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PrimCAM registration form

Please fill in this registration form and

- Fax it (0041-55-418 49 50) or
- Send it (fold and glue together)

Then you will receive

- Information about the new versions
- Technical support for questions about PrimCAM

Company:	
Name:	
Street, Number (or p.o. box):	
State, City:	
Country:	
Phone:	
Fax:	
Email address:	
Licence no. (see label on dongle):	

Suggestions: _____

_____

✍ We **don't** want our company to be on the reference list!

_____ ✍ _____

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Switzerland

PrimCAM®

User's Guide

Contents

Important information about PrimCAM

General

- Not all parts of PrimCAM are programmed for a variable screen resolution yet. Choose **1024x768** pixels and 256 (or more) colors for the moment.
- The function key **F12** allows you to switch on and off the title bar of the main PrimCAM window. The title bar is off by default, giving you more space for your drawings.
- Switch off the task bar, giving the whole screen to PrimCAM. This is done in Startmenu/Settings/Task bar - automatic in the background.
- Windows **NT/2000/XP**: Make sure that you are logged in as a system administrator when installing PrimCAM, because that's the only way the hardlock driver can be installed. You can also install the hardlock driver by calling HLDINST.EXE in the PrimCAM directory: hldinst –install

Reprogramming the dongles for PrimCAM 2.x

- Old dongles (DOS) have to be reprogrammed for the windows version. This can be done by entering a password. Ask your dealer or PRIMUS DATA for your password.

Fonts

- True Type fonts can be imported for milling now. They can be milled axis parallel, not contour parallel yet.

Postprocessors

- Postprocessoren now come as precompiled versions (*.DLL). They translate much faster than the old ones (*.PP), but they cannot be changed by the user.
- Interpreted postprozessors (*.PP) are only processed by PrimCAM M3 (with open postprocessor system).
- PP's can be compiled to DLL's by PRIMUS DATA.

Preface

The demand for flexible, in due time, high-quality, and at the same time **profitable fabrication** is permanently increasing. With the introduction of new C-technologies an essentially higher productivity is perceptible not only in the construction but also in the tool manufacture. Particularly medium sized and small scale enterprises are challenged intensely, as they often aren't able to install expensive CAD/CAM-solutions for financial reasons.
PrimCAM is an efficient and user-friendly software package for **creating CNC-programs for manufacturing machines** which will satisfy these requirements. You can already create the NC-programs for the next workpieces while the machine is still working. By use of PrimCAM the machines work more productively and **without stop periods**. The simulation on the screen before creating the program eliminates collisions of tools with the clampings or the workpiece.

Motivation for the development of PrimCAM

In 1991, we were, as a little software company active in the field of industrial automation, confronted with the job of evaluating a NC-programming system for a close machine construction enterprise. The following requirements were essential to this system:

- Simple user interface quickly learned by a mechanic
- Software costs of maximum CHF 20'000.--
- It should run on a PC
- Large saving of time during the programming

During evaluation it turned out that no system available on the market satisfied our requirements. For that reason we decided to develop a system according to the requirements enumerated above by ourselves.
Thereby we profited from various marginal conditions. Because we are ourselves NC-machine-users, we are familiar with the practical aspects. Besides we could profit from our former extensive experiences in software development for related projects.

Structure and use of this manual

This paragraph should briefly show the structure and give hints for optimal use of this manual.

1. The **system overview** gives a short technical description of PrimCAM and should show the possibilities and capabilities of the system. It is very useful especially during the evaluation phase or simply helps you to get a rough survey about the features.

2. The chapter **installation** describes the exact procedure for starting with PrimCAM on your computer. If your distributor made the installation, you can skip this paragraph.

3. The **introduction** explains you briefly the most important basics of PrimCAM. The helpsystem, the coordinates input, the mouse control, and the selection of the elements are mentioned here.

4. It is proved that you learn essentially faster by examples than by working through exhausting theory. The new knowledge can also be transferred quicker to practice. In the **tutorial** you'll get to know the extensive possibilities of the software step by step. In this chapter we set a high value on simple legibility, many examples, drawings, and pictures. We recommend you to work through this chapter one unit after another. You can also look up certain topics you are interested in in the index (chapter 8) and then work through the corresponding unit.

