Pro Art & Pro Lace

Reference Manual
Release 2.0

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Starting with ProArt/ProLace

This chapter contains information about the installation and the basic concepts of the ProArt/ProLace programs

Introduction

ProArt/ProLace provides an ideal aid for the embroidery designer and/or puncher. It helps you to create technical embroidery designs. With ProLace you can also punch directly on the designs.

Installation

System requirements

The system requirements are as follows:

- Pentium IV based Windows2000 or WindowsXP computer
- minimum 512MB RAM
- 10GB free hard-disk space
- CD-ROM drive
- graphic screen with a minimum resolution of 1024x686 dots
- Mouse or WinTab digitizer as input device
- plotter or printer
- Scanner

The program is delivered on CD-ROM.

How to install ProArt / ProLace

- Insert the **ProArt/ProLace** CD and run the **SETUP** program. The setup program will guide you through the installation steps. After the installation you will get a start button on your desktop.
- Plug the the **Rainbow Sentinel SuperPro lock** on the printer port. The lock is included in the **ProArt/ProLace** package.
- Install the SuperPro driver before you start **ProArt/ProLace**. (See following section)
- Next you can change the user information, which appears in the header of each design. Therefore open the file **C:\Icad\User.ini** with the **Notepad** and modify the entries (e.g. **AMD_USER=AllCAD Technologies Ltd.**).

  ```
  Example:
  amd_enlarger = Loi Sakaeo
  amd_puncher = Günter Heinzle
  amd_user = Embroidery Connection Co. Ltd.
  amd_user2 = 442/93 M.5, T.Naklua, Chonburi 20150
  amd_user3 = email: office@embroideryconnection.net
  amd_copyr = Embroidery Connection Co. Ltd.
  ```

- Now you can start **ProArt/ProLace**. If you install **ProArt/ProLace** for the first time a dialog-box opens where you must enter the user data and the **ProArt/ProLace Serial Number**. You can find the serial number on the CD.

How to install the Rainbow Sentinel SuperPro lock for Windows2000® or WindowsXP®

Before you can start the software on a Windows2000® or WindowsXP® system, you must install the driver for the **Rainbow Sentinel SuperPro** lock.

- You can find the driver in the directory **\Tools\Ntlock** of the **ProArt/ProLace CD**.

- Run the command **setupx86.exe** to start the installation program.

- Call up **Functions / Install Sentinel Driver**.

- After having installed the driver you must restart the system.

Note: To be capable to install the Sentinel driver, you must have Administrator rights!

Directory structure and file description

In this section the directory structure of ProArt/ProLace is described.

ProArt/ProLace directory structure

The base directory for all ProArt/ProLace sub-directories is the program directory **ICAD**. The **BIB** subdirectory contains the symbols and definition tables.

```
C:\ICAD Program directory
    \MNU Menus and Bitmaps for the toolbars
    \BIB
        \DATA           Data files
        \S              System block library
        \SLD            System and user slides
        \U              User blocks library
        BIB.DIR         Identification files
        DATA.DIR
        STD.DIR
    \MNU                ProArt/ProLace menu and bitmaps
    \FONTS              Text styles
    \PATTERNS           Filling patterns
```

Configuration files in the program directory ICAD

File	Description
CMDLINE CSV	Context menu prompts
DIALOGS CSV	Dialog-box prompts
MESSAGES CSV	Other system messages
EDS INI	Definition file for system
ICAD INI	variables
USER INI	Definition file for system
DEFAULT INI	variables
	User information
	Definition of default values

The DATA-directory

File	Description
BORPROG CSV	Bore program definition file
FRWPROG CSV	CRC program definition file
SYMLIB CSV	Symbol library contents

Symbol library

The **system library** is filed in the subdirectory **S** of the library path. All symbols

start with the characters **AMD** and have the extension **DWG** (e.g. AMDCIR.DWG).

The **user library** is filed in the subdirectory **U**, the system slide libraries and the user slides are filed in subdirectory **SLD**.

Attention: User-blocks must not start with **AMD**!

Files and file types

To memorize/save a file on a floppy-disk or hard disk it must be given a filename. A filename name consists of the name and the extension, which are separated by a full stop, e.g. EDS.DLL or COMPLEX.SHX.

ProArt/ProLace uses for system and user files several file extensions, which define the type of file. Below you can find a list of **ProArt/ProLace** file extension in alphabetical order:

File	Description
.BAK	Safety copy of a design
.CFG	Configuration file
.DWG	Drawing file
.DLL	Program files
.LIN	Line type definitions
.LSP	Lisp program files
.MNU	Menu files
.PAT	Fill pattern definitions
.SHX	Character set file
.SV$	Periodically saved drawing file
.SLD	Slide file
.SLB	Slide library
.INI	Initialization files
.CSV	Data tables

Data integrity concept

When you start a new design a file with the chosen name is being created in the file extension **.DWG**, i.e. **91356.DWG**.
If you edit this design a copy of the design under the same file name, however, with the file extension **.BAK** will be filed prior to starting the design editor.

During the work process the design data is memorized on a regular basis on the hard disk. The time interval, defined with the **File/Options…** command, can be determined in the configuration.

Should the design file **name.DWG** have been erased for any reason you may search first for the file i.e. **name.SV$** and rename it to **name.DWG**. Should this file also not be available you can rename the file **name.BAK** to **name.DWG** and start work with this old version.

In order to save space on the hard disk you should periodically clean the **BAK** files from the hard disk.

How to start ProArt/ProLace

How to start with a new design

The design system is being started with a double-clicking onto start-button on the Windows Desktop.
The system is ready when the following information appears on the command prompt area of the screen:

```
AllCAD Technologies Ltd., ProArt 2.0, Build 2004-01-01
Command:
```

Now specify a name for the design. Therefore call up the command **File / Save as...**.

Next call up the command **SETUP**. A dialog-box opens where you can specify the repeat, the height of the design, the standard stitch density, the yarn etc. After leaving the dialog-box, the basic design with the header, repeat and the repeat grid is being constructed on the screen.

How to start with an existing design

The design system is being started with a double-clicking onto start-button on the Windows Desktop.
The system is ready when the following information appears on the command prompt area of the screen:

```
AllCAD Technologies Ltd., ProArt 2.0, Build 2004-01-01
Command:
```

Now call up the command **File / Open...** and choose the file, which you want to edit. The selected file is being loaded and you can continue designing.

The ProArt/ProLace screen

When working in the design editor, hence when you are working on a new design or editing one, the video areas will be defined as follows:

	Name	Description
A	**Command prompt area**	consisting of three lines
B	**Status line**	on the button of the screen
C	**Pull down menu**	on the topside of the screen
D	**Toolbars**	user definable position. Normally below the pull down menu
E	**Context menu**	appears on right click
F	**Graphics area**	in the center
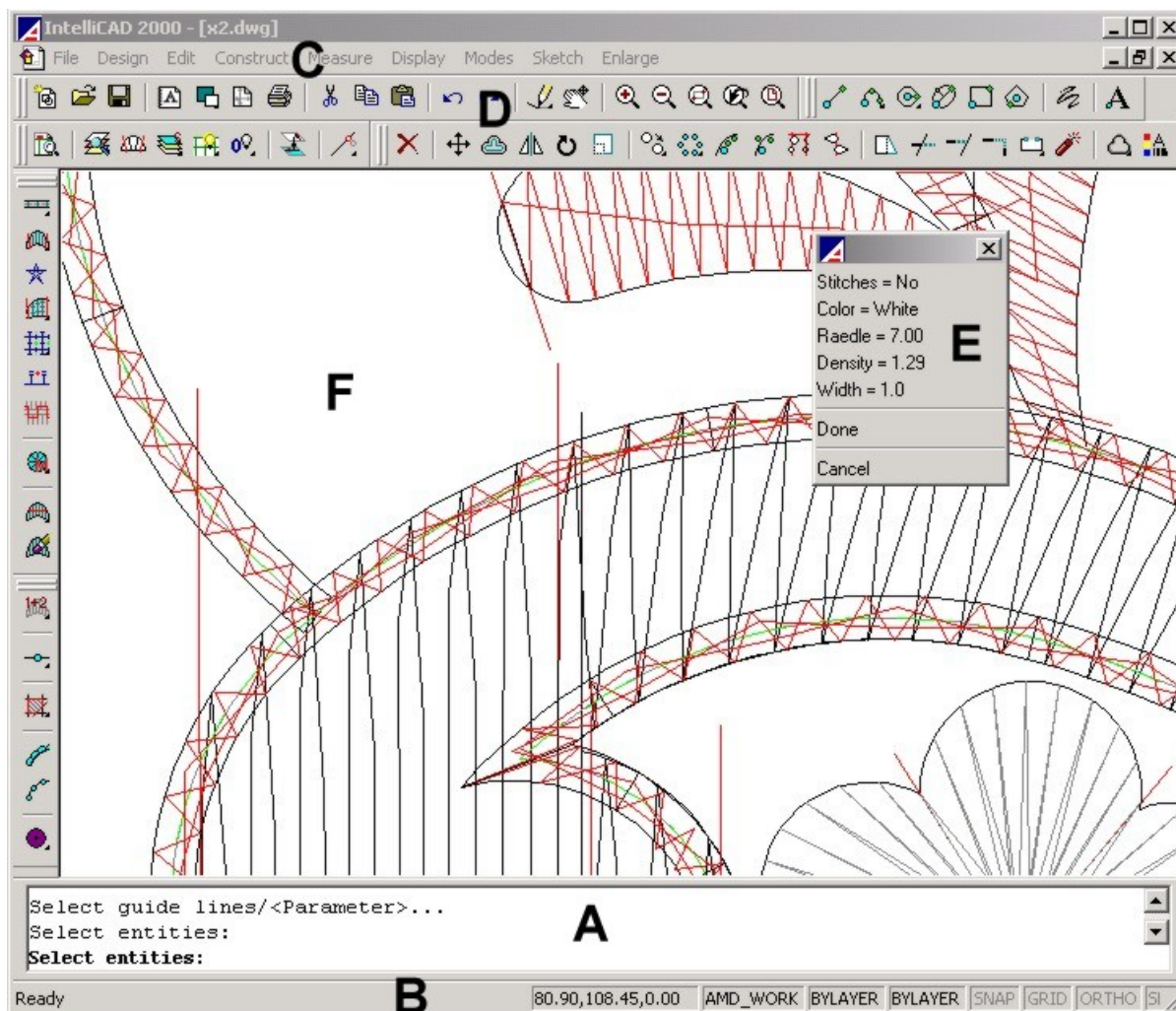
- On the **command prompt area** the last three lines of the command dialog will be displayed. It can be switched on and off with the **F3-key**.

- The **status line** shows the actual layer, the active design helps and the coordinate of the crosshair cursor.

- The **pull down menu** serves as command input. If you click on of the pull-down menu items the corresponding menu will be scrolled down and you then can select the desired command.

- The **Toolbars** serve as command input. To execute a command move the crosshair onto the corresponding button and click with the pick button of the mouse.

- The **context menu** appears on right click with the mouse and it offers normally command options.

- In the center of the screen (**graphics area**) the actual part of the design is being displayed. All changes are immediately shown. A crosshair cursor, controlled by the mouse, shows the position of the input.

Design entities

A large part of designing work consists of placing entities in the design. Entities are predefined design elements which can be placed in the design with the relevant command.

The system makes available the following entities:

Object type	Description
Line	A line is defined by two points
Circle	To construct a circle there are different methods available.
Point	A point defines a coordinate and is displayed as a small circle.
Polyline	A polyline is a string of lines composed of straight lines or arcs. Special forms of polylines are ellipses and regular polygons. A polyline will always be considered as one element.
Text	Text can appear in various forms, various sizes and directions.
Block	Blocks are entities composed of other random elements.

What are layers

Different parts of a design can be allocated to different layers.

A layer is similar to a transparent sheet of foil. Each singular foil contains certain design elements. Different foils put on top of each other result in the complete design. This technique allows to make visible or not visible certain parts of the design, such as the header.

Each layer has its color. Therefore if the layer is being changed the actual color will change and the relevant elements belonging to each layer can easily be checked.
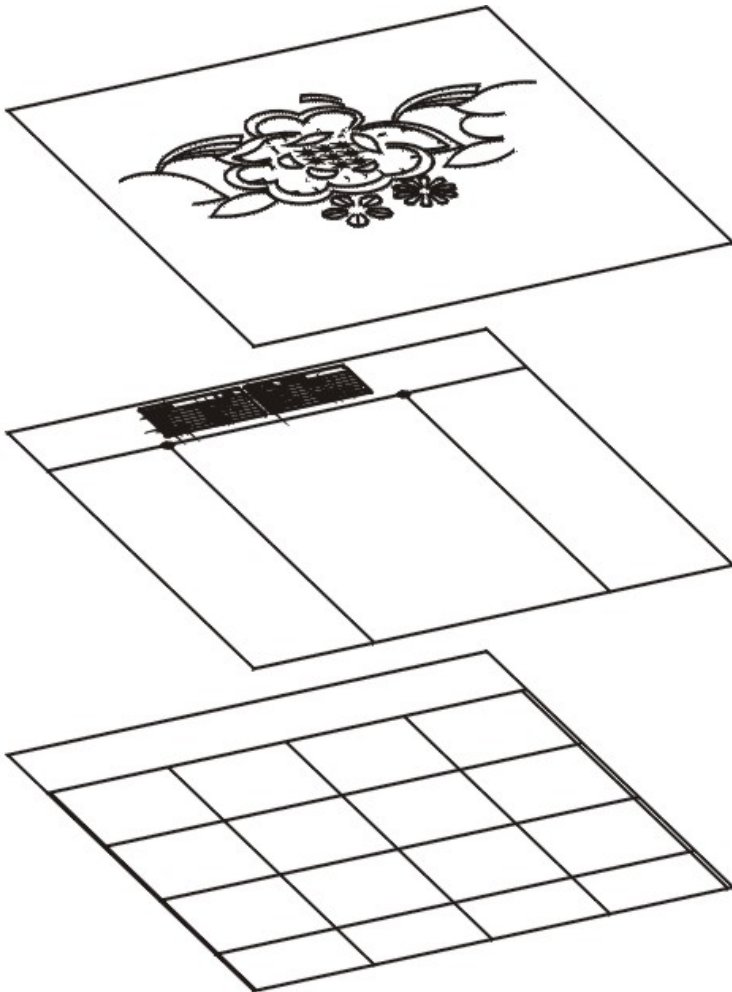
Layers are predefined in the system and are distinguished as follows:

- **User layers** used by the designer to draft and construct the design, and

- **System layers** used by the design system by itself for automatic embroidery functions.

Attention: You may define your own layers. However, existing layers may not be canceled or given a new name because a lot of functions depend on these layers.

Colors

The system offers 255 colors. The colors are numbered from 1 to 255. The first 8 colors have standard names.

How to call up commands

A Command can be entered as follows:

- using the keyboard
- using the pull down menu
- using the toolbars
- using the button menu (on the digitizer mouse)
- using the digitizer menu

Command input via the keyboard

Write the desired command and press <**Return**>.

Some commands are defined on the function keys of the keyboard.

F1 - Request Help

F2 - Change over from text to graphic screen

F3 - On/Off of command prompt area

F4 - Change direction of polyline

F5 - Snap nearest

F6 - Snap endpoint

F7 - Snap center

F8 - Ortho mode On/Off

F9 - Snap intersection

F10 - Snap mid point

F11 - Snap perpendicular

F12 - Snap tangent

Command input via the pull down menu

If you move the crosshair cursor into the area of the status line on the top area of the screen, different menu lines with a variety of menu titles are shown. If you move the cross hair cursor on to a menu title within the menu line, the corresponding name will be illuminated. If you press the pick button on the mouse while a menu title is illuminated, the menu will be pulled down below.
Now you can choose within the pull down menu and the corresponding function which is illuminated can be selected by pressing the pick button of the mouse.

Command input via the toolbar buttons

If you move the crosshair cursor on a button of the screen menu, a short description of the function is displayed. To activate the function, press the pick button on the mouse.
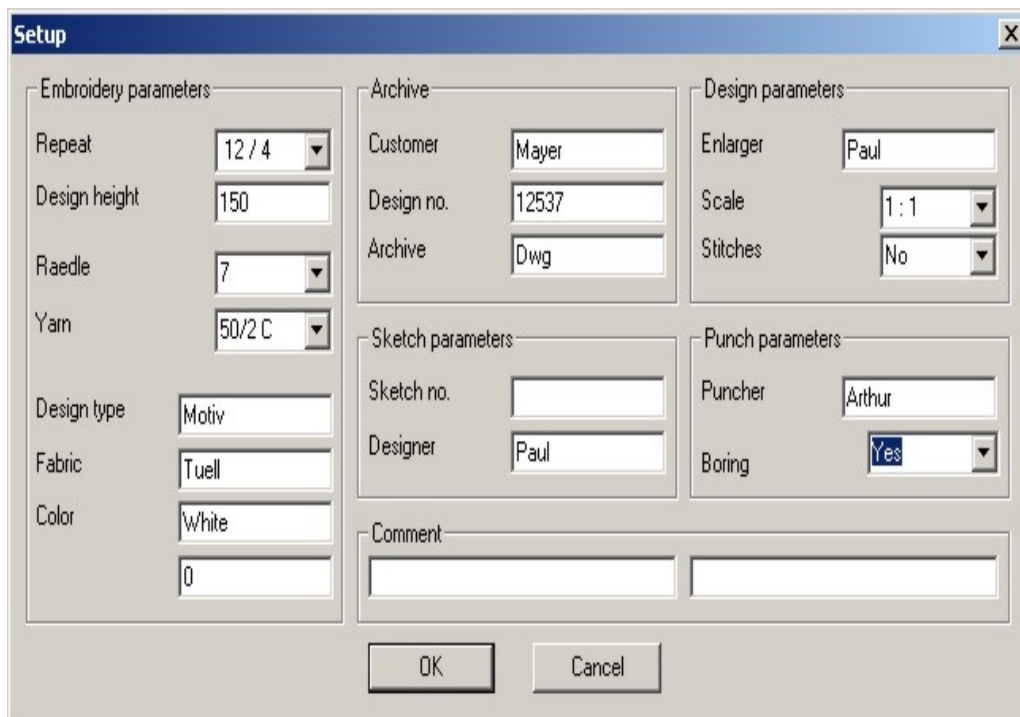
Command input via tablet menu

If a digitizer is connected with the system, the commands can be chosen from the tablet menu. A tablet menu is a printed template occupying a defined area of the digitizer. To select a command, place the crosshair cursor on the desired command on the menu area and press the pick button.

Dialog-box

The setting of various parameters can be carried out in so-called dialog-box. The dialog-box show status, value or name of a size and can be changed by picking the relevant area.

Each dialog-box has a field called "**OK**" which serves to carry out the correction and an area named "**Cancel**" which will end the dialog without changes.

Units and scale

This design system uses millimeter as measuring unit. The scale is 1:1. This means all measurements must be indicated in the original size of the embroidery.
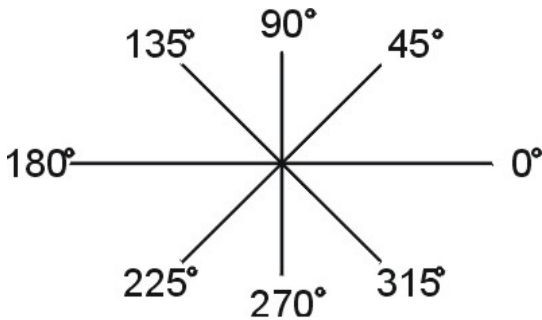
How to input data

Most commands need additional parameters, such as points, angles and distances

Input of angles

The input of angles is effected as follows:

- Zero degrees is towards east

- and the angles are defined anti-clockwise.

Input of angles using the keyboard

If the system asks for an angle the input can be carried out using the keyboard.

```
Command: INSERT
Block name (or ?): *blume1
Insertion point: 100,50
Scale factor <1>: <Ret>
Rotation angle <0>: 45
```

Angle input using two points

In most cases angles can be defined with two points. The system in this case calculates the defined angle and shows the calculated value.
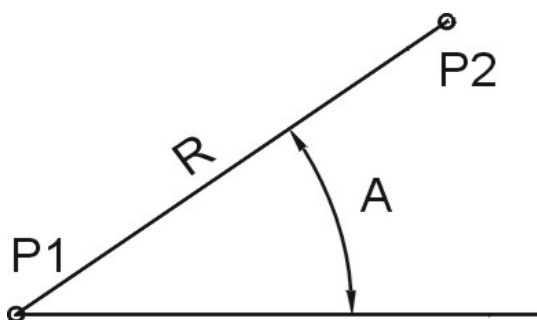
Displacements

Commands such as **MOVE, COPY** etc. are defined with displacement distances.

Using the keyboard

The displacement can be defined with a pair of coordinates, whereby x and y values

correspond to displacement of the x and y direction. If the system asks for a second point after input of this coordinate pair simply press **<Return>**.

```
Command: MOVE
Select entities: select
Basepoint of displacement: 10,0
Second point of displacement: <Ret>
```

Angle definition with two points

Displacement can also be defined with two points. The system thereby calculates the shown x and y distance and uses the corresponding result.

Point input using the keyboard

In order to localize a defined point in a drawing the CAD programs underlay an absolute Cartesian coordinate to the design. A pair of coordinate consisting of an x and y coordinate defines the position of each point in a design. The position is always being calculated from the origin of the coordinate system (0,0) which is the lower left-hand corner of the design.

Whenever you want to give the exact value of a coordinate point you may use the command input via the keyboard. Several possibilities are available.

Input with absolute Cartesian coordinates.

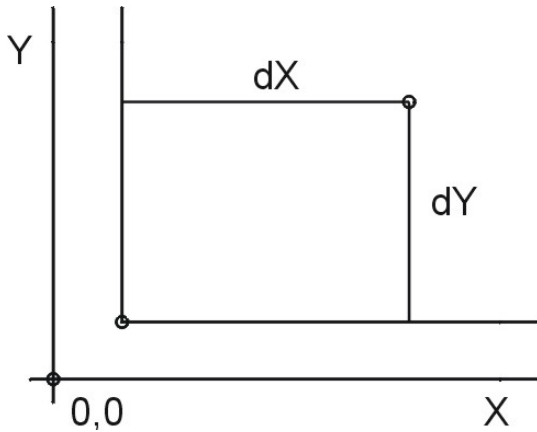A point will be defined by the distance in x and y direction starting from the origin of the coordinate system.
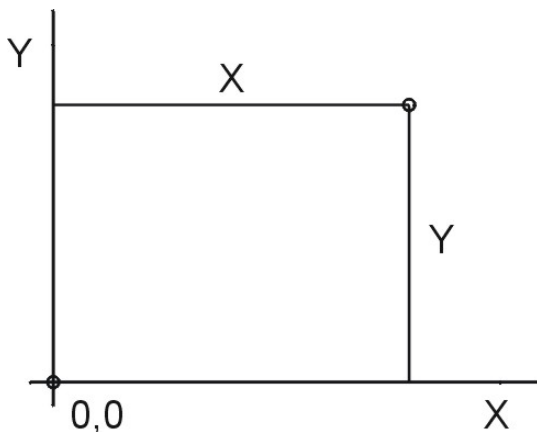

```
Command: LINE
From point: 4.2,-3.1
To point: 4.2,-10
```

Input in relative Cartesian coordinates.

A point will be defined using the distance of the x and y direction in relation to the last point. These two values are characterized with a @.
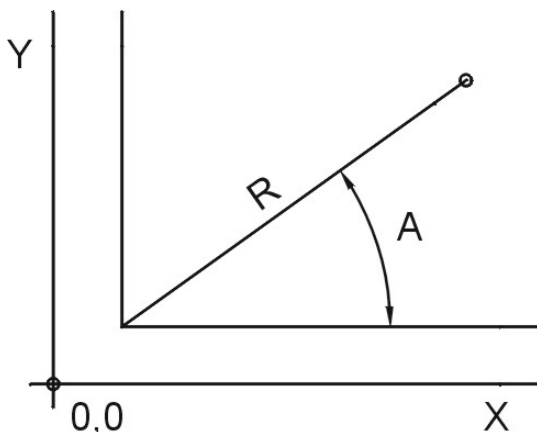
```
Command: LINE
From point: 4.2,-3.1
To point: @0,-6.9
```

Input using relative polar coordinates.

A point is being defined with the distance and the angle from the last point. An **@** character must be put in front of the command. Distance and angle are divided with the **<** character.


```
Command: LINE
From point: 4.2,-3.1
To point: @6.9<45
```

Using the point to point method

The input of coordinates of point can also be carried out by positioning the crosshair cursor with the mouse. To make a positioning easier several design helps are at your disposal.

- coordinate display in the status line
- visible dot grid (dot grid with definable distances)
- invisible snap grid (the crosshair cursor only locks with defined snap points)
- orthogonal designing

How to work with object snap

The object snap (to localize a geometrical reference point) can be used when designing lines, editing or inserting blocks etc. This feature allows to snap points on geometrically defined points of already existing entities. For example if you want to pull a line right up to an arc or if you want to draw a line through the center of the circles, the object snap makes work a lot easier.
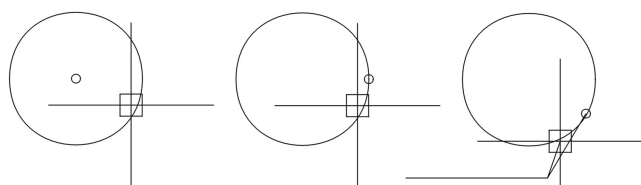
The object snap can be recalled from the **Snap Toolbar** or by activating the middle mouse button. After defining the object snap method the center of the cross hair cursor will be changed to a square. This square defines the search area used to search for a point on the object.
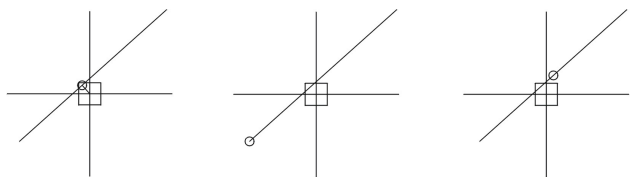
NEAr   ENDpoint   MIDpoint
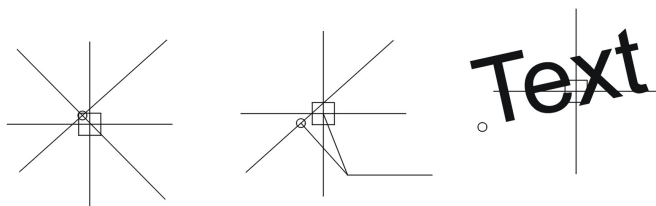
INTersect. PERpend.   INSert

CENter   QUAdrant   TANgente

The following object snap methods are at your disposal...

Icon	Key	Description
NEAr	F5	Snaps the point on a line, a circle or a polyline, which is nearest to the crosshair cursor.
ENDpoint	F6	Snaps the next end point of a line, a polyline.
CENter	F7	Snaps the center of a circle, an arc or a polyarc. The object snap square must be placed on a visible part of the circle.

INTersect or **PLA**nview	**F9**	Snaps the intersection point of lines, circles, arcs or polylines. Arcs and circles, belonging to blocks, are not recognized.
MIDpoint	**F10**	Snaps the middle of a line or a polyline segment.
PERpend	**F11**	Snaps a point on a line, a circle, or a polyline, which is perpendicularly to the last defined point.
TANgent	**F12**	Snaps the point on a circle, an arc or a polyarc, which forms a tangent with the last defined point.
QUAdrant		Snaps the nearest quadrant of a circle or a polyarc.
INSert		Snaps the insertion point of texts or blocks.
NODe		Snaps a point.

How to select entities

With most editing- as well as with some design- and block commands, design elements must be chosen for manipulation. Selected entities are highlighted by the system. With the following dialog the system asks you to select entities, which must be erased:
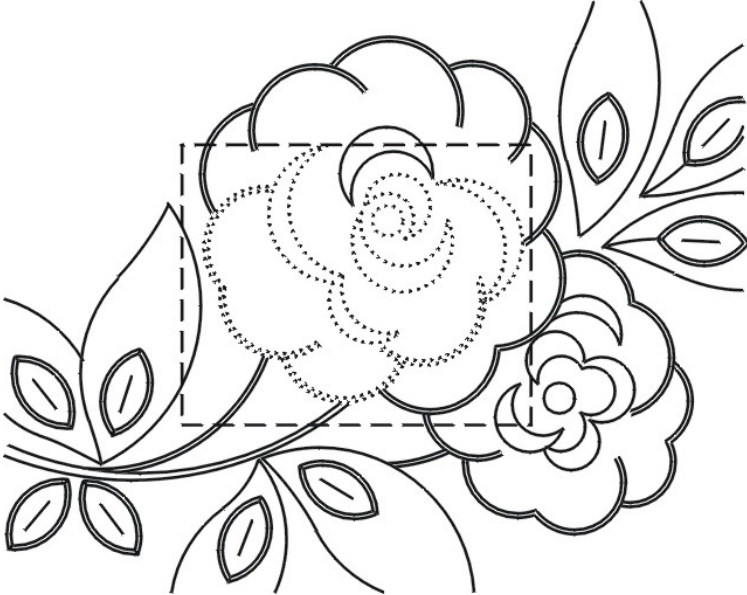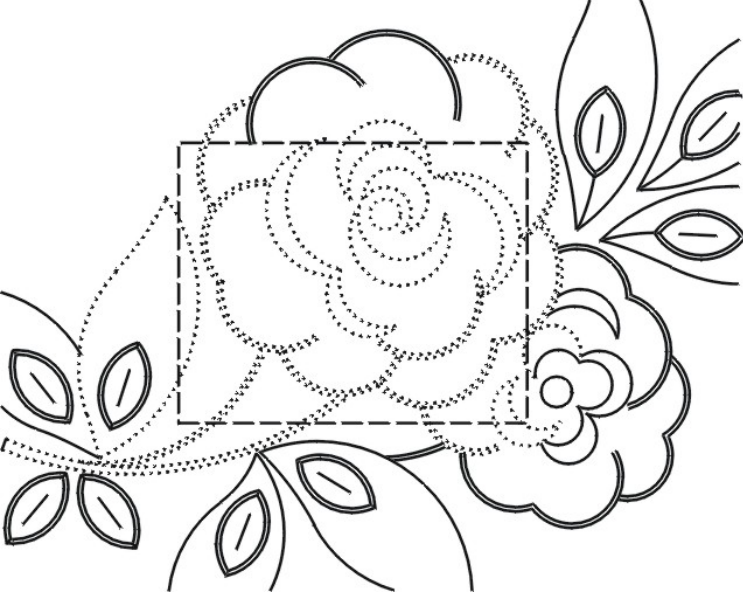
```
Command: ERASE
Select entities:
```

The crosshair cursor is being replaced by the pick-box, which can also be positioned with the mouse. From the upcoming menu you may choose the desired entity selection method.

Following possibilities of entity selection are at your disposal:

Mode	Description
Point input	The entity within the pick box is joining the selection set.
Select all	All entitys in the drawing are selected.
Add to set	With Add you can switch back from the remove mode to the add mode. Selected elements will then be added to the selection.
Subtract from set	If Subtract from set is used, newly selected elements are erased from the selection.
Last entity	Defines the last designed entity for the selection set.
Previous selection	The same selection set as with the previous entity selection is being taken up again.
Window-Inside Window-Polygon Window-Circle	Define a rectangle (**Inside**), a **Polygon** or a **Circle** and all entities, fully placed within the area, are selected.

Crossing-Window Crossing-Polygon Crossing-Circle	Same technique as Window, however all entities either fully or partially within the area are selected.
Outside-Window Outside-Polygon Outside-Circle	Same technique as Window, however all entities fully outside the area are selected.
Fence	With this option you define a cutting line. All elements touched by it are selected.
Undo	The last joint element is being erased from the selection set.

Note: If you pick an empty area with the pick box, a window is automatically created. If you pull the window to the right , the selection method window is activated if you pull it to your left, the selection method crossing is being activated.

If no further entity has to be added to the selection set, press **<Return>** or the space key. The selected action will then be carried out with the selected elements. If you want to cancel the entity selection or the active command, press **ESC.**

```
Command: ERASE
Select entities: W
First corner: point 1
Other corner: point 2
Select entities: <Ret>
```

Files, Plot, Scan, Display

This chapter describes functions to handle a design. All these functions are located in the Toolbar **Standard** or in the pull down menus **File** and **Display**.

The **Standard Toolbar** contains the following commands...

	Standard Toolbar
	Standard Toolbar
	[NEW - Create a new design](#)
	[OPEN - Open existing design](#)
	[QSAVE - Save a design to disk](#)
	[DRAWORDER - Change the draw order of image](#)
	[SETUP - Initialize a design](#)
	[FORMATS - Select paper format](#)
	[PLOT - Plot/Print a design](#)
	Cut to clipboard
	Copy to clipboard
	Past from clipboard
	[UNDO - Undo a command](#)
	[REDO - Recover the previously undone command](#)
	Flyout Redraw
	[PAN - Move the display](#)
	[Change display](#)
	Redraw
	[REDRAW - Clean the screen](#)
	[REGEN - Regenerate the screen](#)

Other commands in the pull down menu

The following commands cannot be selected from a toolbar. They must be called up from the pull down menu.

▤ [END - How to terminate ProArt/ProLace session](#)

▤ [SAVEAS - How to save design under another name](#)

See also...

▤ [How to initialize a new design](#)

▤ [ESC - How to interrupt command with ESC key](#)

NEW - How to create a new design

Toolbar: **Standard > New**

Menu: **File > New**

Keyboard: **NEW**

With the command **New** a new design is started. The design actually on the screen will be terminated, whereby the system will ask, whether you want to save the modifications or not.
Thereafter you have to give a name to the new design. The system will now start with a new design, this means you must now define the standard values of the design and the data for the header.
If the command line states **command**, the system is ready.

OPEN - How to open an existing design

Toolbar: **Standard** > **Open**

Menu: **File** > **Open**

Keyboard: **OPEN**

With the command **Open** you load an already existing design. The design you are
actually working on, will then be terminated whereby the system will ask, whether
you want to safe the modification or not.
Thereafter you will be asked the name for the other design and the system then will
load it. If the command line states **command**, the system is ready.

QSAVE - How to save a design to disk

Toolbar: **Standard** > **Save** 💾

Menu: **File** > **Save**

Keyboard: **QSAVE**

With the command **QSAVE**, the actual design will be filed. Should a technical error occur or the program be interrupted due to an operator error, you may continue the work at the point where the last save has been made.

SAVEAS - How to save a design under a new name

Menu: File > **Save As ...**

Keyboard: SAVEAS

With the command **Saveas**, the actual design is being filed.

After calling up the command, the file dialog-box opens, where you must specify the new name.

Note: If you want to work out a variant of the design, you can also use this command to copy the design to another name.

END - How to terminate the program

Menu: File > **Terminate program**

Keyboard: **END**

The design is filed on hard disk and the system jumps back to the operating system. If you modify an existing design, the last version of the design will be filed with the file extension **.BAK**.

Prior to leaving the program, the system will ask you, whether you want to save the modifications.

How to initialize a new design

Before you can work out a design you must put a sketch or a sample into the background of the screen. Sketches and samples can be read with a scanner and superimposed on the screen for design work.

Following these steps to initialize a new design:

- Scan the sketch using **Corel PhotoPaint**.
- Adjust the Brightness and Contrast of the image. Eventually invert the image.
- Copy the image to the **Clipboard**.
- Start ProArt/ProLace.
- Save the drawing under the desired name. Therefore use the command **File/Save as...**
- Run ▦ **SETUP** and specify the embroidery parameters.
- Use the command **PAST** to insert the image in the design.
- Click onto the border of the image, call up 🗔 **DRAWORDER** and send the image to the back.
- Finally move the image to the desired position.

Note: The best resolution for scanning is **150 dpi**.
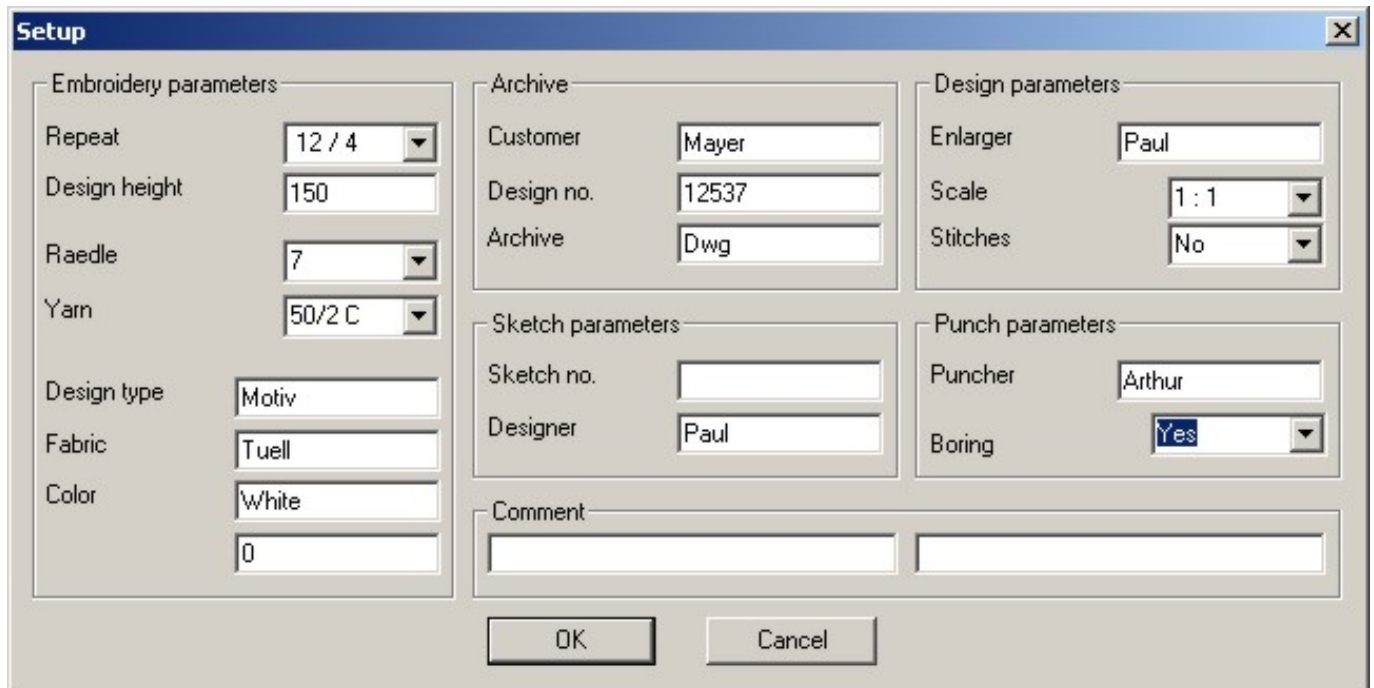
SETUP - How to initialize a design

Toolbar: Standard > Setup

Menu: File > Setup...

Keyboard: SETUP

This function must be called first when you start with a new design. In a dialog-box you can specify all important parameters like repeat, design-height, raedle, yarn etc. After having defined all the parameters, the repeat, the repeat grid and the header are inserted in the design.

DRAWORDER - How to change the display order

Toolbar: **Standard** > **Draworder**

Menu: **Sketch** > **Draworder**

Keyboard: **DRAWORDER**

The **DRAWORDER** command changes the drawing and plotting order of any object in the drawing database in a way that it appears behind or in front of all other elements.

After calling up the command you can choose the entities, which should be modified, then you can choose from a menu one of the following options...

Bring to front

Choose this option to change the drawing and plotting order of any object in the drawing database in a way that it appears in front of all other elements.
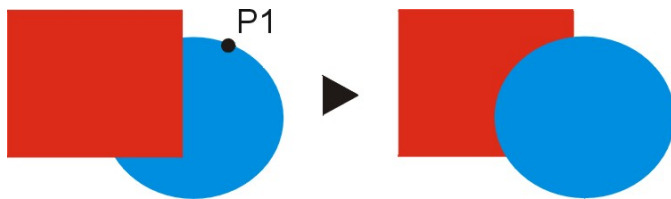
```
Command: DRAWORDER
Select entities: P1
Select entities: <Ret>
> Select Bring to front from menu <
```

Send to back

Choose this option to change the drawing and plotting order of any object in the drawing database in a way that it appears behind of all other elements.
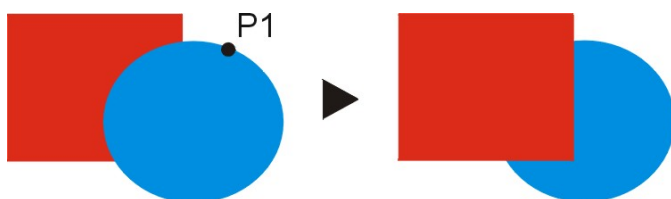
```
Command: DRAWORDER
Select entities: P1
Select entities: <Ret>
> Select Send to back from menu <
```

FORMATS - How to select the paper format

Toolbar: Standard > Paper format
Menu: File > Formats
Keyboard: FORMATS

Prior to plotting, you must determine how the design will be positioned on the paper. With larger sized designs it often is necessary to divide the design into several segments. This is being done with the function **FORMATS**.

After calling up the command you must define the orientation. You can choose between the following options:

- **Portrait**
- **Landscape**
- **Free**

If you select **Free**, you can define the paper width and paper length.

If you have selected Portrait or Landscape you can choose the desired paper from a menu.

Finally you must specify the scale factor. You can choose between

- **1:1**
- **3:1**
- **6:1**
- **Free**

If you select **Free** you can define any desired scale factor.

Now a rectangle will appear marking the plot boundaries. This rectangle can be positioned using the normal **MOVE** command in order to optimize the plot area.

If the complete design cannot be placed within this rectangle, copy the rectangle and position the new rectangle beside the previous one. Continue until the complete design is covered with rectangles. Each rectangle represents to a sheet of paper on the plotter.

Attention: The selected format can never be bigger than the maximal plot area of the plotter.

PLOT - How to plot a design

Toolbar: **Standard** > **Plot/Print** 🖨

Menu: **File** > **Plot/Print...**

Keyboard: **PLOT**

After the plot format has been selected, you may start plotting.

After calling- up the plot command, a dialog-box opens, where the following settings must be made:

- In the sub-menu **Print Setup...** select desired device (Plotter, Printer) and select the paper format as well as the orientation.

- In the **Color/Width Map** tab you can adjust the plot colors and line-width.

- In the **Scale/View** tab select the area, which should be plotted. The selected area should cover the area of a rectangle, which was designed with the command **FORMATS** 📄.

