two objects axes which have a minimal misalignment of axes.

To set "Additional rotation degree", use **Service >> Settings >> General settings**, setting "**Additional rotation degree**".

Note. This command doesn't work with additional object points, which were created with **Operations >> Part operations >> Point of contour**

Note. This command is assigned to align products. Therefore if you select object included in the Assembly to be magneted, the Assembly will be selected automatically to be magneted.

List of prompts

Object to be moved: Select an editable object.

Gap (if faces are found) <0>: Enter a gap value between objects in the command line (if it is necessary) move the editable object to the base object and execute mouse left-click, see fig. (If the objects will not be snapped, you should increase "**Magnet searching area**".)

Note. The gap current value is displayed in the command line. To set a new value, enter it in the command line and press **Enter**.
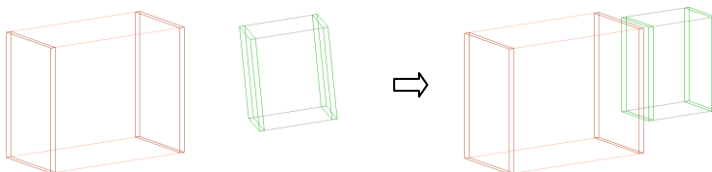
Fig. 5.18   The small assembly was magneted to the biggest assembly with gap 20 mm.

Attention! You can rotate the editable object on 90 degree around Z-axis of WCS by pressing **Space** button, while the object is moving.

5.11.5. In a corner (for assemblies)

Menu: **Edit >> In a corner (for assemblies)**
Toolbar: **Edit**
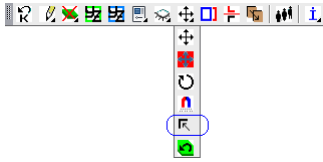
Summary

Places a specified Assembly in the corner of the wall (you can specify a corner or an end of the wall to place object). The Assembly back is aligned along specified edge of the wall.

List of prompts

Select object to be setted up: Specify object, see. fig. 5.19, specifying "1". If you specify a Part included in Assembly, the Assembly will be selected automatically.

Select a wall: Specify the wall edge to align Assembly X-axis along specified edge, see. fig. 5.19, specifying "2". Enter placement parameters in the dialog window.

From a wall

Defines the distance between the specified wall edge and the Assembly back.

From a corner

Defines the distance between the nearest wall corner (end) to the specified point and the Assembly face.

From a floor

Defines the distance between a floor and the bottom Assembly face. It's comfortable to use it to place kitchen wall unit.
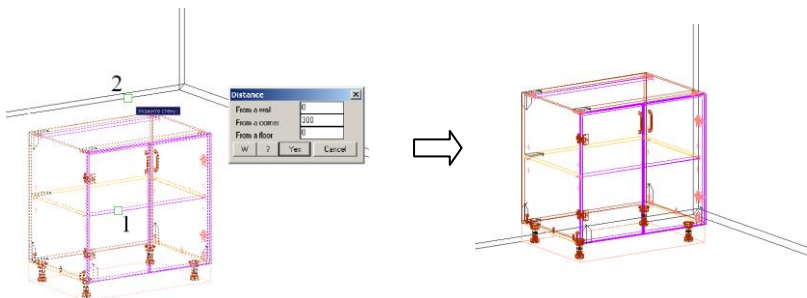
Fig. 5.19   The placement unit in the corner

5.11.6. Place

Summary

Align edges of the specified objects. When you specify the first and the second objects, the coordinate system (thereinafter CS) of operation will be created and displayed for both objects, see fig. 5.20. The specified point on the object determines the CS origin and edges to be aligned.

Note. As edge to be aligned may be specified any edge of XY-plan of object. If you specify the edge which not belong to XY-plan of object, the nearest suitable edge will be taken automatically. For example, if you specify one of vertical Assembly edge, it will be taken nearest horizontal edge.

List of prompts

*product #B / *list #C / Object to be setted up*: Specify an edge of the first object.

*product #B / list #C / *move #G / Base object*: Specify an edge of the second object. Enter in the dialog window offset parameters.

Note. You can enter "#" sign as X-offset parameter, to center the first edge concerning the second edge.

Product

Selects a product by specifying object included in the product.

List

Selects an object from the object list. The object list is created concerning specified object.
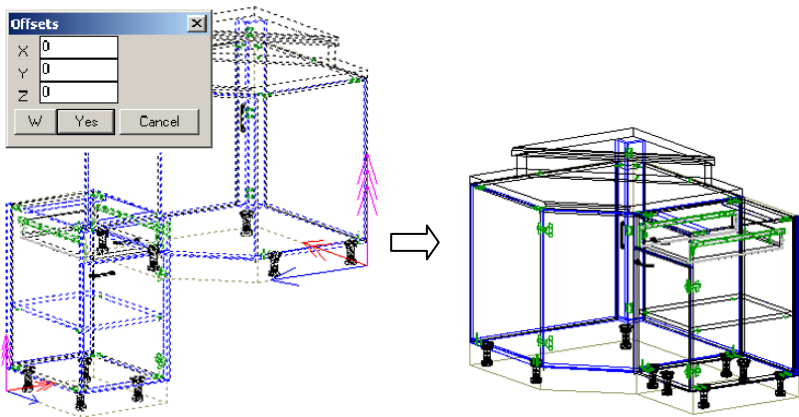
Fig. 5.20   The placement of one object concerning another object

5.11.7. Move, Rotate, Shift

Menu: **Edit >> Move, Rotate, Shift**
Toolbar: **Edit**
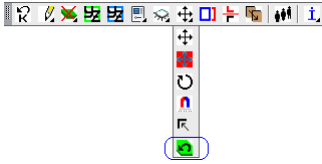
Summary
Places a specified object concerning another object or WCS (world co-ordinate system). It is comfortable to use this command before an object reference if it is difficult to imagine how to get a desired result with the reference command.

Note. If you select to object (object to be placed and base object) you will be able to rotate and align the first object. If you select only one object (object to be placed) you will be able to move, rotate and align specified object concerning WCS.

List of prompts
Object to be moved: Specify the first object.
Base object for aligning: Specify the second object or press **Enter** to place the first object concerning WCS.
• If you specify two objects, the next prompt will be:
Object axis
 *x_rotate / y / z / X_align / Y / Z / *Exit:* Activate an option to orient the object.
• If you specify only one object, the next prompt will be:
*x_rotate / y / z / X_align / Y / Z / *Exit / Point 1 of moving (or an object axis)*: Activate an option to orient the object.

Exit
Complete the command.

x_rotate / y / z
Determines the axis of the first specified object around which will be rotated the object.

X_align / Y / Z
Determines the axis of the first specified object which will be aligned with axis of the second object or WCS.

5.12. Axis reverse

<u>Access methods</u>
Menu: **Edit >> Axis revers >> X**
       Edit >> Axis revers >> Y
       Edit >> Axis revers >> Z
Toolbar: **Edit**

(Mirror axis Z)

<u>Summary</u>
Replaces a specified object with a mirrored object. To create the mirrored object, its axis (selected with a command) is reversed.

Attention! Don't use this command for an Assembly, which has included objects (this command mirrors only the Assembly frame). To mirror an Assembly frame and all included objects, use **Edit >> Mirror**. See fig. 5.21.
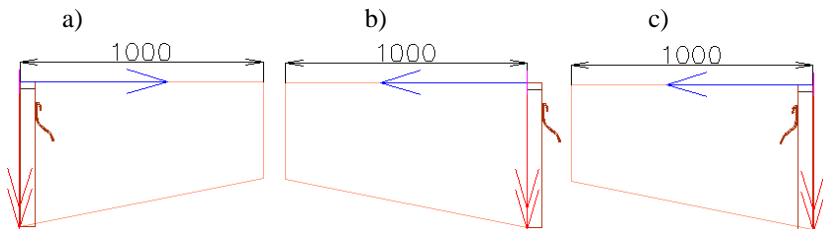
Fig. 5.21   a) Initial state of objects

               b) The frame of Assembly is edited with **Edit >> Axis revers >> X**

               c) The Assembly is edited with **Edit >> Mirror**

               The dimension (1000 mm) is set to the Assembly frame.

5.13. Mirror

<u>Access method</u>
Menu: **Edit >> Mirror**

<u>Summary</u>
Creates a mirrored copy of selected object, see fig. 5.21 a) and c) page 244.

Note. The mirroring plane always perpendicular to the XY plan of an editable object. It may be defined with two points, or object X-axis, or object Y-axis. This command works with one object at same time.

*product #B / *list #C / Select object*: Specify an object to be mirrored
*reverse X #X / reverse Y #Y / 1-st point of mirror line: Specify the first point of mirror line on the screen.

2-d point of mirror line: Specify the second point of mirror line on the screen

Delete original object?

*no #Y / yes #L: Select "Yes" to delete the initial object or "No" to keep it in the project

Product

Selects a product to be mirrored by specifying any object included in the product.

List

Selects an object to be mirrored from the object list. The object list is created concerning specified object.

Reverse X

Defines the mirror plane as the object YZ-plane. The mirror plan is coincide with the base point of object.

Reverse Y

Defines the mirror plan as the object YZ-plane. The mirror plan is coincide with the base point of object.

5.14. Copy

Access methods
Menu: **Edit >> Copy**
Toolbar: **Edit**

Summary

Creates a copy of selected object. The copy of object can inherit all properties including the code of initial object. To define whether the copy inherits the code, use **Service >> Setting >> General setting**, checkbox "**Make new code of copied objects**".

Attention! When you generate part drawings, quotation documents, two objects will be determined as equal if they will have identical overall dimensions and properties marked with sign "+" in the "Properties" dialog window. (The shape of objects and their machining doesn't matter).

List of prompts
*product #B / *list #C / Select object*: Specify an object to be copied.
New point: Specify a point on the screen to place a copy of object.

Note. It may be comfortable to activate "Ortho Mode" (press F8) when you place a copy of object.

Note. It is possible to activate "Base point" enquiring for this command. To enable it, use **Service >> Settings >> General settings**, checkbox "**On copy and move do not ask point 1**".

Note. The initial object and its copy will be referenced to the same base object if the initial object was referenced before copying.

5.15. Stretch

Access methods
Menu: **Edit >> Stretch**
Toolbar: **Edit**

Summary
Stretches specified object edge. You can stretch Part, Assembly frame, Informational size.

If you stretch an Assembly frame edge
If the Assembly variable is determined by a parameter, the value of parameter will be edited. If variable is determined with an expression, the expression will be increased (or decreased) according to stretch distance.

If you specify a vertical frame edge to stretch, the height of Assembly will be edited.

If you stretch a Part edge and Part hasn't got a box
(More details about Part box see in section 3.1.1, page 30).

If you stretch a Part edge one of its parameter will be edited. If parameter is determined by a value, the value will be edited. If parameter is determined by an expression the expression will be increased or decreased according to stretch distance.

If you stretch a Part edge and Part has got a box
The stretch distance will be entered in the box variable (SX1, SX2, SY1, SY2). Note, each of box variable correspond to some of Part edge.

Attention! If you stretch a referenced object, it will be kept with a reference. Therefore you can get unexpected result, see fig. 5.22.

List of prompts
Edge of object to stretch: Specify edge.
Define offset (value or point): There are two ways to determine a new edge position:
- move mouse and watch the new highlighted contour of object. Create left-mouse click on the screen to complete operation.
- determine the stretch direction by mouse on the screen, enter the stretch value in the command line and press **Enter**.
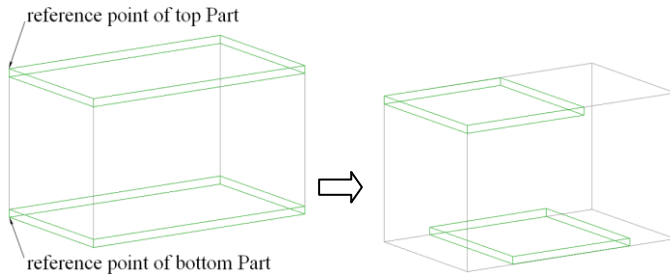
Fig. 5.22 Example of stretching objects.

The right and left edges of two Parts were short-ened to the same distance. The top Part hasn't a box; the bottom Part has a box.
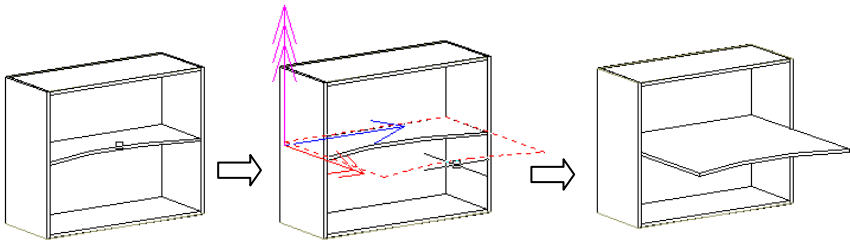
Fig. 5.23. Example of stretching object

Note. When you stretch edge, the cursor is moved with a step. To set step, execute **Service >> Setting >> General setting**, setting "**Drag step**".