5. **PrimCAM intern** is meant for PrimCAM-professionals. The chapter explains internal system settings, used formulas and tables for feed rate calculations and so forth. The details supported by import and export formats are also specified.

6. The **appendix** contains faxforms for suggestions in any regard, like altering, correction or extending and so forth.

7. The **glossary (dictionary)** explains words of the computer language present in this user's guide but probably not familiar to everybody. Whenever you don't understand a word in the manual, consult the glossary.

8. Because - from previous experience - the most accesses to a manual are made via **index**, we set a high value on its completeness.

PrimCAM's further development continues very quickly. Therefore it's impossible to always have present the most actual description of all the functions in the user's guide (we would have to print a new manual every two weeks). Because of that the **reference** to the invidual functions of PrimCAM has been moved to the helpsystem, which you can always pop up by pressing F1. So you have access to hundreds of pages of context sensitive help text.

These symbols in the manual point to functions that are only available in certain versions of PrimCAM, such as M1, M2 or M3.

This symbol marks the end of a paragraph that is only valid for a certain version of PrimCAM.

Extending suggestions / errors

Because the development of PrimCAM goes on permanently, we need ideas and extending suggestions from our customers. Depending on the possibility and the demand the extentions will be realized in the next update. All the large software packages are not totally free of errors. PrimCAM will be no exception. Should you notice a malfunction of the software, we are thankful to your exact description and probably a testfile on disk. For ideas and correction suggestions please contact your distributor or our head office. The appendix contains faxforms for error information and extending suggestions.

1. System review PrimCAM

The system review will provide a brief overview of the possibilities and capabilities of PrimCAM. This has been written especially for people who already have some experience with other CAD/CAM-systems and would like to obtain a quick overview of the features that PrimCAM has to offer, for instance during the evaluation phase for a product like this.

> Features marked with a star (*) are either not yet completely implemented or are currently being tested.

General aspects

The primary objective during the development of PrimCAM was to provide a tool for the mechanic that allows him to create NC-programs in significantly less time then possible manually. This objective should be attained with justifiable expenses, especially for medium sized and small scale companies, meaning cost effective hardware (PC) and software. These requirements had a large effect on the development of PrimCAM summarized as follows:

- Simple and intuitive operation, which is specially important for a non-computer scientist.
- Quickly understood.
- Large capabilities are hierarchically organized and therefore clearly arranged
- Cost effective hard- and software.
- Short training time: 1-2 days.

PrimCAM is an extensive software package consisting of different modules. Some of these modules will be described briefly in the following sections.

User interface

During the development of the user interface we have always considered how the mechanic sets about his task, how can we minimize his effort to attain his goals. A comprehensive user-friendly interface resulted, which guarantees quick access to all functions without working through a lot of pull-down menus. A simple, intuitive interface is possible by using the mouse driven user interface with useful and meaningful icons. We also paid attention that the drawing area stays as free as possible, so that a good overview is always guaranteed. This may be summarized as followed:

- Hierarchical icon menu bar for quick access to the extensive functions
- Icons for selecting the functions (A picture says more than thousand words)
- Clear menus with dialogue boxes, filled with default values or recent inputs. You have to change only what is necessary.

- A context sensitive help system with cross references and a built in search utility offers extensive help for each function.
- Availability of different languages for the user interface.

CAD part for creation of the drawing

The CAD part of PrimCAM has got all the functionality necessary for the professional drawing. So there is no need for an additional CAD-program. Should it however be necessary to import drawings from other CAD-systems (for example customer drawings), PrimCAM features the usual graphics exchange formats (for example DXF or HPGL).