- Also in the **Scale/View** tab you must define the scale factor with the fields **Printed Millimeters = Drawing Units** e.g. **6 = 1**.
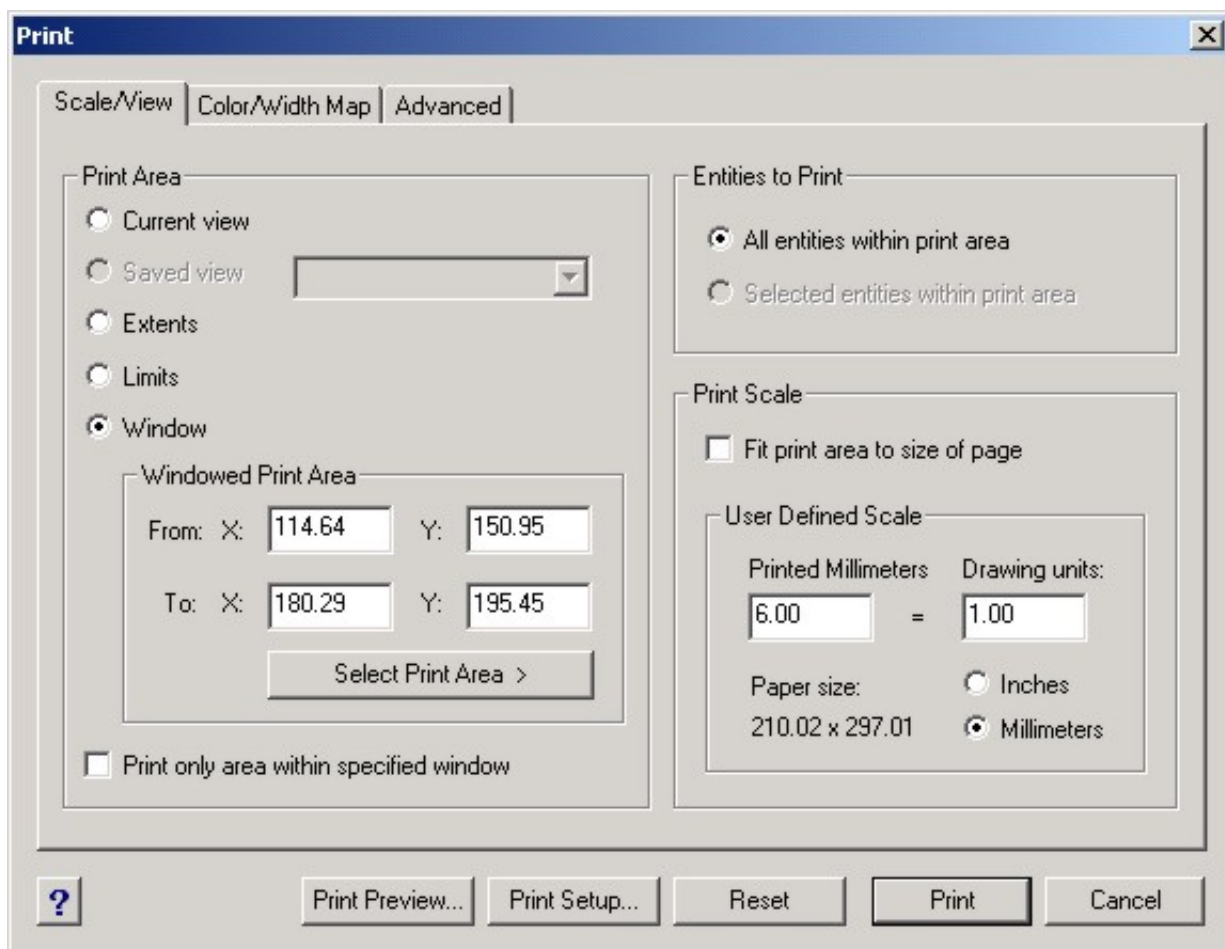
All parameters will be saved. This means, whenever you are working with the same paper, the same scale and with, the same pens, you only have to select the plotting area prior to plotting.

Note: With the option **Print Preview...**, you can superimpose on the screen the positioning of the design on the paper. This is important at the beginning, if you have to check the selected parameters.

PAN - How to move the display

Toolbar: **Standard > Pan**

Menu: **Display > Pan**

Keyboard: **PAN**

If you imagine your computer screen to be a window, through which you look at a design, then you can display with the command **PAN** the design behind the window by moving the window.
The displacement must be defined with two points.

```
Command: PAN
Pan base point: 1st point
Pan displacement point: 2nd point
```

Notice: The **PAN** command can be called-up while carrying out another command.

ZOOM - How to modify the view

The zoom command gives you the possibility to enlarge the part of the design you are working on to a screen-filling size. The actual size of the entities will remain unchanged. They only appear larger on the screen.

The different zoom commands have following effects:

Command	Description
Zoom In	Enlarge the display size by factor two.
Zoom Out	Reduce the display size by factor 0.5.
Zoom Window	Selection of the area by using two diagonal corners of a rectangle.
Zoom Previous	Will bring back the previous area on the screen.
Zoom All	Display of the complete design on the screen according to defined boundaries.

How to use Realtime Zoom and Pan

With **Real time Zoom/Pan** can dynamically change the display content.

Real time Zoom can be activated by simultaneously pressing the **Ctrl** key, the **Shift** key and the **Pick**-button on the mouse, and moving the mouse up/down. On moving the mouse down, the display size will be reduced (like Zoom Out), on moving up the display size wil be enlarged (like Zoom In).

Real time Pan can be activated by simultaneously pressing the **Ctrl** key, the **Shift** key and the **Enter**-button on the mouse and moving the display with the mouse to

the desired position.

Note: The **ZOOM** commands can be called-up while carrying out another command as long as no regeneration is necessary.

REDRAW - How to clean the screen from construction points

Toolbar: **Standard > Redraw**

Menu: Display > Redraw

Keyboard: REDRAW

With the command **REDRAW**, the actual screen content will be newly designed. The image will thereby be cleaned from any construction points.

> **Notice:** The **REDRAW** command can be called-up while carrying out another command.

REGEN - How to regenerate the screen

Toolbar: Standard > Regen

Menu: Display > Regen

Keyboard: REGEN

With the command **REGEN** you can newly regenerate a design from the drawing database. Use **REGEN**, when changes could not be made visible with the command **REDRAW**.

Attention: The **REGEN** command cannot be called-up while carrying out another command! The command will be interrupted.

U / Undo - How to undo a command

With this command the last command can be canceled. The command **Undo** (or **U** with the key board) can be repeated indefinite until the start of the design is being reached.

REDO - How to recover the previously undone command

Toolbar: **Standard** > **Redo**

Keyboard: **REDO**

This command is used to recover an action, which was canceled with an **Undo** command.

ESC - How to cancel an active command

Keyboard: **ESC key**

With the **ESC** key on the keyboard you can interrupt an active command or you can deactivate the handles of elements, which were activated by a previous selection or modification.

Basic design functions

This chapter describes commands to design basic design entities like curves, lines and circles etc. All these commands are located in the Toolbar **Design** or in the pull down menu **Design**.

The **Design Toolbar** contains the following commands...

	Design Toolbar
	[LINE - How to design single lines](#)
	Flyout Polyline
	Flyout Circle
	[ELLIPS - How to design an ellipse](#)
	[RECTANG - How to design a rectangle](#)
	[POLYGON - How to design a polygon](#)
	[SKETCH - How to design freehand lines](#)
	[TEXT - How to insert text](#)
	Polyline Toolbar
	[PARC - Polyarc, starting with a 3 point arc](#)
	[PARCDIR - Polyarc, with starting direction](#)
	[PLINE - Polyline, starting with a straight segment](#)
	[SPIRALE - How to design a spiral](#)
	[DROP - How to design a drop](#)
	Circle Toolbar
	[CIRRAD - Circle, defined by radius](#)
	[CIR2P - Circle, defined by 2 points](#)
	[CIR3P - Circle, defined by 3 points](#)

LINE - How to design single lines

Toolbar: **Design > Line**

Menu: **Design > Line**

Keyboard: **LINE**

You design a line by defining the two end points of the line. Independent line segments are being designed until you end this function by pressing **<Return>**.

```
Command: LINE
From point: 1,1
To point: 1,5
From point: 2,1
To point: 2,5
From point: <Ret>
```

PARC - How to design a polyarc starting with a 3 point arc

Toolbar: **Polyline** > **Polyarc 3 points**

Menu: **Design** > **Polyarc 3 points**

Keyboard: **PARC**

The arc is being defined via the starting point, a second point on the arc and the end point. The subsequent arcs are defined tangentially to the last arc and only the endpoint has to be defined, whereby the arc form is shown dynamically.

You can choose from the following options...

Draw Line

If you are actually designing in the arc mode you can change to the line mode.

Direction

If an arc should not start tangentially to the last arc you can select the option **Direction** to define a starting direction for the next segment.

Second point

With this option you can specify the second point on a 3-point-arc, thereby changing the direction.

Close

The curve line is automatically closed with the starting point of a polyline.

Undo

The last arc segment is canceled.

Example:


```
Command: PARC
Start of polyline: starting point P1
Second point: second point P2
```

```
End of arc: end point P3
Line/Direction/.../Undo/<End of Arc>: option Second point
Second point: second point P4
End of arc: end point P5
Line/Direction/.../Undo/<End of Arc>: end point P6
Line/Direction/.../Undo/<End of Arc>: <Ret>
```

```
End of arc: end point P3
Line/Direction/.../Undo/<End of Arc>: option Second point
Second point: second point P4
End of arc: end point P5
Line/Direction/.../Undo/<End of Arc>: end point P6
Line/Direction/.../Undo/<End of Arc>: <Ret>
```

PARCDIR - How to design a polyarc with starting direction

Toolbar: **Polyline** > **Polyarc direction**

Menu: **Design** > **Polyarc direction**

Keyboard: **PARCDIR**

The arc is defined by a starting point, starting direction and arc endpoint. The starting direction is shown as kind of rubber band, starting from the first point of the arc. The subsequent arcs are defined tangentially to the last arc and only the endpoint has to be defined, whereby the arc form is shown dynamically.

You can choose from the following options...

Draw Line

If you are actually designing in the arc mode you can change to the line mode.

Direction

If an arc should not start tangentially to the last arc you can select the option **Direction** to define a starting direction for the next segment.

Second point

With this option you can specify the second point on a 3-point-arc, thereby changing the direction.

Close

The curve line is automatically closed with the starting point of a polyline.

Undo

The last arc segment is canceled.

Example:


```
Command: PARCDIR
Start of polyline: starting point P1
```

```
Direction from start: start direction P2
End of Arc: endpoint P3
Line/Direction/.../Undo/<End of Arc>: option Direction
Line/Direction/.../Undo/<End of Arc>: start direction P4
End of Arc: endpoint P5
Line/Direction/.../Undo/<End of Arc>: P6
Line/Direction/.../Undo/<End of Arc>: <Ret>
```

PLINE - How to design a polyline, starting with a straight segment

Toolbar: Polyline > Polyline

Menu: Design > Polyline

Keyboard: PLINE

Polylines are elements, which can contain line segments and arcs. They are however considered as one element.

This function is the same as the two Polyarc functions described before. The only difference is, that the starting element is a line and not an arc.

You can choose from the following options...

Draw arcs

If you are actually designing in the line mode you can change to the arc mode.

Close

The curve line is automatically closed with the starting point of a polyline.

Undo

The last arc segment is canceled.

```
Command: PLINE
Start of polyline: Start point
Arc/Distance/.../<Next point>: Endpoint 1
Arc/Distance/.../Undo/<Next point>: Endpoint 2
...
Arc/Distance/.../Undo/<Next point>: <Ret>
```

SPIRALE - How to design a spiral

Menu: Design > Spiral

Keyboard: SPIRALE

To construct a spiral you must define a center, the quantity of turns and the distance between the turns.
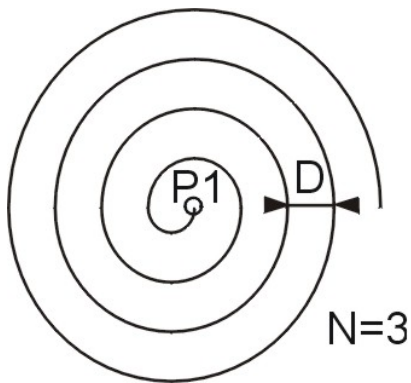

```
Command: SPIRALE
Center point: point P1
Number of turns <0>: N=3
Distance: D=10
```

If you define **number of turns = 0**, you can now specify the outside radius of the spiral and the distance between the turns. The number of turns is calculated.
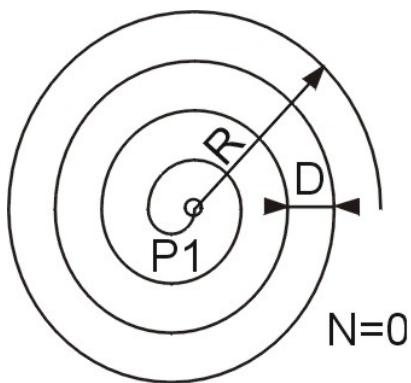

```
Command: SPIRALE
Center point: point P1
Number of turns <0>: <Ret>
Outside radius: radius R
Distance: D=10
```

DROP - How to design a drop

To construct a drop you must first specify the diameter of the circle and then the point of the drop.
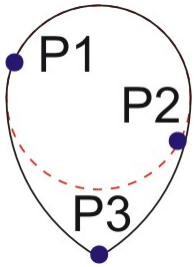
```
Command: DROP
Point 1: P1
Diameter: P2
Select paddle top: P3
```

CIRRAD - How to design a circle, defined by radius

Toolbar: **Circle > Circle radius** ⊕

Menu: **Design > Circle radius**

Keyboard: **CIRRAD**

The circle is constructed by defining the center and the radius of the circle.
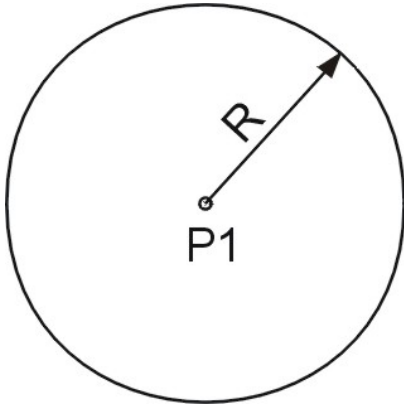

```
Command: CIRRAD
Center point: center P1
Radius: radius R
```

CIR2P - How to design a circle, defined by 2 points

Toolbar: Circle > Circle 2 points

Menu: Design > Circle 2 points

Keyboard: CIR2P

This command defines the circle with the 2 end points of the diameter.
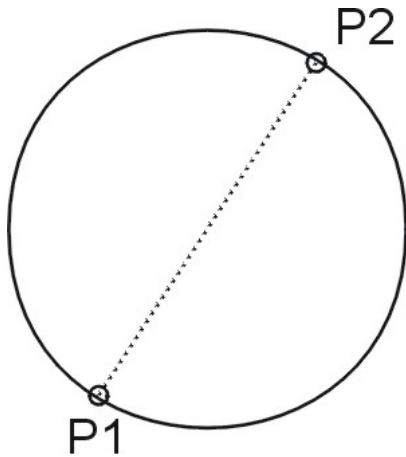

```
Command: CIR2P
Point 1: point P1
Point 2: point P2
```

CIR3P - How to design a circle, defined by 3 points

This command defines the circle with 3 points on the periphery of the circle.
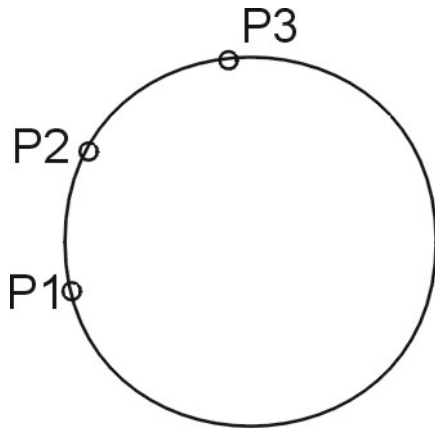
```
Command: CIR3P
Point 1: point P1
Point 2: point P2
Point 3: point P3
```

ELLIPSE - How to design an ellipse

Toolbar: **Design > Ellipse**

Menu: **Design > Ellipse**

Keyboard: **ELLIPSE**

An ellipse is being designed by defining one axis and the width of the second axis.
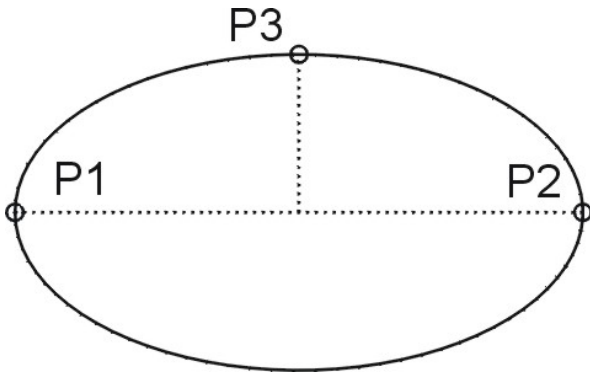

```
Command: ELLIPSE
Center/<First endpoint of axis>: P1
Second endpoint of axis: P2
Other side: P3
```

Another possibility is to specify the center and points on the two axis. In this case you must choose the option **Center** instead of defining the first endpoint of the axis.

```
Command: ELLIPSE
Center/<First endpoint of axis>: C
Center point: center of ellipse
Second endpoint of axis: P2
Other side: P3
```

RECTANG - How to design a rectangle

With this function you can design a rectangle.

Two diagonal points will define the rectangle. Thereafter the orientation of the rectangle can be determined.
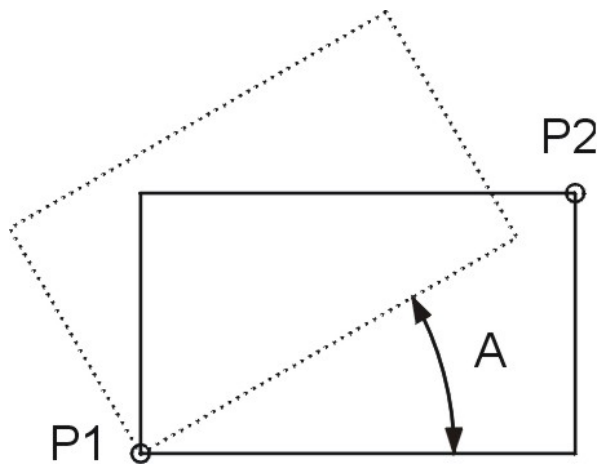

```
Command: RECTANG
First corner: point P1
Other corner: point P2
Angle<0>: Angle A
```

POLYGON - How to design a regular polygon

Toolbar: **Design > Polygon**

Menu: **Design > Polygon**

Keyboard: **POLYGON**

Using the command **Polygon** you can design a polygon with 3 up to 1024 sides. First the system will ask the number of sides, then it will request the center of the polygon and finally it will ask for the outside radius of the polygon.
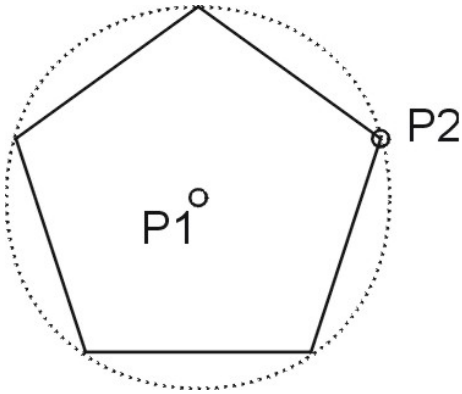

```
Command: POLYGON
Number of sides<4>: 5
Inside/<Center point>: P1
Radius: P2
```

You can also define the polygon via the inside radius of the polygon. In this case you must choose the option **Inside** before you define the center point.

```
Command: POLYGON
Number of sides<4>: 5
Inside/<Center point>: I
Inside/<Center point>: P1
Radius: point in the middle of a segment
```

SKETCH - How to design freehand lines

Toolbar: **Design > Freehand**

Menu: **Sketch > Freehand**

Keyboard: **SKETCH**

With the command **SKETCH** you can design freehand sketches. To sketch, trace with the mouse the desired contours. The system creates a series of lines. The length of these line segments and thereby the resolution can be defined after calling up the command. Please limit yourselves to the absolute necessary precision, as a lot of data will accumulate.

While sketching, the normal commands are not available, and the button menu is being replaced by an own button menu for sketching.

Button definition:

- **Pick button** - The pick button (left mouse button) is used to raise and sink the pen (Pen up / Pen down).

- **Right button** - By pressing the right button, the freehand command is terminated and the freehand lines are memorized.

On sketching, the curser is changed to a pen.

TEXT - How to insert text

Toolbar: **Design** > **Text** A

Menu: **Design** > **Text**

Keyboard: **TEXT**

With the command **TEXT** you can position text in the design. Text will be inserted in the actual text style.

First specify the insertion point of the text. The text will be positioned left justified to this point. Next the text height and finally the insertion angle have to be defined prior to text input.

```
Command: TEXT
Insertion point: Basepoint
Height <1.50>: <Ret> if not defined with style
Angle <0>: <Ret>
Text: This is a text
```

Editing functions

In this chapter you find the commands to manipulate design elements. All editing functions are located in the Toolbar **Modify** or in the pull down menu **Edit**.

	Modify Toolbar
	Modify Toolbar
	[ERASE - How to erase selected entities](#)
	[MOVE - How to move selected entities](#)
	[OFFSET - How to construct a parallel](#)
	[MIRROR - How to mirror selected entities](#)
	[ROTATE - How to rotate selected entities](#)
	[SCALE - How to change the size of entities](#)
	Flyout Copy
	Flyout Array
	[COPYROT - How to copy, rotate and scale entities](#)
	[COPYMIR - How to copy, mirror and scale entities](#)
	[COPYREP - How to copy entities to the repeat](#)
	[COPYALOV - How to copy an allover](#)
	[STRETCH - How to stretch points of entities](#)
	[TRIM - How to trim entities to a boundary](#)
	[EXTEND - How to extend entities to a boundary](#)
	[JOIN - How to join elements in their corners](#)
	Flyout Break
	[EXPLODE - How to break a polyline or a block apart](#)
	[PLEDIT - How to modify a polyline](#)
	Flyout Change
	Copy Toolbar
	[COPY - How to make a copy of entities](#)
	[COPYM - How to make multiple copies of entities](#)
	Array Toolbar
	[PARRAY - How to arrange entities in a polar array](#)

	[RECARRAY - How to arrange entities in a rect. array](#)
	Break Toolbar
	[BREAK - How to erase a part of an entity](#)
	[CUT - How to break an object apart](#)
	Edit polyline Toolbar
	[PLMAKE - How to join entities to a polyline](#)
	[PLADDPT - How to add vertices to a polyline](#)
	[PLWIDTH - How to change the width of entities](#)
	[PSPLINE - How to lay a spline through vertices](#)
	[CURVE - How to lay a curve through vertices](#)
	[DECURVE - How to erase curves](#)
	Change Toolbar
	[DDCHPROP - How to change characteristics of entities](#)
	[XCOLOR - How to change the entity color](#)
	[TOFRONT - How to bring entities on top](#)
	[TOBACK - How to bring entities to back](#)
	[DDATTE - How to edit the header](#)

ERASE - How to erase selected entities

Toolbar: **Modify** > **Erase** ✖

Menu: **Edit** > **Erase**

Keyboard: **ERASE**

The command **ERASE** allows erasing undesired entities in the design. All object selection methods are available to define elements to be erased.

> **Attention:** You should not use this command to erase automatically generated embroidery elements. To erase embroidery elements use the 🖼 **DELEMB** command.

MOVE - How to move selected entities

Toolbar: Modify > Move ⊕

Menu: Edit > Move

Keyboard: MOVE

The command **MOVE** allows to move entities by a definable displacement.

First select the entities to be moved. The displacement can be defined with two points.
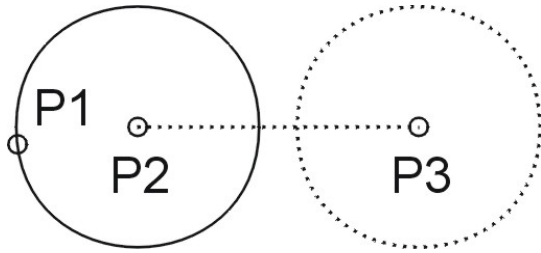

```
Command: MOVE
Select entities: select P1
Select entities: <Ret>
Base point of displacement: base point P2
Second point of displacement: insertion point P3
```

PARALLEL - How to construct a parallel line to an entity

Toolbar: **Modify > Parallel**

Menu: **Edit > Parallel**

Keyboard: **PARALLEL**

If you want to design a parallel line to an entity with a defined offset distance or to a point select the command **PARALLEL**.

First define the parallel distance, next select the element to be offset and finally define the side to offset.
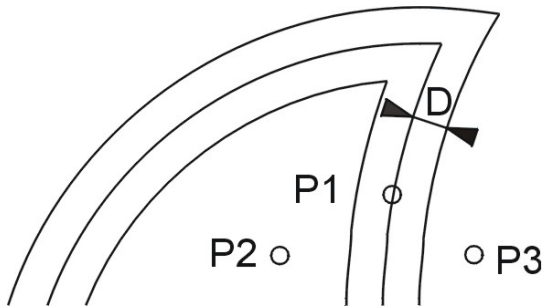

```
Command: PARALLEL
Distance <last>: D
Select entity: P1
Point on side: P2
Select entity: P1
Point on side: P3
Select entity: <Ret>
```

Attention: With intersecting polylines or acute corners undesired results may happen. With the trimming commands, possible corrections can be made. Sometimes it is useful to break the polyline apart and offset the single segments individually.
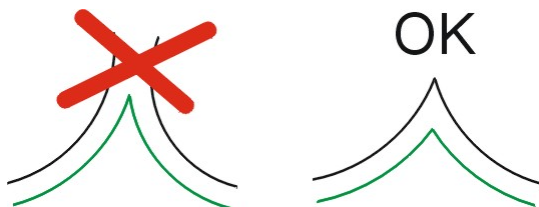
MIRROR - How to mirror selected entities

Toolbar: **Modify** > **Mirror**

Menu: **Edit** > **Mirror**

Keyboard: **MIRROR**

If you want to mirror entities in a design you can use the command **MIRROR**.

First select the entities to be mirrored and then specify the mirror line. The mirror line can be defined with 2 points or by selecting an existing line.

Finally you have to define whether the original entity should be erased or not.

The mirror line is defined with 2 points

If the system cannot find a line or a polyline at the first point of the mirror line, the system asks for the second point of the mirror line.
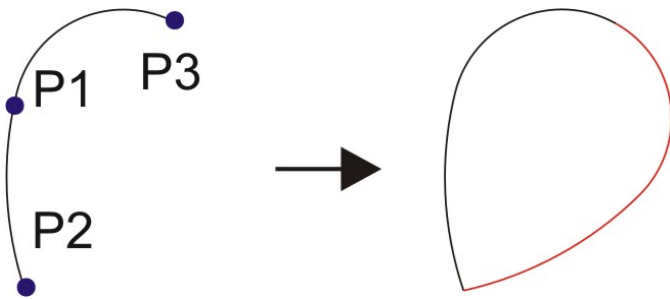
```
Command: MIRROR
Select entities: P1
Select entities: <Ret>
Select mirror line or first point of mirror line: P2
Second point: P3
```

Hint: In this example you should find the points **P2** and **P3** with the entity snap function **F6 Endpoint**.

The mirror line is an existing line or polyline

If the system finds a line or a polyline at the point where you specify the first point of the mirror line, the system uses this line or polyline as mirror line.
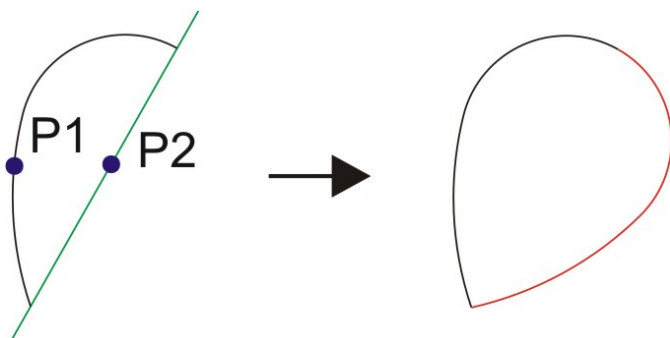
```
Command: MIRROR
```

```
Select entities: P1
Select entities: <Ret>
Select mirror line or first point of mirror line: P2
```

ROTATE - How to rotate selected entities

Toolbar: **Modify** > **Rotate** ⟳

Menu: **Edit** > **Rotate**

Keyboard: **ROTATE**

If you want to rotate entities of your design you can use the command **ROTATE**. The selected entities can be rotated around a specified base point. The dialog is as follows.
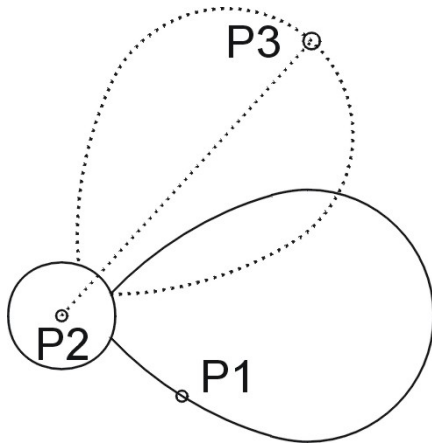

```
Command: ROTATE
Select entities: P1
Select entities: <Ret>
Base point: P2
<Rotation angle>/Reference: P3
```

If you enter **R** for **Reference**, instead of defining a rotation angle, the actual insertion angle of an entity can be corrected to a given value. The dialog for such a case is as follows.
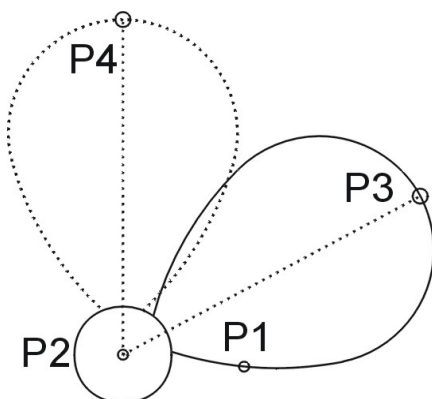

```
Command: ROTATE
Select entities: P1
Select entities: <Ret>
Base point: P2
<Rotation angle>/Reference: R
Reference angle <0>: P2
```

Second point: **P3**
New angle: **P4**

SCALE - How to change the size of entities

Toolbar: **Modify > Scale** 🔲

Menu: **Edit > Scale**

Keyboard: **SCALE**

With the command **SCALE** the size of entities can be changed.

After selecting the entity, a base point and a scale factor must be defined.

```
Command: SCALE
Select entities: P1
Select entities: P2
Select entities: <Ret>
Base point: P3
<Scale factor>/Reference: scale factor
```

If you enter **R** for **Reference**, instead of defining a scale factor, the actual size of an entity can be corrected to a given value. The dialog for such a case is as follows.
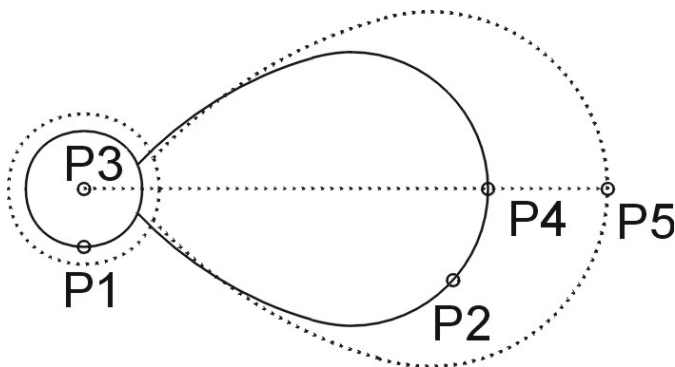

```
Command: SCALE
Select entities: P1
Select entities: P2
Select entities: <Ret>
Base point: P3
<Scale factor>/Reference: R
Reference length <1>: P3
Second point: P4
New length: P5
```

COPY - How to make a copy of entities

With the command **COPY** you can make a copy of one or more entities.

After object selection the system requests the base point. Next the system requests the insertion point.
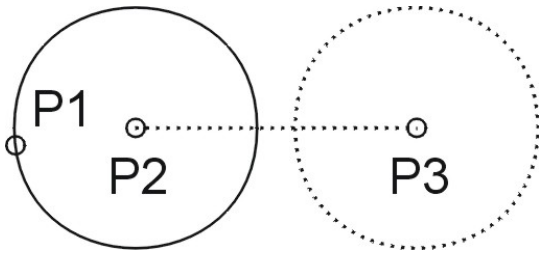
```
Command: COPY
Select entities: select items P1
Select entities: <Ret>
Base point or displacement: base point P2
Second point of displacement: insertion point P3
```

COPYM - How to make multiple copies of entities

Toolbar: Copy > Copy multiple

Menu: Edit > Copy > Copy multiple

Keyboard: COPYM

With the command **COPYM** you can make multiple copies of one or more entities.

After object selection the system requests the base point. Next the system requests insertion points until you press <Return>. On each point a new copy will be placed.
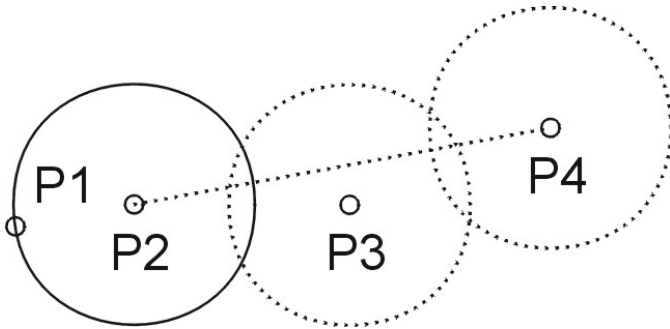

```
Command: COPYM
Select entities: select items P1
Select entities: <Ret>
Base point or displacement: base point P2
Second point of displacement: insertion point P3
Second point of displacement: insertion point P4
Second point of displacement: <Ret>
```

POLARRAY - How to arrange entities in a polar array

Toolbar: **Array** > **Polar array**

Menu: **Edit** > **Copy** > **Polar array**

Keyboard: **POLARRAY**

To copy elements in a polar array, select the entities, which you want to have arranged, then define the center point of the array, the number of elements and finally the angle, on which the elements must be copied.
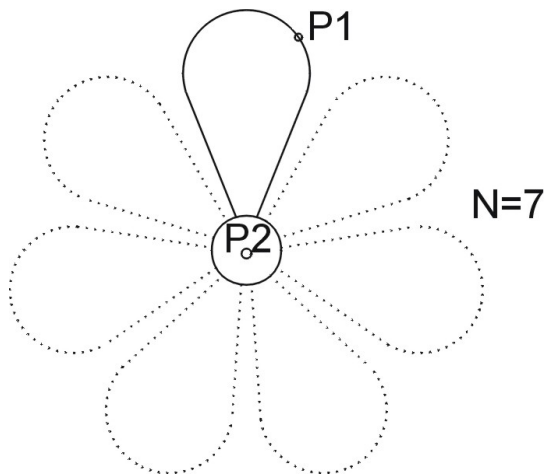

```
Command: POLARRAY
Select entities: P1
Select entities: <Ret>
Center point: P2
Number of items: 6
Angle to fill (+=CCW,-=CW) <360>: <Ret>
```

Note: If you enter a negative angle to fill, the rotation will be clock wise otherwise anti clock wise.

RECARRAY - How to arrange entities in a rectangular array

To copy elements in a rectangular array, select the entities, which you want to have arranged, then specify number of rows and columns and finally specify the distance between rows and column.
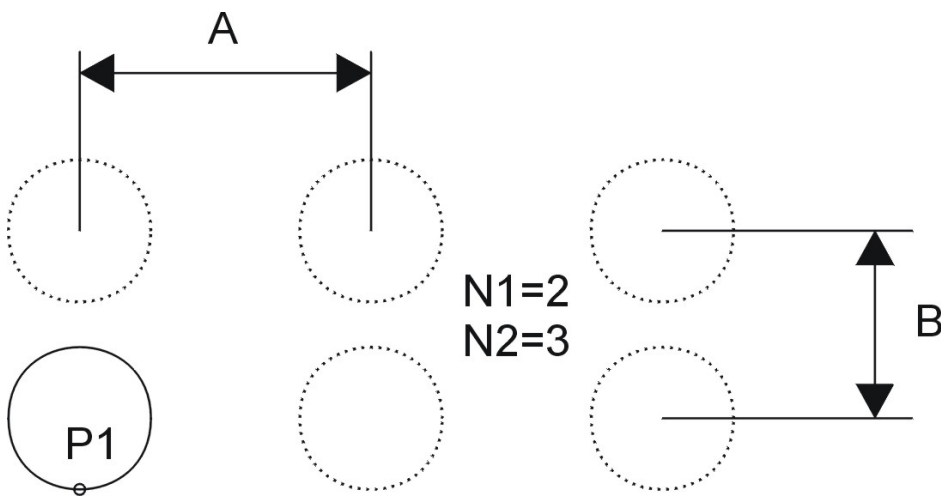

```
Command: RECARRAY
Select entities: P1
Select entities: <Ret>
Number of rows (—-) <1>: N1=2
Number of columns (|||) <1>: N2=3
Distance between rows (—-): B
Distance between columns (|||): A
```

COPYROT - How to copy, rotate and scale entities

Toolbar: Modify > Copy rotate 🖼

Menu: Edit > Copy > Copy rotate

Keyboard: COPYROT

With this command you can copy, scale and finally rotate selected elements several times.
After selecting the object, you must define the displacement by picking two points.

The second point is also the rotation point, around which the entities can be rotated dynamically. If you enter **R** for **Reference**, you can first specify a reference angle and then the new angle.

Finally you can scale the copied entities by picking the opposite side of the entities and stretching them dynamically.
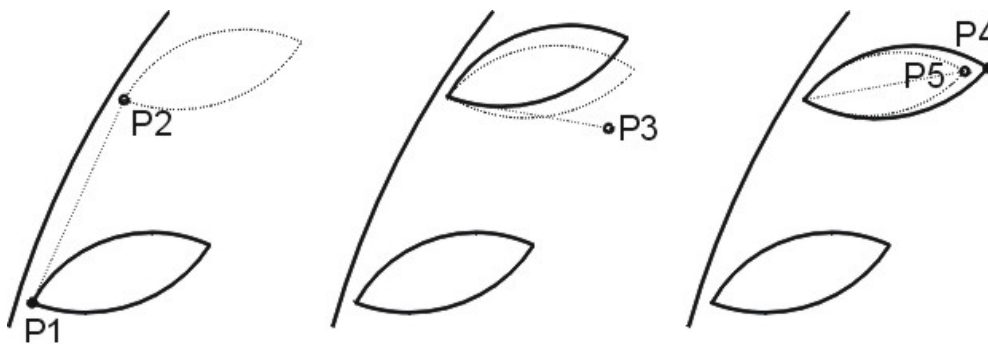

```
Command: COPYROT
Select entities: select leave
Select entities: <Ret>
Base point of displacement: P1
Second point of displacement: P2
Reference/<Rotation angle>: P3
Reference length <1>: P4
New length: P5
Second point of displacement: <Ret>
```

Example with rotation by Reference:

```
Command: COPYROT
Select entities: select leave
Select entities: <Ret>
Base point of displacement: P1
Second point of displacement: P2
Reference/<Rotation angle>: R
Reference angle: specify actual angle
New angle: specify new angle
Reference length <1>: P4
New length: P5
Second point of displacement: <Ret>
```

See also the description of the command **ROTATE** .

COPYMIR - How to copy, mirror and scale entities

With this command you can copy, scale and finally mirror selected elements several times.
After selecting the object, you must define the displacement by picking two points.
Next you can mirror the entities dynamically to the desired position.
Finally you can scale the copied entities by picking the opposite side of the entities and stretching them dynamically.
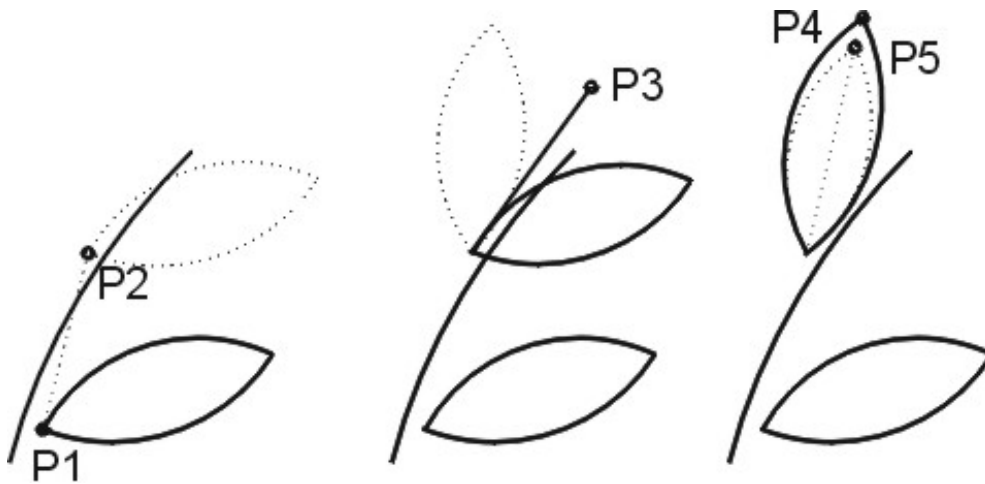

```
Command: COPYMIR
Select entities: select leave
Select entities: <Ret>
Base point of displacement: P1
Second point of displacement: P2
Second point of mirror line: P3
Reference length <1>: P4
New length: P5
```

COPYREP - How to copy entities to the repeat

Toolbar: Modify > Copy repeat

Menu: Edit > Copy > Copy repeat

Keyboard: COPYREP

With this command you can copy the selected elements to the repeat. The copied elements are laid on the layer **AMD_RAPPORT_LR**.

How to copy selected elements to the repeat

After calling up the command you must first select the elements, which you want to copy to the repeat. After the selection a menu opens where you can specify weather you want to copy to the right or to the left repeat.

With the option **Stitches** you can additionally define if you want to copy with or without stitch data. If **Stitches=Yes**, selected elements are copied with stitch data, if **Stitches=No** only the shape of the element will be copied.

```
Command: COPYREP
Select entities: select
Select entities: <Ret>
```


How to copy the whole design

After calling up the command you can first select elements. If you terminate the selection with **<Ret>**, the system automatically copies all elements on layer **AMD_WORK** to the right and to the left repeat and all embroidery data are removed from the copied elements.

```
Command: COPYREP
Select entities: <Ret>
```

Note: Before automatically copying elements to the right and to the left, **COPYREP** erases all elements, which are on Layer **AMD_REPEAT_LR**. Hence it is possible to bring the repeat up to date with this command after having made changes on the design.

COPYALOV - How to copy an allover

Keyboard: **COPYALOV**

With this command you can copy and mirror the selected elements around a vertical or a horizontal mirror line. This function is used to mirror and copy elements of an allover.