5.16. Overall dimensions

Access method
Menu: **Edit >> Overall dimensions**

Summary
Determines a specified object size as a distance between points of exist-ing objects. It creates a distance-variable and enters it in the appropriate Part parameter. In any words this command parameterizes an object size.

Note. If you edit the object size which has been determined with a dis-tance-variable, this command will only redefine distance-variable value. (All expressions and parameters defined with this distance-variable will be recalcu-lated)

List of prompts
*product #B / *list #C / Select object*: Specify object to edit.

To change direction - any key / ENTER - apply: Specify object size to edit. The object size is highlighted with arrow. To select another object size, press any key. To accept size to be edited, press Enter.

*ODO (box) + / *menu #V / Distance between points - first point*: Define the first point to determine the object size.

*ODO (box) + / *change point #N / Second point*: Define the second point to determine the object size.

*change point #N / *apply selected #G*: Press **Enter** to complete the command.

Product

Selects a product to be edited by specifying any object included in the product.

List

Selects an object to be edited from the object list. The object list is created concerning specified object.

ODO (box) +

Determines what point will be taken. It may be object point or box point (detail about box see in section 3.1.1, page 30). To specify point of an object, must be displayed "*ODO (box) +*". To specify box point of an object, must be displayed "*ODO (box) -*". Note, if "*ODO (box) -*" is displayed and the specified object hasn't box, it will be taken the object point.

Menu

Activates a special menu, where you can set different types of distance-variable, more details see in section 6.2.5.3, page 295

Change point

Redefine specified point if it was specified incorrectly.

Note. More details about object variables see in section 6.2.5, стр. 291.

5.17. Special commands

5.17.1. Copy fitting in defined place

<u>Access method</u>
Menu: **Edit >> Special commands >> Copy fitting in defined place**

<u>Summary</u>
Copies and places a specified fitting in a new place. It transmits a reference to a copy of object from object that you copy. Note, a copy of object will be referenced to a new base object according to the reference which has been determined for a fitting that you copy.

For example, using this command you can place one leg on the object and quickly copy it in other places (on the same object or other objects). Note, each leg copy is referenced to own corner of the object, see fig. *5.24* b).

<u>List of prompts</u>

Select object to be copied: Specify a fitting

ODO (box) + / Define insert point <ENTER - execute>: Specify an object edge to place the fitting copy.

*ODO (box) + / reference parameters #G / reverse Z #H / *exit #D / Define insert point*: Specify next object edge to place the fitting copy or press Enter

ODO (box) +

Determines what point will be taken. It may be object point or box point (detail about box see in section 3.1.1, page 30). To specify point of an object, must be displayed "*ODO (box) +*". To specify box point of an object, must be displayed "*ODO (box) -*". Note, if "*ODO (box) -*" is displayed and the specified object hasn't box, it will be taken the object point.

Reference parameters

Edits reference parameters of the last copied fitting.

Reverse Z

Reverses Z-axis of the last copied fitting.

Exit

Completes the command.

Note. Depending on the reference type, which was used to place an initial fitting, this command may give the result required additional editing. To edit placement result in the copying command, use "Reference parameters" and "Reverse Z" options. If the initial object is referenced with "Point to direction", you will get correctly result at once.

a)   b)
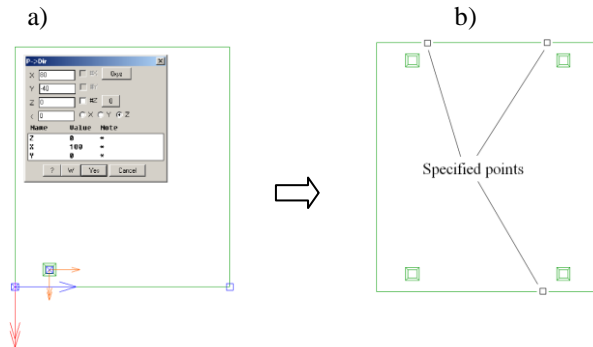
Fig. 5.24   Example of leg copying

(Top view on the cabinet)

Note. When you specify an edge to copy fitting, you should take into account the reference direction of initial fitting.

5.17.2. Adapt frame of Assembly by line

Summary

There are some frame types which can be adapted according to an AutoCAD polyline. These frames are named as "Adaptive contour *" in the "Database of assembly forms" (where sign "*" is any words), see fig. *5.25*.
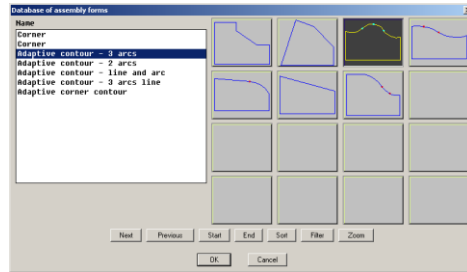
Fig. 5.25   Database of assembly forms

This tool helps you to create a set of furniture with splined doors which looks like single soft front line, see fig. *5.26*.
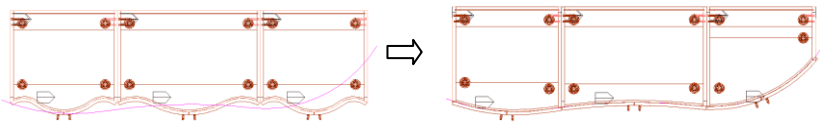
Fig. 5.26   Adapting units by polyline

Attention! Each adaptive frame has a unique front edge. Front edge can include one, two or three primitives (lines or arcs). To adapt a frame successfully, it has to be limited by piece of polyline which includes primitives quantity equal to or less than the unit edge.

Note. It is very difficult to create universal adaptive unit, therefore you can create some unites to close some typical situation.

List of prompts

product #B / list #C / Select assembly to be adapted: Select an Assembly frame (you may specify any object included in the Assembly).

product #B / list #C / Select assembly to be adapted: Select next frame.

product #B / list #C / Select assembly to be adapted: Press **Enter** to complete selection.

Select face line: Specify the polyline.

Product

Selects a product frame by specifying any object included in the product.

List

Selects an object to be edited from the object list. The object list is created concerning specified object.

Note. This command adapt only Assembly frames. To redraw all object included in the Assembly, use an appropriate command (for example **Edit >> Redraw >> Product**).

5.17.3. Replace of texture for a set of objects

Access method
Menu: **Edit >> Special commands >> Replace texture for a set of objects**

Summary

Replaces object texture in current project without editing properties of blocks and materials in the 3D-Constructor database.

It helps you quickly present some photorealistic variant of project to the customer.

List of prompts
Select objects to replace textures <project>

Select objects: Press Enter to select all object in the project or use "Window selection" or "Crossing selection" to select some objects (More details about object selection see in section 1.13.5, page 21).

*all objects #D / doors #A / excluding doors #R: This is an additional filter to objects which have been selected in a previous step. "All objects" selects all objects. "Door" selects only doors from previous selection. "Excluding doors" excludes doors from previous selection.

When objects to edit have been selected exactly, will be open special dialog window (fig. 5.27). Here you can select type of material (or type of block) and redefine texture of selected item in the bottom window part.

Note! The texture redefining is not equal to the material replacement. You will not see the operation result in the drawings or quotation documents.
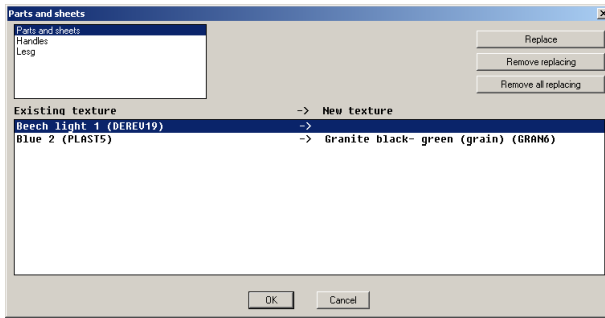
Fig. 5.27 Texture replacement

To restore materials textures according to database settings, use **Edit >> Special command >> Restore default texture**.

5.17.4. Replace of object texture

This command corresponds to it considered above, but it edits only one selected object (it can be Part or Assembly which includes some parts).

5.18. Copy and Paste with the Clipboard

Edit >> Copy object 3DC in clipboard
Copies some specified products to the Clipboard. The product is selected by specifying any object included in the product. It is possible to use "Window selection" or "Crossing selection" to specify objects, more details see in section 1.13.5, page 21.

Edit >> Paste object 3DC in clipboard
Pastes the Clipboard contents into a project.
If objects, you are going to paste from the Clipboard, have external links, you will get a notice "Pasted objects have external links" in insertion moment. The external links will be destroyed (variables will be replaced with their values; references will be replaced with reference to WCS). Therefore pasted 3D-model may not be redrawn correctly.

Note. AutoCAD primitives may be copied and pasted with Clipboard according to above considered way.

5.19. Replace of objects

Access method
Menu: **Edit >> Replace objects**

Summary

Replaces object(s) with another single-type object. (You can replace a handle with another handle, a door with another door).
Note. The fastener may be replaced with any fastener which has identical insertion method. For example it is possible to replace Confirmat with minifix, or dowel, or cam, or carcass connector.

List of prompts

*product #B / *list #C / Select object example*: Specify an object to replace (here you define a sample). Clarify your selection in the dialog window, see fig. 5.28 page 253.

Product

Selects a product to replace by specifying any object included in the product.

List

Selects an object to replace from the object list. The object list is created concerning specified object.
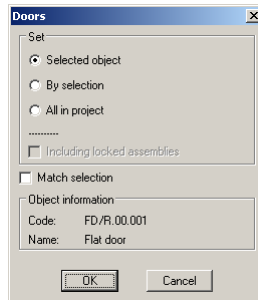
Fig. 5.28   Dialog window "Object replacement"

Selected object

Defines replacement only selected object. Other settings in the window don't matter at all.

By selection

You will able to select additional objects in the next step to replace them. Note, if your additional selection will include some objects not equal by type to the sample, they will be automatically excluded from selection.

 All in project

Defines replacement objects in the project.

Match selection

Selected objects equal to the sample (you specified it before) will be replaced. If the control is disabled, selected objects which type is equal to the sample will be replaced.

Including locked assemblies

Controls whether objects from locked assemblies will be replaced

Note. The sample code and name are displayed in the bottom part of the dialog window, see fig. 5.28

Attention! The model replacement presupposes the partial or full inheritance of model variables. (The variables are inherited from the project in the replaced model). It is impossible to replace a model with another one if inheritance rules are not taken into account.

The rules of inheritance in case of model replacements

Conditionally the list of variables is split into two areas: a top area and a bottom area. The first system variable is a border, see fig. *5.29*. The system variables are marked with "*" symbol in a note field. The bottom area of the variables list includes variables which determine the model. They are localized inside of model (they are never inherited when you replace the model). The variables from the top area determine a space where the model is placed. These variables may be inherited when you replace the model.

If a typical model or drawer is replaced, the variables "H", "L", "W", "X", "Y", "Z" and other from the top area of the variables list are inherited from the typical model which you are going to replace in the project to the new model which will be placed instead of the first one. Note, the variables marked with "_" sign never are inherited. For example "H_", "W_", "V1_", see fig. *5.29*.

If a Door model is replaced, the box variables "Lb", "Wb", "SX1", "SX2", "SY1", "SY2" and variables from the top area of the variable list are inherited to the exclusion of variables marked with "_" sign.

Note. If there are the same variables, which can be inherited, in different models and these variables have different functions (for example "V1" is a plinth height in one model and "V1" is a cabinet height in other one) these models can't be replaced correctly. Therefore it is recommended to create own variables system and keep it in different models.
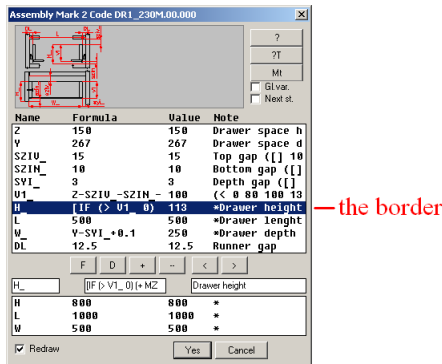
Fig. 5.29 "Variables" dialog window

5.20. Replace of materials

Access method
Menu: **Edit >> Replace materials**

Summary

Replaces materials and objects which has class "standard product". This command controls materials and standard objects of all selected objects including locked Assemblies.

Note. This command is a unique means to adjust project to the current database if it was created in another database (in another computer).

List of prompts

Select objects <project>

Select objects: Press **Enter** to select all object in the project or use "Window selection" or "Crossing selection" (more details see in section 1.13.5, page 21.) to select some objects.