After calling up the command you must select the elements and specify the displacement via 2 points. Finally you specify the mirror axis with a 3rd point. If you specify the 3rd point above or below the insertion point of the copy P2, the elements are mirrored vertically. Is the 3rd point right or left from the insertion point P2, the elements are mirrored horizontally.
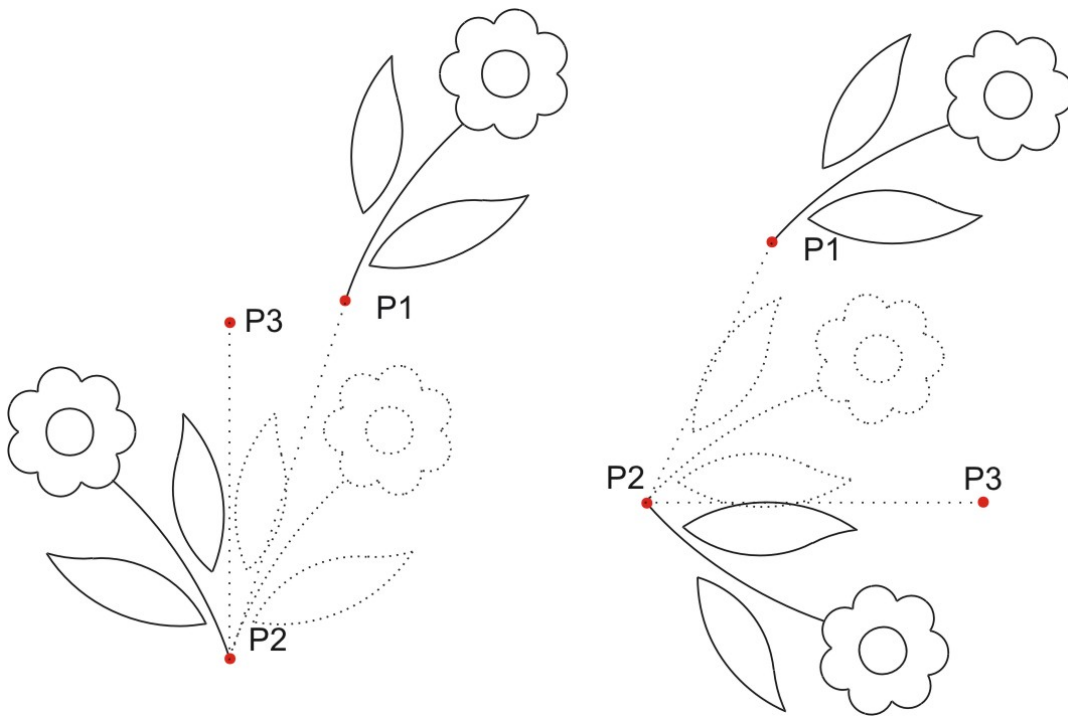
```
Command: COPYALOV
Select entities: select
From point: P1
Second point of displacement: P2
Second endpoint of axis: P3
```

STRETCH - How to stretch points of entities

Toolbar: **Modify > Stretch** 🖉

Menu: **Edit > Trim > Stretch**

Keyboard: **STRETCH**

With this command you can move defined points of an element, whereby the remaining part of the elements remains unchanged. Defined points are end points of lines, arcs and polylines as well as cross points of polylines.

First mark the corners which have to be displaced with Crossing window.
Then the displacement is determined with the base point and the new point.
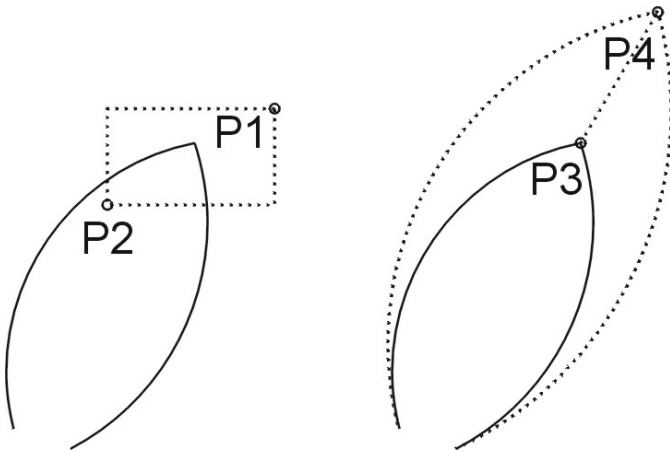
```
Command: STRETCH
Select entities to stretch by crossing-window...
Select entities: P1
Other corner: P2
Select entities: <Ret>
Base point of displacement: P3
Second point of displacement: P4
```

TRIM - How to trim entities to a boundary

Toolbar: **Modify** > **Trim**

Menu: **Edit** > **Trim** > **Trim**

Keyboard: **TRIM**

This command is the complement of the command **EXTEND**. With this command entities can be trimmed in such a way to end exactly at the boundary of other entities.
First define the trimming edge on which the elements must be trimmed.
Then select the entities to be trimmed by picking the part of the object that has to be trimmed. With **<Ret>** you can access the parameter menu. To terminate the command select **End command** from the menu.
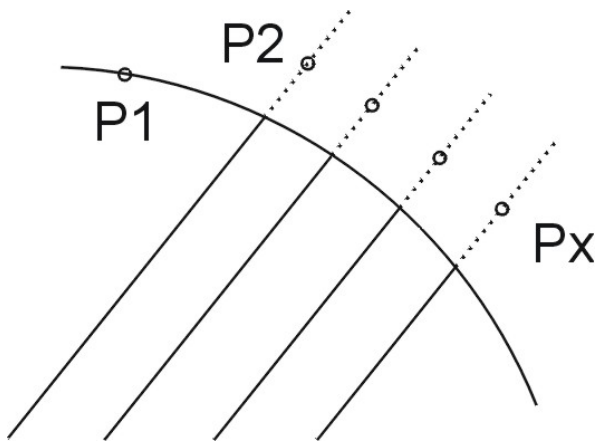

```
Command: TRIM
Select cutting edge (s)...
Select entities: P1
Select entities: <Ret>
Select object to trim<Parameters>: P2
...
Select object to trim<Parameters>: Px
Select object to trim<Parameters>: <Ret>
```

In the parameter menu you can choose from the following options...

Actual mode

Switch between **Trim** and **Extend** mode.

Extend edge

Choose **Extend edge** to trim selected entities to an implied boundary.

New boundary

You can choose a new boundary as trimming edge.

Fence

With this option you define a cutting line. All elements touched by it are trimmed.

Done

End the menu and continue trimming.

End command

Terminate the command.

EXTEND - How to extend entities to a boundary

Toolbar: **Modify** > **Extend**

Menu: **Edit** > **Trim** > **Extend**

Keyboard: **EXTEND**

This command is the complement of the command **TRIM**. You can extend existing entities in order to end at boundaries, defined by other entities.
First select the boundary (or boundaries) up to which the elements should be extended. Then select the object to be extended by picking the side of the object that must be extended. With **<Ret>** you can access the parameter menu. To terminate the command select **End command** from the menu.
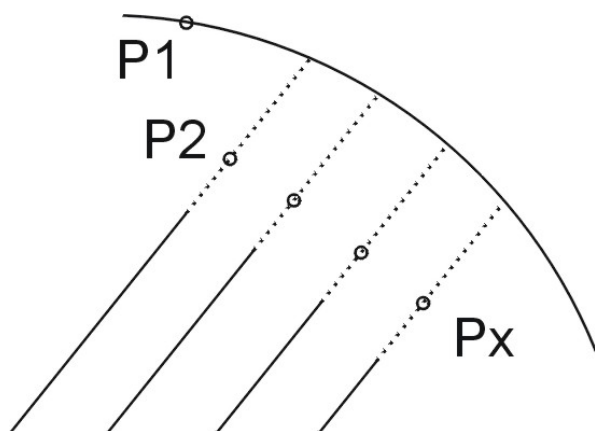

```
Command: EXTEND
Select boundary edge.(s)...
Select entities: P1
Select entities: <Ret>
Select object to extend<Parameters>: P2
...
Select object to extend<Parameters>: Px
Select object to extend<Parameters>: <Ret>
```

In the parameter menu you can choose from the following options...

Actual mode

Switch between **Trim** and **Extend** mode.

Extend edge

Choose **Extend edge** to extend selected entities to an implied boundary.

New boundary

You can choose a new boundary up to which the elements should be extended.

Fence

With this option you can define a cutting line. All elements touched by it are extended to the boundary.

Done

End the menu and continue extending entities.

End command

Terminate the command.

JOIN - How to join elements in their corners

With this command you may join arcs, lines and polylines. The entities will be extended or trimmed if necessary.
The joined elements will automatically be changed into a polyline.

After launching the **JOIN**-command, select the entities at the endpoints, which have to be joined.
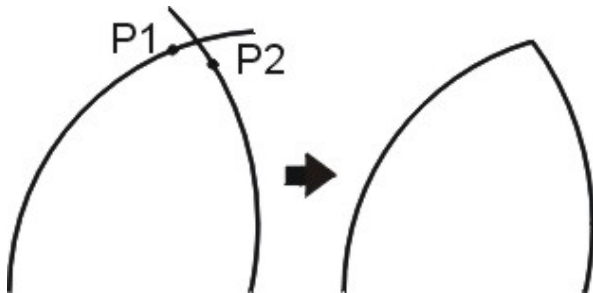

```
Command: JOIN
Select polyline/<Radius=0.0>: P1
Select object: P2
```

With the **JOIN** command you can also creates a fillet, or rounded corner. To define the radius click the right mouse button instead of selecting the first polyline. Now a menu opens where you can define the fillet radius. After having defined the radius select the entities at the endpoints, which have to be filleted.

BREAK - How to erase a part of an entity

With this command you can erase a part of a line, a polyline, a circle or an arc. First select the object to be broken, then define with two points the section to be erased.
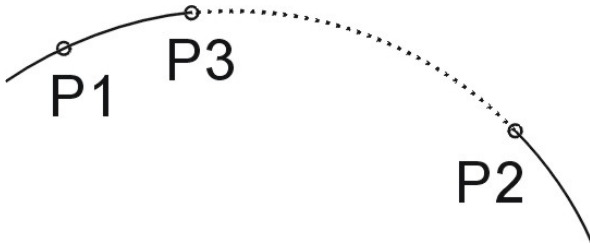

```
Command: BREAK
Select object: P1
First point: P2
Second point: P3
```

Attention: When you want to erase a section of an arc or a polyline, the two points must be defined counterclockwise.

CUT - How to break an object apart

Toolbar: **Break > Cut**

Menu: **Edit > Trim > Cut**

Keyboard: **CUT**

With this command an element will be broken into two parts. The screen image will not change.
First select the object to be broken, then define the break point.

```
Command: CUT
Select object: select object to break
Break point: point on element
```

This command is mainly used with fillings and hatches whereby exact contours are necessary. Line overlap results in hatch error.

EXPLODE - How to break a polyline or a block apart

Toolbar: **Modify** > **Explode**

Menu: **Edit** > **Trim** > **Explode**

Keyboard: **EXPLODE**

If you want to break a block or a polyline into its original segments, choose the command **EXPLODE**. The image of the broken elements on the screen will not change.
The command **EXPLODE** only affects the last level. If the polyline is part of a block and the command origin is used with the block, the block will be broken, however, the polyline will not be affected. The command must, if so desired, also be used with the polyline.

You can also convert embroidery objects like blattstitches, edges, etc. to manually punched sequences or manually punched sequences to polylines.

Attention: Blocks inserted with different x and y factors cannot be broken-up again!

PLEDIT - How to modify a polyline

Toolbar: **Modify** > **Edit polylines** 🖱

Menu: **Edit** > **Polyline** > **Edit**

Keyboard: **PLEDIT**

With the command **PLEDIT** you can modify polylines. After calling up the command a menu is opened.

In the menu you can choose from the following options...

Join polylines

From individual lines, arcs and polylines you can make one single polyline. You only have to select the elements to be joined.

Attention: The end points of the single elements must be joined exactly.

Add point

You can insert vertices in an existing polyline.

```
Select entity: select polyline
Insertion point: new vertice
```

Invert

The direction of a polyline can be changed. This means, the polylines start and end will be inverted.

After selecting a polyline, the actual start of the polyline will be indicated by a cross and from a popup menu you can choose weather you want to change the direction or not. If you select **Invert**, the direction of the polyline will be changed and the new endpoint will be indicated by a cross. If you select **Cancel** from the menu, you terminate the command without changing the polyline.

Note: This command is mainly used in conjunction with the command
🖉 **CHAIN2P**.

Wide polyline

You can define the width of polylines and circles. This is specially useful when you want to represent edges as solid elements in a sketch representation.
After calling up the command you must select the polylines, for which you want to define a width. Next you must specify the width. Finally the system makes a copy of all selected elements. The copied elements get the specified width and are moved to Layer 2.

```
Select entities: select
```

```
Select entities: <Ret>
Width <0.00>: 0.7
```

> **Note:** With **Toback** you can move the area below all other elements, and with **ToFront** you can move the area to the top of all other elements.

Spline

After selecting a polyline, a spline will be laid through the vertices of the polyline.
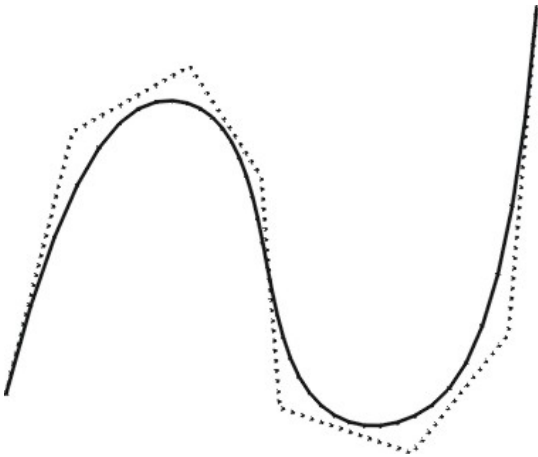
Curve

After selecting a polyline, a curve will be laid through the vertices of the polyline. The curve will be laid directly through each vertices.
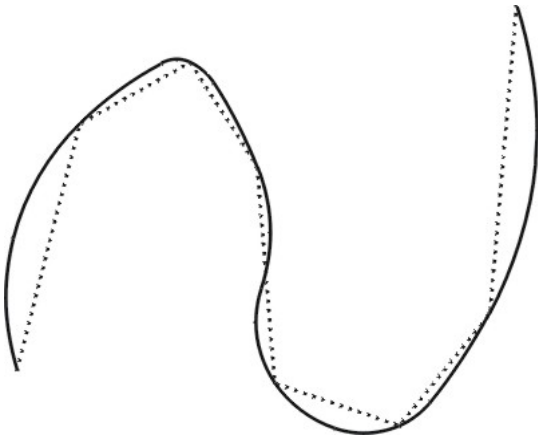
Erase curve

A polyline, through which a spline or a curve line has been laid, is being changed to a normal polyline with straight line segments.

PLMAKE - How to join entities to a polyline

Toolbar: **Change polyline > Join polylines**

Menu: -

Keyboard: **PLMAKE**

From individual lines, arcs and polylines you can make one single polyline. You only have to select the elements to be joined.

Attention: The end points of the single elements must be joined exactly.

PLADDPT - How to add vertices to a polyline

Toolbar: Change polyline > Add point

Menu: -

Keyboard: PLADDPT

With the command **PLADDPT** you can insert vertices in an existing polyline.

```
Command: PLADDPT
Select entity: select polyline
Insertion point: new vertice
```

PLWIDTH - How to change the width of entities

Toolbar: Change polyline > Polyline width

Menu: -

Keyboard: PLWIDTH

With this function you can define the width of polylines and circles. This is specially useful when you want to represent edges as solid elements in a sketch representation.
After calling up the command you must select the polylines, for which you want to define a width. Next you must specify the width. Finally the system makes a copy of all selected elements. The copied elements get the specified width and are moved to Layer 2.

```
Command: PLWIDTH
Select entities: select
Select entities: <Ret>
Width <0.00>: 0.7
```

Note: With **Toback** you can move the area below all other elements, and with **ToFront** you can move the area to the top of all other elements.

PSPLINE - How to lay a spline through vertices

Toolbar: **Change polyline > Spline**

Menu: -

Keyboard: **PSPLINE**

After picking a polyline, a spline will be laid through the vertices of the polyline.
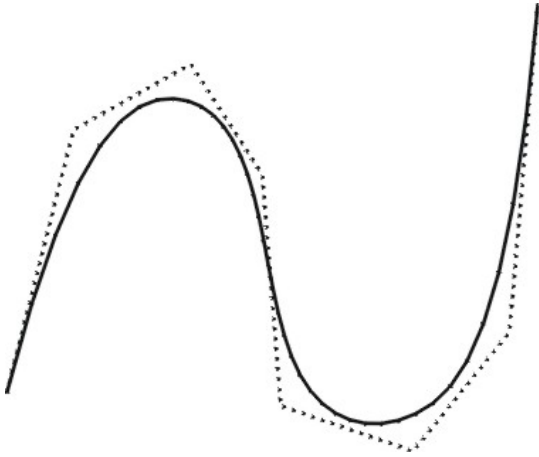
CURVE - How to lay a curve through vertices

Toolbar: **Change polyline > Curve** ⌒

Menu: -

Keyboard: **CURVE**

After picking a polyline, a curve will be laid through the vertices of the polyline. The curve will be laid directly through each vertices.
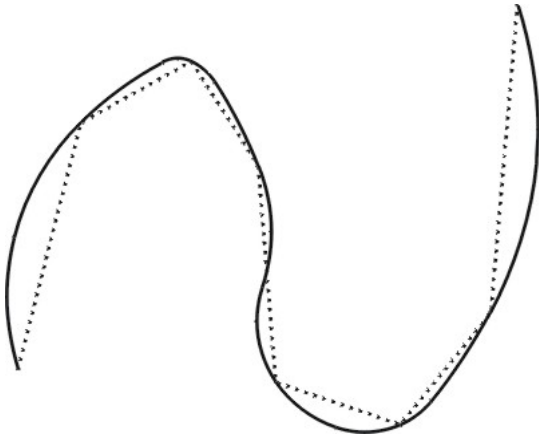
DECURVE - How to erase curves

Toolbar: **Change polyline > Erase curve**

Menu: -

Keyboard: **DECURVE**

A polyline, through which a spline or a curve line has been laid, is being changed to a normal polyline with straight line segments.

DDCHPROP - How to change characteristics of entities

Toolbar: Change > Properties

Menu: Edit > Change > Properties

Keyboard: DDCHPROP

With this command element characteristics, such as color, linetype, layer etc. can be modified.

After selecting the command and the entities, which must be modified a dialog-box will appear. The desired changes can then be carried out interactively.

Attention: Element characteristics of automatically created entities may not be changed. Changes of these characteristics may result in system error.

XCOLOR - How to change the entity color

Toolbar: **Change > Change color**

Menu: **Edit > Change > Change color**

Keyboard: XCOLOR

With the command **XCOLOR** the color of selected entities can be changed.

```
Command: XCOLOR
Select entities: select
Select entities: <Ret>
New color: red
```

Attention: Element characteristics of automatically created entities may not be changed. Changes of these characteristics may result in system error.

TOFRONT - How to bring entities on top

Menu: Edit > Change > To front

Keyboard: TOFRONT

The **TOFRONT** command changes the drawing and plotting order of any object in the drawing database in a way that it appears on top of all other elements.
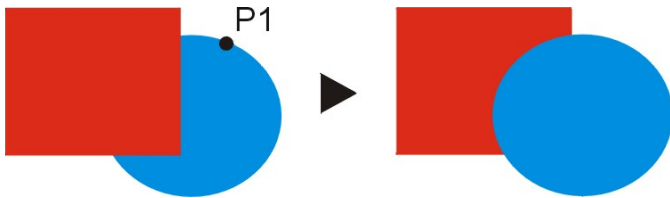

```
Command: TOFRONT
Select entities: P1
Select entities: <Ret>
```

TOBACK - How to bring entities to back

The **TOBACK** command changes the drawing and plotting order of any object in the drawing database in a way that it appears behind of all other elements.
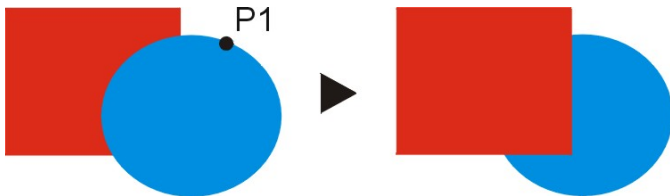
```
Command: TOBACK
Select entities: P1
Select entities: <Ret>
```

DDATTE - How to edit the header

Toolbar: **Change** > **Change header**

Menu: **Edit** > **Change** > **Header...**

Keyboard: **DDATTE**

The command **DDATTE** allows you to correct the texts, entered in the header. After calling up this command a dialog-box will appear with all actual data of the header. This data can be edited as normal text.

After having corrected the texts close the dialog-box with **OK**. If you do not want the changes to be carried out close with **Cancel**.

Construction aids

In this chapter commands are described, which can serve to construct a design or parts of it. All these functions are located in the Toolbar **Construct** or in the pull down menu **Construct**.

The **Construct Toolbar** contains the following commands...

	Construct Toolbar
	Flyout Inquiry
	[AUXLIN - Auxiliary lines](#)
	Flyout Fill
	[CHAIN2P - How to arrange 2 point blocks](#)
	[CHAINC - How to arrange center blocks](#)
	[MODULE - How to work with modules](#)
	Flyout Block see Chapter 6
	Inquiry Toolbar
	COUNTSTI - How to count stitches of a design (not yet punched)
	[DIST - How to measure the distance between points](#)
	[ELENGTH - How to determine the length of an element](#)
	[DIMLINEAR - How to insert a dimension](#)
	Auxiliary lines Toolbar
	[HHORI - How to insert a horizontal auxiliary lines](#)
	[HVERT - How to insert a vertical auxiliary lines](#)
	[HPERP - Auxiliary line, perpendicular to](#)
	[HGEN - How to insert a rotated auxiliary lines](#)
	[HGRID - How to insert an auxiliary grid](#)
	[HFLOWER - How to insert a construct aid for flowers](#)
	[HDEL - How to erase auxiliary lines](#)
	Fill Toolbar
	[PATFILL - How to fill an area with a pattern](#)
	[ARCFILL - How to fill an area with arcs](#)

	[AREAFILL - How to fill an area with color](#)
	[BORDER - How to design a polyline around an area](#)

Other functions in the pull down menu

The following command cannot be selected from a toolbar. It must be called up from the pull down menu **Construct**.

 [MANSTI - How to design single stitches](#)

DIST - How to measure the distance between points

Menu: Measure > Distance

Keyboard: DIST

With the command **DIST** you can measure the distance between two points. After having defined two points the distance between the points and the angle is shown.

```
Command: DIST
From point: point 1
To point: point 2
```
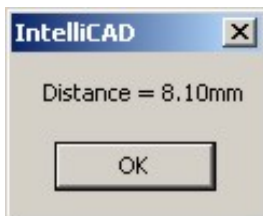
ELENGTH - How to determine the length of an element

Toolbar: Inquiry > Element length

Menu: Measure > Element length

Keyboard: ELENGTH

With the command **ELENGTH** you can determine the length of an object. After calling up the command, you must select the element, which must be measured.

```
Command: ELENGTH
Select entity: select
```
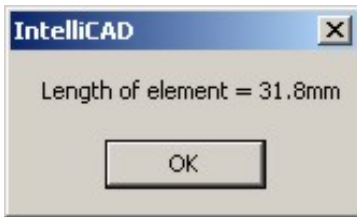
DIMLINEAR - How to insert a dimension

Menu: Measure > Dimension

Keyboard: DIMLINEAR

With this command you can insert a horizontal or vertical dimension. After calling up the command, you must specify two points, then the position of the dimension.

```
Command: DIMLINEAR
From point: P1
To point: P2
Dimension line location: P3
```

AUXLIN - How to design auxiliary lines

Toolbar: Construct > Auxiliary lines

Menu: Construct > Auxiliary lines

Keyboard: AUXLINE

With the command **AUXLINE** you can design and erase various auxiliary lines. Additionally con switch on/off the layer auxiliary lines. After calling up the command a menu is opened.

In the menu you can choose from the following options...

Horizontal

With this option a horizontal auxiliary line is inserted, which is limited by the design boundaries. Definition is made by means of the insertion point.
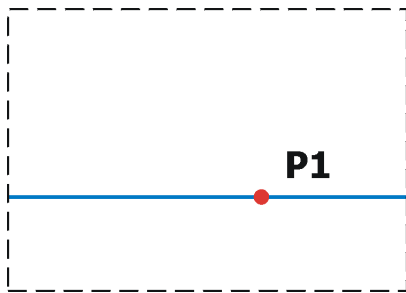
```
Insertion point: P1
Insertion point: <Ret>
```

Vertical

With this option a vertical auxiliary line is inserted, which is limited by the design boundaries. Definition is made by means of the insertion point.
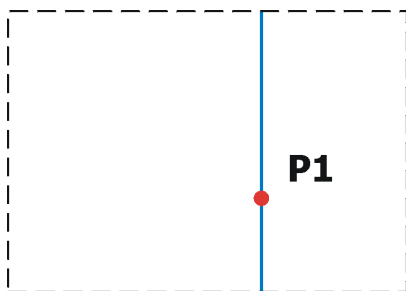
```
Insertion point: P1
Insertion point: <Ret>
```

Angle

With this option an auxiliary line with a definable insertion angle can be inserted. The line is limited by the design boundaries. Definition is being done using an angle and the insertion point.
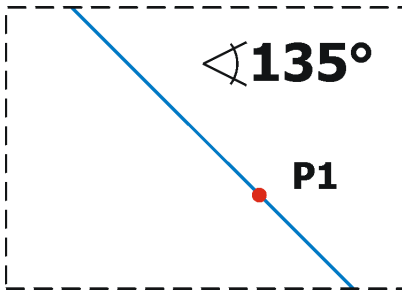
```
    Angle <45>: 135
    Insertion point: P1
    Insertion point: <Ret>
```

Perpendicular

With this option an auxiliary line, perpendicular to another line, can be inserted. The line is limited by the design boundaries. Definition is being done by picking the reference line and the insertion point.
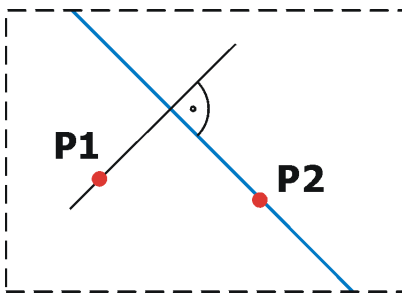

```
    Select object: P1
    Insertion point: P2
    Insertion point: <Ret>
```

Divide polyline

You can divide a polyline in equal segments. The division points are marked with small circles.

After having selected this option you can choose the polyline, which you want to have divided. Next you can specify how many segmentation points you want to have.

Define number of segments

Just specify the number of segmentation points **N**.


```
    Select guide line: P1
    Number of Segments<0>: 6
```

Define size of a segment

Instead of specifying the number of segments confirm with **<Ret>**. A menu opens, where you can specify the distance between the segmentation point **A**.

Additionally you can define via the option **Corner** how the system handles corners. If **Corner=Yes**, the system divides from corner to corner to make sure that every corner has a segmentation point. In curves with a small radius the distance between the segmentation point will automatically be reduced.
If **Corner=No** the system ignores corners and the distance between the segmentation points is always same.
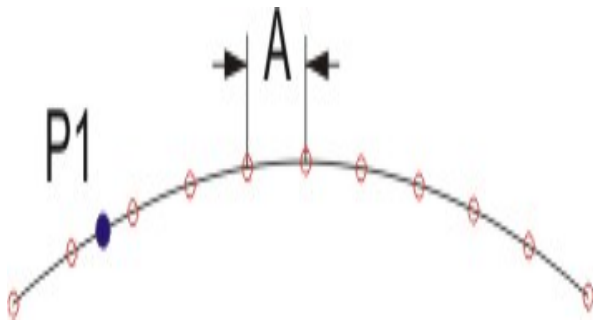
Corner = Yes   Corner = No

```
Select guide line: P1
Number of Segments<0>: <Ret>
```
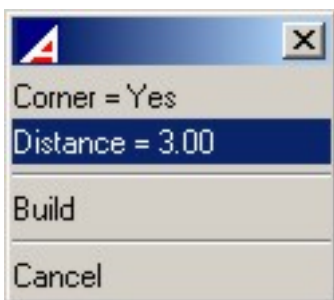

Flower

With this option you can draw three segmented circles. This grid can be used to construct a flower.

First define the center point of the construction, then the corresponding radius of the three circles and finally specify the number of segments.

```
Center: P1
1. radius: P2
2. radius: P3
3. radius: P4
No. of segments: 10
```
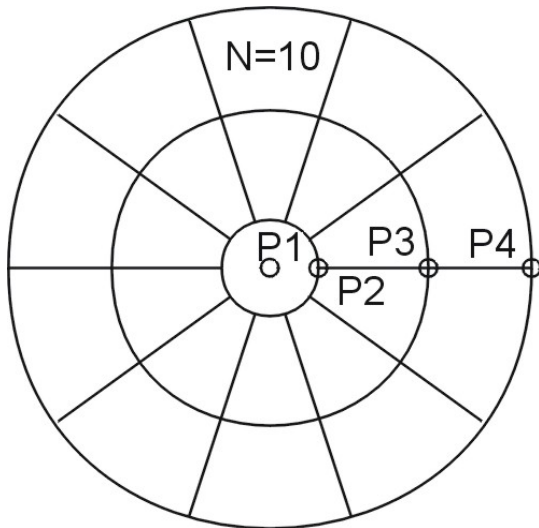
Note: If you want to design a flower with 5 paddles we recommend to specify 10 segments. In this way you have also the center line for each paddle.

Grid

With this option you can fill a defined area with a grid.

The area is defined using the diagonal points of a rectangle. Thereafter you select number of rows and columns, in which the rectangle has to be divided.
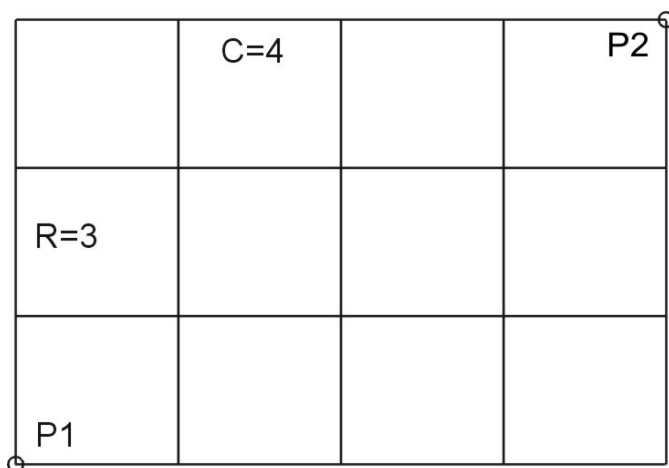

```
First corner: P1
Second corner: P2
Number of rows (−−) <1>: 3
Number of columns (|||) <1>: 4
```

Layer on

Switch the auxiliary layer on.

Layer off

Switch the auxiliary layer off.

Delete

With this option you erase auxiliary lines that originate from this command, or elements, which are on the layer **AMD_AUXLIN**.

Done

Terminate the command

HHORI - How to insert a horizontal auxiliary lines

Menu: -

Keyboard: **HHORI**

With this command a horizontal auxiliary line is inserted, which is limited by the design boundaries. Definition is made by means of the insertion point.
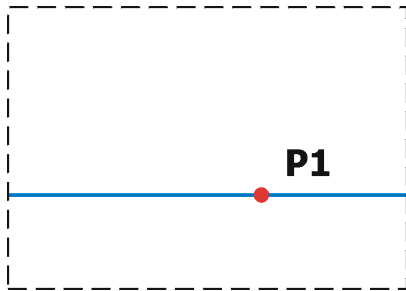

```
Command: HHORI
Insertion point: P1
Insertion point: <Ret>
```

HVERT - How to insert a vertical auxiliary lines

Keyboard: **HVERT**

With this command a vertical auxiliary line is inserted, which is limited by the design boundaries. Definition is made by means of the insertion point.
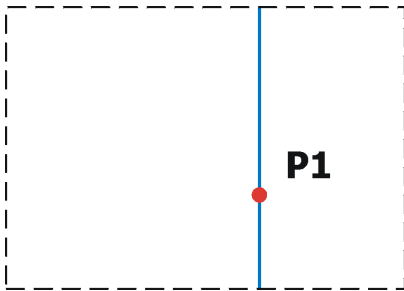

```
Command: HVERT
Insertion point: P1
Insertion point: <Ret>
```

HPERP - How to insert an auxiliary line, perpendicular to another line

Toolbar: Auxilary lines > Perpendicular

Menu: -

Keyboard: HPERP

With this command an auxiliary line, perpendicular to another line, can be inserted. The line is limited by the design boundaries. Definition is being done by picking the reference line and the insertion point.
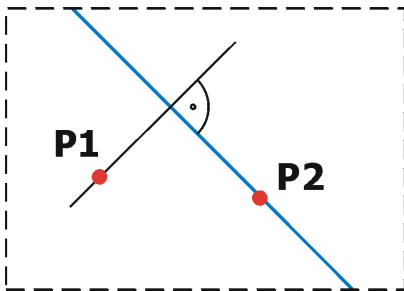

```
Command: HPERP
Select object: P1
Insertion point: P2
Insertion point: <Ret>
```

HGEN - How to insert a rotated auxiliary lines

Toolbar: Auxilary lines > Angle

Menu: -

Keyboard: HGEN

With this command an auxiliary line with a definable insertion angle can be inserted. The line is limited by the design boundaries. Definition is being done using an angle and the insertion point.
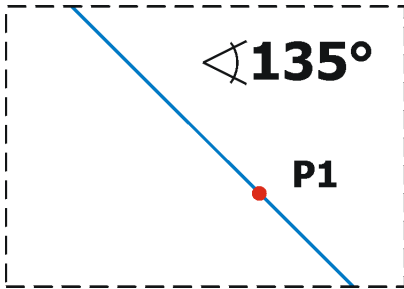

```
Command: HGEN
Angle <45>: 135
Insertion point: P1
Insertion point: <Ret>
```

HGRID - How to insert an auxiliary grid

Toolbar: **Auxilary lines > Grid**

Menu: -

Keyboard: **HGRID**

With the command **HGRID** you can fill a defined area with a grid.

The area is defined using the diagonal points of a rectangle. Thereafter you select number of rows and columns, in which the rectangle has to be divided.
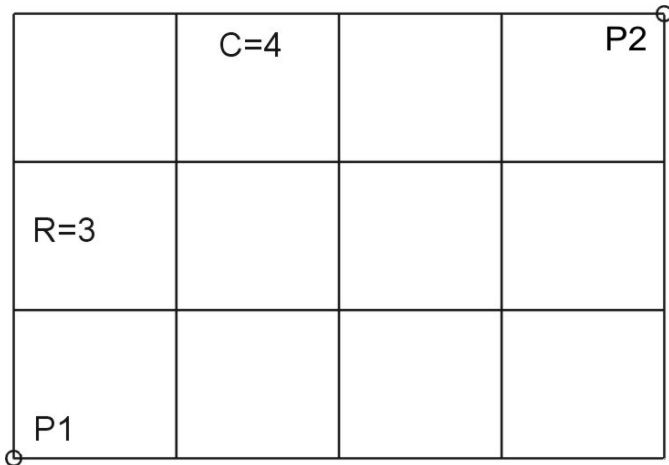

```
Command: HGRID
First corner: P1
Second corner: P2
Number of rows (—-) <1>: 3
Number of columns (|||) <1>: 4
```

HFLOWER - How to insert a construct aid for flowers

Toolbar: **Auxilary lines > Flower** ⊞

Menu: -

Keyboard: **HFLOWER**

With this command you can draw three segmented circles. This grid can be used to construct a flower.

First define the center point of the construction, then the corresponding radius of the three circles and finally specify the number of segments.
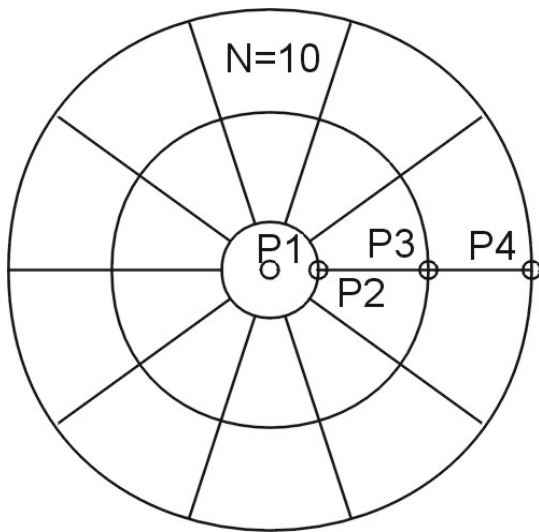

```
Command: HFLOWER
Center: P1
1. radius: P2
2. radius: P3
3. radius: P4
No. of segments: 10
```

Note: If you want to design a flower with 5 paddles we recommend to specify 10 segments. In this way you have also the center line for each paddle.

HDEL - How to erase auxiliary lines

Toolbar: **Auxilary lines** > **Erase auxilary lines**

Menu: -

Keyboard: **HDEL**

With this command you erase auxiliary lines that originate from commands such as **HFLOWER**, **HGRID**, **HVERT** etc. Or elements, which are on the layer **AMD_AUXLIN**.

PATFILL - How to fill an area with a pattern

Toolbar: **Fill > Fill with pattern** ⊞

Menu: Construct > Fill with pattern...

Keyboard: PATFILL

With the command **PATFILL**, filling effects can be exactly positioned in a determined area.

An area can be filled in 3 steps:

- After calling-up the command select the area to be filled.
- Selection of the desired filling effect from the dialog-box.
- Fit in the filling effect in the desired area.
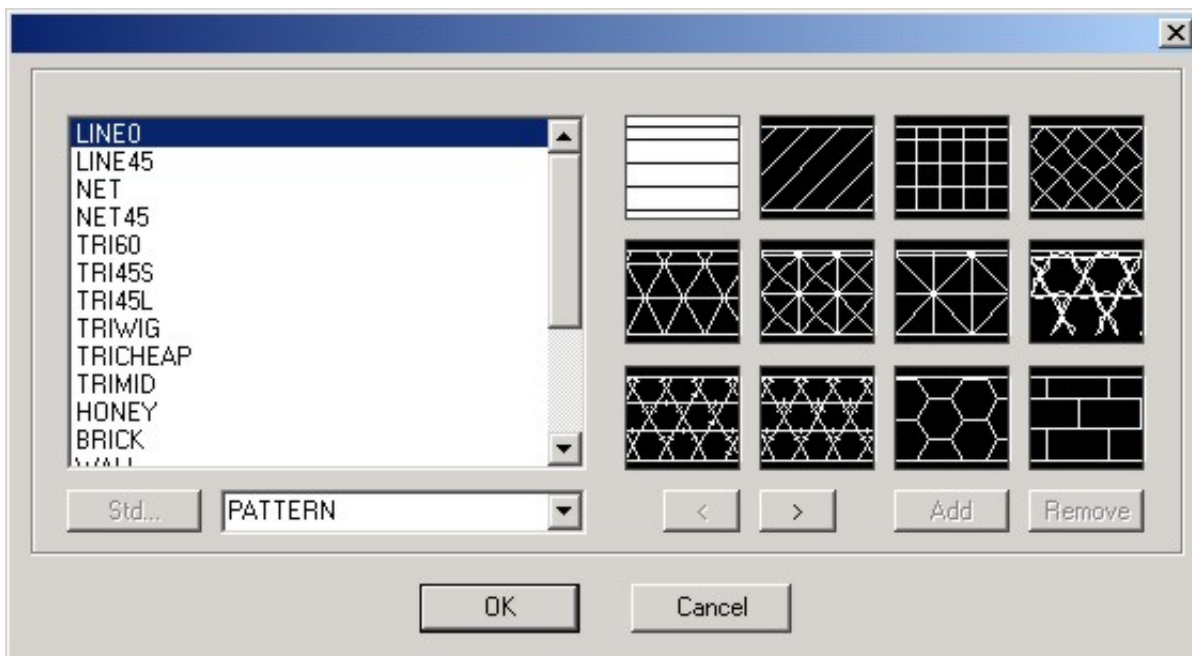
After having selected the filling effect, you can position the effect with the keyboard or you can enter a popup menu, where you can specify the parameters for the filling effect. You enter the parameter menu with **<Ret>** or by clicking with the right mouse button.

In the parameter menu you can choose from the following options...

Size

Pattern size in mm. This value defines the length of a single element of the effect.

Angle

Insertion angle of the pattern in degrees

Size step

With the + and - keys you can increase or reduce the size of the filling effect by a certain value. **Size step** defines the change in size for one step.

Angle step

With the / and * keys you can rotate the effect by a certain value. Angle-step defines the rotation angle for one step.

Displacement

Displacement, by which the filling effect will be moved by pressing the key once (7,8,9,4,6,1,2,3).

Done

Leave the menu. You can continue moving the filling effect with the keyboard.

Build

Build the effect and terminate the command.

Cancel

The effect will be removed and the command is terminated.

How to adjust the effect with the keyboard

Using the number block on the key board, the hatch can be moved, enlarged, reduced and rotated.

Attention: The number block must be set up for number entry.

Key functions

+   Enlarge the effect according to the defined value.

-   Reduce the effect according to the defined value..

*   Rotate the effect according to the selected angle clockwise.

/   Rotate the effect according to the selected angle counterclockwise.

7   Move the effect according to the selected value to the left up.

8   Move the effect according to the selected value up.

9   Move the effect according to the selected value to the right up.

4   Move the effect according to the selected value to the left.

6   Move the effect according to the selected value to the right.

1   Move the effect according to the selected value to the left down.

2   Move the effect according to the selected value down.

3   Move the effect according to the selected value to the right down.

```
Command: PATFILL
Select boundary edge(s)...
Select entities: select
Select entities: <Ret>

>> Select pattern from dialog-box <<

The pattern will be inserted. Now it must be positioned with the
keyboard and the popup menu.
```

Attention: The area must be closed, otherwise it cannot be filled!

Hexagon cheap and middle version

First fill the area with the corresponding filling effect.
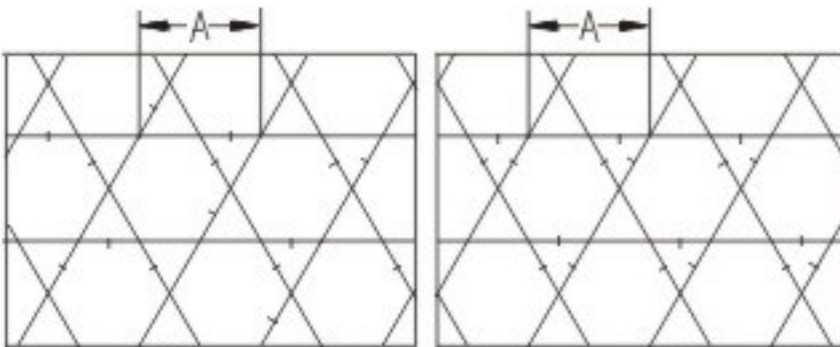
To design the stitches, you must use stitch automatic **STEP**. When you apply the **STEP** automatic, the number of stitches per segment must be set to 1.

Hexagon with wiggle stitch

First fill the area with the corresponding filling effect.
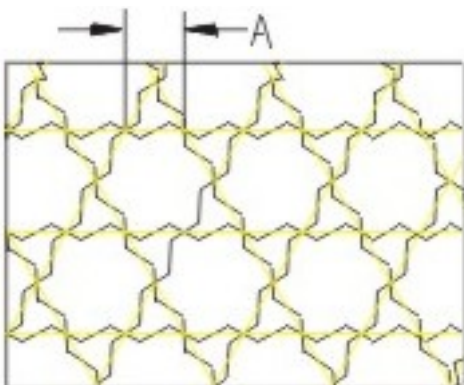
To design the wiggle stitches, you must use stitch automatic **WIGGLE**. When you

apply the automatic, the stitch distance must be set to the length of a hexagon side.

ARCFILL - How to fill an area with arcs

Toolbar: **Fill > Fill with arcs**

Menu: **Construct > Fill with arcs**

Keyboard: **ARCFILL**

With the command **ARCFILL** you can fill an area with polyarcs or polylines. The inclination polylines must be constructed before calling up the command.

The area can be represented by a single closed polyline or by a circle, or it can be represented by two polylines. In case of two polylines, the area must be between the two polylines.

Hint: If the boundary of the area is made up of more then two polylines, you can use the command **BORDER** to generate a single polyline, which is surrounding the desired area.
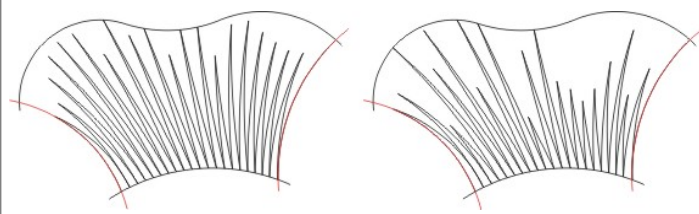
Before you can fill the area you must design the inclination polylines.
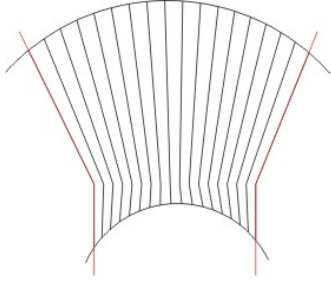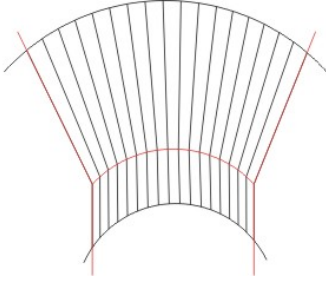
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: ARCFILL
Select boundary edge 1/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Ray distance **A** in millimeter
Random	The rays can have a random length. The random part is always outside (at the end of the inclination polylines). The parameter defines the relation between full ray length and random ray length. This value must be specified in percent (See upper illustration). **Example:** If the ray length is 100 mm and the value is 80%, the random ray length can vary from 20 to 100 mm.

	Random=30%	Random=70%
Guide line(s)	With additional guide lines you can get a smoother distribution of the arcs. The guide lines must also be constructed before you can call up the command.	
Done	Exit from the menu and continue with selecting boundaries.	
Cancel	Interrupt the command	

If the parameters are defined correctly select first the area boundaries. Thereafter select the previously designed inclination polylines.

To fill an area with polylines, the following rules must be respected:

- All inclination polylines **B** must be designed in the same direction.
- All inclination polylines **B** must be designed with the same number or line respectively arc segments.
- You cannot mix arc and curve segments in an inclination polyline.
- An inclination polyline made of arcs must not have corners.
- The inclination polylines **B** must be selected in the proper order.
- If you want to fill an area with rays of random length, the random part is always on the side of the end point of the inclination polylines **B**.
- if you fill a closed area **A,** the imaginary line between the intersection points of the inclination lines **B** with the area **must not intersect** the area **A**.

dashed line intersects
not OK!   dashed line does not intersect
OK!

If you use guide lines, the following rules must be respected:

- The guide lines must be designed in the same direction as you select the inclination polylines **B**.
- The vertices of the inclination polylines must be exactly on the guide line **C**.
- If you select the guide lines **C**, you must select them in the same direction as the inclination polylines **B** were designed.

To fill an area with polyarcs with random ray length you can proceed as follows...
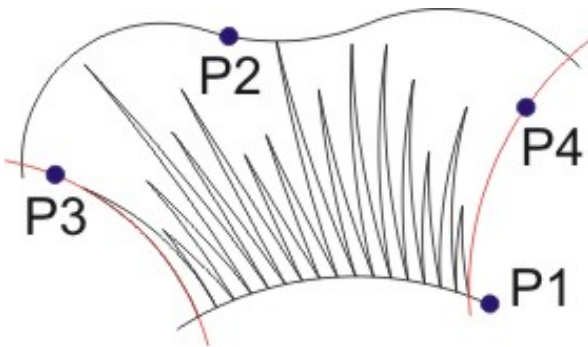

```
Command: ARCFILL
Select boundary edge 1/<Parameter>: <Ret>

>> Define Random=70 in the menu <<

Select boundary edge 1/<Parameter>: P1
Select boundary edge 2: P2
Select inclination lines...
Select entities: P3
Select entities: P4
Select entities: <Ret>
```

AREAFILL - How to fill an area with color

Toolbar: **Fill > Solid filled area**

Menu: **Sketch > Solid filled area**

Keyboard: **AREAFILL**

This command is also used to fill an area with a solid color.

After calling up the command, you can choose a point within an area, where you want a border to be constructed. You can fill areas until you terminate the command with **<Ret>**.

```
Command: AREAFILL
Boundary/<Point in area>: Point in area
Boundary/<Point in area>: <Ret>
```

If the design is getting bigger it can take long time to find the boundary. In this case you can use the **Boundary** option to select the elements, which are recognized for the boundary calculation.

```
Command: AREAFILL
Boundary/<Point in area>: B
Select boundary edge (s)...
Select entities: select boundary 1
...
Select entities: select boundary x
Select entities: <Ret>
Boundary/<Point in area>: Point in area
Boundary/<Point in area>: <Ret>
```

The resulting solid fill will be placed on layer **1**, the area boundery in layer **3**.

Attention! If the fill should be successful, the selected area must be closed!

BORDER - How to design a polyline around an area

This command is used to create a closed border. After calling up the command a menu is opened.

In the menu you can choose from the following options...

Select boundary

With this option you can manually create a closed area.

First select the elements, which are necessary to create the desired area. The selected elements are copied to the layer **AMD_BORDER** and all other layers are switched off, so that you can see only the which are necessary for the construction of the boundary.
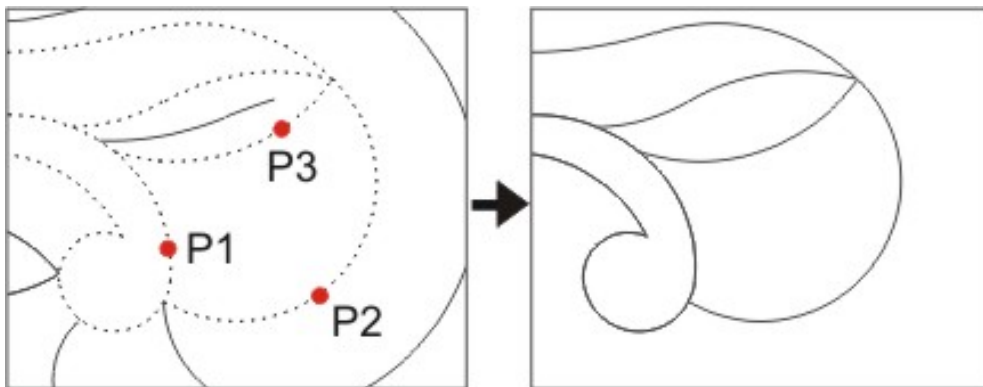
Now you can use the normal trimming commands like **JOIN** , **TRIM** etc. to construct the desired area.
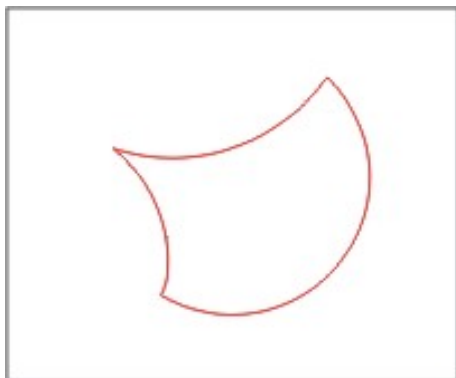
Finally you can use the command **TOVIEW** to switch back to the normal work view.

The resulting polyline will be placed on the layer **AMD_BORDER**. It can be erased with the option **Delete**.

Find boundary

After calling up this option, you can choose a point within an area, where you want a border to be constructed.
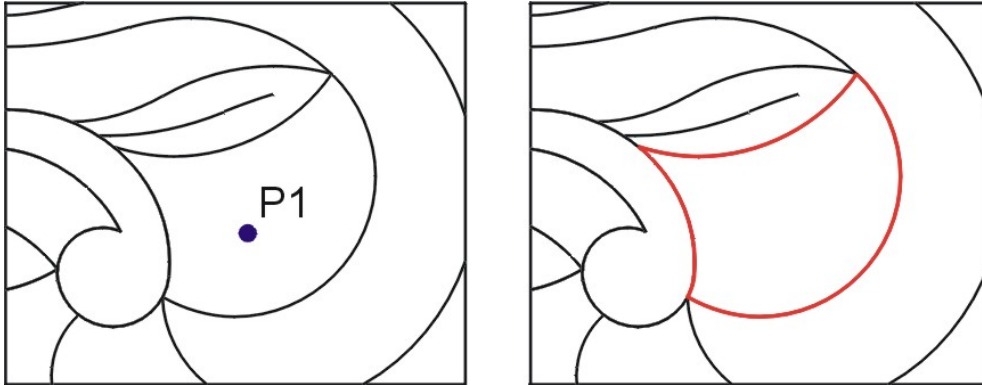

```
Boundary/<Point in area>: P1
```

If the design is getting bigger it can take long time to find the boundary. In this case you can use the **Boundary** option to select the elements, which are recognized for the boundary calculation.
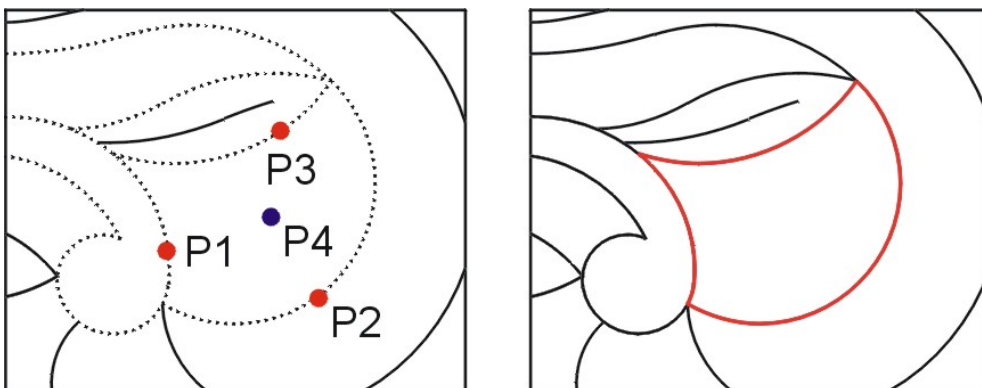

```
Boundary/<Point in area>: B
Select boundary edge (s)...
Select entities: select P1
Select entities: select P2
Select entities: select P3
Select entities: <Ret>
Boundary/<Point in area>: P4
```

The resulting polyline will be placed on the layer **AMD_BORDER**. It can be erased with the option **Delete**.

> **Attention:** If the search for boundary edges should be successful, the selected area must be closed!

Delete

With this option you can erases borders constructed with the previous options.

Done

Terminate the command

CHAIN2P - How to arrange 2 point blocks

Toolbar: **Construct > 2 point chain**

Menu: **Construct > 2 point chain**

Keyboard: CHAIN2P

Blocks, which are defined with 2 points (i.e. arches, pointed arches, dots etc.) can be arranged along a guide line with this command, whereby either the quantity of elements or the size of elements must be selected.

First select a guide line, along which the blocks must be arranged. Then a menu with predefined blocks will appear. Select the desired block, and finally you may define the quantity of the blocks and the block size.

```
Command: CHAIN2P
```
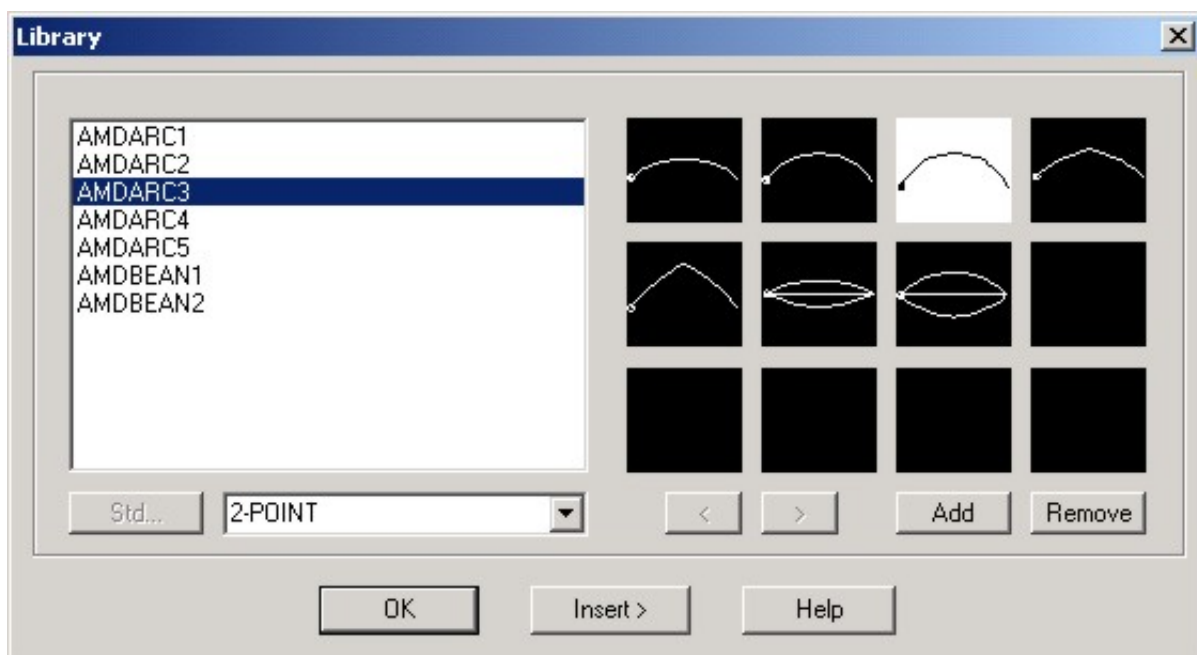


```
Select guide line (s) ...
Select entities: P1
Select entities: <Ret>
Number of items <0>: 5
Size of block <0.00> : <RET>
```
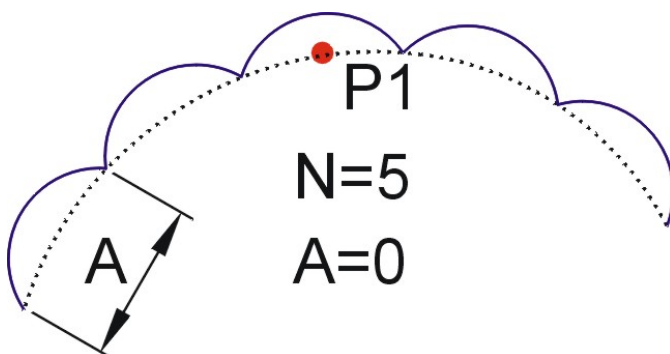

If you define **Number of items = 0**, you may define the block size and the distance between the blocks. The quantity of the blocks will then be calculated based on this

size of the block and the length of the guideline.

```
Command: CHAIN2P

>> Block dialog-box <<

Select guide line(s) ...
Select entities: P1
Select entities: <Ret>
Number of items <0>: <Ret>
Size of block <1.00>: 5
Distance between blocks <0.00>: <Ret>
```
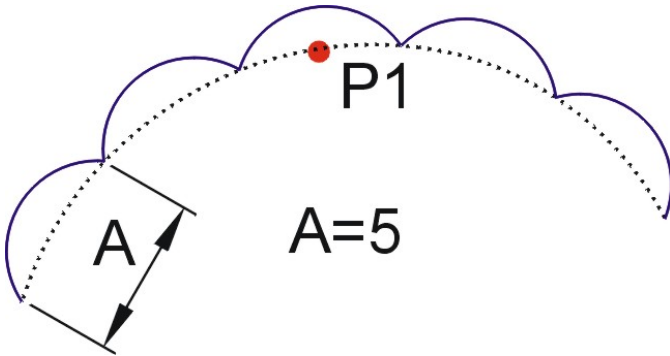

Attention: To get a correct result, the blocks must be
standardized on 1 mm. (see blocks)

CHAINC - How to arrange center blocks

Toolbar: **Construct > Center chain**

Menu: **Construct > Center chain**

Keyboard: **CHAINC**

Blocks, which are defined by their center (i.e. circles, stars, asterix etc.) can be arranged along a guide line, whereby you can select the quantity or the size of the elements.

First select the guide line, along which the blocks must be arranged. Then a menu with predefined blocks will appear. Select the desired block.

Now you may define the quantity of blocks and the block size. If no block size is defined, a chain of blocks will be constructed without distance between the blocks.

```
Command: CHAINC
```
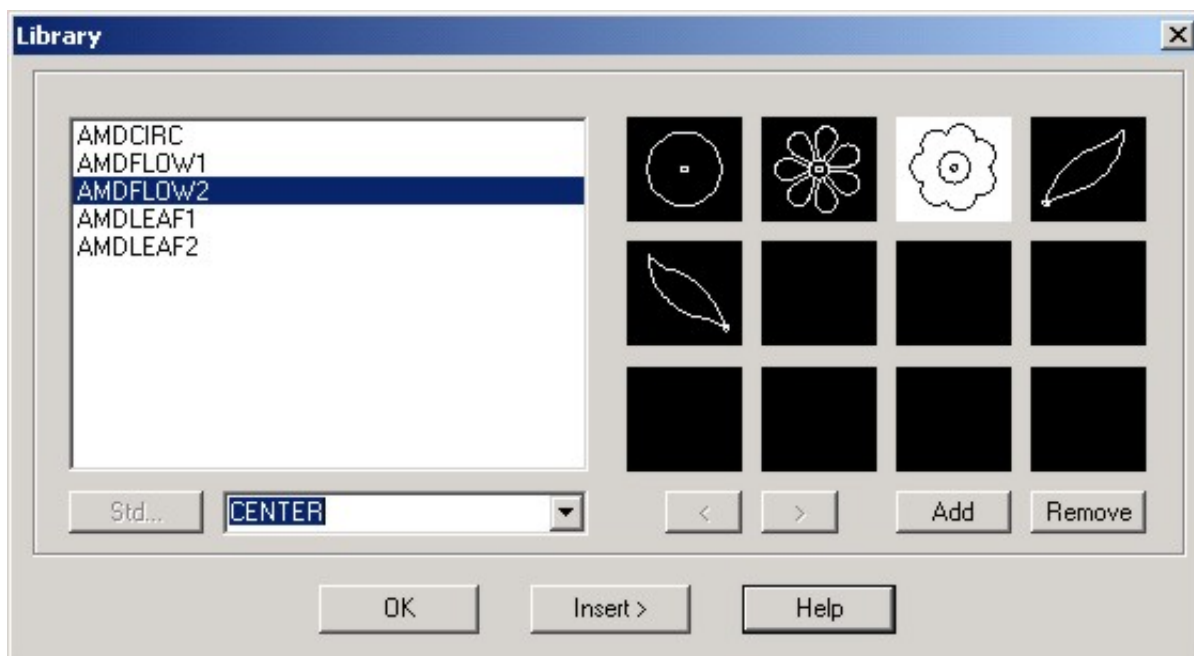


```
Select guide line(s) ...
Select entities: P1
Select entities: <Ret>
Number of items <0>: 4
Size of block <1.00>: 3
```
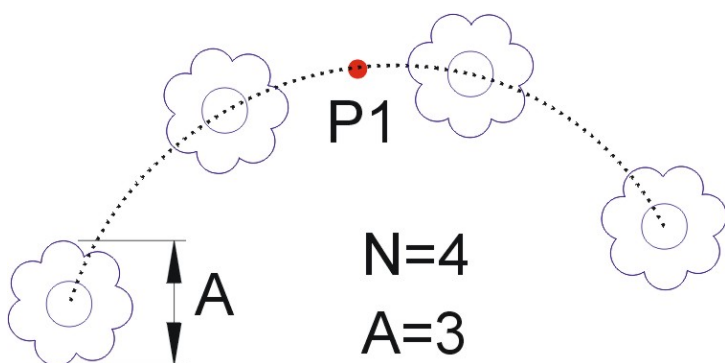

If you define **Number of items = 0**, you may define the block size and the distance

between the blocks. The quantity of the blocks will then be calculated based on this size of the block and the length of the guideline.

```
Command: CHAINC

>> Block dialog-box <<

Select guide line(s) ...
Select entities: P1
Select entities: <Ret>
Number of items <0>: <Ret>
Size of block size <1.00>: 3
Distance between blocks <0.00>: 3.5
```
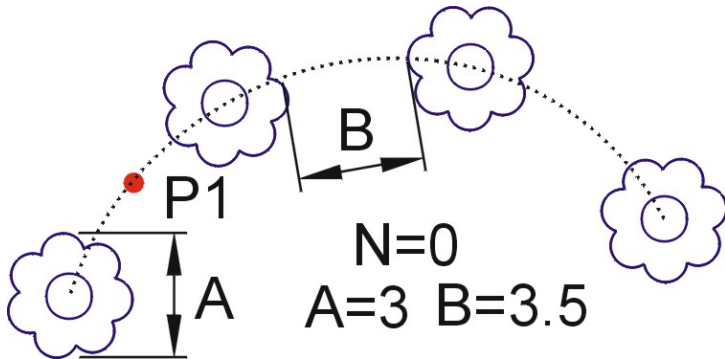

Attention: To get a correct result, the blocks must be standardized on 1 mm. (see blocks)

MODULE - How to work with modules

Toolbar: **Construct** > **Module** ▦

Menu: **Construct** > **Module**

Keyboard: **MODULE**

A module is a pattern of stitches, which can be recorded under a certain name and arranged along a polyline (running stitch).
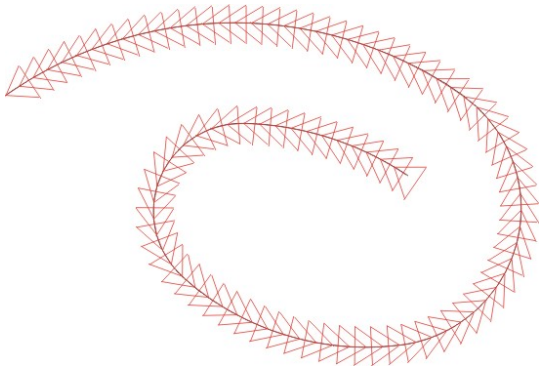
See more...

▤ [How to use modules](#)

MANSTI - How to design single stitches

Menu: **Construct** > **Manual stitches**

Keyboard: MANSTI

With this function you may design stitches or inclination lines.

First select the boundary edges, where the stitches must be cut, next design the stitches or inclination lines, finally the stitches will be cut at the selected boundary edges.

Attention: Design the stitches in a way that the stitches intersect the boundary edges at both ends.
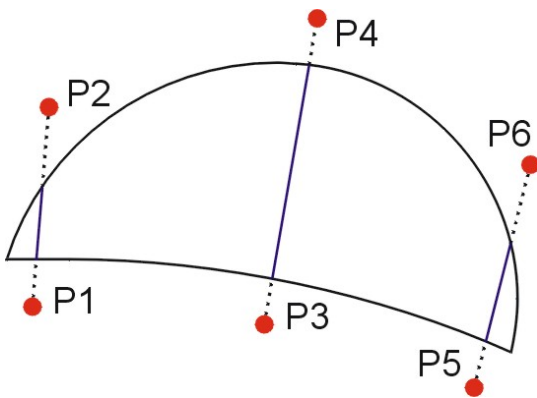
```
Command: MANSTI
Select boundary edge(s) ...
Select entities: select
Select entities: <Ret>
Direction of stitches...
From point: P1
To point: P2
...
From point: P5
To point: P6
From point: <Ret>
```

All stitches or inclinations are saved on the layer **AMD_BLATTSTITCHES.**

How to work with blocks

Block commands can be called up from the **Construct toolbar**.

A block is a set of lines, arcs, circles etc. which are joined to form a complex object. This group of elements will be given a block name. A block can be recalled at any time, at any place, in any size and under any angle and inserted into the design. A block will be handled as a single object, it can be displaced, copied, deleted etc. It makes sense to create and memorize blocks of often used elements such as leaves, flowers, stars etc. These blocks can later be recalled and inserted in any desired design.
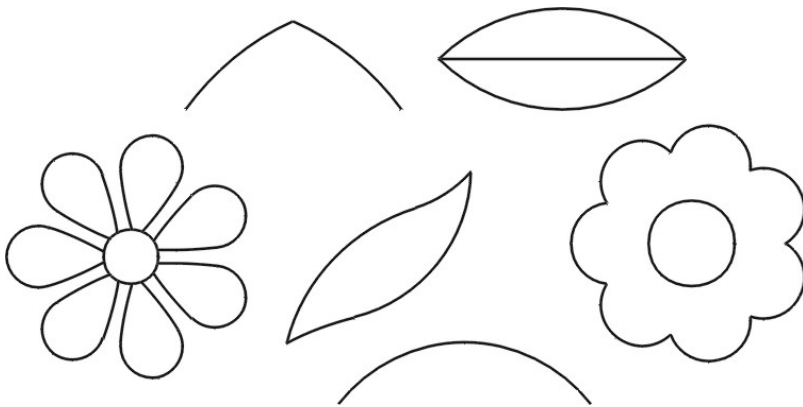
There are three types of blocks:

- Blocks, delivered with the system. These blocks are filed in the subdirectory: **C:\ICAD\BIB\S** (S for system) and the file name of these blocks starts with **AMD**.

- Blocks created by the operator and filed on hard disk. These blocks are filed in the subdirectory: **C:\ICAD\BIB\U** (U for user). The block name may not start with **AMD**.

- Blocks created by the operator, however only used with the actual design. These blocks may not be filed on the hard disk.

The subdirectory is being defined using the system variable **AMD_BIB**. In the library subdirectory the directories **U**, **S**, **SLD** and **DATA** are located. This system variable can be initialized in the file **EDS.INI**.

```
AMD_BIB=C:\ICAD\BIB
```

Attention: When inserting filed blocks you have to define an enlargement factor. This enlargement factor is based on the original size of the block. If a block in original has a size of i.e. 6 mm and you define a factor 3, the inserted block will have a size of 18 mm. For this reason all blocks should be filed with the size 1 mm. In this case the enlargement factor will correspond to the real size. With the ⊞ **LIBMAN** command you may create blocks, which are normalized on 1 mm.

The **Block Toolbar** is a flyout of the **Construct Toolbar** and contains the following commands…

	Block Toolbar
	LIBMAN - How to use the library manager
	INSERT - How to insert blocks by name
	DEFBLOCK - How to define a new block
	PAILLETT - How to insert paillettes
	BLOCKNR - How to count number of blocks

LIBMAN - How to use the library manager

Menu: **Construct > Blocks > Librarian...**

Keyboard: **LIBMAN**

With this command you can insert blocks from an icon-menu.
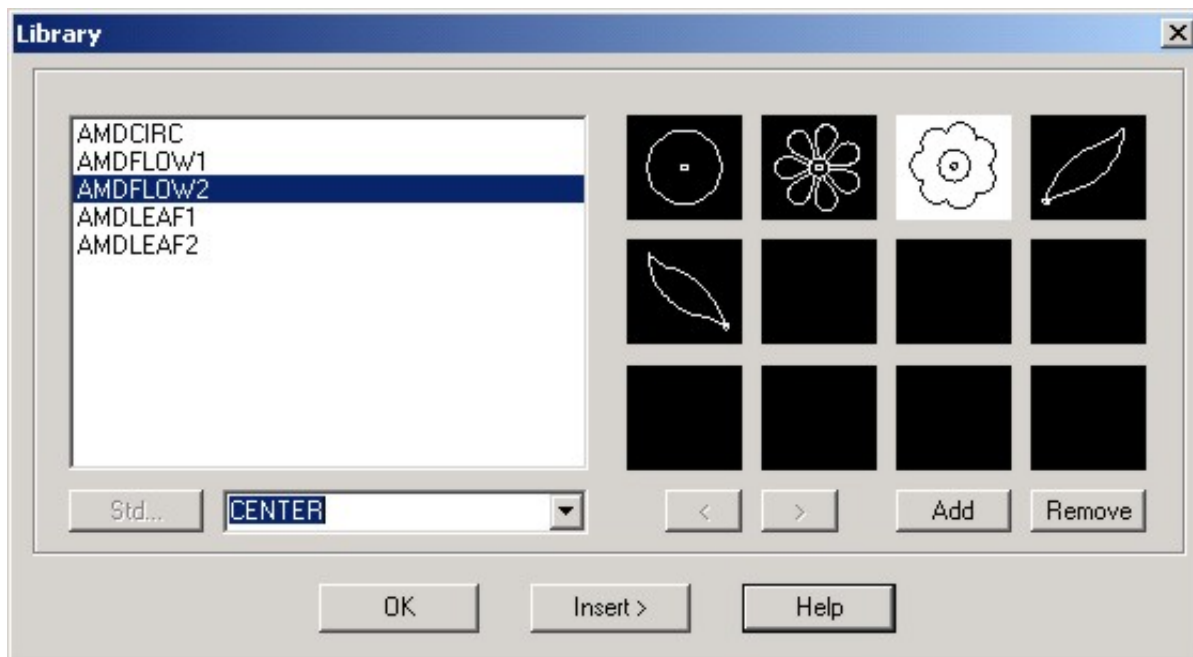
Icon menus are divided into libraries. The library can be selected from the popup list below the symbol list. Libraries can be added by editing the **SYMLIB.CSV** file in the **DATA** directory.

The symbols can be selected by picking an icon or by selecting the symbol description in the symbol list. If there are more than 12 symbols within one library, you can go to the next or to the previous page by picking one of the arrow buttons.

If you activate the **Insert** field, the selected symbol can be inserted in the drawing as often as you need it. When you are ready with inserting you can use **<Ret>** to come back to the dialog-box.

Add - How to add a symbol to the library

Via the button **Add** you can create a new block and add it to the icon menu. The new block is inserted at the position of the actually selected symbol.

First, as in the standard block definition, you have to specify a block name and the insertion point. Then you can define the description, displayed in the dialog-box.

If you want to use the block with the command **CHAIN2P** and **CHAINC**, the reference length must be defined. The Reference length corresponds to the width of the object in X direction.

Finally you must select the elements joint in the block.

To redefine an existing block named **FL1** with its original size you can proceed as follows...
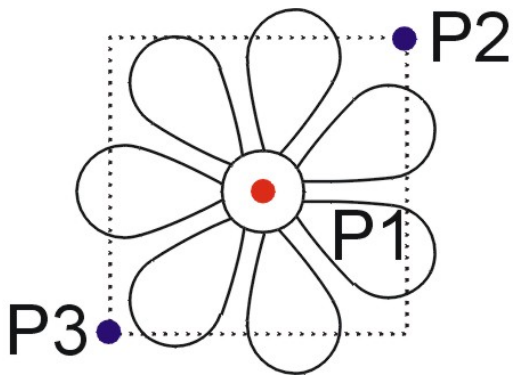

```
Block name: FL1
Block FL1 already exists.
```



```
Description: Flower 7 paddles
Insertion point: P1
```



```
Select entities: P2
Select entities: P3
Select entities: <Ret>
```

To redefine an existing block named **FL1** as standard block with 1mm size you can proceed as follows...


```
Block name: FL1
Block FL1 already exists.
```
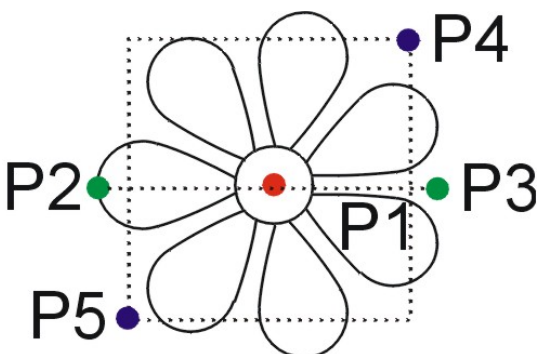
```
Description: Flower 7 paddles
Insertion point: P1
```



```
Reference length: P2
Second point: P3
Select entities: P4
Select entities: P5
Select entities: <Ret>
```

Attention: If a block already exists, you will be asked, if you really want to overwrite the existing block. If you overwrite the block, the existing block in the library will be changed and the new block will be inserted at the desired position.

Attention: When you create a block, the actual view on the screen will be displayed in the slide of the icon menu. Therefore it is important, that you display the block as big as possible, before you create a block.

Remove - How to remove a symbol from the library

Via the button **Remove** you can remove the selected symbol from the library. On deletion of the block, the slide as well as the entry in the icon-menu will be erased.

Before the symbol will be erased, you will be asked, if you really want to erase it.

INSERT - How to insert blocks by name

Toolbar: **Block > Insert by name**

Menu: **Construct > Blocks > Insert by name**

Keyboard: **INSERT**

With the command **INSERT** you may insert blocks from the library.

After calling up this command, you must first enter a block name. Your next step is to determine the insert point. This point corresponds with the base point of the block. Finally you have to define the scale factor and the insertion angle.

To insert a block named **flower1** with scale factor **10** and rotation angle **0** proceed as follows:

```
Command: INSERT
Block name <last>: flower1
Insertion point: base point of block
Scale factor <1>: 10
Rotation angle <0>: <Ret>
```

Note: If * is being placed ahead of the block name, the block will directly be exploded upon insertion.

DEFBLOCK - How to define a new block

Toolbar: **Block > Define block** 🗊

Menu: **Construct > Blocks > Define block**

Keyboard: **DEFBLOCK**

With the command **DEFBLOCK** you can create a block definition, this means, you combine parts of an existing design to blocks.

First you define a block name which can have up to **8 characters** (characters, numbers as well as "_" and "-"). If you have already used the name for another block the system will ask you if you want to redefine this block. If you answer the question with yes, blocks, already inserted in the design, will be replaced with the new block.

After you have defined the block name you have to specify the base point. This is the point, on which the block has to be inserted into the design. Around this point the block can also be rotated on insertion.

The elements, which form the block, can be determined with the usual object selection methods.

To redefine an existing block named **FL1** with its original size you can proceed as follows...
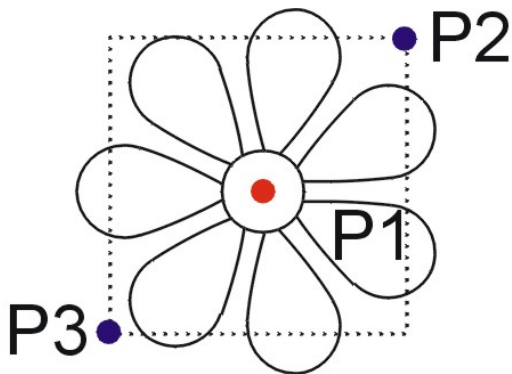

```
Command: DEFBLOCK
Block name: FL1
Block FL1 already exists.
```



```
Insertion point: P1
Select entities: P2
```
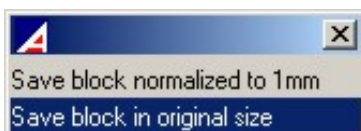
```
Select entities: P3
Select entities: <Ret>
```

DEFBLOCK also allows for simple definition of standard blocks (blocks with 1 mm size) used with the command 🖊 **CHAIN2P** and 🖊 **CHAINC**.
After the definition of the insert point, a reference length must be defined. The Reference length corresponds with the width of the object in horizontal direction.

To redefine an existing block named **FL1** as standard block with 1mm size you can proceed as follows...
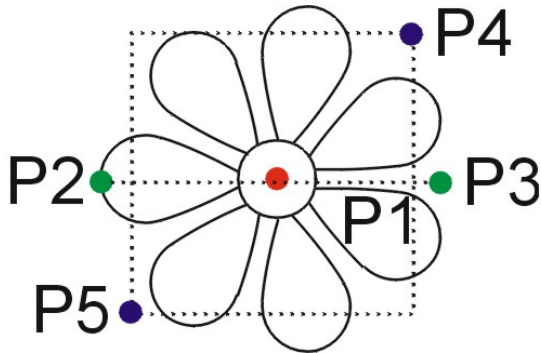

```
Command: DEFBLOCK
Block name: FL1
Block FL1 already exists.
```



```
Reference length: P2
Second point: P3
Insertion point: P1
Select entities: P4
Select entities: P5
Select entities: <Ret>
```

PAILLET - How to insert paillettes

Toolbar: **Block > Paillettes**

Menu: **Enlarge > Paillettes...**

Keyboard: **PAILLET**

With this command you can insert paillettes from an icon-menu.

The paillettes can be selected by picking an icon or by selecting the symbol description in the symbol list. If there are more than 12 symbols within one library, you can go to the next or to the previous page by picking one of the arrow buttons.

If you activate the **Insert >** field, the selected paillette can be inserted manually or it can be distributed along a polyline.

Distribute paillettes along a polyline

To insert paillettes along a polyline just select one or more polylines, along which the paillettes should be distributed, and then specify how many paillettes you want to have inserted on a single polyline.

To arrange **7** paillettes along a polyline you can proceed as follows...
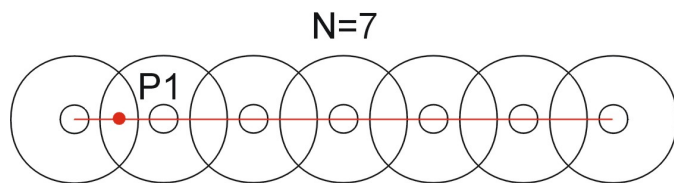

```
Select entities: P1
Select entities: <Ret>
Number of items <0>: 7
```

You can also specify the distance between the paillettes. In this case the system calculates the quantity for you. To specify the distance confirm the request for number of items with **<Ret>**, then you can specify the distance between the paillettes.

To arrange paillettes with a distance **A** along a polyline you can proceed as follows...
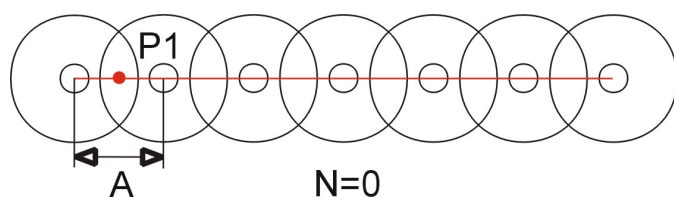

```
Select entities: P1
Select entities: <Ret>
Number of items <0>: <Ret>
```

```
       Distance between block <last>: specify distance A
```

Insert paillettes manually

To insert paillettes manually just click **<Ret>** instead of selecting polylines. Now the paillette hangs at your cross-hair and you can insert as many paillettes as you need it. When you are ready with inserting you can use **<Ret>** to come back to the dialog-box.

```
       Select entities: <Ret>
       Insertion point: center of paillette 1
       ....
       Insertion point: center of paillette x
       Insertion point: <Ret>
```

'Add' field - Add a paillette to the library

Via the button **Add** you can create a new paillette and add it to the icon menu. The new paillette is inserted at the position of the actually selected paillette.

First, as in the standard block definition, you have to specify a name and the base point. Then you can define the description, which is displayed in the dialog-box. Finally select the elements joint in the block.

To add a paillette with the name **PA5** and description **5mm** proceed as follows...
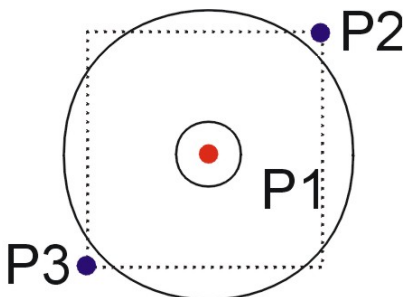

```
       Block name: PA5
       Insertion point: P1
       Description: 5mm
       Select entities: P2
       Select entities: P3
       Select entities: <Ret>
```

Attention: If a paillette already exists, you will be asked, if you really want to overwrite the existing one. If you overwrite the paillette, the existing block in the library will be changed and the new block will be inserted at the desired position.

Attention: When you create a paillette, the actual view on the screen will be displayed in the slide of the icon menu.
Therefore it is important, that you display the block as big as possible, before you create the paillette.

'Removel' field - Remove a symbol from a library

Via the button **Remove** you can remove the selected paillette from the library. On deletion of the paillette, the slide as well as the entry in the icon-menu will be erased.

Before the paillette will be erased, you will be asked, if you really want to erase it.

BLOCKNR - How to count number of blocks

Toolbar: **Block > Count blocks**

Menu: **Construct > Blocks > Count blocks**

Keyboard: **BLOCKNR**

The command **BLOCKNR** allows you to find out, how many blocks with a certain name are inserted in a design. This can be used to count e.g. pailettes.

```
Command: BLOCKNR
Select block: Select block to be counted
```

DDINSERT - How to insert other blocks

Toolbar: -

Menu: Construct > Blocks > Insert...

Keyboard: DDINSERT

The command **DDINSERT** is the universal command to insert a block. This command allows you to insert any block, filed in any directory.
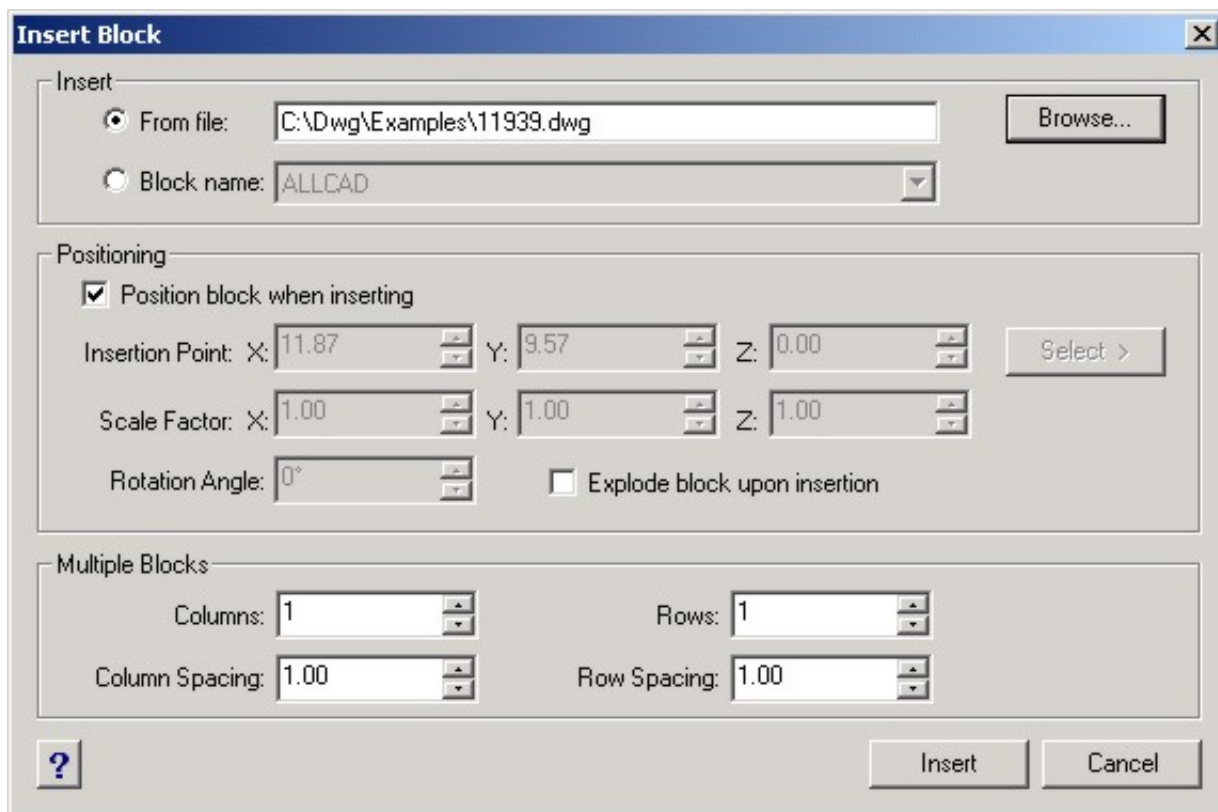
This command makes sense, when a complete design has to be inserted in the actual design, or if you want to insert a block with different XY factors.