Execute materials replacement in the opened dialog window, see fig. 5.30.
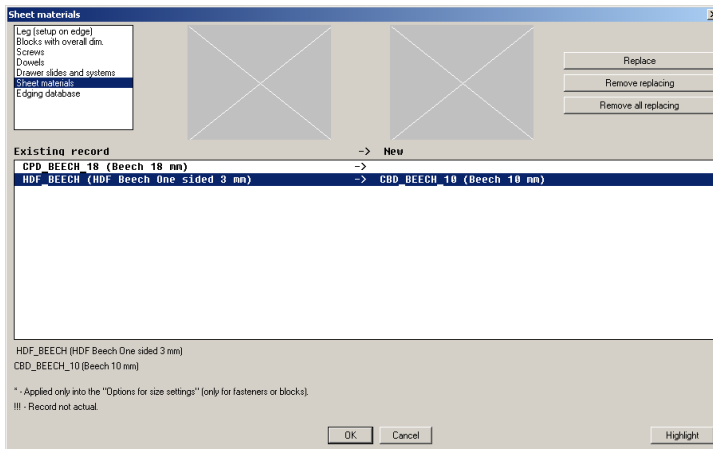
Fig. 5.30. "Control materials" dialog window

Database sections in the left top corner of the window (Leg, Blocks, Screws, Dowels and other in this example) filter the material according database sections.

Execute double click on the material in the bottom part of the window or specify it and press "**Replace**" button to replace it. If a material has been replaced, you see old material in the right window part and a new material in the left window part, for example "HDF_BEECH" and "CBD_BEECH_10"

Remove replacing

Cancels replacement for specified material

Remove all replacing

Cancels all replacements which have done in the dialog window.

Note. If item is marked with "*" sign in the dialog window, it means that fastener or block is absent in the project, but it is present in the "Options for size settings". More details about "Options for size settings" see in sections 3.5.1.8, page 76 and section 3.8.12, page 145.

Note. If item is marked with "!!!" sign in the dialog window, it means that item of database was edited after item insertion in the project. Press "Ok" to close window and item will be automatically adjust in the project according to database.

Attention!
- If a Part is included in an Assembly and its material is assigned as an Assembly logical material, this command doesn't control the Part material.
- To control the Assembly logical material with this command, you should select the Assembly frame. Note, if you pressed Enter to select all object in the project, the all existing Assembly frames are selected automatically.

If you replace materials, it will lead to redrawing of objects. Their thickness, size, textures, costs will be updated.

Attention! Pressing on "**Highlight**" button cancels the materials replacement and highlights objects in the project which are matched to the specified material in the dialog window. (Visibility of other objects (which are not matched) will be disabled). To restore objects visibility, redraw the project or execute **Edit >> Visibility >> On**.

Using the "**Highlight**" button, it is convenient to execute a visual control of object materials and standard objects. For example, if you select an edging and click "Highlight", you will see all objects with the edging.

5.21. Control of construction order

The construction order is very important in 3D-Constructor.

The program saves the construction order of 3D-model and repeats it when you redraw the project according the input parameters. (If you change the Assembly sizes, the model will be rebuilt automatically according the new input parameters and construction order).

When you create a model, it is prohibited to create links from top of the construction order to bottom (the object created earlier can't be linked on object created later). For example, if there are two horizontal Parts and the first Part is created earlier, it is impossible to reference the first Part to the second one, it is

impossible to create a variable-distance for the first Part, which will use points or edges of the second Part.

If you try to break the construction order, you will get the message "Violation of construction order!"

You can correct the construction order sometimes. To control it, use commands:

Menu: **Edit >> Construction order of main objects**
Toolbar: **Edit**

(Controls the constructional order of Parts, Bent Sheets, Bench Tops, Profiles and Blocks)

Menu: **Edit >> Construction order of fittings and hardware**
(Controls the constructional order of Fasteners and Furniture fittings)
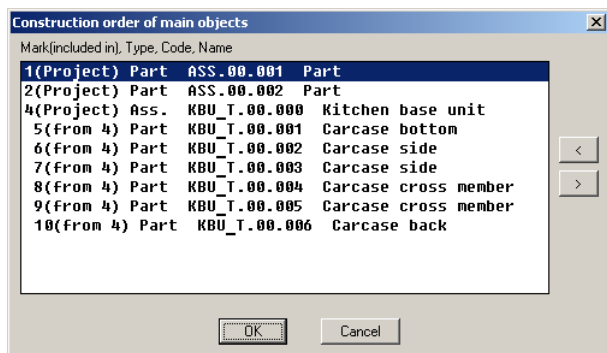
Fig. 5.31. "Construction order" dialog window

To edit the construction order, specify an object and click "<", ">" buttons. The object will be moved if it is possible. If it is impossible, you will get the message "Object can't be moved"

To avoid the construction order editing, take into account next:
• Create your furniture 3D-model like you mount the furniture;
• Create products independently from each other (it is not a good, if objects of one product are linked to object from another product);
• It is impossible to delete main object if another object are linked to it.

5.22. Shortcut Keys

There are some Shortcut Keys by default in the program.

CONTROL+"T"	– insertion of typical model,
CONTROL+"D"	– generation of drawing set,
CONTROL+"1"	– edging thing,
CONTROL+"2"	– edging thick,
CONTROL+"N"	– edit reference of N Parts in opening,

CONTROL+"H"   – insert fastener "Edge to surface" method from preset database,

CONTROL+"G"   – projection fastener on an edge,
SHIFT+"C"   – copy fitting in defined place,
CONTROL+"C"   – copy objects to Clipboard,
CONTROL+"G"   – paste objects from Clipboard.

To edit Shortcut Keys by default, use standard AutoCAD means (command "CUI").

Chapter 6. Reference and variables of objects

6.1. Reference of object

6.1.1. Generalities

The reference is a constraint which blocks an object position concerning another object.

Note. "Move", "Rotate", "Place", "Magnet" commands (see section 5.11, page 237) define an object position. The references define and fix an object position in 3D-model (reference is a tool of parametric drawing)

Hereinafter an object which is referenced to another object is named "referenced object"; the object, to which another object is referenced, is named "base object".

Note. The base object has to be situated in the *Construction order* before the referenced object, otherwise you will get message "Violation of construction order"

If we consider the reference concept deeper, the reference is a binding of the coordinate system (hereinafter CS) defined on the referenced object and the CS defined on the base object. The type of reference determines the way of CS definition. There are three type of reference CS.

- If you reference a point or reference to a point of an object, the reference CS is defined with CS of the object (The object CS is moved in the specified point);
- If you reference a direction or reference to a direction of an object, the reference CS is defined with two specified points. The first specified point assigns the reference CS origin, the second point assigns X-axis. Z-axis of the reference CS is assigned with Z-axis of the object CS. Y-axis is created so to get the right-handed CS;
  Note. Two specified points (which determine direction) have to belong to one object.
- If you reference an object to a surface assigned with three specified points, the first surface point assigns the reference CS origin, the second point assigns X-axis, the third point assigns Y-axis, Z-axis is created so to get the right-handed CS.

Attention! When you reference an object, it is available "ODO (box)" option. If it looks as "ODO (box)+", you will specify the object point. If it looks as "ODO (box)-", you will specify the object box point (if box is absent and you see "ODO (box)-", an object point will be specified). The object points and object box points have different colour. To control colour, use **Ser-**

vice >> Settings >> General settings, control "Point marker" and control "Marker of point of overall object sizes". More details about object box (overall object sizes) see in section 3.1.1, page. 30.

6.1.2. Reference parameter

To enter or edit reference parameters, the special dialog window is used, see fig. 6.1. Reference parameters may include operations and functions considered in section 6.2.5.2 page. 293.
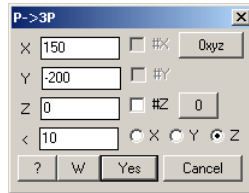
Fig. 6.1.   Reference parameters definition

The window controls are considered below.

"**X**", "**Y**", "**Z**" fields
Determine an offset of the reference CS origin of the referenced object concerning the reference CS origin of base object. The values which are in these fields by default show the current offsets at the time of a reference execution.

"**<**" field
Assigns a rotation angle around one of the reference CS axis. To define the axis use ⟨X ⟨Y ⟨Z controls.

"**0xyz**" button
Assigns nil value in the "**X**", "**Y**", "**Z**" fields.

"**0**" button
Assigns nil value in the "**X**", "**Y**", "**Z**" and "**<**" fields.

"**#X**", "**#Y**", "**#Z**" controls
Assign centering of the referenced object concerning the reference base point of the base object. The centering is considered below. These controls are available if the centering is possible.
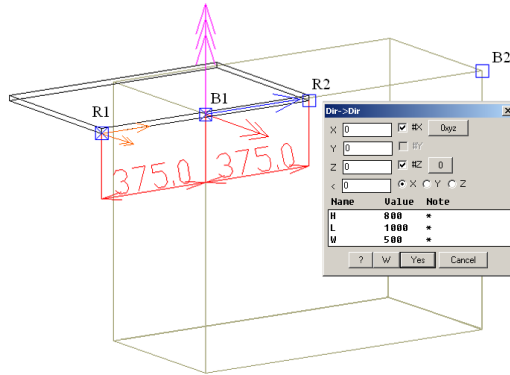
Fig. 6.2   Edge centering of referenced object concerning the origin of reference CS of base object

See the example above. The direction "R1R2" (edge of sheet Part) of the referenced object is bound to the direction "B1B2" (edge of the Assembly frame) of the base object.

Control "**#X**" sets the "R1R2" midpoint in the "B1" point.

Control "**#Z**" sets the referenced Part thickness midpoint in the "B1" point.

Note if you clear "**#X", "#Z"** controls, "R1" and "B1" points will be coincided.

Set "#" sign in the "**X**", "**Y**", "**Z**" fields, to center the base point of the referenced object (see fig. *6.3*) concerning the base object. The "#" sign in the "**X**" field sets R1 point in the middle of the "B1B2". The "#" sign in the "Z" field sets R1 point in the middle of the Assembly thickness.
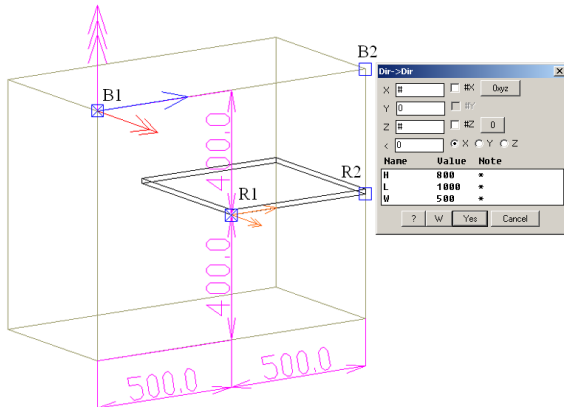
Fig. 6.3   Centering of origin CS of the referenced object (point "R1") concerning base object.

Use the fraction centering in the "**X**", "**Y**", "**Z**" fields, to place some objects at equal intervals along a direction, see fig. *6.4* and *6.5*.
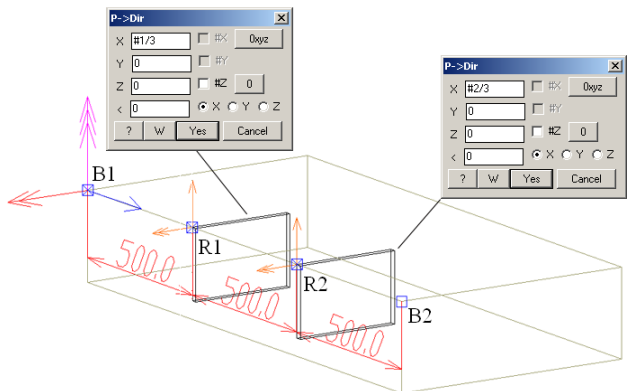
Fig. *6.4*   *Two Parts fraction centering along the "B1B2" direction.*
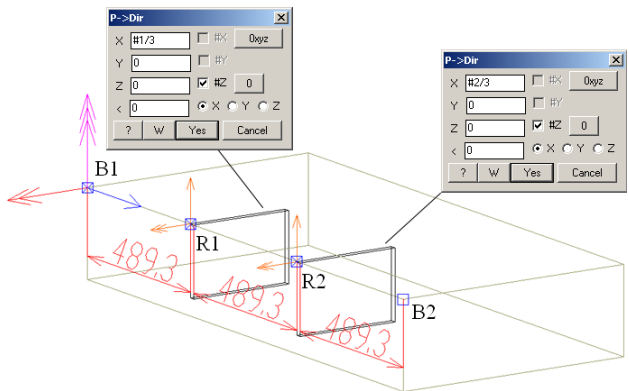
Fig. *6.5*   *Two Parts thickness centering and fraction centering along the "B1B2" direction.*

Attention. You can set an offset for fraction centering, for example enter "#1/3T+50" in the "X" field to move the first Part on 50 millimeters.

Note. The fraction centering is usually set automatically (very seldom manually) with special commands, see section 3.7.1, page 89 and section 3.7.2, page 95.

Note. You can assign a centering, if you reference a direction or reference to a direction. In other words you can center a point concerning a line.

Attention! Any reference command applied to an object cancels its previous reference.

When you reference an object, it is necessary to watch for reference axis directions. It was mentioned above that X-axis are marked with one arrow, Y-axis– with two arrows, Z-axis– with three arrows. The arrows color and size are set with command **Service >> Settings >> General settings**, area "**Dialogue elements colours**" and area "**Sizes of dialogue elements**"

All reference commands are grouped in the **Edit >> Object reference** and "Reference" toolbar.