After calling up the command, a dialog-box appears, where you can choose the desired block.

```
Command: DDINSERT

>>> Dialog-box <<<

Insertion point of block: insertion point
Corner/XYZ/X-scale factor <1>: <Ret>
Y-scale factor<Equal to X scale=(1)>: 2
Rotating angle for block <0>: <Ret>
```

Note: If you activate the option **Explode block** within the dialog-box, the block will be exploded directly on insertion.

Note: With this command, blocks with different XY factors can also be inserted. This can be used to scale a complete design in perceptually in one direction.

Attention: Blocks with different X and Y factors cannot be exploded with the command **EXPLODE**.

System settings

In this chapter we talk about various system settings. All setting functions are located in the Toolbar **Object properties** or in the pull down menu **Modes**.

The **Object properties Toolbar** contains the following commands...

	Object properties Toolbar
	[DDLMODES - How to manage layers](#)
	[TOVIEW - How to switch layer groups](#)
	[LSET - How to activate layers](#)
	[LON - How to switch layers on/off](#)
	[LX - How to change the layer of an element](#)
	Flyout Object Snap (See chapter 1)

Other settings in the pull down menu

The following commands cannot be selected from a toolbar. They must be called up from the pull down menu.

[DDRMODES - How to adjust settings of drawing aids](#)

[DDGRIPS - How to adjust grips](#)

[DDSELECT - How to adjust entity selection](#)

[DDOSNAP - How to adjust entity snap](#)

[SETDEF - How to save and load embroidery parameters](#)

[TABLET - How to configure the digitizer command and screen area](#)

Layer administration

Parts of a design can be laid on different layers. The layers **AMD_WORK**, **AMD_BORDER**, **AMD_AUXLIN** and the layers **5** to **99** are available for the user. All other layers are automatically managed by the system.

DDLMODES - How to manage layers

Toolbar: Object properties > Layers

Menu: Modes > Layer > Layer

Keyboard: DDLMODES

With **DDLMODES** a dialog-box is activated, which will give you control of all layers.
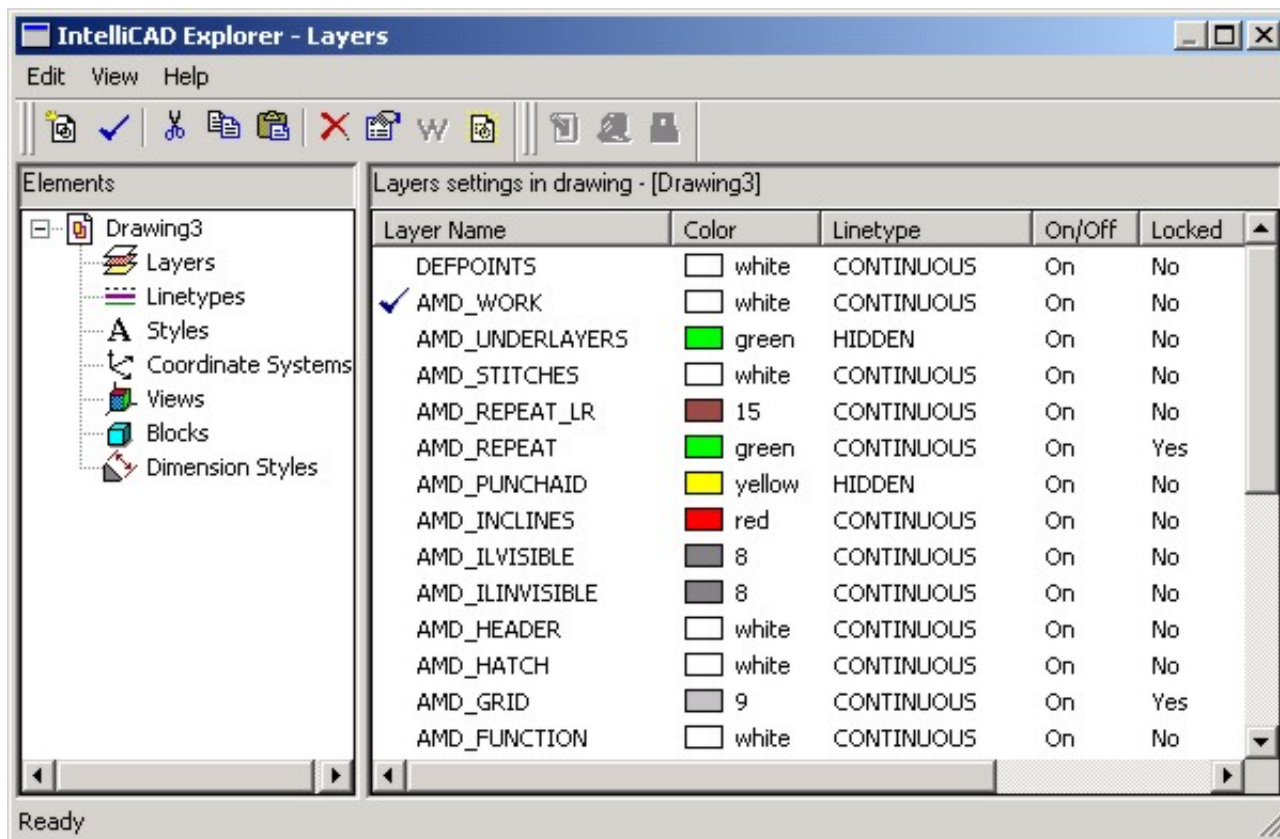
You can...

- activate a layer by double clicking on the desired layer name. The active layer has a tick left from the layer name.

- create new layers with **Edit > New > Layer**

- delete a layer with all elements, laying on the layer, with **Edit > Delete**

- rename a layer with **Edit > Rename**

- changing the color of a layer by clicking on the color rectangle in the **Color** column and choosing a new color from the upcoming dialog-box.

- switching On/Off a layer by clicking on the On or Off in the **On/Off** column.

- locking or unlocking a layer by clicking on the No or Yes in the **Locked** column.

Attention: Only change settings of your own layers. Do not change layer names or layer settings of layers, managed by the system! System layers start with the characters **AMD_**. Also the layers **0** - **9** are system layers

Hint: If you should work on a light image, it can be very difficult to see the white tracing line when you are taking over the image. To change the tracing color click on the color rectangle right from the layer **AMD_WORK** and choose another color from the color dialog-box.

TOVIEW - How to switch layer groups

Toolbar: Object properties > Layer group

Menu: Modes > Layer > Layer group

Keyboard: TOVIEW

The command **TOVIEW** makes layer groups visible or invisible, depending on the job to be carried out.
After calling up the command a menu opens, where you can select the layer group, which you want to have activated.

In the parameter menu you can choose from the following options...

Sketch

All layers, necessary for the output of a 1:1 sketch, are activated. Stitches and other elements of an enlargement are not visible.
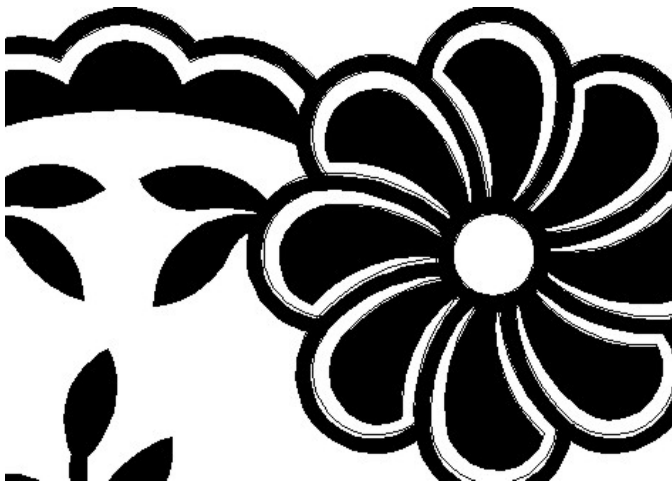
Enlargement

All layers, necessary for the output of a correct enlargement, will be activated. Inclination lines, infolines etc. are not visible.
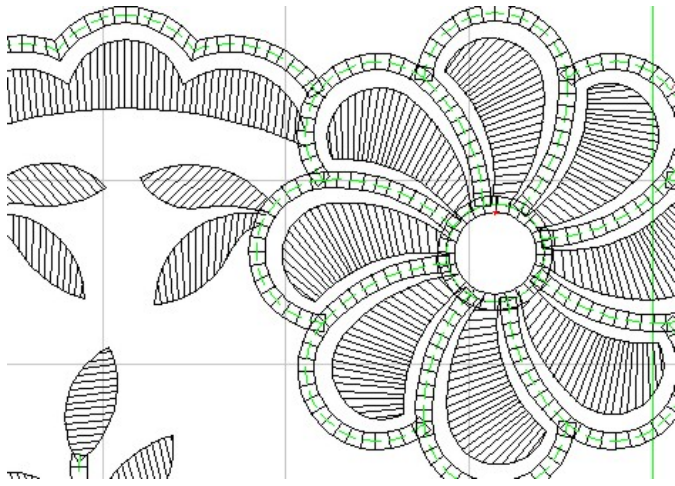
Punch

Only the layers with the infolines, which represent the punch stitches, plus the repeat and the header are activated.
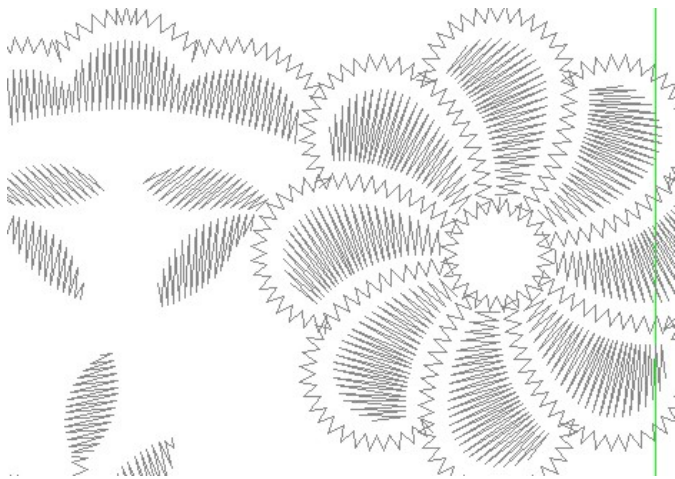
Work

Layers, necessary to work on an enlargement, are activated. Infolines, inclination lines and all automatically created embroidery elements are included in the group.
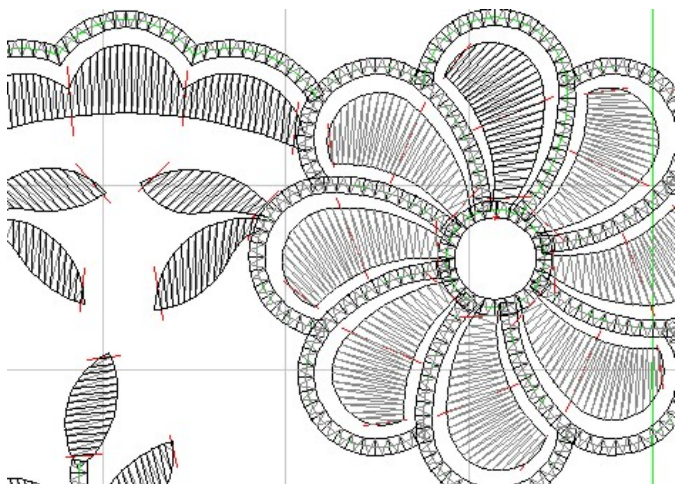
LSET - How to activate layers

Toolbar: **Object properties > Layer Set** 🖼

Menu: **Modes > Layer > Layer Set**

Keyboard: **LSET**

A layer is defined as active, when new design elements are filed on it. After calling up the command **LSET** a menu is displayed, where you can choose the layer, which you want to activate.
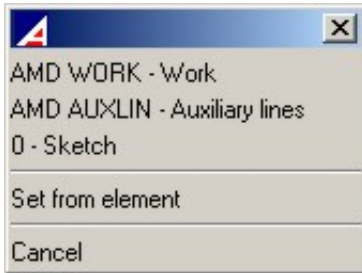
By choosing the option **Set from element** you can taken over the layer, which you want to activate, from an element in your design.

```
Select element on desired layer: select item
```

LON - How to switch layers on/off

Toolbar: Object properties > Layer On/Off

Menu: Modes > Layer > Layer On/Off

Keyboard: LON

Elements on a layer can be visible (On) or invisible (Off). Only Elements on visible Layers can be displayed on the screen or can be plotted.

After calling up the command a menu is displayed, where you can see the status of certain layers. Layers with a << behind their name are actually switched on, layers without << are switched off. By clicking the desired layer you can change the status of the layer from on to off or from off to on.
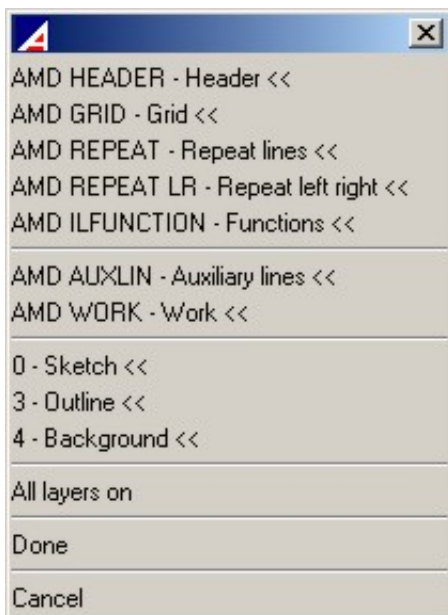
By choosing the option **All layers on** you can switch on all layers in the actual design and hence make visible all elements of the design.

LX - How to change the layer of an element

Toolbar: Object properties > Change layer

Menu: Edit > Change > Change layer

Keyboard: LX

With the command **LX** you can change the layer of selected elements. There are three ways to change the layer. The new layer can be the actual layer, the layer, another element belongs to, or the layer, selected from a dialog-box.

Actual

With the option **Actual** the selected elements will be placed on the actual layer.

```
Command: LX
Select entities: selection
Select entities: <Ret>
Actual / From element/ From List: select Actual
```

From element

With the option **From element** you can move the selected elements to the layer, which a picked element belongs to.

First you have to select the elements you want to change, then you have to pick an element, that belongs to the desired layer.

```
Command: LX
Select entities: selection
Select entities: <Ret>
Actual / From element/ From List: select From element
Element on desired layer: pick element
```

From List

You can also define the desired layer by selecting it from a dialog-box. First you have to select the elements, which you want to have changed, then you have to select the destination layer from a dialog-box.

```
Command: LX
Select entities: selection
Select entities: <Ret>
Actual / From element/ From List: select From List
```

Layers

```
4
5
6
7
9
AMD_AUXLIN
AMD_BORDER
AMD_CONSTLIN
AMD_DEFBLOCKS
AMD_FUNCTION
AMD_GRID
AMD_HATCH
AMD_HEADER
AMD_ILINVISIBLE
AMD_ILVISIBLE
AMD_INCLINES
AMD_PUNCHAID
AMD_REPEAT
```

OK   Cancel
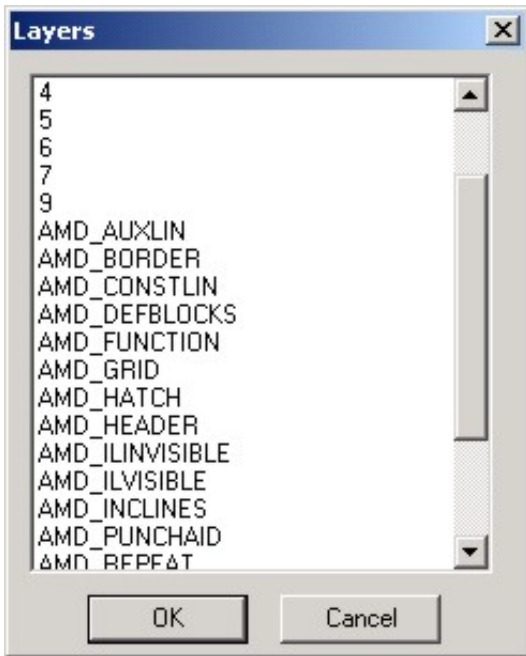
DDRMODES - How to adjust settings of drawing aids

Menu: Modes / Drawing aids ...

Keyboard: DDRMODES

Using **DDRMODES** a dialog-box will be activated, which allows to control the drawing aids.

The following drawing aids can be controlled:

- **Grid** For easier positioning of design elements, you can define a visible point grid. The grid is only a designing aid and is no part of the design itself. The grid distance can be controlled and the grid itself can be switched on and off.

- **Snap** With the snap function you can define an invisible grid. With the cross hair you can only snap points on the grid. The distance of these grid points can again be defined and the snap mode can be switched on and off.

- **Orthogonal** The orthogonal mode allows for absolute vertical and horizontal designing independent from the input with the design device. This option can be switched on and off.
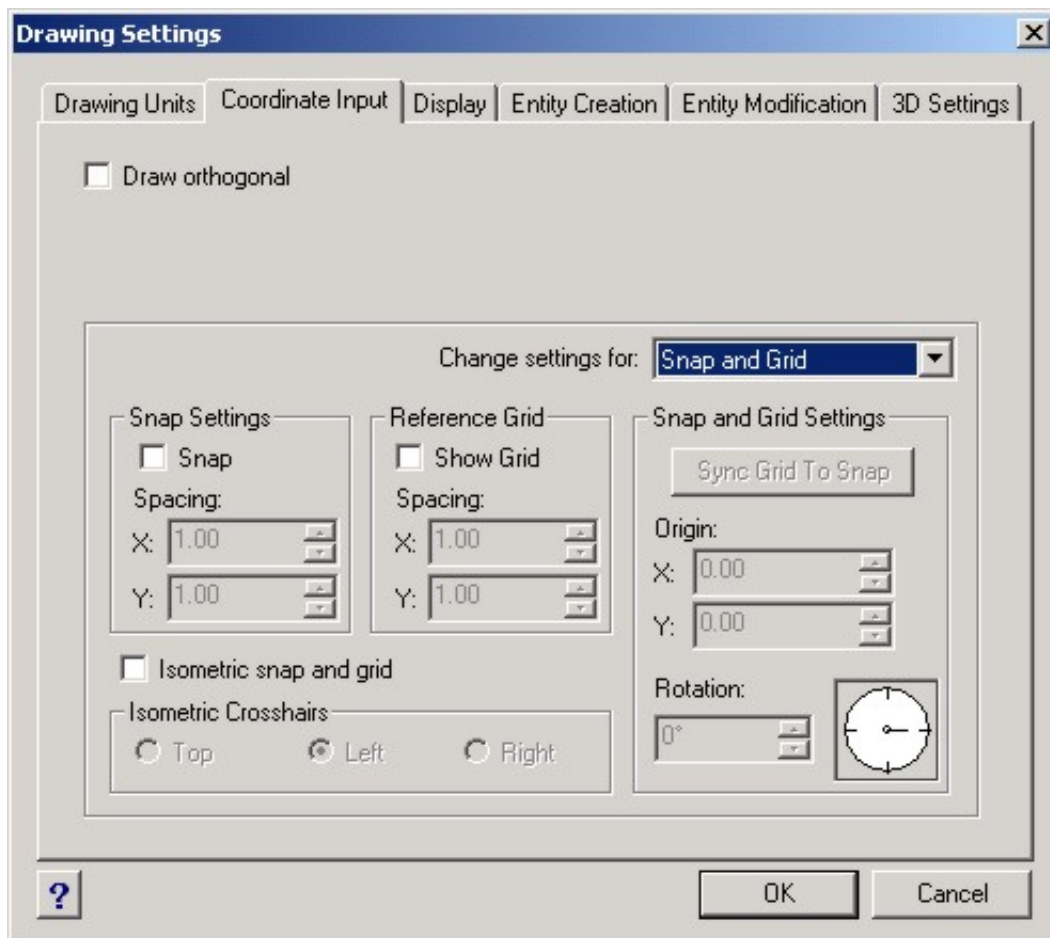
DDGRIPS - How to adjust grips

Menu: **Modes / Grips...**

Keyboard: **DDGRIPS**

Selected entities can be edited using the handles, which appear on the definition points of the object. The dialog-box **DDGRIPS** allows for modification of handle parameters, such as color and size of the handle.
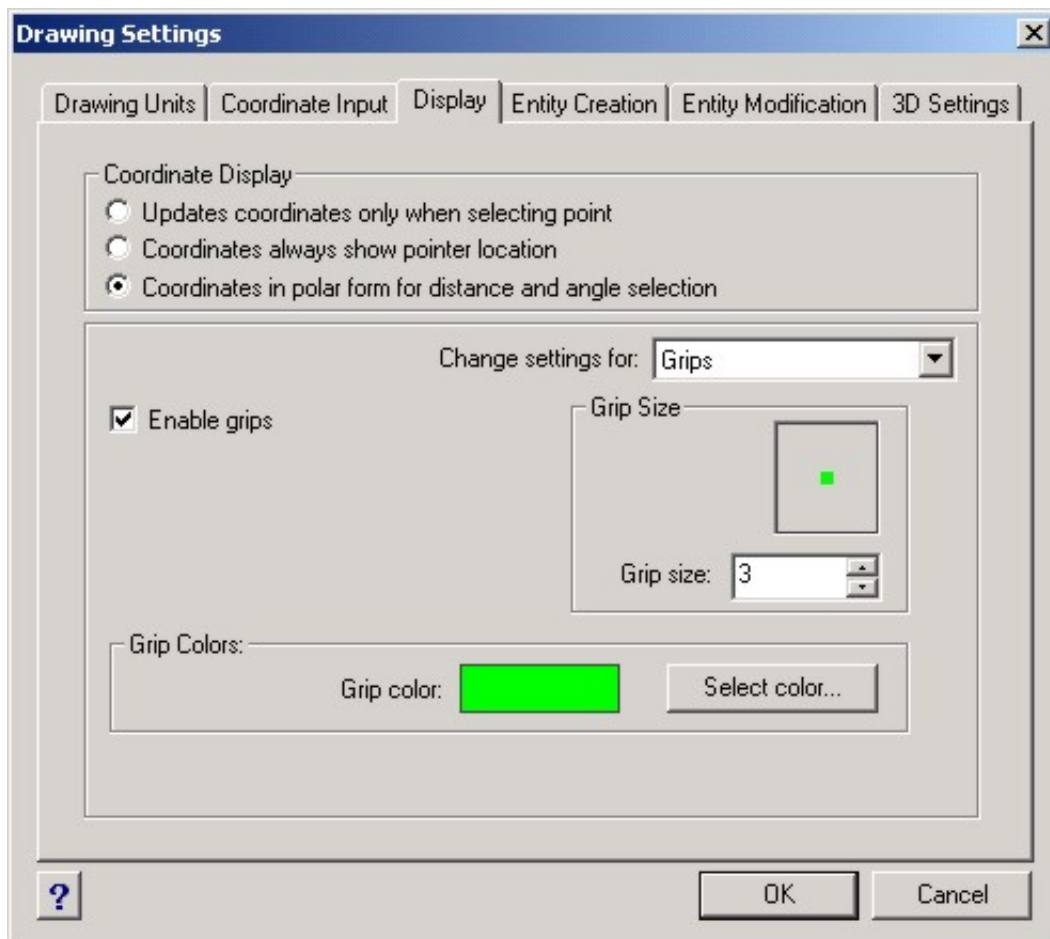
DDSELECT - How to adjust entity selection

[Modes / Entity selection ...] - If you want to modify elements you must first select them. Using the dialog-box DDSELECT you can set object selection parameters, such as size of the pick box.

TABLET - How to configure the digitizer command and screen area

[Modes / Tablet] - With this command, you can align the digitizer overlay.

After calling up the command a popup menu with options is displayed. Enter CFG by keyboard to start the configuration. Now you are asked, if you want to align the overlay. Answer with Yes. Finally you must pick the 5 points shown below in the correct order.

```
Command: TABLET
Command: TABLET
```

```
Tablet: ON/OFF/CALibrate/ConFiGure/<On>: CFG
Do you want to align the tablet overlay? Yes/No/<No>: Y
Digitize upper left corner of overlay: P1
Digitize lower left corner of overlay: P2
Digitize lower right corner of overlay: P3
Digitize lower left corner of screen pointing area: P4
Digitize upper right corner of screen pointing area: P5
```

DDSELECT - How to adjust entity selection

Menu: Modes / Entity selection ...

Keyboard: DDSELECT

If you want to modify elements you must first select them. Using the dialog-box **DDSELECT** you can set object selection parameters, such as size of the pick box.
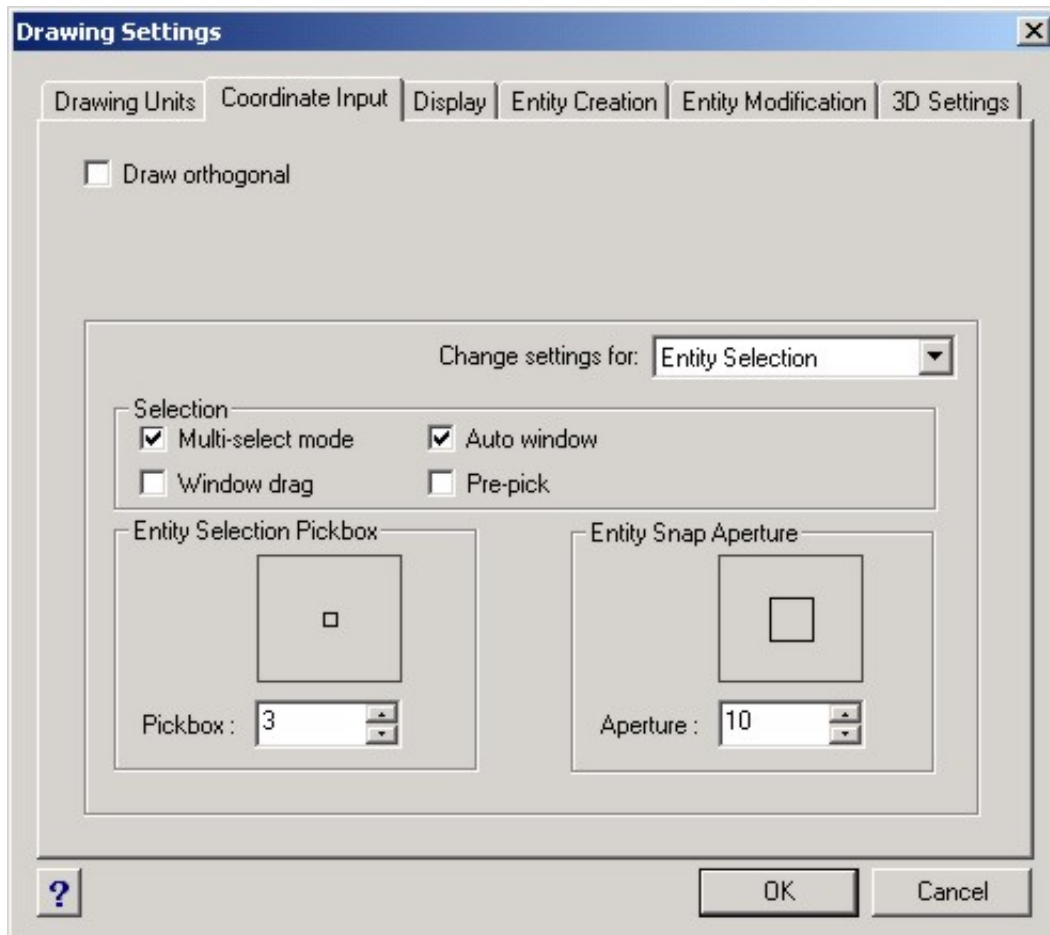
DDOSNAP - How to adjust entity snap

Menu: **Modes / Entity snap ...**

Keyboard: **DDOSNAP**

One or more entity snap methods can be switched on permanently. In this dialog-box you can control the use of entity snap methods.
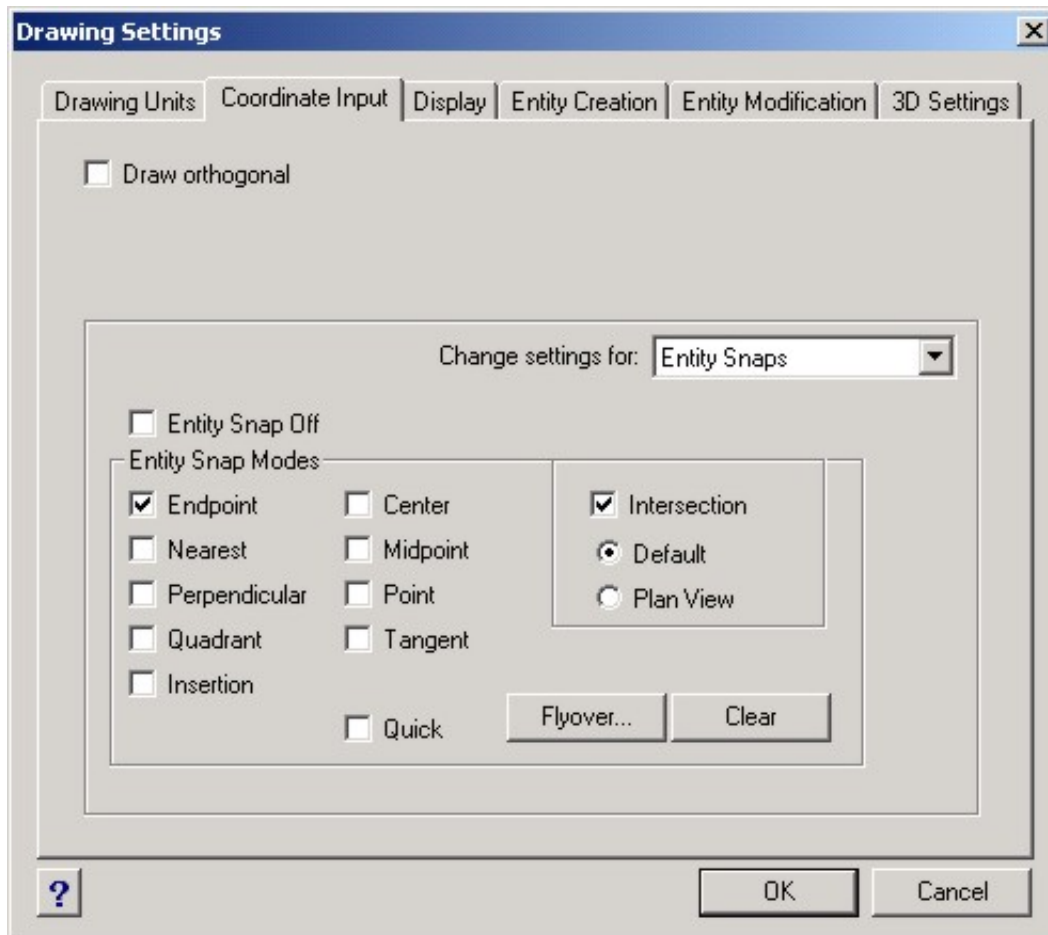
Note: To switch off all entity snap methods you can also use the menu function **Modes > Clear entity snap.**

SETDEF - How to save and load embroidery parameters

Menu: Modes / Embroidery defaults

Keyboard: SETDEF

With this command, you can save, load and reset the parameters of all automatic stitches. So you can define settings for different kind of fabrics and yarns and recall these settings whenever you need them again.

After calling up the command a menu is opened.

In the menu you can choose from the following options...

Save to file

With this option you can save all parameters of the various automatic stitches in a file. After choosing this option the file dialog-box opens where you can specify the file name and the place where you want to save the parameters.

Read from file

With this option you can initialize all parameters of the various automatic stitches from a previously written file. After choosing this option the file dialog-box opens where choose the desired parameter file.

Set default

Sets the parameters of all automatic stitches to the system default value.

TABLET - How to configure the digitizer command and screen area

Menu: Modes / Tablet

Keyboard: TABLET

With this command, you can align the digitizer overlay.

After calling up the command a popup menu with options is displayed. Enter **CFG** by keyboard to start the configuration.
Next you are asked, if you want to align the overlay. Answer with **Yes**.
Finally you must pick the 5 points shown below in the correct order.
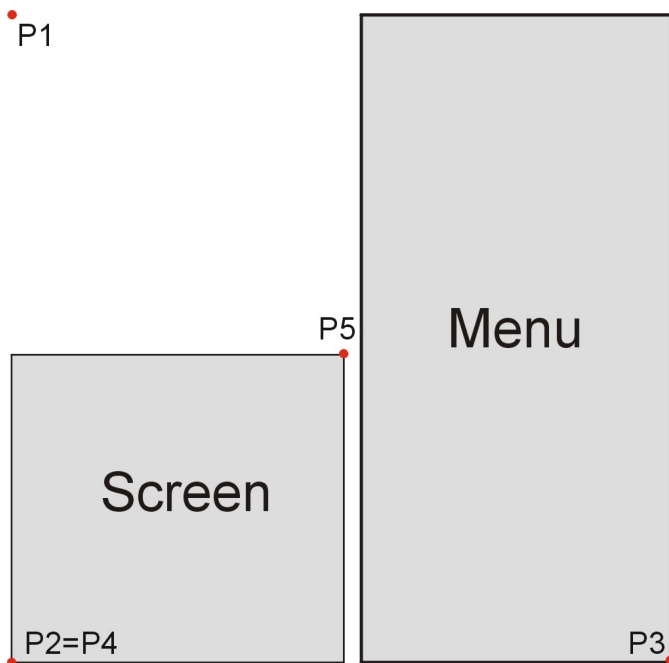

```
Command: TABLET
Tablet: ON/OFF/CALibrate/ConFiGure/<On>: CFG
Do you want to align the tablet overlay? Yes/No/<No>: Y
Digitize upper left corner of overlay: P1
Digitize lower left corner of overlay: P2
Digitize lower right corner of overlay: P3
Digitize lower left corner of screen pointing area: P4
Digitize upper right corner of screen pointing area: P5
```

Automatic embroidery functions

This program allows for construction of embroidery elements automatically as technical enlargements. The exact number of turns can be determined for elements, which are designed in this way and you can use ProLace to directly take over these elements.

How to enlargement with automatic functions

If you want to make an enlargement with basic elements, such as lines, polyarcs, polylines and circles as follows:

- Select the desired automatic functions from the embroidery toolbox or from the pull-down menu. You now will be asked to select the guide line or to input a point.
- Before the selection or before entering the first point you can call up the menu to specify the embroidery parameters. This can be done by clicking the right mouse button.
  Set the embroidery parameters, such as raedle, color, density, etc.

Note: Length values have to be defined in mm, scale 1:1.

- After having specified all the parameters, you can leave the menu via the **Done** option.
- Now you can continue with the next steps. In the following section you will find the explanation for each single function.

Attention: Automatic effects can only be used on polylines and circles.

The **Embroidery Toolbar** contains the following commands...

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	Flyout Blattstitch
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	Flyout Geflect
	Flyout Boring
	Flyout Borer holes
	[ZUCHOLE - How to construct zucholes](#)
	[PICO - How to design lock stitches](#)

	[UNDER - How to add underlayers](#)
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	Edge Toolbar
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	Boring Toolbar
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	[**Borer holes Toolbar**](#)
	[GBORER - How to embroider round a hole with an irregular shape](#)
	[CBORER - How to embroider round a circular hole](#)
	[RBORER - How to embroider round a rectangular hole](#)
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Embroidery macros Toolbar	
	[MFLOWER - How to construct a flower](#)
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Modify embroidery objects Toolbar	
	[MODIFY - How to modify embroidery effects](#)
	[UPDATE - How to update stitches of effects](#)
	[DENSITY - How to modify the density of embroidery objects](#)

SEDGE - How to design a single edge

Toolbar: Edge > Single edge

Menu: Enlarge > Edge > Single edge

Keyboard: SEDGE

With the command **SEDGE** single edges can be automatically constructed by picking the guide line. The guide line represents the center of the edge.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: SEDGE
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined....

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only corner stitches will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Width	Edge with **B** in millimeter
Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

To arrange an edge on a guide line you can proceed as follows...
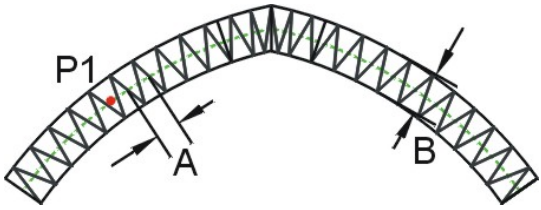
```
Command: SEDGE
Select guide lines/<Parameters>...
Select entities: P1
Select entities: <Ret>
```

Attention: If a corner of the guide line is too accurate, the system might not be capable to construct a parallel line of the edge.
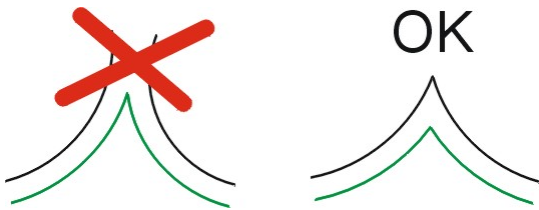
Attention: The guide line must not be closed!

CEDGE - How to design a circular single edge

Toolbar: Edge > Circular edge ▦

Menu: Enlarge > Edge > Circular edge

Keyboard: CEDGE

With the command **CEDGE** circular single edges can be automatically constructed by defining the center and the radius. The given radius represents the radius of the center line of the edge.

The position, where you click the radius, is the position, where the edge starts. The start is marked with a small red cross. If you realize on punching, that the start is at the wrong position, you can use the ⟳ **ROTATE** command to rotate the element.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: CEDGE
Center point/<Parameters>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only corner stitches will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Width	Edge with **B** in millimeter
Done	Exit from the menu and continue with the definition of the center.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To construct a circular edge you can proceed as follows...
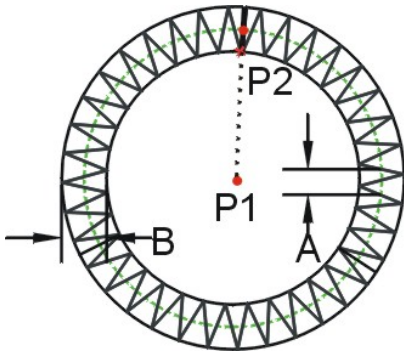

```
Command: CEDGE
Center point/<Parameters>: P1
Radius<actual value>: P2
Center point/<Parameters>: <Ret>
```

DEDGE - How to design a double edge

Toolbar: Edge > Double edge ▦

Menu: Enlarge > Edge > Double edge

Keyboard: DEDGE

With the command **DEDGE** double edges can be automatically constructed by picking the guide line. The guide line represents the center of the edge.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: DEDGE
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only corner stitches will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Width	Edge with **B** in millimeter
Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

To arrange a double edge on a guide line you can proceed as follows...
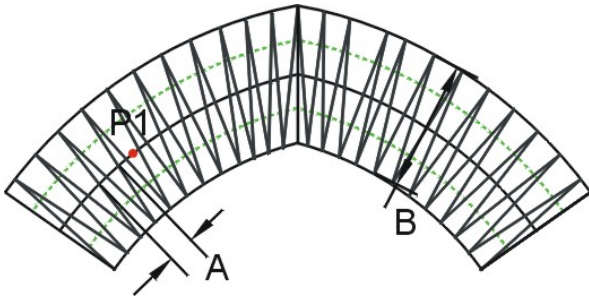
```
Command: DEDGE
Select guide lines/<Parameters>...
Select entities: P1
Select entities: <Ret>
```

Attention: If a corner of the guide line is too accurate, the system might not be capable to construct a parallel line of the edge.
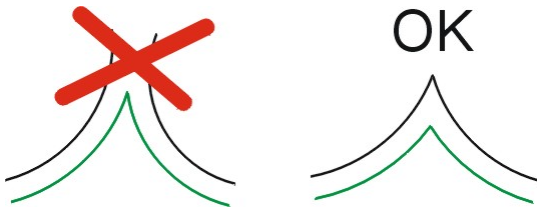
Attention: The guide line must not be closed!

WIGGLE - How to design wiggle stitches

Toolbar: Edge > Wiggle stitch ∿

Menu: Enlarge > Edge > Wiggle stitch

Keyboard: WIGGLE

With the function **WIGGLE** wiggle stitch can be designed along guide lines.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: WIGGLE
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Width	Wiggle width **B** in millimeter
Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, you can continue with the selection of guide lines. If you have drawn the guide lines with a filling function like 🔲 **PATFILL**, you can use the entity selection option **Previous** to select all elements of the effect. Finally the effect is constructed by the system.

To arrange wiggle stitches on guide lines you can proceed as follows...


```
Command: WIGGLE
Select guide lines/<Parameters>...
Select entities: P1
```
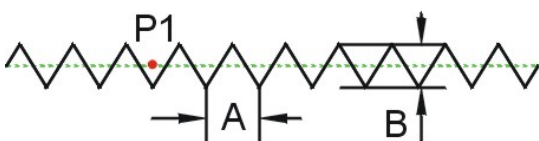
Select entities: **<Ret>**

BLATTSTI - How to design a rotated blattstitch

Toolbar: **Blattstitch > Rotated blattstitch** 🏛️

Menu: **Enlarge > Blattstitch > Rotated blattstitch**

Keyboard: **BLATTSTI**

With this command you can fill any area with a blattstitch. The area can be represented by a single closed polyline or by a circle, or it can be represented by two polylines. In case of two polylines, the area must be between the two polylines (see below).

> **Hint:** If the boundary of a blattstitch is made up of more then two polylines, you can use the command ▢ **BORDER** to generate a single polyline, which is surrounding the desired area.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: BLATTSTI
Select boundary edge 1/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter

Transition	With the transition option you can define variable densities for a single object. Off Random 0.3>0.9>0.3 **See also...** 📄 [How to define the transition parameters](#)
Define pattern...	Define how the system generates the stitch pattern. **See also...** 📄 [How to define the stitch pattern](#)
Random	The rays of the effect have a random length. The random part can be Inside, Outside or on both sides. **See also...** 📄 [How to define a random pattern](#)
Over	Here you can specify how many percent of the total stitch length the system sticks behind the boundary. After clicking this option you can define the overstitch value in %.

Density correction	If the blattstitch is getting too narrow, the stitch distance should be increased up to the double value of the regular blattstitch distance. After clicking this option you can define the low value **L** and the high value **H** to define the section **S**, between which the system corrects from normal stitch distance to double stitch distance.
Shorten stitches	To prevent, that stitches are getting too dense, stitches can be moved to the inner side by a certain value. After clicking this option you can first define the minimum stitch distance **A** and then the distance **B**, by which stitches should be moved. If the density is getting smaller than the defined distance, the system starts to move stitches by the defined value.
Done	Exit from the menu and continue with selecting boundaries.
Cancel	Interrupt the command

If the parameters are defined correctly select first the blattstitch boundaries. Thereafter define the inclination lines, whereby the inclination lines must cross the blattstitch boundary lines.

Important! The starting point of the first inclination line specifies the side, on which the blattstitch starts. The endpoint of the last inclination line defines the side, on which the blattstitch ends.

See also...

📄 How to define the transition parameters

📄 How to define the stitch pattern

- 📄 [How to define a random pattern](#)

- 📄 [How to define inclination lines](#)

Finally, after having constructed all inclination lines and after having left the inclination line menu with the **Build** option, the effect is constructed by the system.

To fill an area with a blattstitch you can proceed as follows...
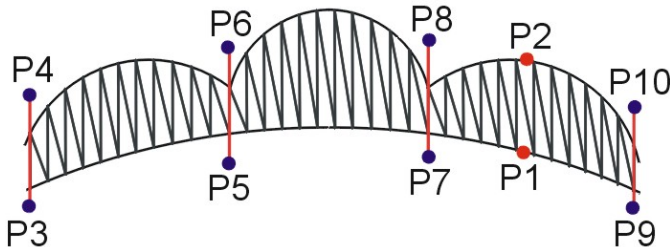

```
Command: BLATTSTI
Select boundary edge 1/<Parameter>: P1
Select boundary edge 2: P2
Direction of stitches...
From point/<Parameter>: P3
To point: P4
Direction of stitches...
....
To point: P10
Direction of stitches...
From point: <Ret>
>> Select Build from the menu <<
```

Note: If the system hangs because of incorrectly defined inclination lines, the calculation can be interrupted by pressing **ESC** one or more times.

DBLATT - How to construct a divided blattstitch

Toolbar: Cgeflect > Divided blattstitch 🔶

Menu: Enlarge > Blattstitches > Divided blattstitch

Keyboard: DBLATT

With the command **DBLATT** you can construct a divided blattstitch.

The area can be represented by a single closed polyline or by a circle, or it can be represented by two polylines. In case of two polylines, the area must be between the two polylines (see below).

> **Hint:** If the boundary of a blattstitch is made up of more then two polylines, you can use the command ⬚ **BORDER** to generate a single polyline, which is surrounding the desired area.

Before you can fill the area you must construct the inclination polylines.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: DBLATT
Select boundary edge 1/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Geflect density **A** in millimeter

Stitch length	If the **Stitch length B** is unequal to 0, the system divides the stitchs in a way that the stitches have approximately the specified length.
	The stitches are distributed unequally. To get this uneven distribution, the first stitch has a random length. The length is between the **Minimum stitch length** and the **Stitch length**. The bigger the difference between this two values is, the more uneven the pattern will be.
Minimum stitch length	If the stitch is being divided by one of the following methods, this value defines the minimum stitch length the system allows.
Guide line(s)	With additional guide lines you can get a smoother distribution of the arcs. The guide lines must also be constructed before you can call up the command.
Done	Exit from the menu and continue with selecting boundaries.
Cancel	Interrupt the command

If the parameters are defined correctly, select first the blattstitch boundaries. Thereafter select the previously designed inclination polylines.

Important! The starting point of the first inclination line specifies the side, on which the blattstitch starts.

To construct a divided blattstitch, the following rules must be respected:

- All inclination polylines **B** must be designed in the same direction.
- All inclination polylines **B** must be designed with the same number or line respectively arc segments.
- The inclination polylines must not have curve segments.
- The inclination polylines **B** must be selected in the proper order.

If you use guide lines, the following rules must be respected:

- The guide lines must be designed in the same direction as you select the inclination polylines **B**.
- The vertices of the inclination polylines must be exactly on the guide line **C**.
- If you select the guide lines **C**, you must select them in the same direction as the inclination polylines **B** were designed.
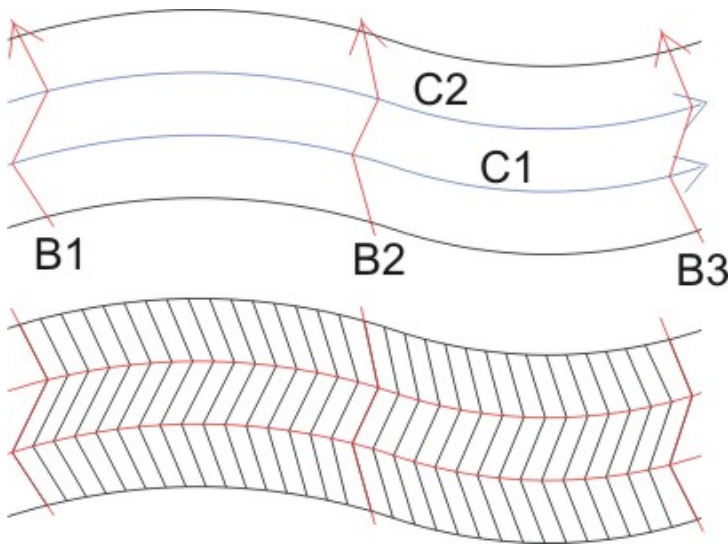
To fill an area with a divided blattstitch you can proceed as follows...
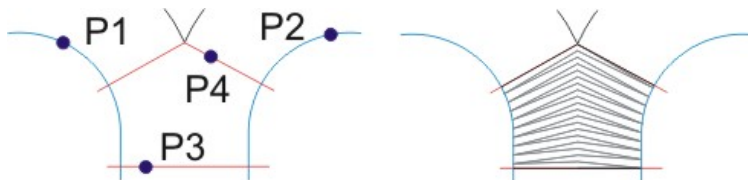
Attention! The lower inclination polyline must also be designed with 2 line segments!

```
Command: DBLATT
Select boundary edge 1/<Parameter>: <Ret>
Select boundary edge 1/<Parameter>: P1
Select boundary edge 2: P2
Select inclination lines...
Select entities: P3
Select entities: P4
Select entities: <Ret>
```

CIRBLATT - How to design a circular blattstitch with hole

Toolbar: Blattstitch > Circular blattstitch ⊕

Menu: Enlarge > Blattstitch > Circular blattstitch

Keyboard: CIRBLATT

With the command you construct a round hole. For construction you must simply define the inside and outside radius of the blattstitch.

The position, where you click the inside radius, is the position, where the blattstitch starts. The start is marked with a small red cross. If you realize on punching, that the start is at the wrong position, you can use the 🔲 **MODIFY** command to rotate the element.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: CIRBLATT
Center point/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter

Over	Here you can specify how many percent of the total stitch length the system sticks behind the boundary. After clicking this option you can define different overstitch values for both sides. ![Over illustration]
Density correction	If the blattstitch is getting too narrow, the stitch distance should be increased up to the double value of the regular blattstitch distance.
Shorten stitches	To prevent, that stitches are getting too dense, stitches can be moved to the inner side by a certain value. ![Shorten stitches illustration with A and B] After clicking this option you can first define the minimum stitch distance **A** and then the distance **B**, by which stitches should be moved. If the density is getting smaller than the defined distance, the system starts to move stitches by the defined value.
Select boundary	This option you can switch **On/Off**. If **Select boundary** is **On**, you can choose an existing circle instead of constructing a new one.
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly and **Select boundary** is not activated, define first the center point and thereafter the inside radius and the outside radius. Finally the effect will be constructed and you can continue with the next round

blattstitch.
If you want to terminate the command, press **Return** or the right mouse button and activate the **Cancel** option of the popup menu.

To construct a bored blattstitch proceed as follows...
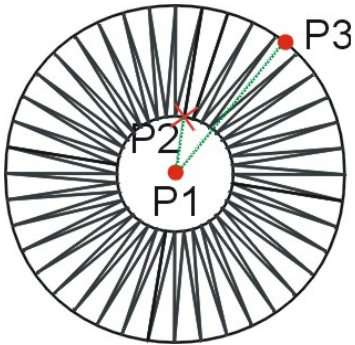

```
Command: CIRBLATT
Center point/<Parameter>: P1
Inside radius: P2
Outside radius: P3
Center point/<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

DOT - How to design blattstitch dots

Toolbar: **Blattstitch > Dot** 🔵

Menu: **Enlarge > Blattstitch > Dot**

Keyboard: **DOT**

With this command you can construct dots, filled with a blattstitch. For construction you simply define the center and the radius of the dot.

The position, where you click the radius, shows into the direction of the stitches. If you realize on punching, that the stitch direction is wrong, you can use the 🏛 **MODIFY** command to rotate the element.
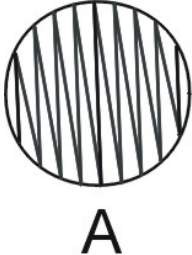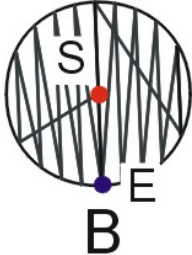
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: DOT
Center point/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Over	Here you can specify how many percent of the total stitch length the system sticks behind the boundary. After clicking this option you can define different overstitch values for both sides.

Start in middle	You can switch On/Off this option. If **Start in middle** is off **A**, the dot starts on one side and ends on the other side. The dot is made up of a single part. If **Start in middle** is on **B**, the dot is divided in two sections, the starting point **S** and the endpoint **E** of the dot are in the middle of the dot.
Select boundary	This option you can switch **On/Off**. If **Select boundary** is **On**, you can choose an existing circle instead of constructing a new one.
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly and **Select boundary** is not activated, define first the center point and then the radius of the dot. Finally the effect will be constructed and you can continue with the next dot.
If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To construct a dot with 10% over stitch proceed as follows...
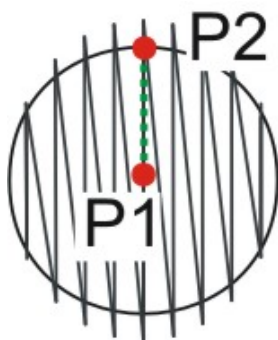

```
Command: DOT
Center point/<Parameter>: <Ret>
>> Specify Over=10 in the menu <<
Center point/<Parameter>: P1
To point: P2
```

```
Center point/<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

```
Center point/<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

BORPAD - How to design a bored paddle

Toolbar: Blattstitch > Bored paddle 🔲

Menu: Enlarge > Blattstitch > Bored paddle

Keyboard: BORPAD

With this command you can construct a bored paddle. The shape of the paddle must be designed as a single polyline.
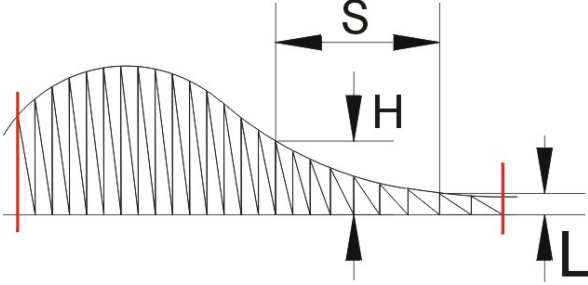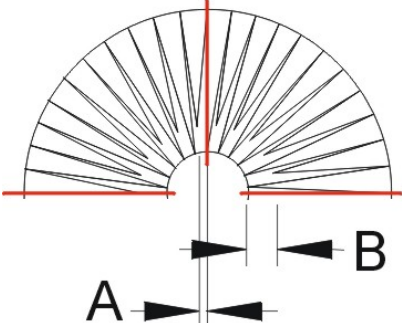
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: BORPAD
Center point/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Inside radius	Radius **R** of the hole
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

For the other parameters see...

📄 BLATTSTI - How to design a rotated blattstitch

If the parameters are defined correctly, you must select the polyline. Next you specify the uppermost point **P2** of the paddle. This point **must be outside of the element** and it defines also the endpoint of inclination line. Finally you must define the base line for the upper part of the bored paddle. If you start on the right side (see below), the upper part rotates counter clockwise, if you start on the left side, the upper part rotates clockwise.
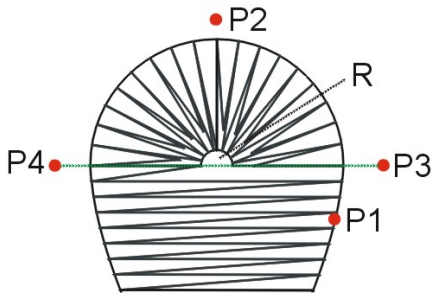
```
Command: BORPAD
Select boundary edge/<Parameter>: P1
Select paddle top: P2
Define base line...
From point: P3
To point: P4
```

If points 3 and 4 of the base line are laid on the start and the end of the polyline, only the rotated part of the effect is constructed. To find the endpoints of the polyline you should use object snap **END**.
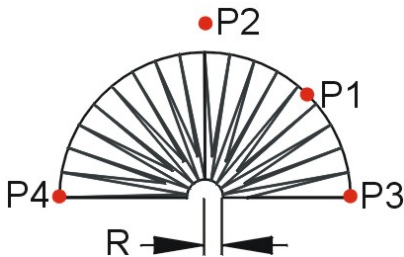

```
Command: BORPAD
Select boundary edge/<Parameter>: P1
Select paddle top: P2
Define base line...
From point: END Snap to endpoint of P3
To point: END Snap to endpoint of P4
```

GEFLECT - How to construct a rotated geflect

Toolbar: **Geflect > Rotated geflect**

Menu: **Enlarge > Geflect > Rotated geflect**

Keyboard: **GEFLECT**

With the command **GEFLECT** a rotated geflect can be constructed.

The area can be represented by a single closed polyline or by a circle, or it can be represented by two polylines. In case of two polylines, the area must be between the two polylines (see below).

> **Hint:** If the boundary of a blattstitch is made up of more then two polylines, you can use the command **BORDER** to generate a single polyline, which is surrounding the desired area.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: GEFLECT
Select boundary edge 1/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Geflect density **A** in millimeter

Transition	With the transition option you can define variable densities for a single object. Off Random 0.3>0.9>0.3 **See also...** ▤ How to define the transition parameters
Define pattern...	Define how the system generates the stitch pattern. **See also...** ▤ How to define the stitch pattern
Random	The rays of the effect have a random length. The random part can be Inside, Outside or on both sides. **See also...** ▤ How to define a random pattern

Over	Here you can specify how many percent of the total stitch length the system sticks behind the boundary. After clicking this option you can define the overstitch value in %.
Turnstitch	If **Yes**, the system creates a short stitch before the border and a long stitch after the border. The short stitch has the minimum stitch length. If **No**, the Geflect turns like a Blattstitch. Turnstitch = Yes Turnstitch = No **Note:** with the system variable amd_geflongfac you can specify the factor between the short stitch and the long stitch of turnstitches. By default

	this factor is 1.3, so the long stitch is 1.3 times longer than the short stitch. You can define this variable in the file **DEFAULT.INI**.
Done	Exit from the menu and continue with selecting boundaries.
Cancel	Interrupt the command

If the parameters are defined correctly, select first the geflect boundaries. Thereafter define the inclination lines, whereby the inclination lines must cross the geflect boundaries.

Important! The starting point of the first inclination line specifies the side, on which the geflect starts. The endpoint of the last inclination line defines the side, on which the geflect ends.

See also...

📄 [How to define the transition parameters](#)

📄 [How to define the stitch pattern](#)

📄 [How to define a random pattern](#)

📄 [How to define inclination lines](#)

Finally, after having constructed all inclination lines and after having left the inclination line menu with the **Build** option, the effect is constructed by the system.

To fill an area with a geflect you can proceed as follows...
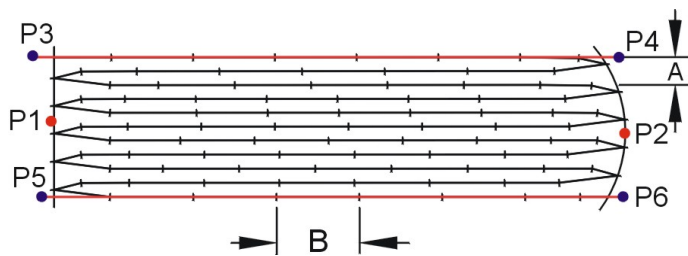

```
Command: GEFLECT
Select boundary edge 1/<Parameter>: P1
Select boundary edge 2: P2
Direction of stitches...
From point/<Parameter>: P3
To point: P4
Direction of stitches...
```

```
From point/<Parameter>: P5
To point: P6
Direction of stitches...
From point: <Ret>
>> Select Build from the menu <<
```

Note: If the system hangs because of incorrectly defined
inclination lines, the calculation can be interrupted by pressing
ESC one or more times.

CGEFLECT - How to fill an area with curves

Toolbar: **Cgeflect > Curvy geflect** ⬚

Menu: **Enlarge > Geflect > Curved geflect**

Keyboard: **CGEFLECT**

With the command **CGEFLECT** you can fill an area with curves.

The area can be represented by a single closed polyline or by a circle, or it can be represented by two polylines. In case of two polylines, the area must be between the two polylines (see below).

> **Hint:** If the boundary of a blattstitch is made up of more then two polylines, you can use the command ⬚ **BORDER** to generate a single polyline, which is surrounding the desired area.

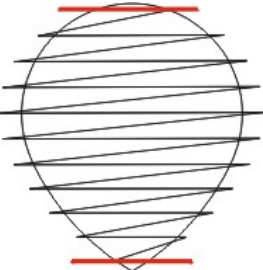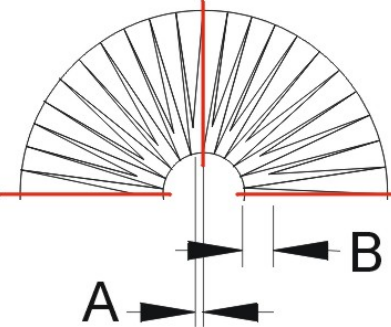Before you can fill the area you must construct the inclination polylines.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: CGEFLECT
Select boundary edge 1/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Geflect density **A** in millimeter
Transition	Not yet supported

Define pattern...	Define how the system generates the stitch pattern. **See also...** 🗏 [How to define the stitch pattern](#)
Random	The rays can have a random length. The random part is always outside (at the end of the inclination polylines). The parameter defines the relation between full ray length and random ray length. This value must be specified in percent (See upper illustration). **Example:** If the ray length is 100 mm and the value is 80%, the random ray length can vary from 20 to 100 mm. Random=30% Random=70%
Guide line(s)	With additional guide lines you can get a smoother distribution of the arcs. The guide lines must also be constructed before you can call up the command.
Turnstitch	If **Yes**, the system creates a short stitch before the border and a long stitch after the border. The short stitch has the minimum stitch length. If **No**, the Geflect turns like a Blattstitch.

	Turnstitch = Yes	Turnstitch = No

Note: with the system variable amd_geflongfac you can specify the factor between the short stitch and the long stitch of turnstitches. By default this factor is 1.3, so the long stitch is 1.3 times longer than the short stitch.

You can define this variable in the file **DEFAULT.INI**.

Done	Exit from the menu and continue with selecting boundaries.
Cancel	Interrupt the command

If the parameters are defined correctly, select first the geflect boundaries. Thereafter select the previously designed inclination polylines.

Important! The starting point of the first inclination line specifies the side, on which the geflect starts.

See also...

📄 [How to define the stitch pattern](#)

To fill an area with polylines, the following rules must be respected:

- All inclination polylines **B** must be designed in the same direction.
- All inclination polylines **B** must be designed with the same number or line respectively arc segments.
- You cannot mix arc and curve segments in an inclination polyline.
- An inclination polyline made of arcs must not have corners.
- The inclination polylines **B** must be selected in the proper order.
- If you want to fill an area with rays of random length, the random part is always on the side of the end point of the inclination polylines **B**.
- if you fill a closed area **A,** the imaginary line between the intersection points of the inclination lines **B** with the area **must not intersect** the area **A**.

dashed line intersects
not OK!

dashed line does not intersect
OK!

If you use guide lines, the following rules must be respected:

- The guide lines must be designed in the same direction as you select the inclination polylines **B**.
- The vertices of the inclination polylines must be exactly on the guide line **C**.
- If you select the guide lines **C**, you must select them in the same direction as the inclination polylines **B** were designed.

To fill an area with curves with random ray length you can proceed as follows...
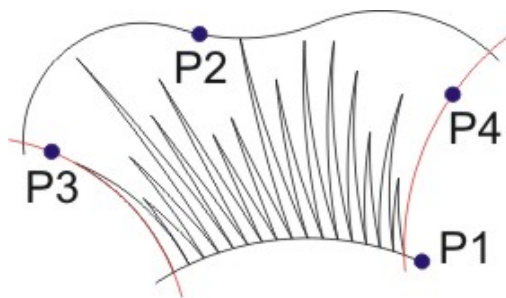

```
Command: CGEFLECT
Select boundary edge 1/<Parameter>: <Ret>

>> Define Random=70 in the menu <<

Select boundary edge 1/<Parameter>: P1
Select boundary edge 2: P2
Select inclination lines...
Select entities: P3
Select entities: P4
Select entities: <Ret>
```

PGEFLECT - How to construct a parallel geflect

Toolbar: Geflect > Parallel geflect

Menu: Enlarge > Geflect > Rotated geflect

Keyboard: PGEFLECT

With the command **PGEFLECT** a parallel geflect can be automatically constructed on an existing guide line. You can specify the stitch distance, or the number of stitches, which you want to have inserted on one segment. The guide line represents the center of the geflect.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: PGEFLECT
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Geflect density **A** in millimeter
Minimum stitch length	If the stitch is being divided by one of the following methods, this value defines the minimum stitch length the system allows.

Turnstitch	If **Yes**, the system creates a short stitch before the border and a long stitch after the border. The short stitch has the minimum stitch length. If **No**, the Geflect turns like a Blattstitch. Turnstitch = Yes Turnstitch = No **Note:** with the system variable **amd_geflongfac** you can specify the factor between the short stitch and the long stitch of turnstitches. By default this factor is 1.3, so the long stitch is 1.3 times longer than the short stitch. You can define this variable in the file **DEFAULT.INI**.
Stitch length	If the **Stitch length L** is unequal to 0, the system divides the stitch in a way that the stitches have approximately the specified length (See illustration of the example below). The stitches are distributed unequally. To get this uneven distribution, the first stitch has a random length. The length is between the **Minimum stitch length** and the **Stitch length**. The bigger the difference between this two values is, the more uneven the pattern will be.
No. of stitches	The value **N** specifies, how many times a stitch must be divided. Each segment has the same length. This method results in an even pattern. A N=2

Width	Specify the width **B** of the geflect
Done	Exit from the menu and continue with selecting boundaries.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

To construct a parallel geflect on guide lines you can proceed as follows...
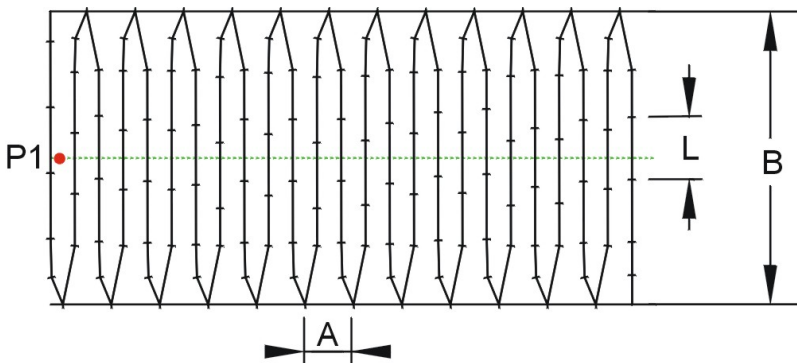

```
Command: PGEFLECT
Select guide lines/<Parameters>...
Select entities: P1
Select entities: <Ret>
```

Attention: If a corner of the guide line is too accurate, the system might not be capable to construct a parallel line of the edge.
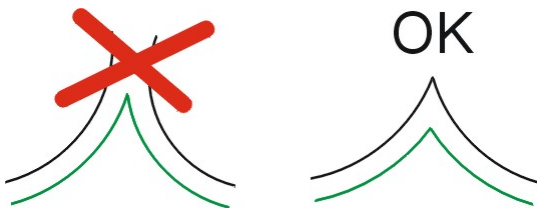
Attention: The guide line must not be closed!

STEP - How to design step stitches

Toolbar: Geflect > Step stitches ![icon]

Menu: Enlarge > Geflect > Step stitches

Keyboard: STEP
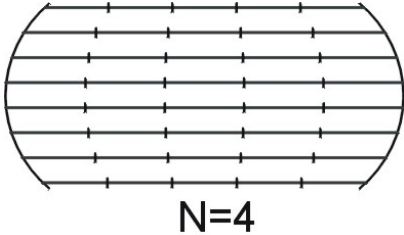
With the command **STEP**, step stitches can be drawn along guide lines. The stitch distance respectively the number of stitches per guide line can be defined.
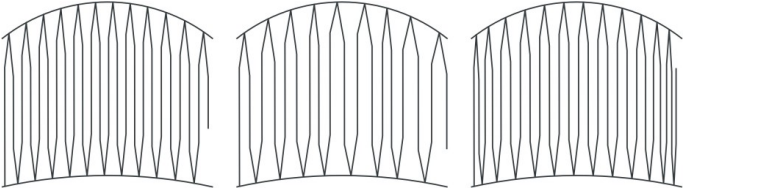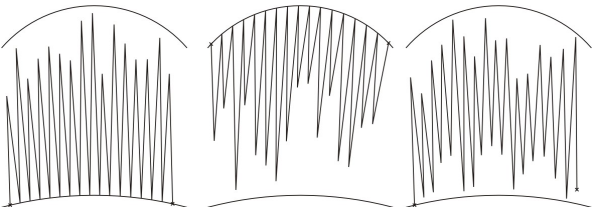
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.
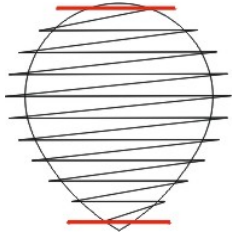
```
Command: STEP
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Corner	You can define how the system treats corners. If **Corner=Yes**, the system divides from corner to corner to make sure that every corner has a segmentation point. In curves with a small radius the distance between the segmentation point will automatically be reduced. If **Corner=No** the system ignores corners and the distance between the segmentation points is always same. Corner = Yes Corner = No
Stitch length	Specify stitch distance **B**. If you specify the stitch distance (See illustration of the example below)

No. of stitches	The value **N** specifies, how many times a guide line must be divided.
	This effect results in an even distribution of stitches.
	 N=4
Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

If you have drawn the guide lines with a filling function like **PATFILL**, you can use the entity selection option **Previous**. Finally the effect is constructed by the system.

To arrange step stitches on guide lines you can proceed as follows...

In the example below the selection of the guide lines is performed with the **Fence** method.
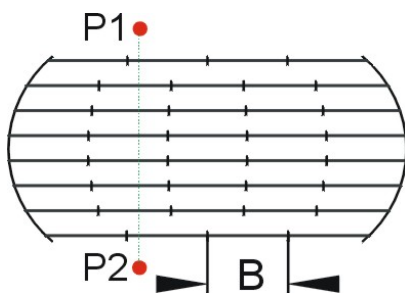

```
Command: STEP
Select guide lines/<Parameters>...
Select entities: Select Fence from menu
First point of fence: P1
Next point: P2
Next point: <Ret>
Select entities: <Ret>
```

HOEHL - How to construct holes

Toolbar: **Boring > Hoehl**

Menu: **Enlarge > Boring > Hoehl**

Keyboard: **HOEHL**

With this command stitches are designed for Hole elements. The Hole guide lines itself can be designed with the function **PATFILL**.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: HOEHL
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

If you have drawn the guide lines with a filling function like **PATFILL**, you can use the entity selection option **Previous**. Finally the effect is constructed by the system.

To arrange hoehl stitches on guide lines you can proceed as follows...

In the example below the guide lines were constructed with the **PATFILL** function and selection of the guide lines is performed with the **Previous** method.

```
Command: HOEHL
Select guide lines/<Parameters>...
Select entities: P
Select entities: <Ret>
```

After all stitches have been designed boring points can be added.

POINT - How to add boring points

Toolbar: **Boring** > **Point**

Menu: **Enlarge** > **Boring** > **Point**

Keyboard: **POINT**

After picking this function you can indicate with the cross hair cursor the position of the Boring points. After insertion of all boring points exit the function by pressing **<Ret>**.

STEFFEL - How to design a steffel

Toolbar: Boring > Steffel 🎬

Menu: Enlarge > Boring > Steffel

Keyboard: STEFFEL

With the command **STEFFEL** a Steffel can be automatically constructed by picking a guide line. The guide line represents the center of the steffel.
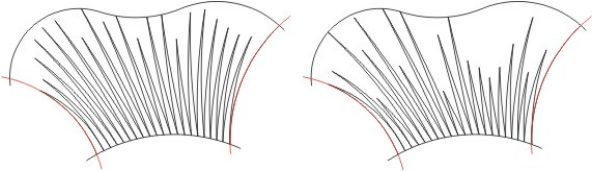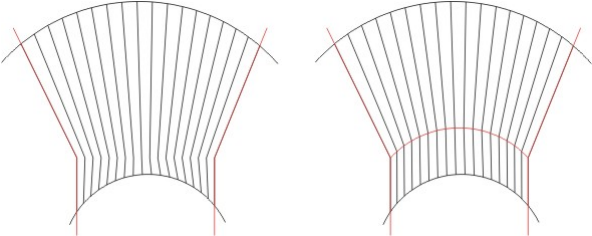
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.
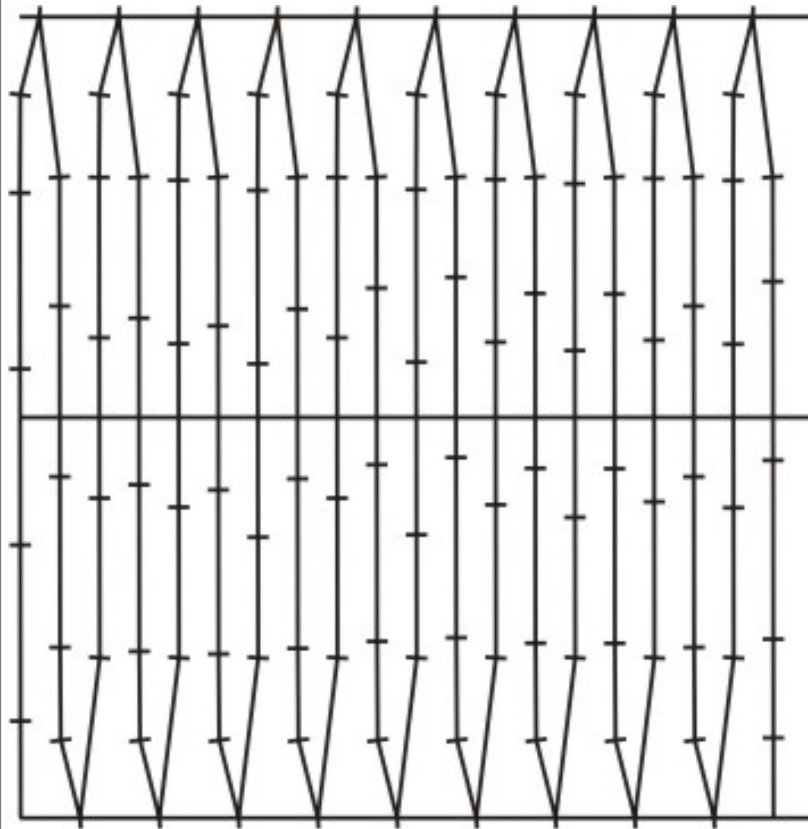
```
Command: STEFFEL
Select guide lines/<Parameters>...
Select entities: <Ret>
```
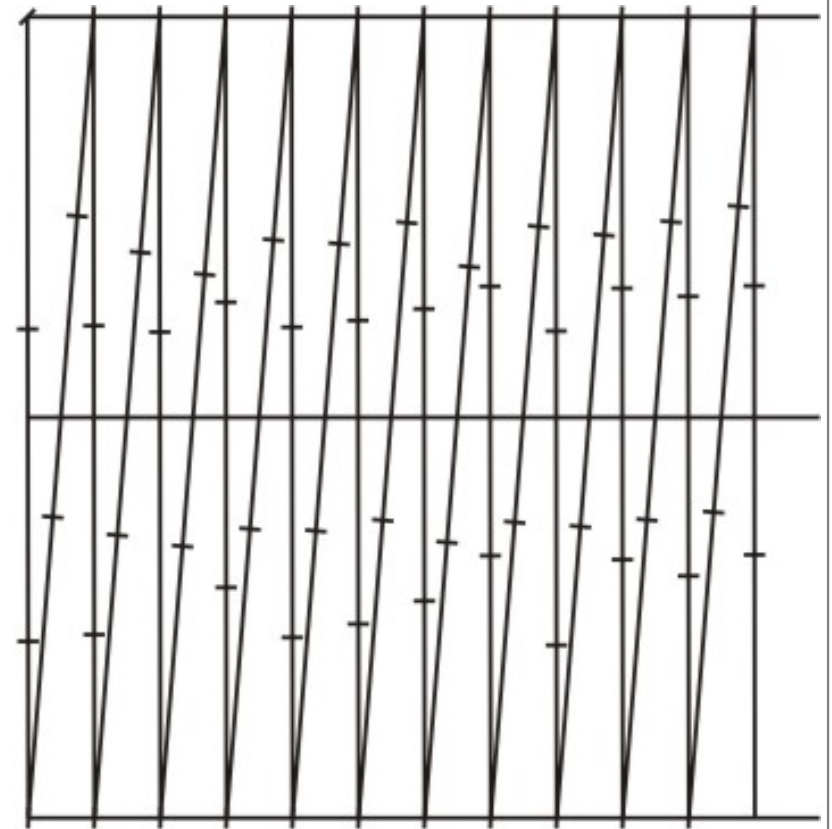
The following parameters can be defined...

Menu entry	Description
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Steffel distance	Distance **A** between Zuchole legs (see illustration below).
Steffel width	Width **B** of a the Zuchole (see illustration below).
Density	Edge density **C** (see illustration below)
Edge width	Edge width **D** (see illustration below)
Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

To arrange a steffel on guide lines you can proceed as follows...
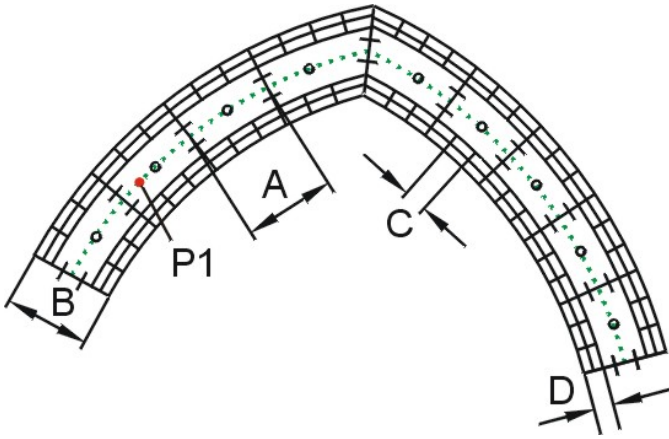
```
Command: STEFFEL
Select guide lines/<Parameters>...
Select entities: P1
Select entities: <Ret>
```

Attention: If a corner of the guide line is too accurate, the system might not be capable to construct a parallel line of the edge.

OK
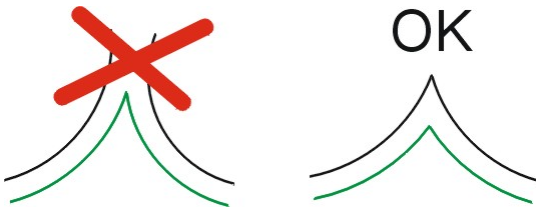
Attention: The guide line must not be closed!

How to embroider round holes

After opening a hole with the borer you must embroider round this hole to remove the fabric. Various commands are available to embroider round holes with different shapes.

GBORER - How to embroider round a hole with an irregular shape

Keyboard: GBORER

With this command you can embroider round an irregular hole. The area of the hole must be represented by two closed polylines.

> **Hint:** If the outside boundary of the hole is made up of more then one polyline, you can use the command ⊡ **BORDER** to generate a single polyline, which is surrounding the desired area.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: GBORER
Select boundary edge 1/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly select first the hole boundaries. Thereafter define the inclination lines, whereby the inclination lines must cross the hole boundary lines.
Important! The starting point of the first inclination line specifies the side, on which the effect starts. The endpoint of the last inclination line defines the side, on which the effect ends.

See also...

Finally, after having constructed all inclination lines and after having left the inclination line menu with the **Build** option, the effect is constructed by the system.

To embroider round an irrregular area proceed as follows...
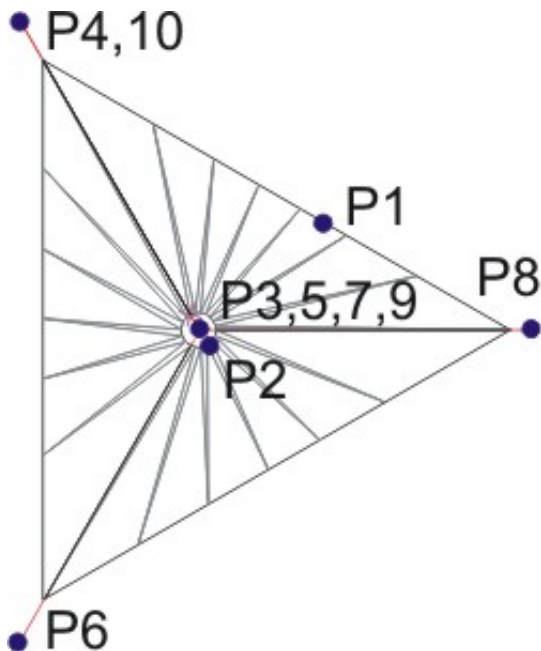

```
Command: GBORER
Select boundary edge 1/<Parameter>: P1
Select boundary edge 2: P2
Direction of stitches...
From point/<Parameter>: P3
To point: P4
Direction of stitches...
.....
To point: Px
From point: <Ret>
>> Select Build from the menu <<
```

Note: If the system hangs because of incorrectly defined inclination lines, the calculation can be interrupted by pressing **ESC** one or more times.

CBORER - How to embroider round a circular hole

Toolbar: Borer holes > Circular hole

Menu: Enlarge > Borer holes > Circular hole

Keyboard: CBORER

With this command you can embroider round a circular hole. For construction you must simply define the center and the outside radius of the hole.