6.1.3. Point reference

6.1.3.1. Point to LCS (remove reference)

Access methods
Menu: **Edit >> Object reference >> Point to LCS**
Toolbar: **Reference**

Summary
Sometimes it is necessary to remove an object reference. In this case you should reference object to the local coordinate system (hereinafter LCS). This command reference one of object points to LCS. (This point will be fixed in space, if you edit object parameters).

Note. There are not unreferenced objects in the project. Every object is referenced to another object or LCS.

List of Prompts
ODO (box) + / Reference point: Specify any point of object
Another reference point (ENTER - selected): Press **Enter** to confirm specified point or specify another point.

ODO (box)+
One of object points is referenced. To reference one of object box point, activate this option. More details about reference to box see in section 6.1.1, page 259.

6.1.3.2. Point to point

Access methods
Menu: **Edit >> Object reference >> Point to point**
Toolbar: **Reference**

Summary
This is more simply way to reference an object. It references a point of one object to a point of another object.

ODO (box) + / Object point: Specify a point of the object which will be referenced.

ODO (box) + / *redefine object point #N / Point of base object: Specify a point of the base object.

Another reference point (ENTER - selected): Press **Enter** to confirm specified point or specify another point.

Enter reference parameters in the dialog window, see fig. *6.6.*
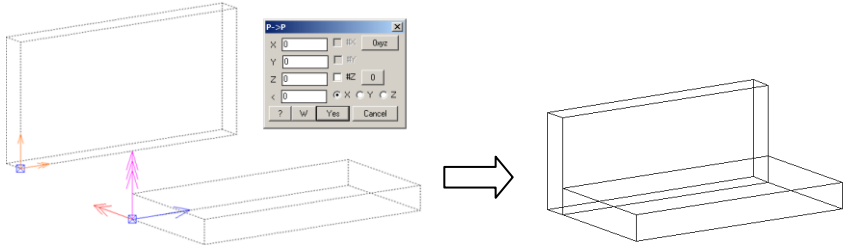
Fig. 6.6.   Reference an object point to a point of base object

6.1.3.3. Point to direction

<u>Access methods</u>
Menu: **Edit >> Object reference >> Point to direction**
Toolbar: **Reference**

<u>Summary</u>
It references a point of one object to a direction of another object. The direction is determined with two specified points on the base object.

The point prompts correspond to the prompts which are considered in section 6.1.3.2, page 263.

Note. This reference allows centering the point of the referenced object concerning the direction of the base object. More details see in section 6.1.2, page 260.

6.1.3.4. Point to surface

<u>Access methods</u>
Menu: **Edit >> Object reference >> Point to surface**
Toolbar: **Reference**

Summary

It references a point of one object to a surface determined with three points on other objects. Note three points may be specified on three different objects.

The point prompts correspond to the prompts which are considered in section 6.1.3.2, page 263.

Note. This reference allows centering the point of referenced object concerning X and Y directions of the surface. More details see in section 6.1.2, page 260.

6.1.3.5. Point to point (Polar CS)

Access methods
Menu: **Edit >> Object reference >> Point to point (Polar CS)**

Summary

It references a point of one object to a point of another object in polar coordinate system.

The reference leg to the round worktop is a good example of this reference type, see fig. 6.7.

The list of prompts corresponds to it which is considered in section 6.1.3.2, page 263.
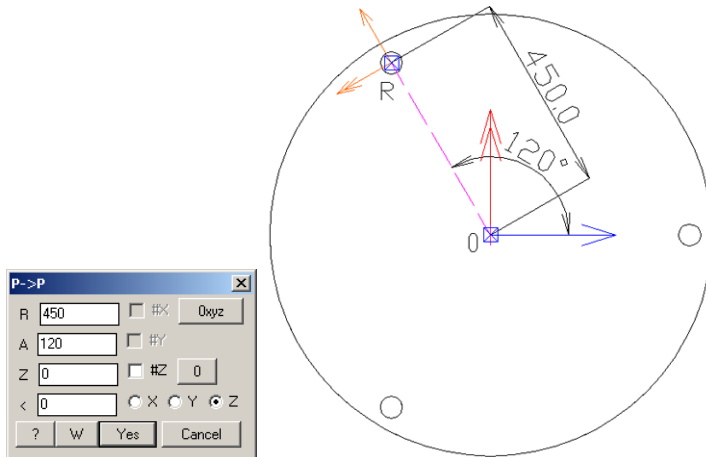
Fig. 6.7   Point to point reference in polar CS

"**R**" field
Defines a distance from the point of the referenced object to the point of the base object (the "R" and "O" points were specified as point to reference).

"**A**" field

Defines an angle between X-axis of the base object and the direction which is defined with two specified points ("RO" direction in the fig. 6.7).

Other elements of the dialog window correspond to elements which are considered in the fig. *6.1*, page 260.

6.1.3.6. Point along path

 Access methods
Menu: **Edit >> Object reference >> Point along path**

Summary

It reference a point of object concerning a path of the base object (it may be taken a path of the Bent Sheet or Profile). A typical example is the handle placing along the Bent Sheet, see fig. 6.8.
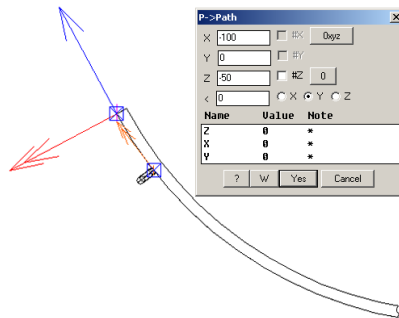
Fig. 6.8   Handle reference along the Bent Sheet path

"**X**" field

Defines a distance from the referenced object point to the base object point along the base object path. Z-axis of the referenced object is set perpendicularly to the path tangent.

Other elements of dialog window (fig. *6.8*) correspond to elements, which are considered in section 6.1.3.2, page 263.

6.1.4. Direction reference

This group of commands determines a direction position specified with two points on the referenced object concerning primitives of the base object.

The rules of creating reference CS are considered in section 6.1.1, page 259.

The list of prompts are identical to it considered in section 6.1.3, page 263.

6.1.4.1. Direction to LCS

Menu: **Edit >> Object reference >> Direction to LCS**

Summary

It references a direction specified with two points on the object to the local coordinate system (hereinafter LCS). In other words the referenced object hasn't a base object it is fixed in LCS.

Note this reference allow centering 3D-Constructor object concerning AutoCAD primitives (it is used very seldom).

6.1.4.2. Direction to point

Access methods
Menu: **Edit >> Object reference >> Direction to CS of object**
Toolbar: **Reference**

Summary

It references a direction specified with two points on the one object to a point of the base object. Note it is possible to center the direction of the referenced object concerning the point of the base object. More details see in section 6.1.2, page 260.

6.1.4.3. Direction to direction

Access methods
Menu: **Edit >> Object reference >> Direction to direction**
Toolbar: **Reference**

Summary

It references a direction specified with two points on the object to a direction specified with two points on another object. Note direction points have to be specified on the one object.

This type of reference allows centering the direction concerning other direction, see fig. *6.9.* The edge of small Part is centered concerning the diagonal of the base Part.
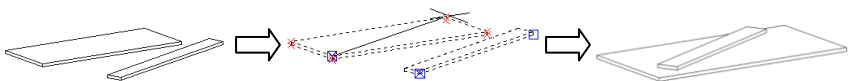
Fig. 6.9.   Direction to direction reference

Note! There is only one angle parameter in the reference parameters (see fig. 6.1 page 260) therefore as default angle value is taken a minimal angle between two any axes of referenced object and base object. In other words to have angle between directions as the reference parameter, you may coincide the directions roughly before reference objects. (To coincide directions roughly, use **Edit >> Rotate** or **Edit >> Move, rotate, shift**)

6.1.4.4. Direction to surface

Access methods
Menu: **Edit >> Object reference >> Direction to surface**
Toolbar: **Reference**
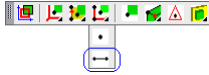
Summary
It references a direction specified with two points on the object to a surface defined with three specified points on base objects. Note three surface points may be specified on three different objects. Two direction points have to be specified on the one object.
Note. This reference allows centering the referenced object concerning X and Y directions of the surface. More details see in section 6.1.2, page 260.

See fig. 6.10. The middle Part is referenced to two Parts with "Direction to surface". Note the middle Part is oriented roughly in the right position before it will be referenced, more details see in section 6.1.4.3, page 267.
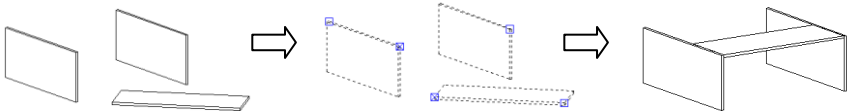
Fig. 6.10   Direction to surface reference

6.1.4.5. Direction to point (Polar CS)

Access methods
Menu: **Edit >> Object reference >> Direction to point (Polar CS)**

Summary
It references a direction specified with two points on the object to a point of the base object in polar CS (you should enter the vector length and an angle between X-axis and the vector, more details see in section 6.1.3.5, page 265)

6.1.5. Side of object
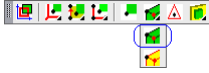
Summary

It redefines a reference point of the referenced object. The new point will be taken from opposite side of the object (to find a new point, the program will move from the referenced point to the new point along Z-axis of object). See fig. 6.11 page 269.

Note the command can't be executed if the object thickness is equal to nil. For example the thickness of *Constructional drawing* is nil.

List of prompts

Select object: Specify a referenced object. The referenced point will be highlighted on the object when object is specified.

To change side - any key / ENTER - apply: Press any key to change the reference point. Press **Enter** to apply highlighted point.
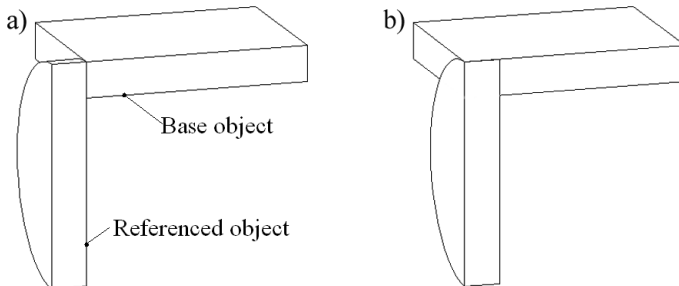
Fig. 6.11. The side of referenced object changing
a) Original position of the objects
b) Objects position after editing

6.1.6. Side of base object

Summary

This command corresponds to the command considered above, but it changes side of reference of the base object, see fig. 6.12

List of prompts

Select object: Specify a referenced object. The referenced point on the base object will be highlighted when object is specified.

Note. You should specify the referenced object, although a side of the base object will be changed.

To change side - any key / ENTER - apply: Press any key to change the reference point. Press **Enter** to apply highlighted point
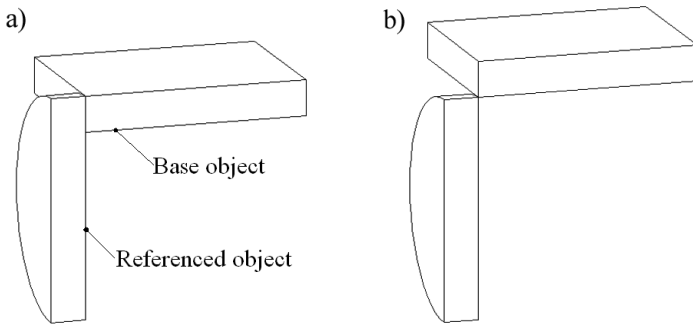
Fig. 6.12. The side of base object changing
a) Original position of the objects
b) Objects position after editing

6.1.7. Put on a surface XY

Access methods
Menu: **Edit >> Object reference >> Put on a surface XY**
Toolbar: **Reference**

Summary

This command places and references a specified object on XY-surface of the base object. To reference objects, one of references considered in sections 6.1.3 and section 6.1.4 is used (this isn't a new reference type). It is important where you specify a referenced object and a base object; the specified primitives (point of block, edge of Part, point of Bent Sheet path) will be taken to reference objects.

ODO (box) + / settings #Y / Referenced object: Specify an object to reference it. Note the specified primitive of object (point or edge) will be taken as point or direction to reference object.

ODO (box) + / Base object: Specify base object. Note the specified primitive (point or edge) will be taken as point or direction to reference the first specified object.

Specify setup surface.

To change direction - any key / ENTER - apply: Press **Enter** to confirm the highlighted surface of the base object (the surface is highlighted with arrow). Press any key to change surface of the base object.

Enter reference parameters in the dialog window.

ODO (box)+

One of object points is referenced. To reference one of object box point, activate this option. More details about reference to box see in section 6.1.1, page 259.

Settings

Opens an additional settings for this command, see fig. *6.13*. Note the selected setting will be saved and used until you will change them.
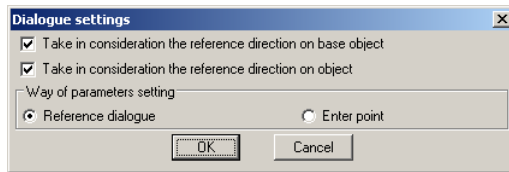
Fig. 6.13   Additional settings for "Put on surface" command

Take in consideration the reference direction on base object

The first specified object will reference to a direction of the base object if control is enable. If it is disabled, the object will be referenced to point of the base object.