The position, where you click the radius, is the position, where the effect starts. The start is marked with a small red cross. If you realize on punching, that the start is at the wrong position, you can use the **MODIFY** command to rotate the element.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.
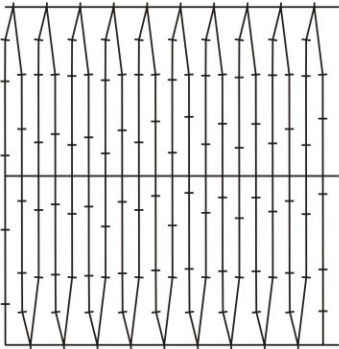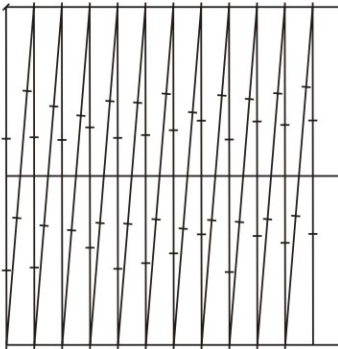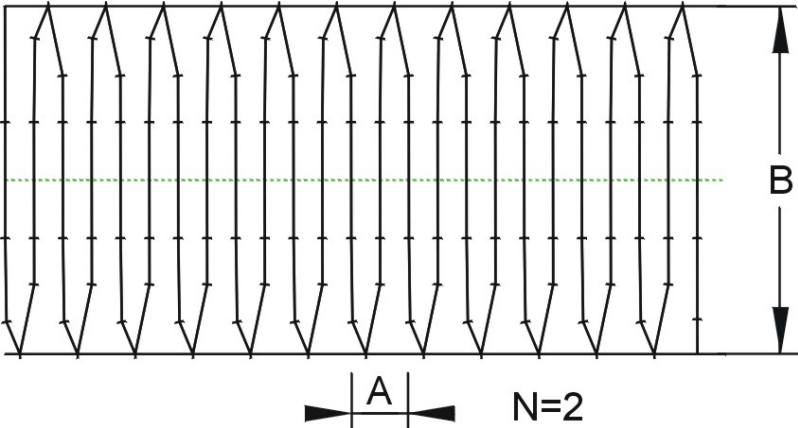
```
Command: CBORER
Center point/<Parameter>: <Ret>
```

## The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Stitch length	You can define the stitch length **A**, with which you want to embroider round the hole. This stitch length defines also the inside boundary of the hole.
Select boundary	This option you can switch **On/Off**. If **Select boundary** is **On**, you can choose an existing circle instead of constructing a new one.
Density	Stitch distance **A** in millimeter
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly and **Select boundary** is not activated, define first the center point and then the radius of the hole. Finally the effect will be constructed and you can continue with the next hole.
If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To embroider round an rectangular hole proceed as follows...
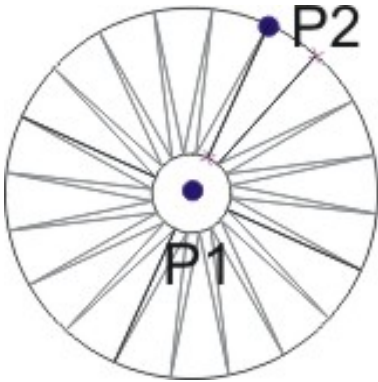

```
Command: CBORER
Center point/<Parameter>: P1
To point: P2
Center point/<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

CBORER - How to embroider round a circular hole

Toolbar: **Borer holes > Rectangular hole** ▪

Menu: **Enlarge > Borer holes > Rectangular hole**

Keyboard: **RBORER**

With this command you can embroider round a rectangular hole.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: RBORER
First corner/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Stitch length	You can define the stitch length **A**, with which you want to embroider round the hole. This stitch length defines also the inside boundary of the hole.
Select boundary	This option you can switch **On/Off**. If **Select boundary** is **On**, you can choose an existing rectangle instead of constructing a new one.
Density	Stitch distance **A** in millimeter
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly and **Select boundary** is not activated, two diagonal points of the rectangle must be entered. Thereafter the orientation of the

rectangle can be determined and the starting point (entry point for the puncher) of the effect can be defined. Finally the effect will be constructed and you can continue with the next hole.

If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To embroider round a rectangular hole proceed as follows...
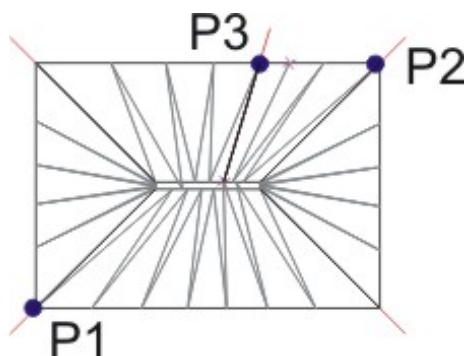

```
Command: RBORER
First corner/<Parameter>: P1
Other corner: P2
Angle<0>: <Ret>
Starting point: P3
First corner/<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

EBORER - How to embroider round an elliptical hole

Toolbar: Borer holes > Elliptical hole ⊕

Menu: Enlarge > Borer holes > Elliptical hole

Keyboard: EBORER

With this command you can embroider round an elliptical hole. For construction you must simply define the two axis of the ellipse and the starting point of the effect.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: EBORER
Center/First endpoint of axis<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Stitch length	You can define the stitch length **A**, with which you want to embroider round the hole. This stitch length defines also the inside boundary of the hole.
Select boundary	This option you can switch **On/Off**. If **Select boundary** is **On**, you can choose an existing ellipse instead of constructing a new one.
Density	Stitch distance **A** in millimeter
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly and **Select boundary** is not activated,

define first the two axis of the ellipse, then the starting point of the effect and finally the effect will be constructed and you can continue with the next hole.
If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To embroider round an elliptical hole proceed as follows...
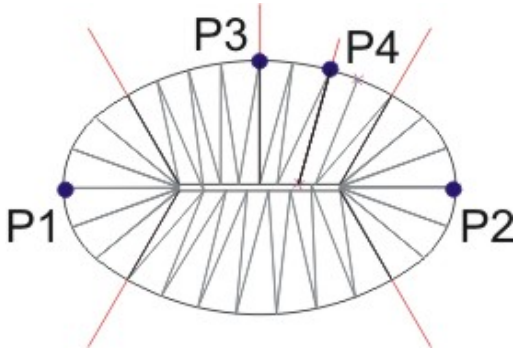

```
Command: EBORER
Center/First endpoint of axis<Parameter>: P1
Second endpoint of axis: P2
Other side: P3
Starting point: P4
Center/First endpoint of axis<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

DBORER - How to embroider round hole like a drop

Toolbar: Borer holes > Hole like a drop

Menu: Enlarge > Borer holes > Hole like a drop

Keyboard: DBORER

With this command you can embroider round an hole with a shape like a drop. For construction you must simply define two points of the circle, the point of the paddle and the starting point of the effect.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: DBORER
Point 1/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	If **Yes**, all stitches will be designed for the technical enlargement. If **No**, only stitches at the inclination lines will be designed
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Stitch length	You can define the stitch length **A**, with which you want to embroider round the hole. This stitch length defines also the inside boundary of the hole.
Select boundary	This option you can switch **On/Off**. If **Select boundary** is **On**, you can choose an existing drop instead of constructing a new one.
Density	Stitch distance **A** in millimeter
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly and **Select boundary** is not activated, define first two points on the circle, the point of the paddle and the starting point of the effect. Finally the effect will be constructed and you can continue with the next hole.

If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To embroider round a hole like a drop proceed as follows...
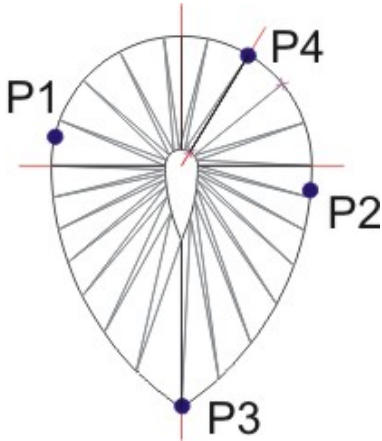

```
Command: DBORER
Point 1/<Parameter>: P1
Diameter: P2
Select paddle top: P3
Starting point: P4
Point 1/<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

Hint: The stitch density on the two sides of the drop is not equal. To correct the density on one side just use the **MODIFY** command to change the density of the corresponding inclination line.

STAR - How to design star dots

Toolbar: Embroidery > Star dot ✳

Menu: Enlarge > Star dot

Keyboard: STAR

This function generates star dots. For construction you must simply define the center and the outside radius of the star dot.

The position, where you click the radius, is the position, where the star dot starts. The start is marked with a small red cross. If you realize on punching, that the start is at the wrong position, you can use the 🏛 **MODIFY** command to rotate the element.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: STAR
Center point/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Stitch distance **A** in millimeter
Select boundary	This option you can switch **On/Off**. If **Select boundary** is **On**, you can choose an existing circle instead of constructing a new one.
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly and **Select boundary** is not activated, define first the center point and then a point on the boundary. Finally the effect is constructed by the system and you can continue with the next dot.

If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To construct a star you can proceed as follows...


```
Command: STAR
Center point/<Parameter>: P1
To point: P2
Center point/<Parameter>: <Ret>
```

ZUCHOLE - How to construct a Zuchole

Toolbar: Embroidery > Zuchole 🔳

Menu: Enlarge > Zuchole
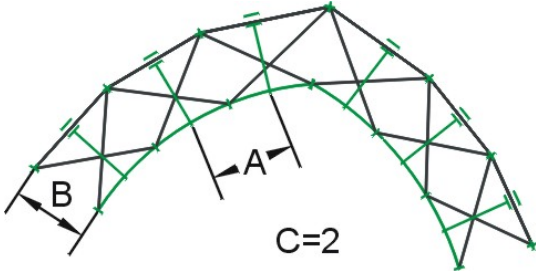
Keyboard: ZUCHOLE
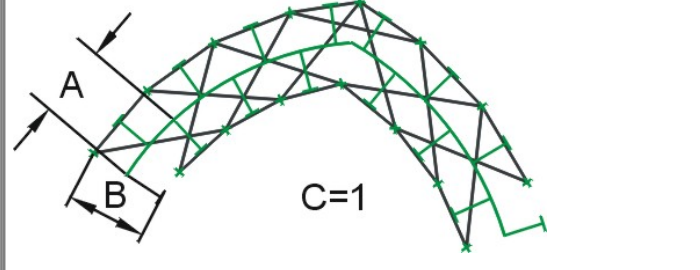
With the command **ZUCHOLE** you can construct a single sided or double sided zuchole by picking the guide line. In case of a single sided zuchole, the guide line represents the base line, in case of a double sided zuchole, the guide line represents the center of the zuchole.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: ZUCHOLE
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Steffel distance	Distance **A** between Zuchole legs (see illustration below).
Steffel width	Width **B** of a the Zuchole (see illustration below).
No. of stitches	Select number of stitches **C** per zuchole leg.
Zuchole type	Choose between single sided and double sided zuchole. **Single sided zuchole** **Double sided zuchole**

Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

To arrange step stitches on guide lines you can proceed as follows...

```
Command: ZUCHOLE
Select guide lines/<Parameters>...
Select entities: select guide line
Select entities: <Ret>
```

Attention: If a corner of the guide line is too accurate, the system might not be capable to construct a parallel line of the edge.
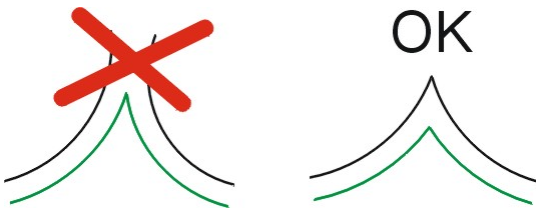
Attention: The guide line must not be closed!

Notice: If the single sided zuchole is drawn on the wrong side of the guide line, you must change the direction of the polyline.
You can do this with the command 📄 **PLCONV**. Then you can use the **UPDATE** or **MODIFY** command to rebuild the zuchole.

PICO - How to design lock stitches

With this command lock or join stitches can automatically be constructed by picking guide lines. A guide line represents the center of the join stitches.

Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: PICO
Select guide lines/<Parameters>...
Select entities: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Color	Display color for the enlargement. **Attention:** This color setting has no influence on punching!
Raedle	You can select the desired **Raedle** from a menu. If you change the **Raedle**, also the **Density** will be adjusted according this value.
Density	Pico stitch distance **A** in millimeter
Width	Width **B** of the pico
Done	Exit from the menu and continue with selecting guide lines.
Cancel	Interrupt the command

If the parameters are defined correctly, select all guide lines and finally the effect is constructed by the system.

To arrange step stitches on guide lines you can proceed as follows...
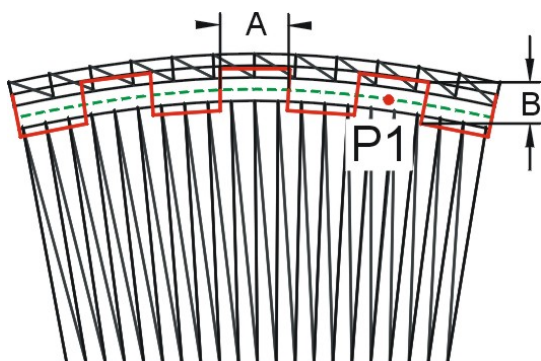
```
Command: PICO
Select guide lines/<Parameters>...
Select entities: P1
Select entities: <Ret>
```

UNDER - How to add underlayers

Toolbar: Embroidery > Underlayer 🔺

Menu: Enlarge > Underlayer

Keyboard: UNDER
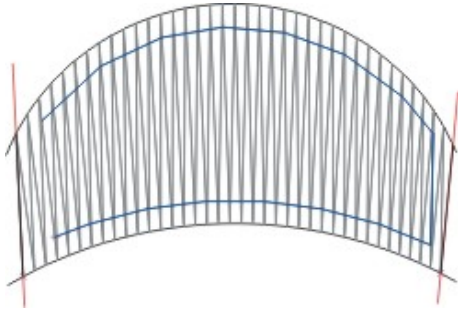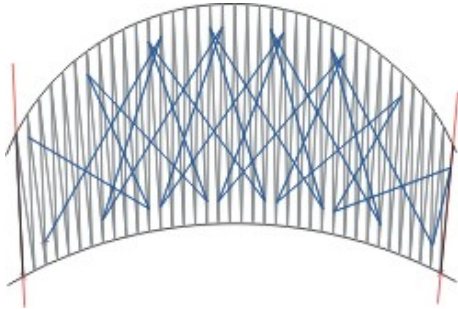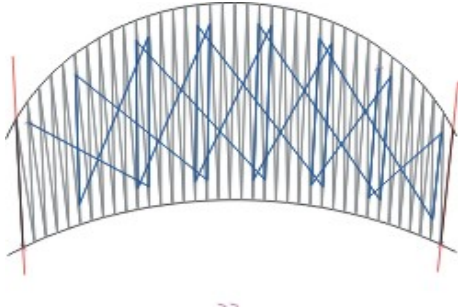
With this command you can underlayers to a Blattstitch. Before you can design underlayers you must have a Blattstitch. To add underlayers just choose the type of underlayer and select the corresponding Blattstitch.
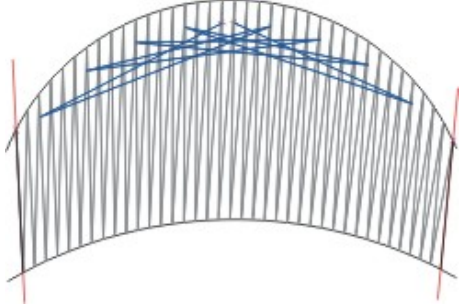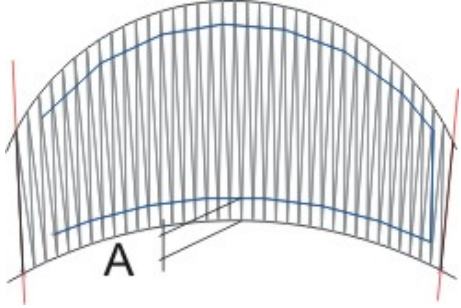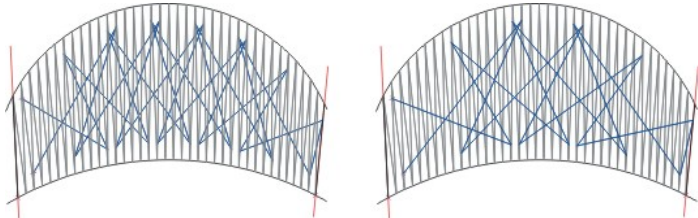
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: UNDER
Select Blattstitch<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Standard	Standard underlayer along the boundaries of the blattstitch.
Chemical 1	
Chemical 2	

Chemical 3	
Offset	Distance **A** of the underlayer from the boundary.
Shift	Shift defines the density of the underlayer. It is the relation between blattstitch density and underlayer density. Values between 1 and 29 are possible.
Done	Exit from the menu and continue with defining the center point.
Cancel	Interrupt the command

If the parameters are defined correctly just select the blattstitches and the underlayers will be constructed.

If you want to terminate the command press the right mouse button and use the **Cancel** option of the upcoming menu.

To add an underlayer Chemical 1 to a blattstitch proceed as follows...
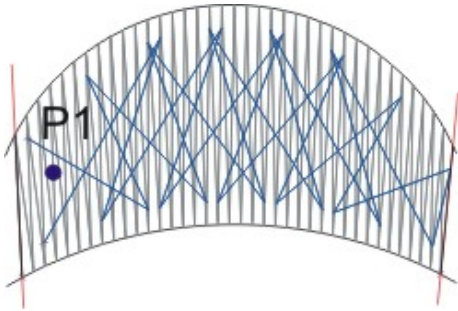
```
Command: UNDER
Select Blattstitch<Parameter>: <Ret>
>> Select Chemical 1 from the menu <<
Select Blattstitch<Parameter>: P1
Select Blattstitch<Parameter>: <Ret>
>> Select Cancel from the menu <<
```

Embroidery macros

A macro is a complex embroidery element like a flower or a border. The whole geometry with all stitches can be constructed by defining a few parameters.

MFLOWER - How to construct a flower

Toolbar: Embroidery macros > Flower macro

Menu: Enlarge > Flower macro

Keyboard: MFLOWER

With the flower macro you can construct almost all kind of flowers. To construct a flower, first setup the parameters and then define the geometry.
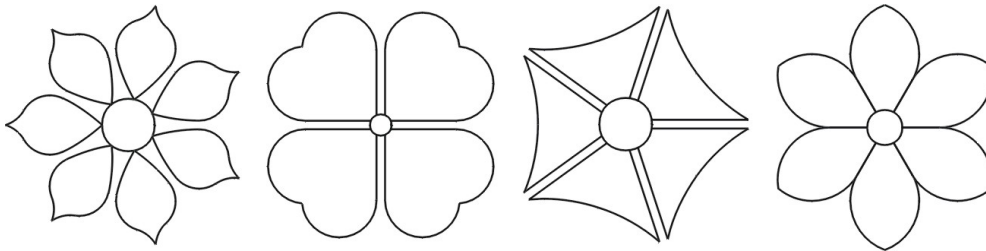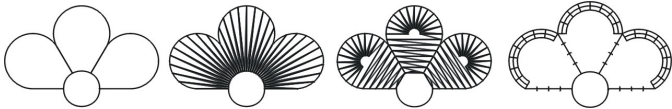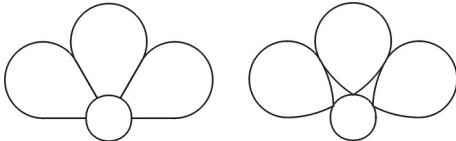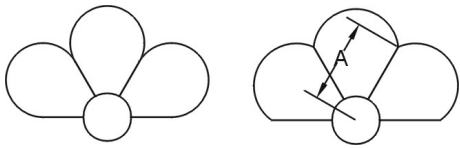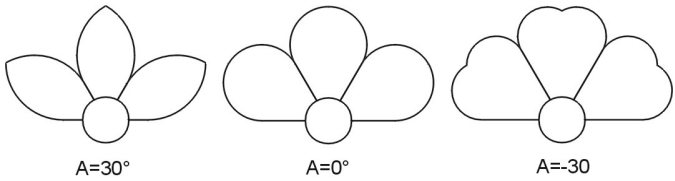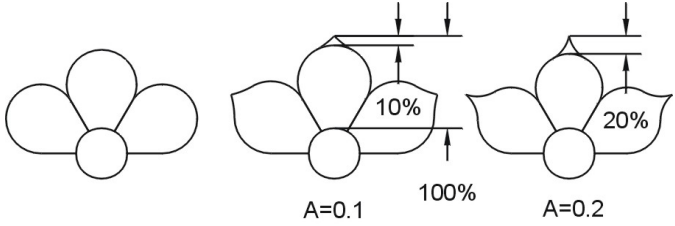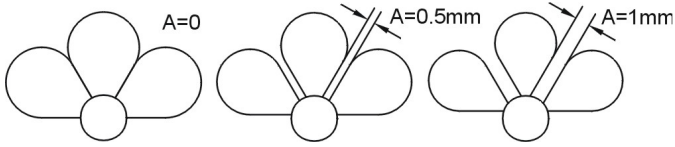
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: MFLOWER
Center point/<Parameter>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	Select the embroidery effect, which must be applied to the flower. You can choose between **None**, **Blatt stitch**, **Abbohr effect** and **Hole**.
Stitch direction	This option appears only in case of stitch type **Blatt stitch**. You can choose between stitch direction **Horizontal** and **Vertical**.
Effect parameters	This option appears only if the stitch type is not **None**. The parameter menu of the selected stitch type is opened and you can setup the desired embroidery parameters.
Bottom closed	If **no**, radial lines lead to the center of the segment. If **yes**, the bottom of the segment is closed with an arc.

Tangentially	If **no**, the lower part of the segment and the upper arcs don't have a smooth, tangential transition. They form a corner at distance **A** from the center. If **yes**, the lower part of the segment and the upper arcs have a smooth, tangential transition.
Top angle	The value defines the deviation angle of the segment top from the tangent. A **positive angle** results in a pointed arch, a **negative angle** results in a heart, a **0** results in an ordinary arch.
Like an arrow	If **yes**, the segment top looks like an arrow.
Hight-factor	This parameter appears only if **Like an Arrow=yes**. It defines the portion of the arrow on the segment height. The value can be between 0 and 0.5.
Segments	Specifies the number of paddles of the flower.
Segment width	If you specify the segment width, the number of segments is derived from this value. You must define the width of a segment at the inside radius.
Segment space	Specify the space between two segments in mm.
Done	Exit from the menu and continue with selecting guide lines.

Cancel	Interrupt the command

If the parameters are defined correctly, define first the center point and thereafter the inside radius and the outside radius of the flower. Finally the flower is constructed by the system.

To construct a flower you can proceed as follows...
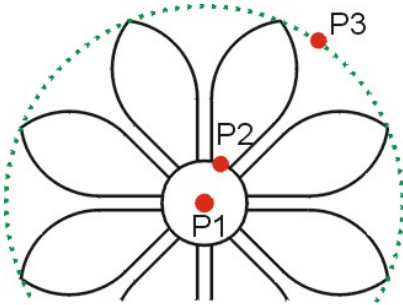

```
Command: MFLOWER
Center point/<Parameter>: P1
Inside radius: P2
Outside radius: P3
```

If **Tangentially=no**, the system will also ask you additionally for the distance to arcs begin.
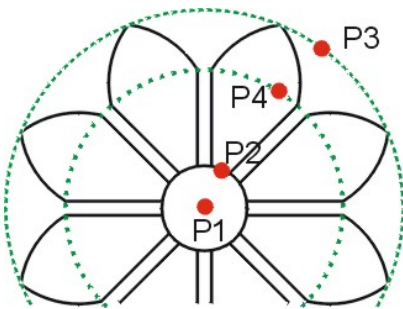

```
Command: MFLOWER
Center point/<Parameter>: P1
Inside radius: P2
Outside radius: P3
Distance to arcs begin<0>: P4
```

MBORDER - How to construct a border

Keyboard: MBORDER

With the border macro you can arrange many different kinds of effects along a polyline. To construct a border, first setup the parameters and then select the guideline, where the effect should be arranged.
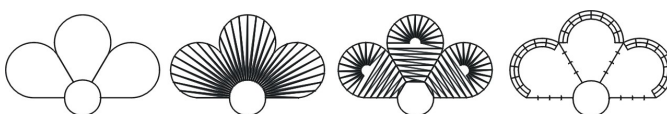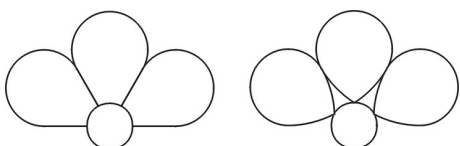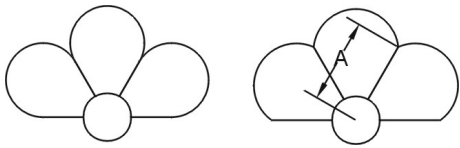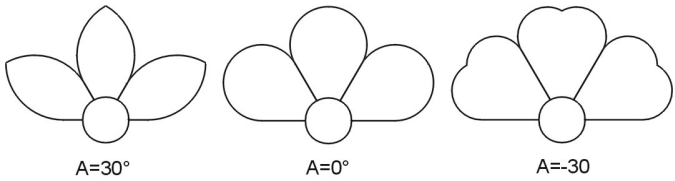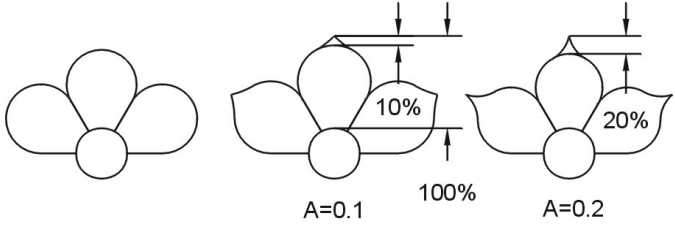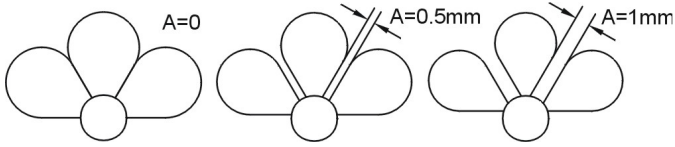
Directly after having called up the command, you can enter the parameter menu with **Return** or with a click on the right mouse button.

```
Command: MBORDER
Select guide line/<Parameters>: <Ret>
```

The following parameters can be defined...

Menu entry	Description
Stitches	Select the embroidery effect, which must be applied to the flower. You can choose between **None**, **Blatt stitch**, **Abbohr effect** and **Hole**.
Stitch direction	This option appears only in case of stitch type **Blatt stitch**. You can choose between stitch direction **Horizontal** and **Vertical**.
Effect parameters	This option appears only if the stitch type is not **None**. The parameter menu of the selected stitch type is opened and you can setup the desired embroidery parameters.
Bottom closed	If **no**, radial lines lead to the center of the segment. If **yes**, the bottom of the segment is closed with an arc.

Tangentially	If **no**, the lower part of the segment and the upper arcs don't have a smooth, tangential transition. They form a corner at distance **A** from the center. If **yes**, the lower part of the segment and the upper arcs have a smooth, tangential transition.
Top angle	The value defines the deviation angle of the segment top from the tangent. A **positive angle** results in a pointed arch, a **negative angle** results in a heart, a **0** results in an ordinary arch.
Like an arrow	If **yes**, the segment top looks like an arrow.
Hight-factor	This parameter appears only if **Like an Arrow=yes**. It defines the portion of the arrow on the segment height. The value can be between 0 and 0.5.
Segments	Specifies the number of paddles of the flower.
Segment width	If you specify the segment width, the number of segments is derived from this value. You must define the width of a segment at the inside radius.
Segment space	Specify the space between two segments in mm.
Done	Exit from the menu and continue with selecting guide lines.

Cancel	Interrupt the command

If the parameters are defined correctly, select first the guide lines and thereafter the heightof the segment. Finally the border is constructed by the system.

To arrange a border macro on a guide line you can proceed as follows...
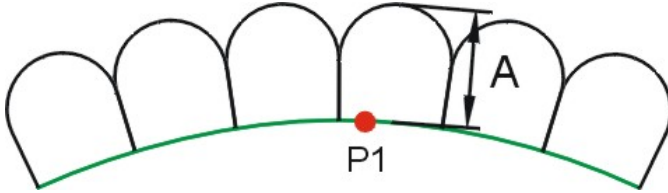

```
Command: MBORDER
Select guide line/<Parameters>: P1
Distance to arcs top <0>: A
```

If **Tangentially=no**, the system will also ask you additionally for the distance to arcs begin.
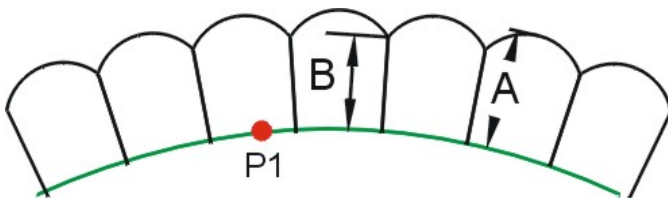

```
Command: MBORDER
Select guide line/<Parameters>: P1
Distance to arcs top <0>: A
Distance to arcs begin<0>: B
```

How to modify embroidery effects

Embroidery effects can easily be modified. There are various commands at your disposal to perform this tasks.

DELEMB - How to delete embroidery objects

Toolbar: **Embroidery > Delete embroidery objects**

Menu: **Enlarge > Delete embroidery objects**

Keyboard: STEP

Elements, generated by embroidery functions, can be erased by picking the guide line or one of the generated elements.

```
Comando: DELEMB
Select guidelines ...
Select entities: select enlarged object
Select entities: <Ret>
```

Hint: It is not necessary to erase embroidery elements if you have selected wrong parameters and if you want to correct the element. In this case use the **MODIFY** command and select the new parameter. The system will automatically update the effect according the new parameters.

MODIFY - How to modify embroidery effects

Toolbar: **Modify embroidery objects > Modify embroidery objects** 🏛

Menu: **Enlarge > Modify embroidery objects**

Keyboard: **MODIFY**

With the **MODIFY** command you can correct the parameters of an embroidery effect or you can also change the effect by itself (e.g. from Steffel to Zuchole or from Single Edge to Double Edge).

How to modify the parameters of an effect

After calling up the command you can select the object, which you want to change. After having selected the element, a popup menu will be displayed with the actual parameters of the effect.

Make the desired changes and apply the changes with **Rebuild**. The effect will be rebuilt according the new parameters.

If you are satisfied with the result you can leave the menu with the option **Done**.

How to change the direction of a Geflect or Blattstitch or the side of a Zuchole

If the selected element is a **Geflect, Blattstitch** or **Zuchole**, you can also find the option **Invert** in the menu.

In case of the **Geflect** or **Blattstitch** you can change the direction. Changing the direction results in moving the starting point to the end. This can be necessary for the puncher in case that he wants to enter the effect at the end.

In case of the single sided **Zuchole** you can change the side, on which the Zuchole will be constructed.

How to modify the effect type

If you want to change the effect e.g. from a single edge to a double edge, you can select the desired effect from the upper part of the popup menu.
After having selected the new embroidery type, the popup menu contains the parameters of the new effect. Make the desired changes and apply the changes with **Rebuild**. The old effect will be removed and the new effect will be generated according the defined parameters.

If you are satisfied with the result you can leave the menu with the option **Done**.

How to rotate circular objects

For some embroidery objects like the **STAR**, the **DOT**, the circular blattstitch **CIRBLATT** and the circular borer **CBORER** you define first the center point and then the radius. The point, where you specify the radius is also the starting point of the punched object. In many cases the puncher must move this starting point to another position by rotating the object around the center point. This rotation can be

performed via the **Rotate** option.

If you select an object of this type you can also find the option **Rotate**. After the selection you can specify the rotation angle or you can choose the option **Reference** to define first a reference angle and then the new angle.

How to modify inclination lines

Like ordinary embroidery objects, also inclination lines can be modified. If you select an inclination line for modification, a popup menu will be displayed with the actual parameters of the inclination line. Make the desired changes and apply the changes with **Rebuild**. The Blattstitch or the Geflect will be rebuilt according the new parameters.

You can also leave the menu with the option **Done**. In this case, the new parameters will be applied, but the effect will not yet be rebuilt.

See also...

📄 [How to define inclination lines](#)

How to move the start- or the endpoint of an effect to the other side

With the option **Invert** you can change the side, on which an effect starts respectively ends. If you want e.g. move the starting point of an effect from inside to outside, just select the first inclination line and click **Invert**. Now the effect will be rebuilt with the starting point on the other side of the effect. The same can be done with the endpoint.

> **Note:** Invert is useful only for the first and for the last inclination line of an effect. In all other cases it has no effect.

How to modify the stitch length of manually punched turns

If the selected object is a group of manually punched stitches you can change the stitch length. Changing the stitch length modifies only the stitch distance of stitches, which are aligned on a straight line. Corner stitches of such a group will not be affected.

Example:

Change the stitch length from 7mm to 3.5mm:

Before change 7mm   After change 3.5mm

Note: This possibility can result in unwanted effects if the user does not know exactly what he does! Therefore this option can be switched On or Off.

To switch this option on you must set the variable **amd_manstiedit=1** otherwise set the variable **amd_manstiedit=0**. The variable can be set in the file **DEFAULT.INI**.

UPDATE - How to update stitches of embroidery effects

Toolbar: **Modify embroidery objects > Update embroidery objects**

Menu: **Enlarge > UPDATE embroidery objects**

Keyboard: **UPDATE**

If you change the position of inclination lines of a blatt stitch, of a geflect etc., or if you change the inclination line type (through, floating, regular), or if you change the size of an element, the stitches are not automatically corrected. This correction can be performed with the command **UPDATE.**

After calling up the command you must select the effect, which must be updated, thereafter the system rearranges the stitches according the new position of the inclination lines or according the new size of the effect.
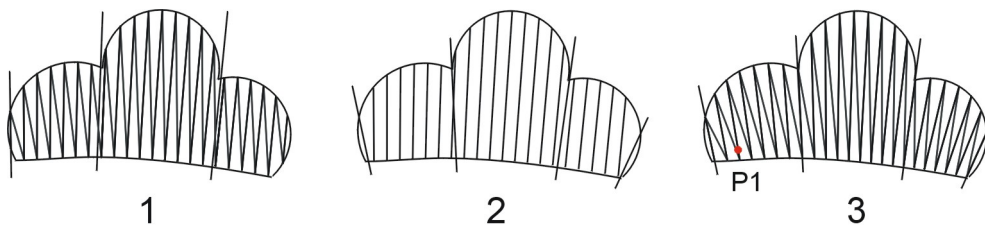
1. Original blattstitch

2. Blattstitch with corrected inclination lines and inclination line types.

3. Blattstitch after update with the **UPDATE** command.

   ```
   Command: UPDATE
   Select guide line: P1
   ```

DENSITY - How to modify the density of embroidery objects

Toolbar: **Modify embroidery objects > Correct density**

Menu: **Enlarge > Density correction**

Keyboard: **DENSITY**

With the command **DENSITY** you can correct the density of all embroidery objects of the design. You can also select the objects, which you want to have changed.

After calling up the command a menu opens.

You can choose from the following options....

Menu entry	Description
All	All embroidery objects of the design are modified.
Select	You can select the embroidery objects, which you want to have changed.

After having selected the objects you can choose in another menu, which embroidery effects should be modified.

You can choose from the following options....

Menu entry	Description
All	The density of all embroidery objects will be corrected.
Blattstitch	If selected, the density of all blattstitch types will be corrected.
Edge	If selected, the density of Edges, wiggle stitches and bored effects will be corrected.
Geflect	If selected, the density of geflect and step-stitches will be corrected.
Steffel	If selected, the density of steffel objects will be corrected.
Borer	If selected, the density of BORER objects will be corrected. (Objects, created with the commands GBORER, DBORER, etc.)

Density correction	Specifies, how much the density is corrected. The value must be defined in %. **Example:** A value of **-20%** means, that an object with a stitch distance of 1mm will have a stitch distance of 1.2mm after the correction.
Done	Exit from the menu and perform the density correction.
Cancel	Interrupt the command

Punching with ProLace

ProLace, the combined Embroidery Design and Punching program allows you to directly punch on ProArt/ProLace designs. The embroidery elements automatically as technical enlargements. The exact number of turns can be determined for elements, which are designed in this way and you can use ProLace to directly take over these elements.

All punching tasks can be performed with the command ![icon] **TURNS**. This command serves to punch manually, to insert underlayers and modules, to insert machine functions and to connect manually punched stitches with previously created embroidery objects like Blattstitches, Edges etc.

To carry out modifications you can use the command ![icon] **MODIFY**. This command allows you to insert stitches, to remove stitches, to break apart punched sequences, to insert and to remove machine functions, etc.

The command ![icon] **OUTPUT** serves to export the punching data to a machine readable file, and the ![icon] **INPUT** command serves to import low level format file for editing.

The **Punching Toolbar** contains the following commands...

	Punching Toolbar
![icon]	TURNS - How to punch and connect with embroidery objects
![icon]	MODIFY - How to modify punched sequences and functions
![icon]	NP - Needle programming
![icon]	OUTPUT - How to export punching data
![icon]	INPUT - How to import punching data for editing

TURNS - How to punch and connect with embroidery objects

Toolbar:

Menu: -

Keyboard: TURNS

With this command you can

- punch a sequence of manual stitches
- you can make underlayers
- place modules along a given line
- insert machine functions or
- connect with embroidery elements, which were designed with ProArt/ProLace automatic stitches.

How to start a punching session

If you call the command for the first time for a design, you have to specify the desired punching parameters and the starting point of the design.

See also...

▤ How to start a punching session

How to define the starting point of a new sequence

If the starting point is just given, you can specify the starting point for a new punching sequence or you can enter a menu from where you can choose from additional options.

See also...

▤ How to define the starting point

How to define vertices

After having defined the starting point of a sequence, you can manually punch stitch by stitch, or you can enter a menu from where you can choose from additional options.

See also...

▤ How to define vertices

How to start a punching session

If you call the punching command for the first time from a certain design, you will get a menu, where you can specify the desired punching parameters. With this parameters you define, which bore respectively color change program will be used and which functions are available during punching.

After having defined all the parameters you can specify the starting point of the design. The starting point is represented by a green triangle. A vertical line will also be inserted in the starting point.

After having defined the starting point you start punching.

See also...

🗒 [How define vertices](#)

How to define the output format

First you have to select the format, which you would like to use.

You can choose one of the following options...

Menu entry	Description
SLC-Plauen	Saurer Low level code for Plauen machines
SLC-Saurer	Saurer Low level code for Saurer machines
SHC	Saurer High level code
Hiraoka DAT	Hiraoka DAT format
Lässer MST	New Lässer MST format
Tajima	Tajima Multihead format
Done	Exit from the menu and continue.
Cancel	Interrupt the command

How to setup punching parameters

After having selected the desired format another menu will be displayed, where you can define boring, color change and function automatic.

You can choose one of the following options...

Menu entry	Description
Color change	Switch color change On/Off and define Needle colors, initial repeat and the color change program.

Borer	Switch Boring On/Off and select the borer program and the initial Borer size in case of SLC-Saurer, SLC-Plauner, Hiraoka or Lässer mode.
Function automatic	Switch Function automatic On/Off. With Function automatic On, the system automatically inserts functions like **BSTI**, **QSTI**, etc.
Done	Exit from the menu and continue.
Cancel	Interrupt the command

Color change

If color change is on, color change functions will be available in the function menu.

The color change settings depend on the previously selected format.

SLC-Saurer, SLC-Plauen and Hiraoka mode

In case of SLC-Saurer, SLC-Plauen and Hiraoka mode a menu will be displayed, where you can select the color change program, the needle colors and the start repeat.

You can choose one of the following options...

Menu entry	Description
Off	Switch color change off
Programs, e.g. Saurer Pentamat old	Select a color change program. Up to 10 programs for different machine types can be defined. The programs are filed in the file **FRWPROG.CSV**, which is located in the directory C:\Icad\Bib\Data
Needle	With this option you can define the display color of each single needle. The number of needles depend on the repeat, which you have defined with the command ▣ **SETUP**. In case of a 12/4 repeat you can define the color for 3 needles.
Start repeat	By default, the start repeat of the design corresponds with the repeat, which you have defined with the command ▣ **SETUP**. With this option you can change the start repeat.

Done	Exit from the menu and continue.

SHC, Lässer and Tajima mode

In case of SHC, Lässer or Tajima format you can just define the needle colors.

You can choose one of the following options...

Menu entry	Description
Needle	With this option you can define the display color of each single needle. The number of needles depend on the repeat, which you have defined with the command ▣ **SETUP**. In case of a 12/4 repeat you can define the color for 3 needles.
Done	Exit from the menu and continue.

Borer

If bore is on, bore functions will be available in the function menu.

The bore settings depend on the previously selected format.

SLC-Saurer, SLC-Plauen and Hiraoka mode

In case of SLC-Saurer, SLC-Plauen and Hiraoka mode a menu will be displayed, where you can select the default bore program for the design.

You can choose one of the following options...

Menu entry	Description
Off	Switch color change off
Programs e.g. STANDARD, SHORT	Select a borer program. Up to 10 programs for different machine types can be defined. The programs are filed in the file **BORPROG.CSV**, which is located in the directory C:\Icad\Bib\Data
Borer size	You can specify the initial borer size
Done	Exit from the menu and continue.

SHC, Lässer and Tajima mode

You can just switch On/Off the bore mode.

Function automatic

In a design there are many functions, for which the system automatically can switch the correct functions. An example is the Blattstitch. If **Function automatic** is on, the system does not offer the **BSTI** function in the function menu and the function is automatically inserted by the system.

In the file **DEFAULT.INI** in the program directory you can specify, which functions should be switched automatically.

Variable	Description
amd_autobsti	If 1 the **BSTI** function will be switched automatically.
amd_bstimin	Specifies the stitch length in mm when the blattstitch function should be switched on. By default this value is set to 3.5mm.
amd_autostg	If 1 the **STG** function will be switched automatically.
amd_autofast	If 1 the system automatically switches to **FAST** mode after Needle On and to **SLOW** mode before Needle off.
amd_autonrempty	If 1 the system inserts an empty stitch before Needle on and after Needle off.

How to define the starting point

After calling up the command you can specify the starting point, or you can enter a menu with **Return** or with a click on the right mouse button.

```
Command: TURNS
From point: <Ret>
```

The following actions can be performed...

Menu entry	Description
Stitches	If yes, a mark will be placed on the stitches
Modules	Define a sequence of stitches and place them along a polyline
Underlayers	Design underlayers on a polyline
Continue at end	Start the new sequence directly at the end of the first punched sequence. This is the sequence, which starts with the START-function.
Connect with endpoint	Connect the starting point of the new sequence with the end point of a previous sequence or with an automatic stitch.
Connect with pickpoint	Connect the starting point of the new sequence with a previous sequence or with an automatic stitch. The connection will be made at a definable point. If necessary the element, with which you want to connect, will be cut.
Done	Leave the menu and continue with the definition of the starting point.
Cancel	Interrupt the command

With **Done** you can leave the menu and continue with the definition of the starting point.

Shortcut keys

Several options are directly available by pressing a key on the keyboard. The following shortcut keys are available...

M - Module

U - Underlayer

E - Continue at end

C - Connect with endpoint

D - Done

How to start at the end of the first punched sequence

With the option **Continue at end** you can start a new punching sequence at the end of the first punched sequence (the sequence, which starts with the START-function). After calling up this option, the system looks for the end of the first sequence, moves the view to this point and you can continue punching.

How to start at the end of a punched sequence or at the end of an automatic stitch

With the option **Connect with endpoint** you can start a punching sequence at the end of another sequence or at the end of an automatic stitch like Blattstitch, Edge, Geflect, etc.

After calling this option you can select the element, from which you want to start.

```
Select guide line: select element
```

If the you want to connect e.g. with a blattstitch, which is not yet included in a punching sequence, select the blattstitch close to the side, where you want to start. The system will snap to the end, which is closer to the pick point (indicated by an arrow).
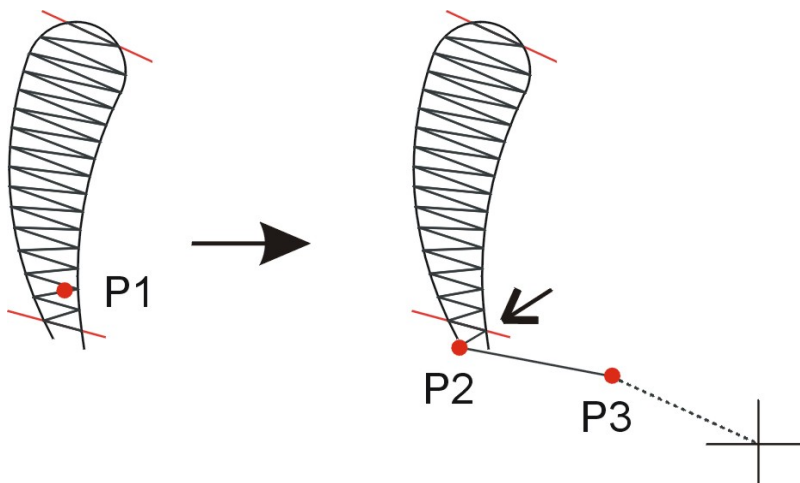

```
Select guide line: P1
To point: Total=123: P2
To point: Total=124: P3
To point: Total=125: Px
```

If you want to connect with a ready punched sequence of elements, just click on an element of the sequence and the system will snap to the end of the sequence (indicated by an arrow).
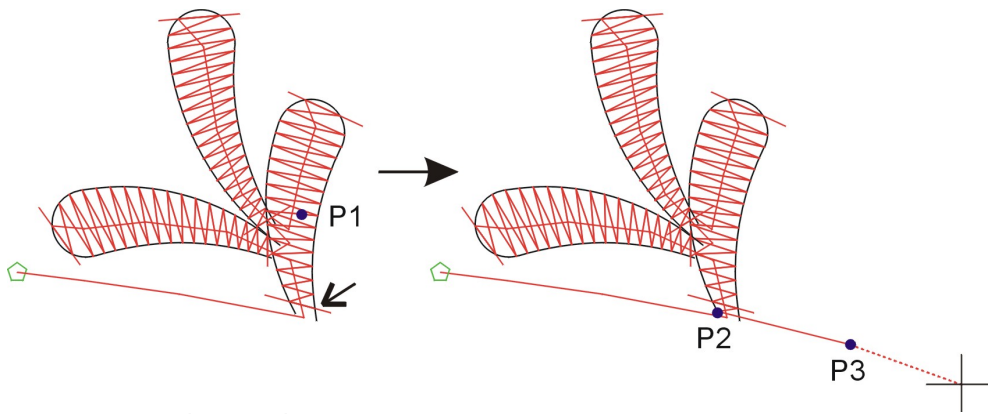
```
Select guide line: P1
To point: Total=123: P2
To point: Total=124: P3
To point: Total=125: Px
```

Hint: For quick access to this option, you can also use the **C**-key on the keyboard.