Take in consideration the reference direction on object

The point or direction of the object which is referenced may be taken to reference. If control is enabled and object has edge, the direction of the object will be referenced.

Way of parameters settings

To reference object to base object, you should define reference parameters. If the "Reference dialogue" checkbox is activated, the reference parameters will be asked through the dialog window, see fig. *6.1*, page 260. If the "Enter point" checkbox is activated, the reference parameters will be defined

with specified point (you will able to edit them after command completion through the **Edit >> Object reference >> Reference Parameters**).
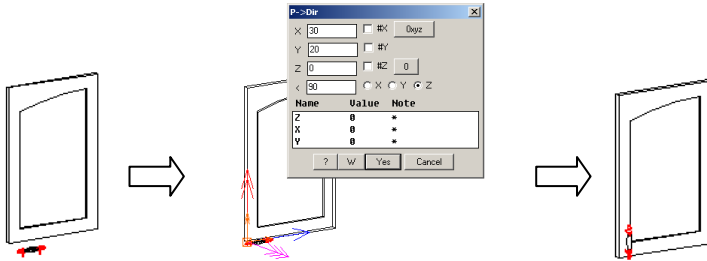
*Fig. 6.14. Example of the "***Put on a surface XY***" reference*

Note. To reference the hand, was specified the horizontal edge of the door therefore hand is align along the horizontal edge in the second picture. If you specify the primitive of base object exactly, the referenced object will be aligned right.

6.1.8. Reference by edges (edge to edge)

This is a special commands group which quickly place and reference one Part about another Part. (Note the object, which will be referenced, should be situated in the *Construction order* after the base object). To reference object, the reference types considered in section 6.1.4 is used (this is not a new reference type).

Access methods
Menu: **Edit >> Object reference >> Reference by edges (edge to edge)**
Toolbar: **Reference**

List of prompts
Object edge: Specify an object to be referenced.
Edge of base object: Specify a base object.
Select join variant <apply>: Select in the special toolbars (see fig. *6.15*) the type of joint and press **Enter** to apply. Note you can press buttons sequentially, and when objects will take the desired position, press **Enter**.

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Fig. 6.15 "Joint types" toolbar

(It is available only when command is executing)

Note. In the "Joint type" toolbar the base object is red, the referenced object is green and the referenced edges are marked with black point.

6.1.8.1. **Joint_1**
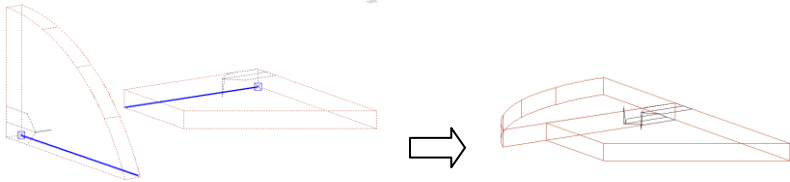
6.1.8.2. **Normal_2**
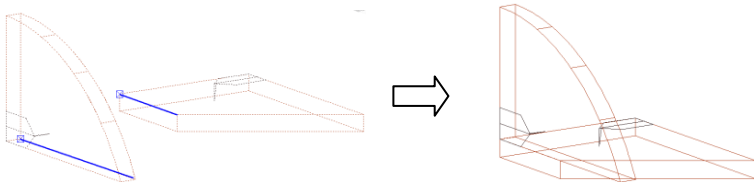
Note. Press "Mirror_7" button to change side of base object, see section 6.1.8.7.

6.1.8.3. **Normal_3**
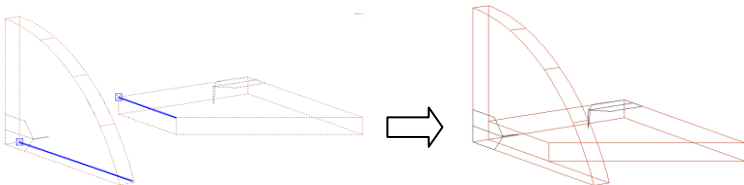
6.1.8.4. **Normal_4**
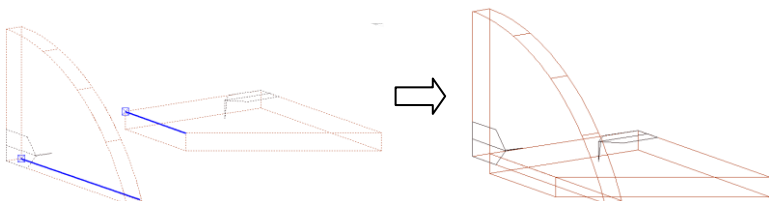
6.1.8.5. Normal_5
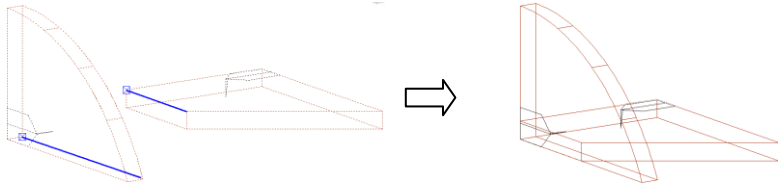
6.1.8.6. Parallel_6
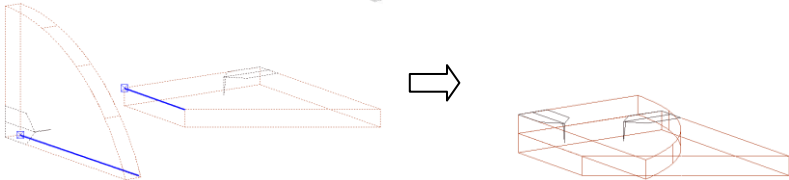
6.1.8.7. Mirror_7

Note this command only change the reference which has been defined. You should use any of above considered buttons to reference an object and then use "Mirror_7" to get the mirrored reference.
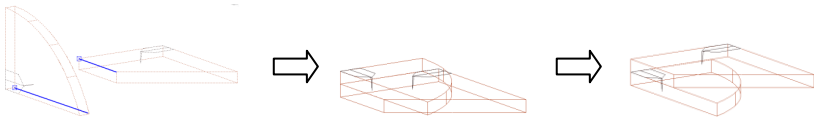
Example 1
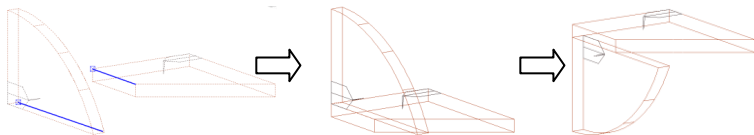
Example 2

6.1.9. Turn

Summary

It turns the referenced object concerning its dimension see fig. *6.16*. The turn direction is highlighted with arrow on the screen and may be determined by user.

Note. The turned object shall be referenced for one of dimensional points to execute the command successfully.

List of Prompts
Select object: Specify object to edit.
To change direction - any key / ENTER - apply: Determine the turn direction and press Enter to complete command.
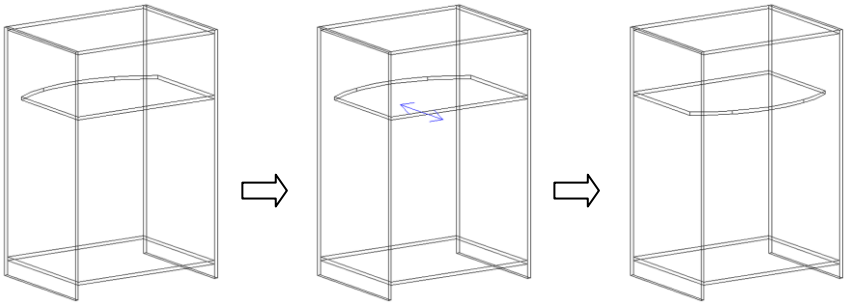
Fig. 6.16. Turn of the referenced object

6.1.10. Additional points (only 0).

Summary

Shifts a reference coordinate system origin of the referenced object concerning specified points of other objects. Note the objects, which define reference additional points, have to be situated in the *Construction order* before

(above) the referenced object. The command allows to define three points to shift reference origin concerning X, Y and Z-axes.

List of Prompts
Select object: Specify object to edit its reference.
*next #L / Offset X - 0 point offset (else base point): Specify an additional point to move the reference origin concerning X-axis or press **Enter** to skip this request.
*next #L / remove existing #C / Offset Y - 0 point offset (else base point): Specify an additional point to move the reference origin concerning Y-axis or press **Enter** to skip this request.
*next #L / Offset Z - 0 point offset (else base point): Specify an additional point to move the reference origin concerning Z-axis or press **Enter** to skip this request and complete the command.

Note. If object has got an additional point for X, Y or Z direction, you will see the "remove existing" option. Activate it to delete an additional point.
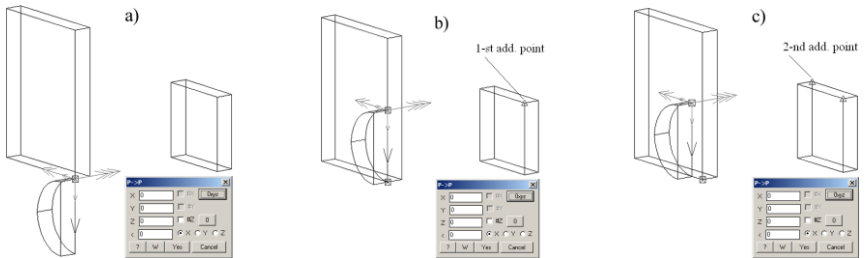
Fig. 6.17.   Entering reference additional points.
            The 1-st point moved reference CS origin along X-axis
            The 2-nd point moved reference CS origin along Y-axis

            Note reference parameters are the same for three case.

Note. The "reference parameters" commands highlights all reference points (reference origin point and additional points)

6.1.11. Additional points

Access methods
Menu: **Edit >> Object reference >> Additional points**

Summary
This command defines an additional reference point to move the reference CS origin (see section 6.1.10, page 275) and to center the referenced object. The list of prompts corresponds to the list considered in section 6.1.10.
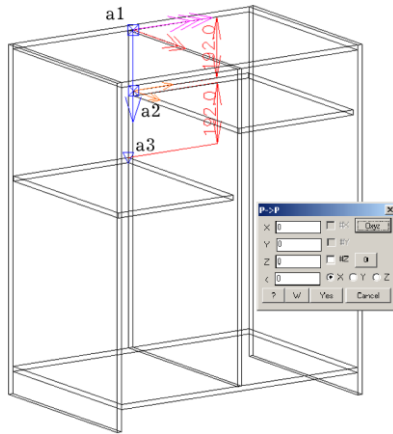
Fig. 6.18. Entering an additional point to center referenced object

"**a2**" is a point of the referenced object which is referenced to a point of the base object.

"**a1**" is a point of base object to which is referenced the shelf.

"**a3**" is an additional reference point which was created to center the shelf along X-axis of the reference CS.

To center the shelf along "**a1a3**"**-**direction:

"#" sign was set in the X**-**field

"#Z" control was enabled.

More details about centering see in section 6.1.2, page 260.

6.1.12. Transmit reference to contour

Access methods
Menu: **Edit >> Object reference >> transmit reference to contour**

Summary

The Part or Door-assembly may have overall dimensions of object (box), more details see in section 3.1.1, page 30. One of object box point may be used to reference it (you should activate "*ODO (box)*" option, to use one of object box point). Other words, you may use an object point or an object box point to reference the object or reference to an object.

To transmit a reference from object box point to the nearest contour point, use "*transmit reference to contour*" command.

List of prompts

*multiple selection #Y / Define object to transfer reference points on contour: Specify an object to edit. Select additional controls in the dialog window. Press "Ok" to complete command.
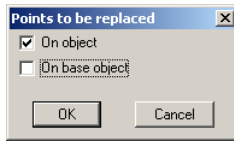
Fig. 6.19   Additional controls to define command

On object
The referenced object is edited. The reference of the referenced object will be transmitted.

On base object
The base object is edited. The reference point will be transmitted on the base object. (Note you specify referenced object but the reference point will be transmitted on the base object)

Multiple selection
Allows you to select some objects in the same time.

6.1.13. Transmit reference to box

<u>Access methods</u>
Menu: **Edit >> Object reference >> transmit reference to box**

<u>Summary</u>
The command corresponds to the command considered in section 6.1.12 page 277 but it transmit the reference of specified objects from contour to box points.

6.1.14. Replace reference point on object

<u>Access methods</u>
Menu: **Edit >> Object reference >> Replace reference point on object**

<u>Summary</u>
Redefines reference points of specified object. Note you specify a referenced object to edit, but reference points may be replaced for the referenced object or its base object. For example you can redefine reference point of the hand (see fig. *6.20*) or of the door to which hand is referenced.

<u>List of prompts</u>
*points of referenced object #J / Specify referenced object to redefine points on the BASE object: Specify object. Note you will replace point on the base object in spite of specifying referenced object.
Select base point (ENTER - use previous) Base point: Specify new base point on the base object. Note if you press **Enter**, the old base point will be taken again.

Change reference point <ENTER - selected>: All object points are being highlighted now. Specify any another point or press Enter to confirm new base point specified before.