How to start at a certain point of a punched sequence or of an automatic stitch

With the option **Connect with pickpoint** you can start a punching sequence at the point, where you selected the object. The selected object will be cut to the end and the new sequence will start at the pick point.

After calling this option you can select the element at your desired starting point.

```
Select guide line: select element
```

If the you want to connect e.g. with a single edge, just select the edge at the desired connection point. The edge will be cut to the end and the new sequence will start at this point.
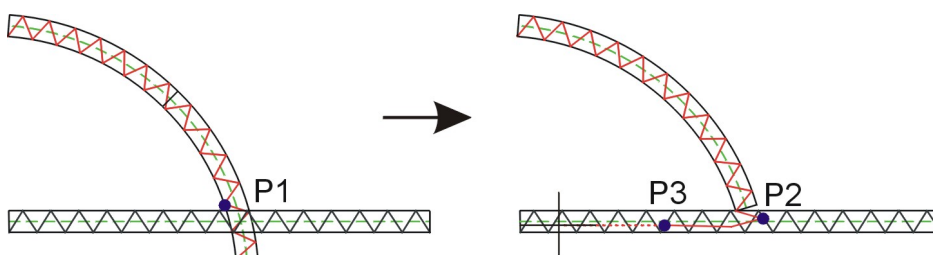

```
Select guide line: P1
To point: Total=123: P2
To point: Total=124: P3
To point: Total=125: Px
```

Note: You can use this option only to connect with objects, which are based on a single polyline. This are e.g. edge, zuchole, step-stitch, wiggle-stitch. You cannot connect with a blattstitch, geflect, etc.

See also...

- [How to use modules](#)
- [How to design underlayers](#)

How to define vertices

After having defined the starting point of a sequence, you can manually punch stitch by stitch, or you can enter an option menu with **Return** or with a click on the right mouse button.

```
Command: TURNS
From point: vertex 1
To point: Total=0: vertex 2
To point: Total=1: vertex 3
...
To point: Total=x: <Ret>
```

The following actions can be performed...

Menu entry	Description
Insert a machine function	You can choose from a selection of possible machine functions. The selection of functions, which you can insert, depends on the actual situation and on the definitions, which you made on start of a punching session. **See also...** [How to insert machine functions](#)
Color	Display color for manually punched stitches. **Attention:** This color setting has no influence on the needle!
Maximum stitch length	Manually punched stitches are divided, if the punched stitch length is longer than the length, defined by this value.
Modules	Define a sequence of stitches and place them along a polyline
Underlayers	Design underlayers on a polyline
Connect with endpoint	Connect the last point of the actual sequence with the starting point of another sequence or automatic stitch.
Connect with pick point	Connect the last point of the actual sequence with another sequence or with an automatic stitch. The connection will be made at a definable point. The object, with which you connect, will be cut.
End command	Terminate the command
Done	Leave the menu and continue with the definition of vertices.

Undo	Undo the last manually punched stitch. Underlayers or modules cannot be undone.
Cancel	Interrupt the command

With **Done** you can leave the menu and continue with the definition of vertices.

Hint: In the lower left corner of the status line you can see the actual number of turns from the start of the actual punching sequence.

Shortcut keys

Several options are directly available by pressing a key on the keyboard. The following shortcut keys are available...

M - Module

U - Underlayer

C - Connect with endpoint

S - Insert Split function

E - End command

D - Done

B - Undo (Back)

1-**9** - Set Maximum stitch length according the pressed digit. 1 is 1mm, 9 is 9mm.

How to connect with the start of a punched sequence or automatic stitch

With the option **Connect with endpoint** you can connect with the start of an already punched sequence or with an automatic stitch like Blattstitch, Edge, Geflect, etc.

After calling this option you can select the element, with which you want to perform the connection.

```
Select guide line: select element
```

If you want to connect e.g. with a blattstitch, select the blattstitch close to the side, where you want to connect. The system will snap to the end, which is closer to the pick point (indicated by an arrow). After having performed the connection, the system jumps to the other end of the blattstitch and you can immediately continue punching.
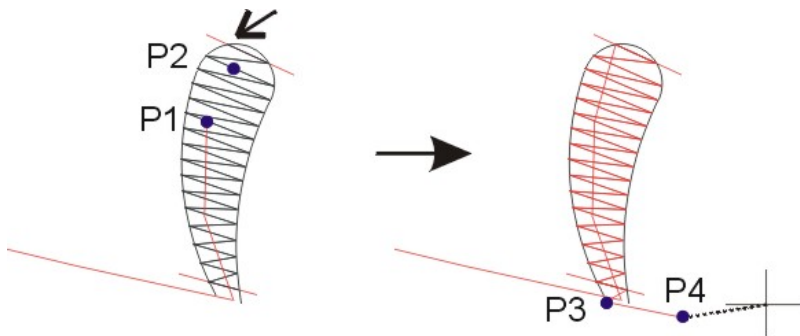
```
To point: Total=123: P1
To point: Total=124: <Ret>

Select Connect with endpoint from the menu

Select guide line: P2
To point: Total=156: P3
To point: Total=157: P4
To point: Total=158: Px
```

If you want to connect with a ready punched sequence of elements, just click on an element of the sequence and the system will snap to the start of the sequence (indicated by an arrow).
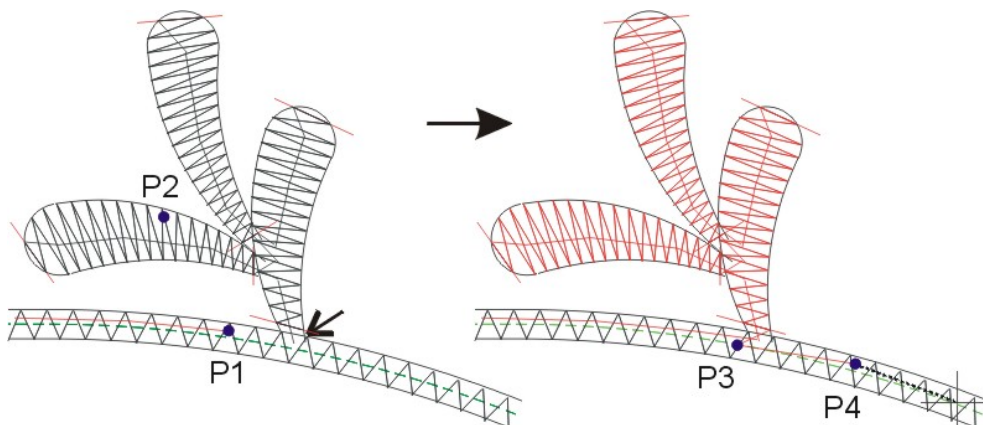

```
To point: Total=123: P1
To point: Total=124: <Ret>

Select Connect with endpoint from the menu

Select guide line: P2
To point: Total=156: P3
To point: Total=157: P4
To point: Total=158: Px
```

Hint: For quick access to this option, you can also use the **C**-key on the keyboard.

How to connect at a point of a punched sequence or automatic stitch

With the option **Connect with pickpoint** you can connect with an already punched sequence at the point, where you select the object. The selected object will be cut at the pick point.

After calling this option you can select the element at your desired starting point.

```
Select guide line: select element
```

If you want to connect e.g. with a single edge, just select the edge at the desired connection point. The edge will be cut to the closer end and the actual sequence will be connected with the edge at the break point. After having performed the connection, the system jumps to the other end of the edge and you can immediately continue punching.
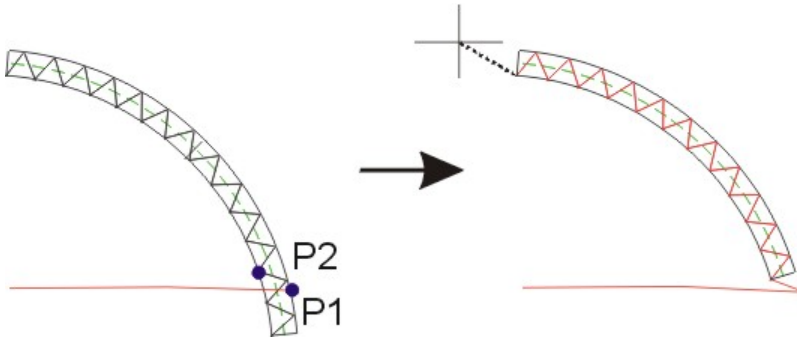

```
To point: Total=123: P1
To point: Total=124: <Ret>

Select Connect with pickpoint from the menu

Select guide line: P2
To point: Total=158: Px
```

Note: You can use this option only to connect with entities, which are based on a single polyline. This are e.g. edge, zuchole, step-stitch, wiggle-stitch. You cannot connect with a blattstitch, geflect, etc.

How to divide long stitches

To prevent too long stitches, you can define the **Maximum stitch length**. If the manually punched stitch is longer than the maximum stitch length, the system automatically divides the long stitch.

How to work with edge pan

If the you insert a stitch close to the border of the graphics area, the display is automatically moved in a way, that the new point is in the center of the screen.

See also...

☰ How to insert machine functions

☰ Description of machine functions

☰ How to use modules

☰ How to design underlayers

How to insert machine functions

In the upper part of the menu you find a selection of machine functions, which can be inserted at the actual position. The selection of machine functions depends on the actual status of the machine and on the definitions, which you made on start of a punching session.
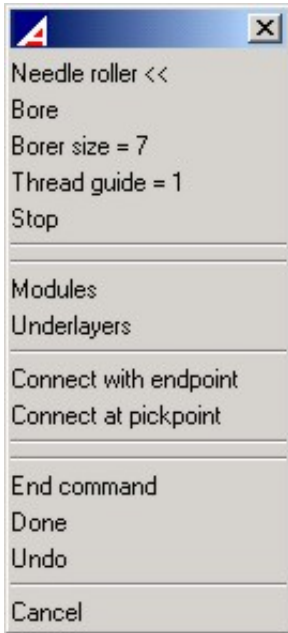
Status of machine functions

The status of functions, which have e certain value, is displayed right from the function description.

```
Borer size = 7
```

To change the value just click on the menu entry and choose the new value. Depending on the function you can choose the new value from a menu...
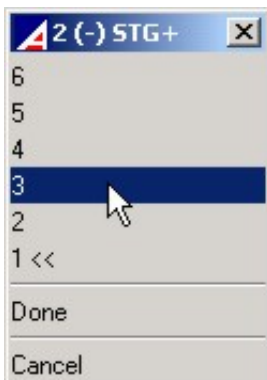
or you just enter the new value by keyboard.

The status of functions, which can be switched on and off, is also visible in the menu entry. A double arrow right from the function description indicates, that the function is on. If the double arrow is missing, the function is off.

```
Needle roller <<
Borer
```

To switch the function just click on the menu entry. If the actual status is On the function will be switched of and vice versa.

Number of turns and more/less display

The actual number of turns are always displayed in the status line. In case of Plauen system also the status of more/less is displayed.

```
To point: Total=0, (+):
```

See also...

 Description of machine functions

How to insert underlayers

Underlayers can be inserted automatically along a polyline or along an embroidery element like an edge. You can insert up to 9 layers. After the first layer, the system produces wiggle stitches.

> **Note:** If you want to insert underlayers along an **edge** you should use the green dashed line of the edge. This line can also be moved from the center to consider the thread tension.

After having selected the **Underlayer** option from the previous menu you should choose the polyline or embroidery element, along which you want to arrange the underlayers. Next you can define, where the underlayers should start and where they should end. If you confirm the enquiries with **Return**, the endpoints of the polyline will be taken over as start- and endpoint for the underlayer generation.

```
Command: TURNS
From point: vertex 1
To point: Total=0: vertex 2
To point: Total=1: vertex 3
...
To point: Total=x: <Ret>

>> Select option Underlayers from the menu <<

Select guide line: select
From point: underlayer start
To point: underlayer end
```

Now the underlayer menu is displayed.

You can choose one of the following options...

Menu entry	Description
Total number of underlayers	Total **T** number of layers, which should be inserted along the selected element.
Start layer	Layer **S**, with which you want to start. By default the start layer is 1.
Actual number of underlayers	Number **N** of layers, which you want to insert now.
Underlayer distance	Stitch length **A** of underlayer stitches.
Width	Wiggle width **B**. This value will only be of meaning if more than one layer is designed.
Build	Generate the underlayers and return to the previous menu.
Cancel	Interrupt designing underlayers.

Example 1:

3 underlayers along a polyline. Start- and endpoint are defined manually.

```
Select guide line: P1
From point: P2
To point: P3
```
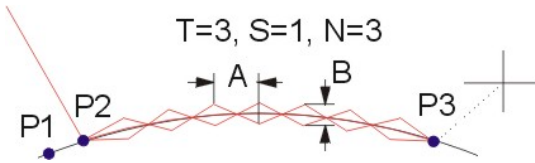

Example 2:

3 underlayers along a polyline. Start- and endpoint correspond with the endpoints of the polyline.

```
Select guide line: P1
From point: <Ret>
To point: <Ret>
```
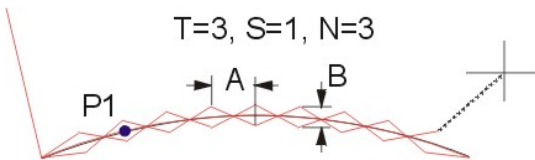

Example 3:

Sometimes it's necessary to insert e.g. 3 layers of underlayers, but after the first layer, you should continue with another element and come back to insert the remaining 2 layers.

In this case you can call up the underlayer option 2 times.

The first time you should define **Total number of underlayers=3**, **Start layer=1** and **Actual number of underlayers=1**. The second time you should define **Total number of underlayers=3**, **Start layer=2** and **Actual number of underlayers=2**.

Now 3 underlayers should be inserted on an edge. Start- and endpoint correspond with the endpoints of the edge. After the first layer, the underlayers of a dot have to be inserted before the two remaining layers can be inserted.

Definition of the first layer:

```
Select guide line: P1
From point: <Ret>
To point: <Ret>
```

T=3, S=1, N=1

P1
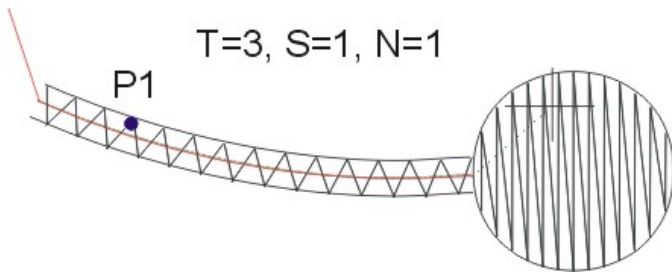
Definition of the remaining two layers:

```
Select guide line: P1
From point: <Ret>
To point: <Ret>
```


T=3, S=2, N=2

P1
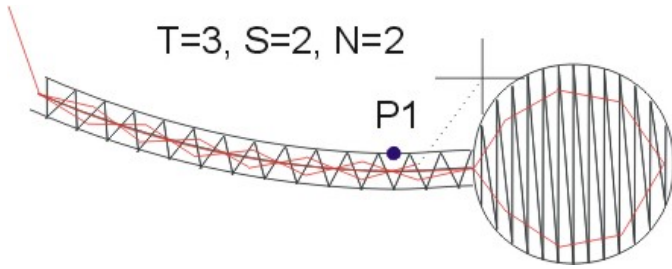
How to work with modules

A module is a pattern of stitches, which can be recorded under a certain name and arranged along a polyline (running stitch).
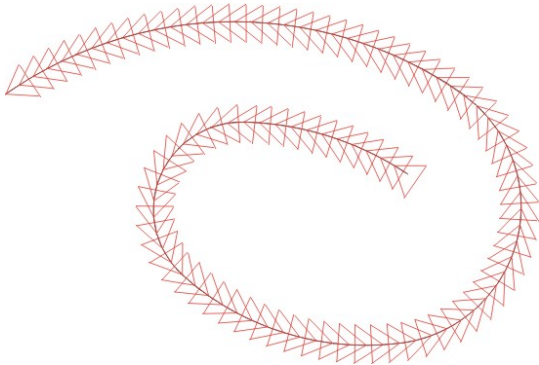
The modules are saved in the drawing and can be called up also in later editing sessions.

> **Attention:** As the modules are saved in the drawing they will not be available in other drawings.

Modules can be inserted automatically along a polyline. After having selected the **Module** option from the previous menu the module menu is opened.

You can choose one of the following options...

Menu entry	Description
Define new	Define a new module and save it under a certain name.
list of modules	A list of existing modules is displayed, from which you can choose the one you want to have arranged along the polyline.
	When you call up the module option for the first time from a certain design, no modules will be listed. Before you can work with a module you will have to define one first.
Cancel	Interrupt the module option and return to the previous menu.

How to define a module

Before you can arrange modules along a polyline you will have to define them. With the option **Define new** you can create new modules.

A module will always be defined on base of an existing polyline. First you have to choose this polyline, next you define the base point of the module. This point is normally a point on the polyline. Now you can continue designing the module. The

last point of the module should be the point, where the next copy of the module should start. This point is normally also on the polyline. After having defined the last point, you can terminate the definition of stitches with **Return**. A menu opens, where you have to choose the **Enter** option to terminate the module definition. In this menu you can also choose **Undo** in case you want to go back one step. Finally you have to define a name for the module.

Example:

With the following dialog you can create a new module named **Triang**.
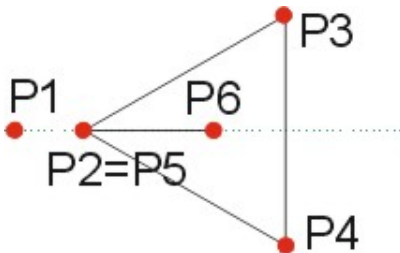

```
Command: TURNS
From point: point
To point: Total=0: <Ret>

>> Select option Module from the menu <<

>> Select option Define new from the menu <<

Select guide line: P1
Insertion point: P2
To point: P3
To point: P4
To point: P5
To point: P6
To point: <Ret>

>> Select option Enter from the menu <<

Module name: Triang
```

How to arrange a module along a polyline

Once a module is defined you can arrange it along a polyline.

To arrange a module choose a previously defined menu from the module menu and choose the polyline, along which you want to arrange the module. Next you can define, where the modules should start and where they should end. If you confirm the enquiries with **Return**, the endpoints of the polyline are taken over as start- and endpoint for the module distribution.

Example:

With the following dialog you can create arrange the previously defined module **Triang**.
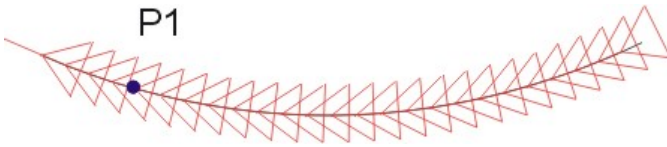
```
Command: TURNS
From point: point
To point: Total=0: <Ret>

>> Select option Module from the menu <<

>> Choose module Trian from the menu <<

Select guide line: P1
From point: <Ret>
To point: <Ret>
```

MODIFY - How to modify punched sequences and functions

Toolbar: **Modify embroidery objects > Modify embroidery objects**

Menu: **Enlarge > Modify embroidery objects**

Keyboard: MODIFY

This command allows you to modify machine functions or punched sequences, created with the **TURNS** command or read in with the **INPUT** command.

You can...

- insert stitches
- remove stitches
- break apart punched sequences
- insert machine functions
- remove machine functions
- edit machine functions
- navigate in the punched design

After calling up the command you can select the object, which you want to change and perform the desired modification.

See also...

How to navigate in the design

How to modify a machine function

How to modify punched sequences

How to navigate in the design

After having selected a punched sequence or a machine function, the selected object will be highlighted, a mark will be displayed at the selected position and a popup menu will be displayed. The upper options of the menu allow you to navigate in the design and are the same for functions, embroidery objects like Blattstitches, Edges etc. and punched sequences, created with the **TURNS** command.

You can choose from the following navigation options...

Menu entry	Description
Previous	One back
Next	One step further
Step	Specify the step width. You can choose between Turns, Objects and Functions

With **Previous** you can go one step back, with **Next** you can go one step further. The mark will be moved to the new position and in case, that you step to another object, the new object will be highlighted.

Note: To step faster through the design you can also use the keys **B** to step to the **Previous** position and **N** to step to the **Next** position.

With **Step** you can specify the step length for the the **Previous** and **Next** option. You can choose between...

- **Turns** - you can step through the design turn by turn. The mark indicates the actual position.

- **Objects** - you can step from embroidery object to embroidery object. An embroidery object can be a machine function, a punched sequence, created with the **TURNS** command, or a high level object like Blattstitch, Edge, Geflect, etc.

- **Functions** - you can step from machine function to machine function.

Note: Embroidery objects like **Blattstitches** and **Edges** are seen as compound objects and single stitches cannot be modified. Also in the **Turns** mode, the program will jump over the object.

How to modify a machine function

After having selected a machine function, the symbol will be highlighted and a popup menu will be displayed. In the upper bar of the menu you can find the number of turns and the name of the selected function.

You can choose one of the following options...

Menu entry	Description
Previous	One step further
Next	One step back
Step	Specify the step width. You can choose between Turns, Objects and Functions
Function value	The value of the selected function is displayed and can be modified. Some functions have no values. In this case no menu entry is displayed.
Connect	Connect the function with an already punched sequence.
Delete	Delete the selected function
Done	Exit from the menu and select the next object to be modified.
Cancel	Interrupt the command

See also...

[How to navigate in the design](#)

How to modify punched sequences

After having selected a punched sequence, it will be highlighted, a mark will be displayed at the actual position and a popup menu will be displayed. In the upper bar of the menu you can find the number of turns at the marked point.

You can choose one of the following options...

Menu entry	Description
Previous	One step further
Next	One step back
Step	Specify the step width. You can choose between Turns, Objects and Functions
Stitches	Switch the display of stitches On/Off.
Highlight previous	If **Highlight previous=Off**, only the actual embroidery object is highlighted. If **On**, also all embroidery objects before the actual object are highlighted.
Pan center	Move the actual position (position of the mark) to the center of the screen.
Zoom +	Enlarge the display size by factor 1.5.
Zoom -	Reduce the display size by factor 0.75.
Insert function	Insert a machine function at the position of the mark.
Break between 2 points	All stitches between two points are removed and the punched sequence between the two points is opened.
Straighten between 2 points	All stitches between two points are removed and the two points are connected with a stitch.
Insert point	Insert stitches in a punched sequence.
Break till the end	Removes all points of a sequence from the actual position to the end of the element.
Connect	Connects the end of the actual sequence with the start on another embroidery object. The endpoint is moved to the starting point of the selected object.
Delete	Delete the selected sequence
Done	Exit from the menu and select the next object to be modified.

Cancel	Interrupt the command

See also...

▤ How to navigate in the design

How to insert a machine function

If you select this option a menu will be opened where you find a selection of machine functions, which can be inserted at the actual position. The selection of machine functions depends on the actual status of the machine and on the definitions, which you made on start of a punching session.

See also...

▤ How to insert machine functions

Shortcut keys

Several options are directly available by pressing a key on the keyboard. The following shortcut keys are available...

B - Previous

N - Next

I - Zoom +

O - Zoom -

P - Pan center

How to break a sequence between 2 points

With the option **Break between 2 points** all stitches between the mark and a defined point are removed and the punched sequence between the two points is opened.

After selecting this option you can pick the end point of the part, which has to be removed. The part between the mark and the defined point will be removed.

How to remove stitches

With the option **Straighten between 2 points** all stitches between the mark and a defined point are removed and the section is connected with a stitch.

After selecting this option you can pick the end point of the section, which has to be straightened. The stitches between the mark and the defined point will be removed and the two points will directly be connected.

How to insert stitches

With the option **Insert point** you can insert one ore more stitches in the actual sequence. First you have to select the stitch, where you want to insert an additional point, then you can define the position of the new point.

Example:

Insert two stitches in a punching sequence.
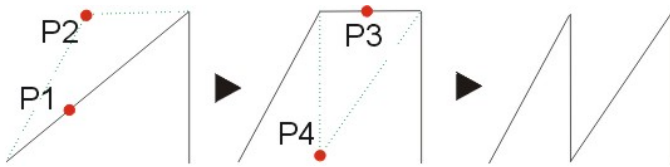

```
Select stitch: P1
Insertion point: P2
Select stitch: P3
Insertion point: P4
Select stitch: <Ret>
```

How to remove stitches from the actual position to the end of the sequence

With the option **Break till end** you can remove all stitches from the actual position to the end of the actual sequence.

This option is only available if the actual element is not yet connected with another element.

How to connect a sequence with another embroidery object

With the option **Connect** the end of the actual sequence can be connected with another embroidery object. The object can be an embroidery object like a Blattstitch, an Edge, a Step stitch etc., a machine function or another punched sequence. The endpoint of the actual sequence is moved to the starting point of the selected object.

This option is only available if the actual element is not yet connected with another element.

OUTPUT - How to export punching data

Toolbar: Write machine code

Menu: -

Keyboard: OUTPUT

With this command you can write the punched data to a machine readable file format. After calling up the command, the output menu will be opened where you can choose the desired format.

You can choose one of the following options...

Menu entry	Description
SLC-Plauen	The data are written in Saurer SLC-Plauen format. The file gets the file extension **pls**.
SLC-Saurer	The data are written in Saurer SLC-Saurer format. The file gets the file extension **sas**.
SHC	The data are written in Saurer SHC format. The file gets the file extension **pat**.
SLC-Plauen	The data are written in Saurer SLC-Plauen format. The file gets the file extension **pls**.
Hiraoka DAT	The data are written in Hiraoka DAT format. The file gets the file extension **dat**.
Lässer	The data are written in Lässer format. The file gets the file extension **mst**.
Tajima	The data are written in Tajima format. The file gets the file extension **dst**.
X Scale factor	Scale factor, by which the design is stretched horizontally.
Y Scale factor	Scale factor, by which the design is stretched vertically.
Generate info-files	If this option is ticked, the system creates the infofiles **name.err** (error messages), **name.log** (list of issued machine commands) and **name.hex** (list of issued machine commands in Hex).
Test functions only	This option allows you to test the design. The system will inform you about too short and too long stitches, more/less errors and machine function errors.
Done	Write the data in the selected format.
Cancel	Interrupt the command.

After having left the menu with the option **Done** the file dialog-box opens, where you can specify the file name and folder. If the option **Generate infofiles** is selected, the system creates also a file named **name.err**. This error file informs you about problem, which the system detected on export. If the system detects too long stitches, they are divided automatically.

When the stitch data are written, the system detects short stitches. After having written all data the systems informs you about short stitches with needle on and short stitches with needle off.

The values for the short stitch detection can be defined in the file **DEFAULT.INI**

```
amd_minsanon = 10     > minimal Saurer needle on stitch
amd_minplnon = 6      > minimal Saurer needle off stitch
amd_minsanoff = 50    > minimal Plauen needle on stitch
amd_minplnoff = 50    > minimal Plauen needle off stitch
```

How to detect punching errors

To test the your design for punching errors you can select the option **Test functions only**. In this case no data are written. The system just scans the whole design and checks it for errors.

The system checks the design for the following problems:

- Too short stitches
- Too long stitches
- More/Less errors in case of SLC-Plauen format
- Machine function errors

When the system detects a problem, an error message will be displayed e.g. **Too long stitch at turn 62 element 10F4H!**. After confirming with OK, the system jumps to the position, where the problem was detected and you can choose between the following options:

- **Continue** if you want to continue testing
- **Edit** if you want to correct the problem immediately
- **Cancel** if you want to stop testing

Note: The minimum and maximum stitch length for needle on and needle off can be defined in the file **DEFAULT.INI**. This file is located in the program directory **C:\Icad**.

How to stretch the design

Sometimes it is necessary to stretch a design horizontally or vertically. With the option two options **Scale factor** you can specify a scale factor, by which the design will be stretched on export. The default value is 1. E.g. if you want to stretch the design vertically by **10%** you have to choose the option **Y Scale factor** and enter a value of **1.1**.

Note: The design will not be changed by this value!

INPUT - How to import stitch data for editing

Toolbar: Read machine code

Menu: -

Keyboard: INPUT

This command allows you to read files in machine code for editing. After calling up the command, you can choose the desired file format from a menu. Actually files of the type **PLS** (SLC-Plauen), **SAS** (SLC-Saurer), **PAT** (SHC). **DAT** (Hiraoka), **MST** (Lässer) and **DST** Tajima are supported.

You can choose one of the following options...

Menu entry	Description
SLC-Plauen	SLC-Plauen file with the extension **PLS**
SLC-Saurer	SLC-Saurer file with the extension **SAS**
SHC	SHC file with the extension **PAT**
Hiraoka DAT	Hiraoka file with the extension **DAT**
Lässer	Lässer file with the extension **MST**
Tajima	Tajima file with the extension **DST**
Color change	Define color change type
Function automatic	If this option is activated, all functions are filtered out from the design, which are inserted by ProLace automatically when stitch data are written. This are e.g. Speed, Blattstitch, Small Thread Guide,...
Cancel	Interrupt the command

After having selected the file type, you can choose the desired file with the file dialog-box and finally you must specify the point, where the start should be inserted.

Note: Importing a design from a low level format like SLC, SHC etc. can result in very long uninterrupted sequences. To prevent this effect the system can split such a sequence in smaller groups of stitches.
With the variable **amd_maxnosti** you can define after how many stitches the system creates a new group.
This variable can be defined in the file **DEFAULT.INI**. By default this variable is set to 1000.

STOP handling in case of Tajima format

If **Function automatic** is **on**, a stop switches automatically to the next needle.

If **Function automatic** is **off** and a stop is detected, you can choose from a menu

in which ProLace function the stop should result.

You can choose one of the following options...

Menu entry	Description
Stop	Insert a **STOP** command
Borer	Switch Borer On/Off
Change needle	Change to another needle. If you choose this option you can select one of 12 needles.

NP - Needle programming

Toolbar: Needle pattern ▨

Menu: -

Keyboard: NP

With this command you can program the needles for Lässer and new Saurer machines. Up to 100 stations can be defined.

After calling up the command for the first time, a dialog-box opens with the standard stations according the actual repeat. An **X** means Needle On a **0** means Needle Off. You can change the programming of a needle of a station by simply clicking onto the corresponding 0/X. When you leave the dialog-box the Needle Pattern file will be created. In case of **Saurer SHC**, the needle information is written to a file with the extension **.npf**, in case of Lässer, the information is written into the header of the **.mst** file on writing the stitch data.

Later, if you call up the command again, the corresponding Needle Pattern file will be read and displayed in the dialog-box. Now you can make the desired modifications. When you leave the dialog-box, the changes are written to the file.
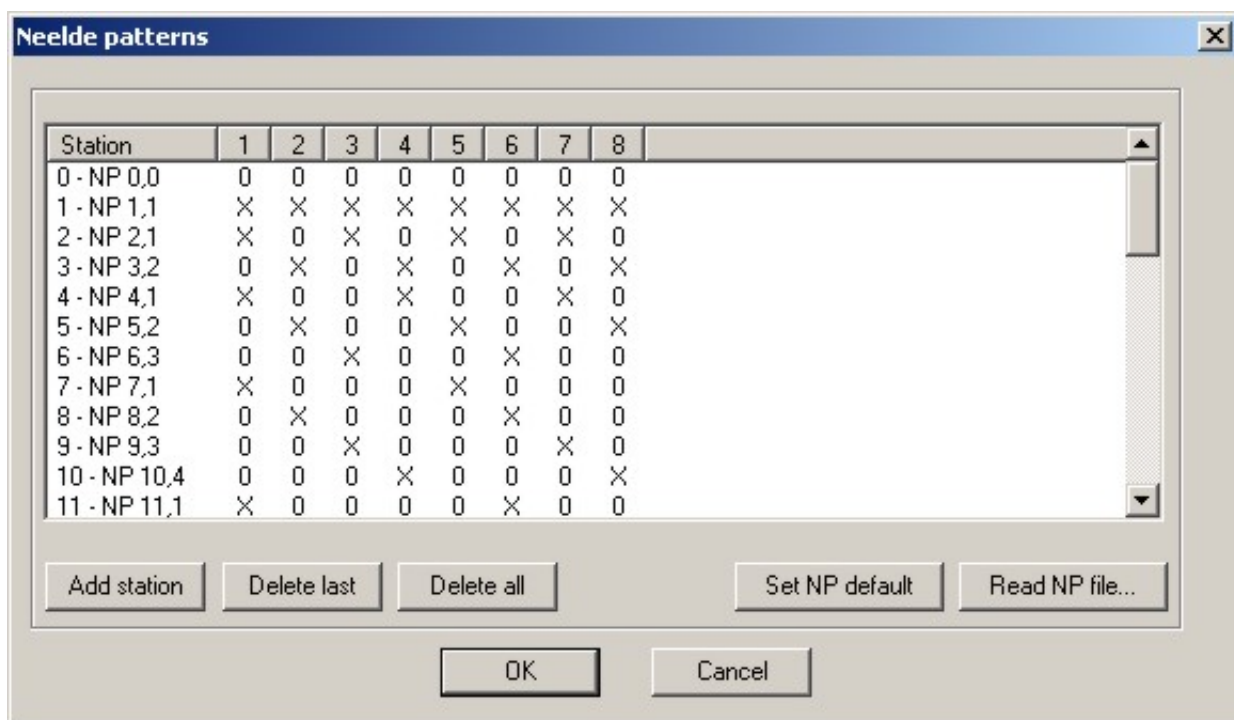
Add station

Add a new station to the end of the list.

Delete last

Delete the last station in the list.

Delete all

Delete all stations in the list.

Set NP default

Set the stations to their initial values.

Read NP file...

Via this button you can read an existing Needle Pattern file.

Description of machine functions

ProLace actually supports Saurer SHC, Saurer SLC, Hiraoka DAT, Lässer MST and Tajima DST formats. The SLC format is divided in SLC 46 Plauen and SLC 22 Saurer formats.

See also...

- The SLC 22 Saurer format
- The SLC 46 Plauen format
- The SHC Saurer format
- The Hiraoka DAT format
- The Lässer MST format
- The Tajima DST format

The SLC 22 Saurer format

The Code type SLC 22 covers all 22 channel Saurer card based machines.

General data

1 machine unit = 1/10mm

Value range in X and Y direction: -171 to +171

Standard functions

In the column **Def.** you can find the default values on startup of the program and in the column **Range** you can find the value range of the function. A rang with + behind the value means, that the value of the function can only be increased (e.g. TB: 1->2->3->4->1->2->...). A range with +/- behind can be increased and decreased within the given values.

SLC-No	Mnem.	Def.	Range	Description
0, 1				no function
2, 3	NR	Off	On/Off	Needle roller
4, 5	TB+	1	1..4 +	Thread break
6, 7	STU	Off	On/Off	Stupfel
8, 9	BSTI	Off	On/Off	Blattstitch
10, 11	SC	On	On/Off	Speed control
12, 13	STOP			Stop
14, 15	STG+	1	1..6 +	Thread guide
16	BS-	6	1..12 +/-	Borer size minus
17	BS+	6	1..12 +/-	Borer size plus
18, 19	CRD	Off	On/Off	Cord laying
20, 21	CLU	Off	On/Off	Clutch
22	PS-	6	1..12 +/-	Production speed minus
23	PS+	6	1..12 +/-	Production speed plus
28, 29	F	Off	On/Off	Feston
30, 31	NOP			No operation
128	COM			Comment
129	END			Block end

130	REP			Repeat start
131	WUP			
255	CODE			Code type

Color change FRW old (STANDARD)

Machine type: Saurer schiffli embroidery machine

Color change system: Pentamat old, FRW Zangs

SLC-No	Mnem.	Original	Description
6, 7	SS	STU	Pentamat
18, 19	FRW-	CRD	FRW station -
28, 29	FRW+	F	FRW station +

The SLC 46 Plauen format

The Code type SLC 46 covers all 46 channel Plauen card based machines.

General data

1 machine unit = 1/6mm

Value range in X and Y direction: -99 to +99

Standard functions

In the column **Def.** you can find the default values on startup of the program and in the column **Range** you can find the value range of the function.

SLC-No	Mnem.	Def.	Range	m/l	Description
2	NR-	x		l	Needle roller off
3	NR+			m	Needle roller on
6	STU+			l	Stupfel on
7	STU-	x		m	Stupfel off
8	BST+			l	Blattstitch on
9	BST- ,BH	x		m	Blattstitch off, Borer
10	SLOW	x		l	Slow on
11	FAST			m	Slow off
12	STOP			l	Stop
13	STOP			m	Stop
14	STG+			l	Thread guide plus
15	STG-	x		m	Thread guide minus
16	BS-	7	1..13 +/-	l	Borer size minus
17	BS+	7	1..13 +/-	m	Borer size plus
18	CRD+			l	Cord laying on
19	CRD-	x		m	Cord laying off
255	CODE				Code type

Color change FRW old (STANDARD)

Machine type: Zangs schiffli embroidery machine

Color change system: FRW Zangs

SLC-No	Mnem.	Original	m/l	Description
6	FRW-	STU+	l	FRW station -
7	FRW+	STU-	m	FRW station +
19	CS-	CRD-	m	Switch axis

The SHC Saurer format

The Code type SHC covers the new Saurer machines like Epoca, Unica.

General data

1 machine unit = 1/10mm

Vector functions

The following table lists all SHC vector functions.

SHC-No	Mnemonic	Description
0	MOV	Move withour thread
1	STI	Stitch
2	BOR	Bore stitch
3	STU	Stupfel
4	BSTI	Blattstitch
5	QSTI	Quilt stitch
6	SSTI	Single hole stitch
7	DSTI	Double hole stitch
8	SUSP	Suspension stitch

In the punching mode you can only Swith On/Off the Needle and the Borer. The real functions **MOV**, **STI**, **BOR**, **BSTI**, **QSTI**, **SSTI** and **DSTI** will be specified by the system when you write the machine code with the command **OUTPUT** .

Parameter functions

In the column **Def.** you can find the default values on startup of the program and in the column **Range** you can find the value range of the function.

SLC-No	Mnem.	Def.	Range	Description
128	COM		0..255	Comment
129	END			End of block
130	REP	1	0..65535	Repetition
131	STOP	0	0..65535	Machine stop
132	CALL	1	1..410E9	Design call
133	DES	1	1..410E9	Design number

134	ARR	1	1..410E9	Arrangement number
246	PRGM		0..255	Function number
247	TH		0..255	Thread length
248	PS	10000	0..10000	Production speed
249	BS		1..13	Borer size
253	UNIT			Vector unit
254	NP			Needle pattern
255	CODE			Code type

Standard settings for the Needle patterns

If no needle patterns are programmed, the following standard needle pattern settings are used by the system:

	1	2	3	4	5	6	7	8
All off	0.1	-	-	-	-	-	-	-
4/4	1.1	-	-	-	-	-	-	-
8/4	2.1	3.2	-	-	-	-	-	-
12/4	4.1	5.2	6.3	-	-	-	-	-
16/4	7.1	8.2	9.3	10.4	-	-	-	-
20/4	11.1	12.2	13.3	14.4	15.5	-	-	-
24/4	16.1	17.2	18.3	19.4	20.5	21.6	-	-
28/4	22.1	23.2	24.3	25.4	26.5	27.6	28.7	-
32/4	29.1	30.2	31.3	32.4	33.5	34.6	35.7	36.8

Example:

To work with **needle 2** in the **20/4** repeat, you can use the function **NP 12.2**.

Change needle and Change repeat

If the needle pattern is set to default you have instead of the command **NP** the commands **Change needle** and **Change repeat** in the the function menu. When you use one of theese command, the system translates them automatically to the corresponding needle pattern command **NP**.

Example:

If the actual repeat is **16/4** and you change to **needle 3** the system generates the command **NP 9.3**.

NP command

If you defined your own needle patterns or if you made changes to the standard needle patterns the **NP** commands appears in the the function menu.

To insert a NP function click on the option NP. Now the system shows you the actual value in the status line. Now you can enter a new value.

Example:

To change from repeat 12/4, needle 1 to 12/4 needle 2 enter the following value:

```
NP<4.1>: 5.2<Ret>
```

Instead of entering a new value you can also right click to open the window with the available stations and choose the desired station.

The Hiraoka DAT format

The Hiraoka's computerized embroidery machine decodes and executes the data stored in this format. The DAT file format is used for standard embroidery machines without the automatic repeat switching mechanism. It has the extension **DAT**.

General data

1 machine unit = 1/6mm

Value range in X and Y direction: -99 to +99

Standard functions

In the column **Def.** you can find the default values on startup of the program and in the column **Range** you can find the value range of the function.

F-No	Mnem.	Def.	Range	Description
00	NOP			No operation
01	BSTI	off		Sperrzeug
02	NR	off		Needle
03	BS	7	1..13 +/-	Borer size
04	STOP			Stop
05	SLOW	on		Slow
06				Not used
07	CRD	off		Cord laying
08	NR+SLOW			Needle + Slow
09	BSTI+BS			Sperrzeug + Borer size
10	NR+BS			Needle + Borer size
11	SLOW+BS			Slow + Borer size
12				Not used
13	CRD+BS			Cord laying + Borer size
14	NR+SLOW+BS			Needle + Slow + Borer size
15				Pause

The Lässer MST format

The Lässer format describes the functions for all computerized Lässer machines. The file extension of Lässer files is **MST**.

General data

1 machine unit = 1/10mm

Value range in X and Y direction: -171 to +171

Standard functions

In the column **Def.** you can find the default values on startup of the program and in the column **Range** you can find the value range of the function.

Mnem.	Def.	Range	Description
NR	Off	On/Off	Needle roller
BSTI	Off	On/Off	Blattstitch
BOR	Off	On/Off	Borer
FAST	Off	On/Off	Fast speed
LTG	Off	On/Off	Large thread guide
CRD	Off	On/Off	Cord laying
CRZ	Off	On/Off	Cord Zig Zag*
BS	6	1..13	Borer size regulation
LTGR	6	1..13	Large thread guide regulation
STG	1	1..4	Small thread guide regulation
TB	1	1..3	Thread break
CRDA	0	0..90	Cord angle
HSR	6	1..9	Head regulation*
PAI	0	0..7	Paillette regulation
NP	1	1..99	Needle program*
STOP			Stop
STOPT			Stop + text*

* not yet supported

The Tajima DST format

Most Multihead embroidery machines can decode and execute the data stored in this format. The file extension of Tajima files is **DST**.

General data

1 machine unit = 0.1mm

Value range in X and Y direction: -121 to +121

Borer displacement: 120mm

Standard functions

In the column **Def.** you can find the default values on startup of the program.

F-No	Mnem.	Def.	Description
03	STI	off	Normal stitch
83	MOV	on	Jump stitch
C3	STOP		Stop / Needle change / Borer on/off
F3	STOP		End of design
43	PAI		Changeover (Paillette)