If you activate "*points of referenced object*" option, the list of prompts will look as presented below.

*points of base object #< / Specify referenced object to redefine points on the REFERENCED object: Specify object to edit.

New base point: Specify new base point on the referenced object.

Note. Depending on a reference method one object to another object (or objects), similar requests of the second and third points can follow.
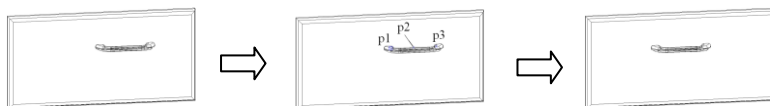
Fig. 6.20   The handle was referenced by the point 1. The reference
            is transmitted to the point 2.

6.1.15. Edit reference N shelves in opening

<u>Access methods</u>
Menu: **Edit >> Object reference >> Edit reference N shelves in opening**

<u>Summary</u>
This command set specified shelves at equal interval along specified line (the line is determined with two points projection on Z-axis of the first specified Part).

To have success the edited object should be:
- referenced
- have the same thickness
- parallel each other

Note. The Parts placing at equal intervals is possible only along Z-axis of Part coordinate system.

Note. To create N-shelves or dividers, use special commands. More details see in section 3.7.1 page 89 and section 3.7.2 page 95.

<u>List of prompts</u>
Select parts to be moved
Select objects: Select Parts to align. Note you can use the multiple selection.

Selected parts have different reference point or different thickness! Note you will get this message if any selected Part is not answer features mentioned above. (This means that something is wrong)

Base *point*: Specify first point to define a line

*change point #N / Second point: Specify the second point or redefine the first specified point.

change point #N / *apply selected #G: Press **Enter** to complete the command. Use "*change point*" option to redefine the second point.

6.1.16. Change part position

Access methods
Menu: **Edit >> Object reference >> Change surface or reference**

Summary
This command is a special case of the command described in the section 6.1.17.

6.1.17. Change part position

Access methods
Menu: **Edit >> Object reference >> Change part position**

Summary
It aligns a referenced object concerning the base object.

List of prompts
Select object: Specify referenced object to align it.

redefine surface #C / mirror #P / rotate #G: Use one of the options to align the specified object. Note the reference CS will be highlighted when you specify referenced object.

To confirm the object position and complete the command press Enter.

Redefine surface
Sets the referenced object in XY, YZ, XZ-surface of reference CS, see figure below.
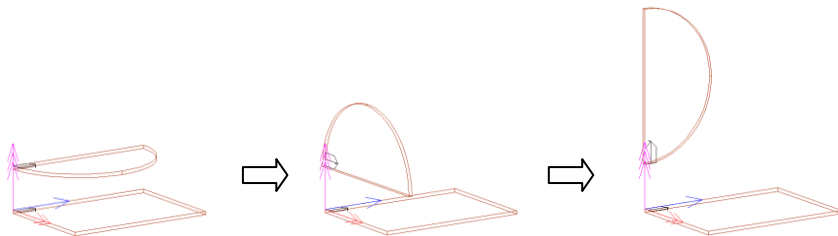
Mirror

Replaces the referenced Part with a mirrored Part. The mirror line is a bisector of the angle which is created with X and Y axes of reference CS, see figure below.

Rotate

Rotates the referenced object around Z-axis of the reference CS on 90 degree.

6.1.18. Block position

Access methods
Menu: **Edit >> Object reference >> Block position**

Summary
It aligns the specified block or fitting (handle, leg, sink and other) concerning base object.

List of prompts
Select block to change orientation: Specify block to edit.
rotate #G / side of setup #C / *next # L: Use one of the options to define a new block position.

Rotate

Rotates the referenced object around Z-axis of the reference CS on 90 degree.
Side of setup
Changes side of the block base Part on the opposite side.
Next
Complete the command.

6.2. Variables

There are next concepts:
- global variables of the project,
- logical materials of the project,
- object variables,
- built-in object variables (!Z, !X, !P…),
- object variables which have fixed names (Lb, Wb, SX1, SX2, KM_L…),
- references on the variables of other objects (@9VT, @2H…)

All of the above mentioned variables, which belong to different objects of the current project, are variables of the current project. The structure scheme of the project variables is presented in the fig. 6.21 page 282.

6.2.1. Project variables
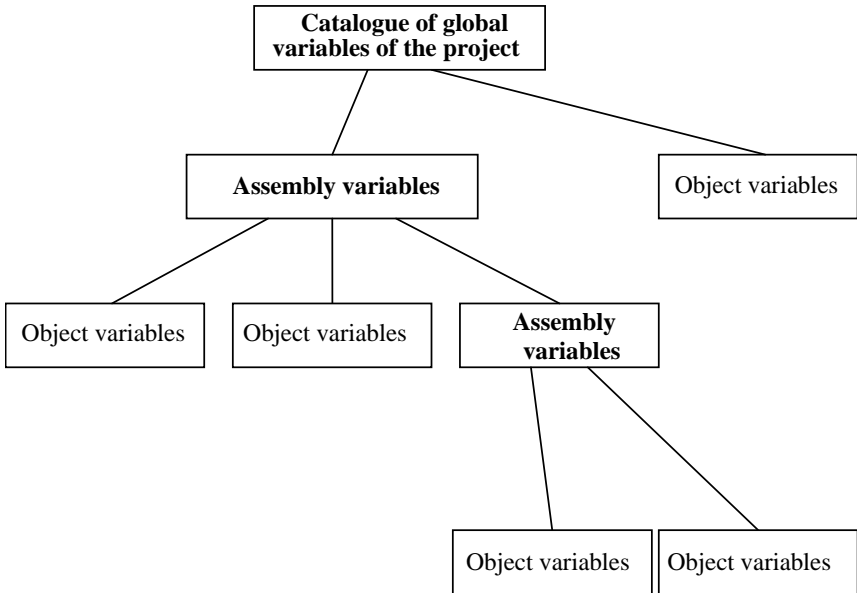
Fig. 6.21. Structure scheme of the project variables

Note. The object is an elementary object (not Assembly) for example Part, Profile, Handle and other.

The interaction between objects variables is created according next rules:

- The catalogue of global variables is the top of the project variables. You can use its variables in any object of the project. Note, you can't utilize its variables directly, you should create references from object or assembly variable to its variables (in other words a value of the object variable can be assigned through one of global variable).
- There are three types of variables. This is a variable-formula, a variable-distance and a variable-logical material.
- The object variables may be used to define object operation parameters or object reference parameters. The object can have variables or not. (In second case its sizes are defined only with parameters)
- The Assembly variables are available for any object included in the Assembly. You may use them to define included object reference and

included object parameters. You may use them to define Assembly reference and Assembly parameters.

- Every object has an access to its own variables and to variables of the Assembly in which it is included.
- Utilizing reference on the variable (for example @9VT, @2H), you can assign a value of the object variable through the variable of any object created before.
- If you edit object parameters or object reference and the object has a variable with the same name as the variable of the Assembly in which the object is included, this variable in the Assembly will be unavailable. You will not see it in the Assembly variable list of the "Parameters" or "Reference" dialog window.

There are some constraints which you should take into account when you edit the variables:

- The name of variable must be defined with English keyboard layout. The first sign of the name must be a letter. The variable name is unique within one object and it can't be same as one of special name (Pi, E or a LISP function name).
- "A" and "a" is the same variable.
- The variables are calculated in the order they are set in the object variable list. At first you should assign a value of the variable and then you can use it in the expression.
- It is impossible to rename or delete a variable if any object parameter, reference parameter or another variable is defined through it.

6.2.2. "Variable" dialog window

To create or edit object variable, you should use "Variable" dialog window, see fig. 6.22 page 283 (command **Edit** >> **Variables**)
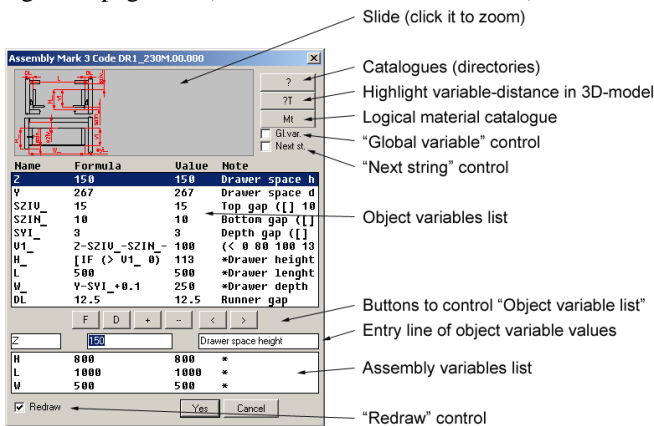
Fig. 6.22   "Variable" dialog window

6.2.2.1. Object variable list

It is a variable list of the selected object. The list string includes field described below.

Name

It is a name of the variable. It is assigned only with Latin letters. The first sign must be a letter.

Formula

It may be a parameter (a number) or an expression or variable-distance. The expression may include parameters, any object variables assigned before in the object variables list and Assembly variables if the object is included in the Assembly. If it is a variable-distance, the field is blocked for editing, more details see in section 6.2.5.3, page 295.

Value

It is a value of the parameter, or a result of the expression, or a value of the variable-distance.

Note. The content of the "Note" field may influence to "Value" field.

Note (field)

It may include a *variable comment*, *service information* and *special expression* which influences to the "Value" field.

The *variable comment* is a text string which explains the variable sense.

The *service information* is "*" and "$" signs. If you see one of these symbols in the "Note" field (for example see "L" variable in fig. *6.22*) this is a system variable. More details see in section 6.2.5, page 291.

The *special expression* is a condition and some values for the variable. The value of the variable will be calculated and then limited through this expression. More details see in section 6.2.5, page 291.

6.2.2.2. Slide

It is a dimension slide of the object, which shows the sense of variables. Note, if there isn't dimensional slide of the object, you will see model slide which will show the object appearance.

Note, the dimensional slide may be created by user for every model in the database. It isn't created automatically. To zoom a slide, execute the left mouse double-click on it.

6.2.2.3. "Catalogues" button

Press "?" button (see fig. 6.22 page 283) to open list of catalogue.
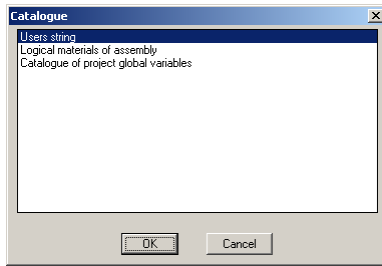
Fig. 6.23   "Catalogue" dialog window

There are next catalogues
- User's string (These are strings which may include any parameters, expressions, variables defined by user to be copy then in Clipboard and paste in the project).
- Logical material of assembly (These are logical materials of Assembly to which object is included. Note, the Assembly may have or not have logical materials).
- Catalog of project global variables.

Open any catalogue and execute left mouse button double click to copy a variable name in the Clipboard and close the catalogue. Set the cursor in the field where the variable has to be inserted and press "Ctrl+V".

6.2.2.4. "Highlight variable-distance" button

Press this button (see fig. *6.22*, page 283) to highlight in 3D-model the selected in the "Variable" window variable-distance. It you select a not variable-distance and press this button, the pressing will be ignored.

6.2.2.5. "Logical material" button

It (see fig. 6.22 page 283) will be preset in the "Variable" window if you edit Assembly variables. Press it to edit logical materials of the Assembly, see fig. *6.27*, page 290.

6.2.2.6. "Global variables" control

Activate this control (see fig. 6.22 page 283) to update values of the variables in the "Variable" dialog window according to "Catalogue of project global variables".

Note. You should reference the variable to the "Catalogue of project global variables" to have a possibility to update it. By default all variables are not referenced to the "Catalogue of project global variables". More details see in section 6.2.3, page 287.

6.2.2.7. "Next string" control

If you assign a value of variable through an expression and the expression can't be calculated, you will be not able to change focus (select another string) in the "Variable" window. You will get an alarm message, if you try to do that, and focus will be set back.

Activate this control (see fig. 6.22 page 283) to change focus in any case. You will be able to select and edit any variable to correct the situation. Note, you will be not able to close "Variable" window through "Yes" button if any variable in the window is not calculated.

6.2.2.8. Buttons to control "Object variables list"

Buttons "<", ">"
Move a selected variable in the list up and down.
Note. System variables may not be moved in the variable list by user. More details see in section 6.2.5, page 291.

Buttons "+", "–"
Add and delete a selected variable in the list.
If you click "+" button a new variable will be added. Its name will be assigned as "Vi", where i=1, 2, 3… A value of the new variable will be copied from a selected before string. Then you will able to edit the variable name and the variable value.
Note. You can delete a selected variable if there isn't reference to it. (If you use a variable in the object parameters or reference parameters, it can be deleted. You will get an appropriate message).

Buttons "F" and **"D"**
Select any variable-distance and press "F" button to convert it into the variable-parameter.
Select any variable-parameter and press "D" button to convert it into the variable-distance. The variable list will be closed and you will be enquired about a distance. (You should define the distance in 3D-model).

6.2.2.9. Entry line

To edit any variable, you should select it in the variable list (see 6.22, page 283) and edit its name, value and note in the entry line.

6.2.2.10. Assembly variable list

If you edit an object which is included in an Assembly, you will see the Assembly variables (see 6.22, page 283) in the object "Variable" window. You are able utilize the Assembly variables in any expressions of the object variable.

6.2.2.11. "Redraw" control

Activate "Redraw" checkbox to close "Variable" window and redraw 3D-model according to new values of the variables.

Note. If "Redraw" isn't activated, object variables will take new values but 3D-model will not be redrawn. In this case you can use one of redraw commands (menu **Edit >> Redraw**) to do it separately.

6.2.3. Catalogue of project global variables

This is a catalogue from which a value of the variable may be inserted in any object variable. This is not a directly reference. A value of the object variable will be updated through special control by user if the global variable is found in the catalog of project global variable. If it is absent in the catalogue, the object variable will save its current value.

Note. This approach to the global variable list allows rebuild model in any case (even through the referenced variable is deleted in the catalogue but is present in the model).

To create reference to a global variable, you should insert its name between < > brackets in the "Note" field of the "Variable" window, for example <KLEG>, see fig. *6.24*.
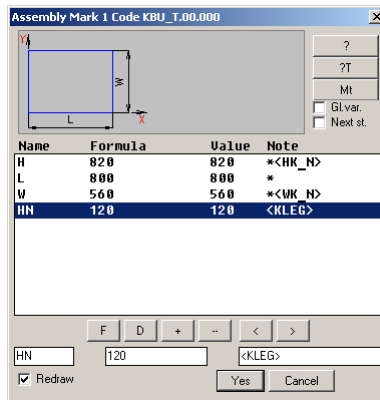
Fig. 6.24   Example of the reference to "KLEG" global variable

Attention! To insert a value of the global variable, you should activate "Gl.var." checkbox before "Variable" window will be closed.

Note. It is comfortable to copy global variable name from the catalog, more details see in section 6.2.2.3, page 284. When copying a name of the global variable from the catalog, it's comfortable to activate "Frame" control to automatically put variable name in the characters "<", ">", see fig. *6.25*.
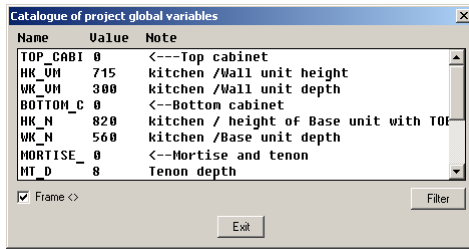
Fig. 6.25 Copying a global variable name from the catalog

To edit the catalog of project global variables, use **Service >> Settings >> Catalogue of global variables >> Current project**, see fig. 6.26. This window consists of two parts. These are global variables of the current project (it is the upper part) and global variables by default.

a)

b)
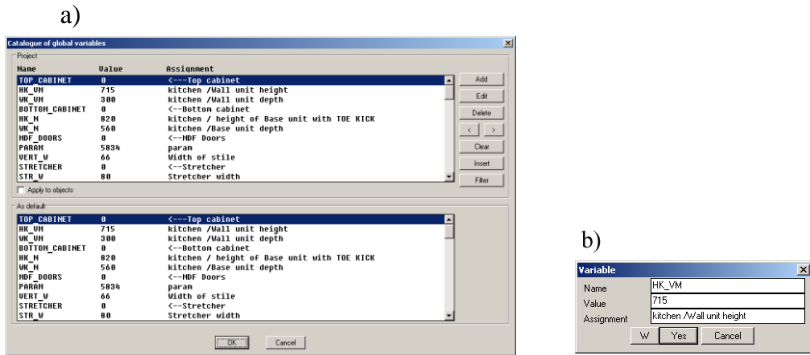
Fig. 6.26. Setting of project global variables
    a) Catalogue of project global variables window
    b) Editing of variable window

Add
Adds the new variable
Edit
Edits the selected variable
Delete
Deletes the selected variable
Buttons "<", ">"
Move the selected variable in the catalog up or down.
Clear
Deletes the all variables from the project catalog.
Insert
Inserts the selected variable from the catalog by default in the catalog of the current project.

Filter

Filters strings of the project catalog. Note, the filter key is the first word of the "Assignment" field.

Apply to object

To update values of the global variables in the 3D-model, you should activate "*Apply to object*" checkbox, press "Ok" and select object to be updated.

Attention! When you insert 3D-model in the project from the database, values of the global variables may be automatically updated. To do this, activate "*After model insert renew global variables*" (command **Service >> Settings >> General settings**)

The "catalog of global variables by default" determines the list of global variables in a new project. To edit it, use **Service >> Settings >> Catalogue of global variables >> By default**

 Note. If you open any existing project, it includes its own list of global variables. (The list of global variables was defined before).

6.2.4. Logical material

6.2.4.1. Generalities

Logical material is an Assembly variable (any other object can't have that variable). Its value is defined with a sheet material thickness (you should assign one of sheet materials to that variable). The name and description of the logical material are determined by user according to total rules.

6.2.4.2. Logical material of Assembly

To create or edit logical materials of an existing Assembly, press "**Mt**" button in the "Variable" window (see 6.22, page 283). Define the list of logical material in the dialog window, see 6.27. You can see in the window a name and a description of the logical material and a code and a name sheet material compared with it.

To define logical materials of Assembly by default, use **Service >> Settings >> Assembly**, area "Logical materials". Every new Assembly will include logical materials defined here.
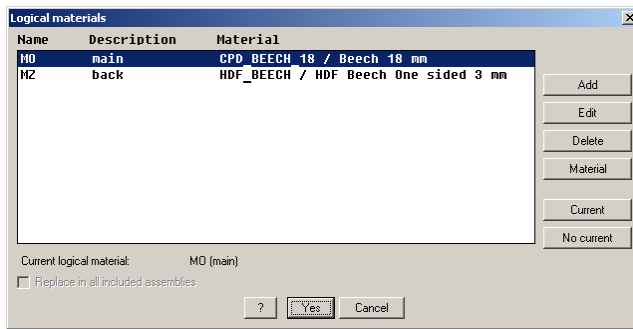
Fig. 6.27 Defining of the selected Assembly logical materials

Add

Adds a new logical material (a new variable)

Edit

Edits the selected logical material (the name and the description may be edited). Note, you can't edit the variable name if it has been utilized (in any expression, or in any object parameter, or in any object reference). You will get alarm message in this case.

Delete

Deletes the selected variable. It's impossible to delete the variable if it has been utilized yet.

Material

Assign a new value of the logical material (compares variable with a new sheet material)

Current

Defines current logical material (see attention below).

No current

Sets a mode "without current"

Attention! If an Assembly has a current logical material, this material will be automatically assigned to all new Parts of the Assembly. Note, the objects created with **Object >> Rectangular part >> Back** are exceptions. "MZ" logical material will automatically assigned these objects if "MZ" is defined in the Assembly. If there isn't "MZ" logical material in the Assembly, the back will assigned through the first material from the sheet material database.

6.2.4.3. Catalog of logical material

To synchronize logical materials with the same name in different objects, you should use a catalog of logical materials.

There is a catalog of logical material of the project and catalog of logical material by default. The first determine materials of the current project. The second fills in the catalog of logical material of new projects.

Attention! If you insert a typical model in the project from database and:

– it has logical materials,
– the current project catalog of logical materials includes some logical materials,
– the names of the typical model logical materials are equal to the name from project catalog
– the control "*After assembly insert apply logical material*" (**Service >> Settings >> Assembly**) is enabled.

that logical materials of the model will be assigned with the logical materials of current project catalog, see fig. *6.28*.
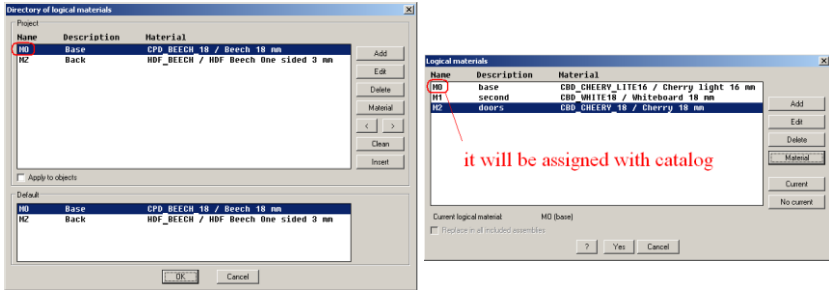
Fig. 6.28   Catalog of logical material of the current project (on the left) and logical materials of the typical model

Note. If you insert an assembly or a typical model in the current assembly, its logical material will be synchronized with logical materials of the current assembly. (The project catalog of logical materials not influences of.)

To edit project catalog of logical materials, execute **Service >> Catalogue of logical materials >> Current project**, see fig. *6.28*. This window consists of two parts. These are logical materials of the current project (it is the upper part) and logical materials by default. The window controls correspond to controls considered for fig. *6.26*, page 288.

The "catalog of logical materials by default" determines the list of logical materials in a new project. To edit it, use **Service >> Settings >> Catalogue of global variables >> By default**. The dialog window corresponds to window presented in fig. 6.27 page 290.

Note. If you open an existing project it will have logical materials determined before.

6.2.5. Object variables

Object variables are created and edited in the "Variables" dialog window. It was considered in detail in section 6.2.2, page 283.

All object variables split into two groups. These are system variables and user variables. The system variables are generated automatically with pro-

gram. You can't delete them and move them in the "Variables" window. They are marked with "*" and "$" signs, see fig. 6.22, page 283. (Note, "*" sign is used to mark Assembly overall dimensions variables.) The user variables are created and edited by user.

There are four types of variables in the program:
- variable-parameter,
- variable-formula,
- variable-distance,
- variable-logical material.

Attention! A value of any variable can be limited with special expression in the "Note" field of the "Variable" dialog window, see fig. *6.29*, page 293.

Special expressions in the "Note" field.

(= 300 400 500)	The variable will be set the nearest value from the number series.
(< 300 400 500)	The variable will be set the nearest less value from the number series.
(> 300 400 500)	The variable will be set the nearest greater value from the number series.
([] 300 500) ([] 300 500 600 800)	The variable will be set any value from the predefined range. If the calculated value of the variable not belongs to the range, the variable will be set the nearest limit value. In the last example the variable will be set value from the two ranges: from 300 to 500 and from 600 to 800. In other words the expression may include some ranges. If you lost one of range limit, that range will be ignored (The program always finds pair of number to get a range).

Note. The number series have to look like this (=□N1□N2□N3□N4□N5), where N is a number, "□" is a space symbol.
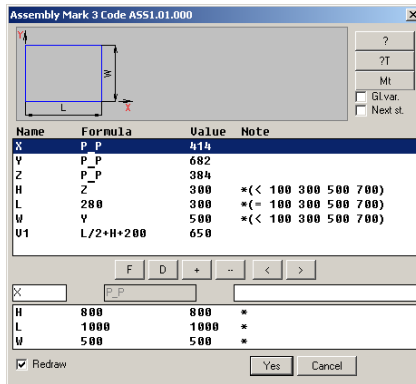
Fig. 6.29   Example for special expression in the "Note" field.

The value of "H" variable is equal to "Z" variable (Z=384). But the value of "H" variable is rounded down with the number series. Therefore the value of "H" is 300.

The value of "L" variable assigned as 280 (see the formula column). But its value is rounded with the number series. Therefore the value of "L" is 300.

The value of "W" is defined with Y variable-distance (Y=682) and rounded down with number series. Therefore the value of "W" is 500.

6.2.5.1. Variable-parameter

A value of the variable defined with number, for example see fig. *6.29*, page 293, L variable (see formula and value columns).

6.2.5.2. Variable-formula

A value of the variable defined with formula. Operands of the formula may be a number, a variable-parameter, a variable-logical material, a variable-distance and a variable-formula. Any variables used in the formula have to be defined in the variable list above the variable which they defined, see fig. *6.29*, page 293, "V1" variable. V1=L/2+H+200, "L" and "H" are variables defined "V1" and situated in the variables list above "V1". The list of allowed mathematical and trigonometric operations is presented in the table below.

+	add
-	subtract
*	multi
/	divide
( )	brackets
SIN(angle value)	Returns the sine of an angle as a real number

	expressed in degrees.
COS(angle value)	Returns the cosine of an angle as a real number expressed in degrees.
TANG(angle value)	Returns the tangent of an angle as a real number expressed in degrees.
ATANG(value)	Returns the arctangent of a number in degrees
SQR(value)	Returns the square root of a number as a real number
SQRT(value)	Returns the square root of a number as a real number
ABS(value)	Returns the absolute value of a number
ROUND(value)	Returns the rounded number of a number as an integer number
TRUNC(value)	Truncates *number* to the nearest integer by discarding the fractional portion.
A1 MAX A2	Returns the largest of the given numbers. Note A1, A2 may be numbers or formulas.
A1 MIN A2	Returns the smallest of the given numbers or formulas result.

You can see an example of the mathematical expression below:
ROUND(X/Y)*SQRT(ABS(V1-X))

It is possible to use some LISP function in the 3D-Constructor. LISP expressions must be framed in [ ] brackets! The next function may be used:

+	add
-	subtract
*	multi
/	divide
( )	Brackets
SIN angle	Returns the sine of an angle as a real number expressed in radians.
COS angle	Returns the cosine of an angle as a real number expressed in radians.
SQRT	Returns the square root of a number as a real number
FIX	Truncates *number* to the nearest integer by discarding the fractional portion.
IF	Conditionally evaluates expressions

=	Equal to
<	Less than
>	Greater than
<=	Less than or equal to
>=	Greater than or equal to
/=	Not equal to
OR	Returns the logical OR of a list of expressions
AND	Returns the logical AND of the supplied arguments

Attention! The logical functions will return 1 (true) or -1 (false) if other are not defined. See examples below.

[-(+ 1 1) 5]– The result of expression is "-3". This Lisp expression are equal to (1+1-5).

[IF (<= W 400) 10]– The result will be "10" if "W" is less than or equal to 400. In any case the result will be "-1".

[IF (= W 400) 10 20]– The result will be "10" if W is equal to "400". In any case the result will be "20".

[IF (AND (= W 400) (= L 1000)) 10]– The result will be "10" if W=400 and L=1000. In any case the result will be "-1".

[IF (= N 1) 300 (IF (= N 2) 500 (IF (= N 3) 700))]– If N=1, the result will be "300", If N=2, the result will be "500", If N=3, the result will be "700". If N not equal to 1 or 2 or 3, the result will be "-1".

[IF (OR (= V2 18) (= V2 19)) 4]– If V2=18 or V2=19, the result will be "4" otherwise the result will be "-1".

[< N1 X N2]– If N1<X<N2, the result will be "1" otherwise "-1".

Attention! You can use usual expression or LISP expression in one place (in one "formula" field of the "Variable" dialog window, in the one parameter of the object or object reference). In other words, you mustn't mix LISP function and mathematical operations in one place, it is prohibited.

6.2.5.3. Variable-distance

Value of the variable is defined with distance between objects primitives in 3D-model. If you change 3D-model, a value of that variable will be automatically updated.

The variable may be defined with:
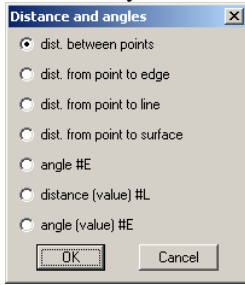
- a distance between two points;
- a distance from a point to an edge;
- a distance from a point to a line;
- a distance from a point to a surface;
- an angle;
- a distance (value);
- an angle (value).

Fig. 6.30   Distance definition window

The angle is defined as the angle between two lines specified with three points.

The distance (value) and the angle (value) only copy current value from 3D-model. They will not be updated if you change 3D-model then.

Points, edges, surfaces, which define a variable-distance, have to belong to objects created before the object being edited.

Note. To highlight objects primitives, which define a variable-distance, you should select the variable in the "Variables" dialog window and press "?T" button, see fig. *6.22*, page 283.

Note. To convert a variable-distance into the variable-formula, select the variable in the "Variables" dialog window and press "**F**" button, see fig. *6.22*, page 283. To define a value of the variable-distance or convert a varia-ble-formula into the variable-distance, select the variable and press "**D**" button. The "Variable" window will be closed and you will be asked for distance. A new value will be gotten from 3D-model and updated in the "Variables" win-dow.

List of propts

*ODO (box) + / *menu #V / Distance between points - first point*: Specify the first point of the distance.

*ODO (box) + / *change point #N / Second point*: Specify the second point of the distance.

Menu

Assigns the distance definition, see fig. *6.30*, page 296.

ODO (box)

Determines what point will be taken. It may be object point or box point (detail about Parts box see in section 3.1.1, page 30). To specify point of an object, must be displayed "*ODO (box) +*". To specify box point of an object, must be displayed "*ODO (box) -*".

6.2.6. Built-in variables

Some objects have built-in variables. Their value are calculated with 3D-Constructor and not influenced by user.

The name of built-in variables have prefix "!". It is prohibited to use this symbol in the variables created by user.

List of built-in variables
Parts variables

!X– Part length (mm). It is a Part size along X-axis of the object coordinate system.

!Y– Part width (mm). It is a Part size along Y-axis of the object CS.

!Z– Part thickness (mm). It is a Part size along Z-axis of the object CS

Note. The variables mentioned above may be used in object parameters, reference parameters and in the "Variables" dialog window.

!P– perimeter (mm)

!PM– perimeter (m)

!SM– one side Part area (m).

!SGBM– one side area of the circumscribed rectangle (m). The rectangle is circumscribed concerning object CS.

!LK– total length of edging (mm).

!LKM– total length of edging (m).

Bent Sheet variables

!LK– total length of edging (mm).

!LKM– total length of edging (m).

!LP– path length (mm)

Profile variables

!SM1Y, !SM1Z, !SM2Y, !SM2Z, !SM3Y, !SM3Z– offset concerning profile path (mm). Detail information see in section 3.3.1.2, page 56.

The Part and Bent Sheet built-in variables may be used to define an object or project composition.

To determine an object composition, execute **Edit >> Properties**, select the object, open the object composition, see fig. *6.31*. Note the quantity of screws is given through the formula (one screw every 99 mm).
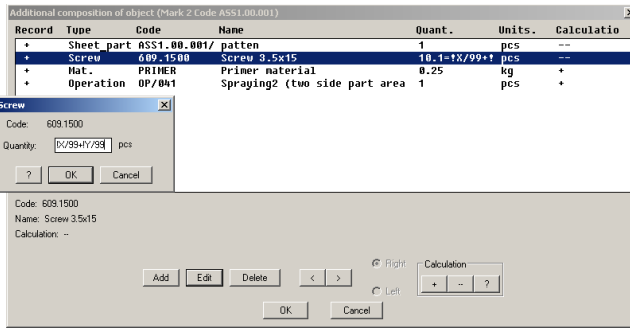
Fig. 6.31 Object composition window. The quantity of "Screws" is calculated through the formula.

Look at the third composition string (see fig. *6.32*). The primer material quantity is defined with one of the typical calculation. The "Quantity" column assigns the primer material quantity for one square meter. The typical calculation calculates the total Part area. The primer material quantity in the quotation document will be calculated as (0.25 x Part area).

Note. To create a typical calculation, execute **Database >> Paid Operations (Jobs)**, select "Typical calculations", add a calculation and enter the calculation parameters.
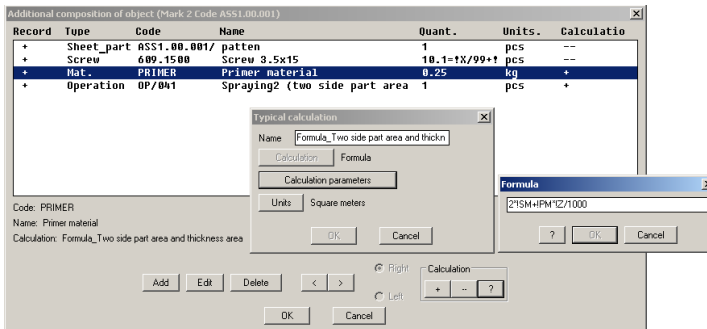
Fig. 6.32 Object composition window. The quantity of "Primer material" is calculated through the typical calculation.

To determine the current project composition, execute **Service >> Catalogue of project composition >> Current project**. The dialog window is equal considered above.

Additional information about composition dialog window see in section 5.6.13.2, page 228.

The built-in variables may be used also into the variable reference, more details see in section 6.2.8, page 301.

6.2.7. Variables with fixed names

Some of the variables entered by the user have the fixed names. Appearance in the object variables or object parameters such names can cause execution of any actions. Moreover, some program algorithms are oriented on use of such names. If the user tries to use these variables not for its intended purpose, the results can be difficult to explain.

6.2.7.1. Insertion of a typical model in the project

There are three concepts which are used to insert typical model.
- Opening (variables X,Y,Z)
- Object box (overall dimensions sizes, variables Lb, Wb, SX1, SX2, SY1, SY2)
- Real object size (variables L, W, H)

Opening is a rectangular parallelepiped in which typical model is built-in. Its dimensions are usually determined through X, Y, Z variables in the model variable list. If you insert a Part or a Door, it is present only two variables X and Y.

The Part and Door-assembly may have an object box. The box is invisible frame of object. The object contour may be situated inside or outside the box. The box points help you to reference the object comfortable, more details see in section 3.1.1, page 30. If the object has box, its variable list includes next variables:

Lb   - box length,

Wb - box width,

SX1 - the 1-st offset along object CS X-axis (the nearest edge to CS origin)

SX2 - the 2-nd offset along object CS X-axis

SY1 - the first offset along object CS Y-axis (the nearest edge to CS origin)

SY2 - the second offset along object CS Y-axis

When you insert a typical model in the opening or execute **Edit >> Overall dimensions**, the program measures distances and automatically enters them in the variables in the following order:
- If the object variables list includes "X", "Y", "Z" variables, the distance will be filled into them.
- If "X", "Y", "Z" are absent but there is "Lb", "Wb", "Hb" variables, the distance will be filled into box variables.
- If "X", "Y", "Z" and "Lb", "Wb", "Hb" are absent, the variables relevant the measured distances will be created and entered into the model parameters.

If you define object size

- along X-axis of the object CS, will be created "X" variable;
- along Y-axis of the object CS, will be created "Y" variable;
- along Z-axis of the object CS, will be created "Z" variable

"X", "Y", "Z" variables will entered into "L", "W", "H" variables. If "L", "W", "H" are absent in the model, "X", "Y", "Z" will be entered in the variables marked with "*" symbol. "Z" will be entered in the first variable marked with "*" in the variable list; "X"– in the second variable; "Y"– in the third variable.

When you insert a Typical Model or a Drawer one of the methods "By points", "By points (face)", "By parts", "By door", "By door with limitations", it will be automatically created "X", "Y", "Z" variables-distance into model and entered into relevant "L", "W", "H" variables or variables marked with "*" symbol.

When you insert a Door one of the methods "By part-cabinet walls (EXT.)", "By part-cabinet walls (INT.)", "By point selection", "LB" and "WB" variables will be automatically converted into variable-distance. "SX1", "SX2", "SY1", "SY2" variables will be automatically filled in with relevant gaps (To determine door gaps by default, execute **Service >> Settings >> Door settings**). More details about Door insertion see in section 3.7.6, page 101.

Note. If you would like to remain an expression or a value defined into the variable, you should use "_" prefix to lock the variable. For example to lock "H" variable, name it as "_H".

Note. You also may influence to insertion result through the global variables. For example, if "W" variable is referenced to one of the global variables and a control "After model insert renew global variables" (**Service >> Settings >> General settings**) is enabled, the "W" variable will be defined with global variable, see fig. *6.33*
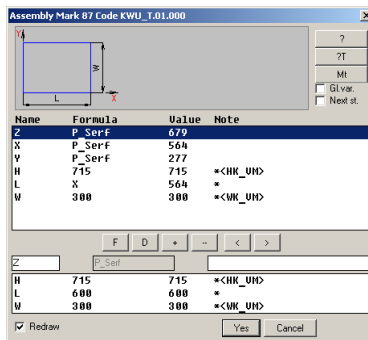

Fig. 6.33 The "Y" and "Z" variables insertion into "W" and "H" variables was suppressed with global variables

Note. When you create a model, you should take into account the rules of inheritance in case of the model replacements, more details see in section 5.19, стр. 252.

6.2.7.2. Fastener insertion

There are some special variables for the Fastener.

"KM_L" is a special variable, which is equal to the edge length of the base object. The fastener fixes two objects. One of them is a base object. You specify its edge to insert the fastener. More details see in section 3.8.1.1, page 125.

"KM_A" variable exists only for hinges. It is an angle (in degrees) between a Door and a Unit side which are fixed with the hinge.

"KM_Z1", "KM_Z2" are thickness of the base fixed Part and the second fixed Part.

6.2.8. Reference to variables of other objects

You can assign a value of the object variable through the variable of any earlier created object. To do it, you should define a mark of object to which you reference and a name of the variable, see example in fig. *6.34*.
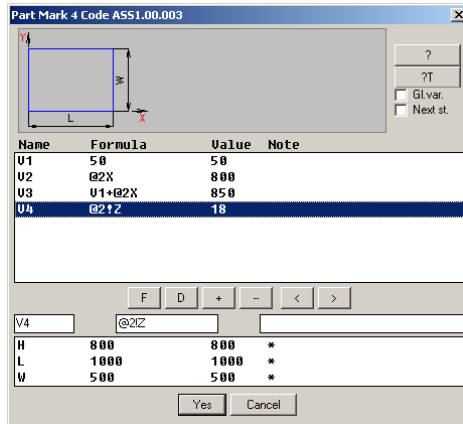

Fig. 6.34   Reference to variable of the earlier created object.

"@" defines the reference beginning.

"@2"	It is a reference to object with mark "2".
"@2X"	It is a reference to "X" variable of the object with mark "2"
"V1+@2X"	It is an expression one of which members is the reference to a variable.
"@2!Z"	It is a reference to "!Z" built-in variable of the object with mark "2", see section 6.2.6, page 296.