- Functions for drawing lines, polylines, rectangles, circles, parallels, arcs, curves, text and so forth
- Layer technique: different components of a drawing can be set on different layers, for example construction, dimension, text, drawing and median layers ...
- Usual zoom functions to inspect details of the drawing
- Lines for orientation (horizontal, vertical, angular, ...) and construction circles on a separate construction layer. They can be hidden or deleted after use
- Editing functions like filleting, chamfering, trimming, expanding, dragging, moving, copying, rotating, scaling, mirroring, breaking, deleting ...
- Special functions like "circle tangent to 2 other circles", "circle with radius tangent to line and circle,, etc.
- Cartesian and polar, absolute and relative coordinate input
- Multiple undo which undoes the last functions step by step
- Input of numbers on the keyboard with the possibility to include arithmetical calculations (functions: +, -, *, /, ( ), cos, sin, tan, acos, asin, atan, exp, ln, sqrt, **(power))
- Symmetric and asymmetric chamfers with selectable lengths
- Positioning of objects in a matrix, along a circle or along other objects
- Various snap functions, for example snap endpoint, intersection point, counter point (line, circle), snap tangential, normal, quadrant point, reference point etc.
- "Intelligent" snap function (selects the point, which is clicked with the mouse, and automatically installs the snap mode the user most probably needs)
- Support of various fonts, which can also be imported from other CAD systems (for example AutoCAD SHP/SHX, TrueType and so forth)
- Possibilities to automatically correct non-CAM-like drawings (overlapping lines or little gaps between the lines makes it difficult to mill a contour)
- Possibility to check or to change the attributes of objects numerically (for example the angles of an arc)
- Dimensioning possibilities for drawings
- Advanced options to change characteristics like color, layer, text heigt for several objects together (e.g. change diameter of all circles to 10mm)

CAM part for programming manufacturing

The CAM part features efficient functions which are necessary for the creation of 2.5D NC-programs. The purpose of PrimCAM is to generate a complete NC-program automatically, so that there's no need for the user to make additional modifications. The option to make modifications is, however, left open. For example, the approach and departure paths for milling contours will be created automatically, but can be altered by the user manually. One of the most significant features, resulting from the object oriented design of PrimCAM, is the fact that geometry and manufacturing are connected to each other. This means you are able to rotate, move, mirror, or copy an already manufactured part without reprogramming the manufacturing again afterwards. The tool paths are automatically changed during modification of the drawing. Parts can also be inserted together with the manufacturing into a library and can be recalled again in later projects, increasing efficiency. The following CAM features are included:

- Contour and path milling.
- Pocket milling with islands.
- Different approach and depart methods (tangential, rectangular, along the contour).
- Different plunging methods (straight down, angular, helical).
- Rough cutting and end milling offsets for contour milling, path milling and pocket milling.
- Efficient algorithms for automatic calculation of tool paths for pockets with islands, paths, contours and so forth.
- *Pockets with optional contour parallel or axis parallel
- Automatical or manual creation of the approach and depart paths for milling
- Modifications in geometry automatically change tool paths. Objects already programmed can be changed in any way without loosing their manufacturing process.
- Various technologies, so for instance spot-drilling, drilling, tapping, reaming, fine drilling cycle, engraving, and digitizing
- Manufacturing processes can be stored in the library
- Printing of tool database, job-schedule, ...
- NC-Programs can be generated for different controls at any time just by selecting other post-processors
- Possibility to program rough cutting with offset and end milling without offset with the same tool in a single job
- The tool offsets for calculation of the tool paths come automatically from the tool library
- Simulation of tool paths and 3D-views of the completed piece
- Customizable libraries for tools, materials, cutting tables
- Automatic selection of the tool and calculation of optimal speed and feed rate relying on material, tool libraries, and cutting feed rate tables.
- Empirical database where the user can store optimal values for spindle speed / feed. These optimal values can be used under similar conditions (material, cutting depth ...)
- *Automatic optimization possibility for NC-programs (path optimization, minimal number of tool changes, ...)
- Use of the capabilities of the machine control by supporting machine specific commands (in the control defined cycles, for example different drill-cycles like high speed peck drilling)

Part libraries with geometry and manufacturing

With PrimCAM you can create part libraries which not only contain the geometry, like common CAD-systems, but also the whole manufacturing for a part. An example: You need repetitively sparings for 9, 15 and 25 Pol Dsub connectors. With PrimCAM you can insert sparings for these connector types in the part library together with the manufacturing, that means with the program for drilling, milling and so forth. If you need these sparrings, they are simply loaded and placed (rotated and scaled appropriately) from the library.

- Parts can be equipped and saved with an icon. Then they can also be reloaded by clicking the same icon.
- An icon editor permits the creation of icons for library parts
- Library parts can be "photographed" and saved as an icon

Technology database

PrimCAM integrates various databases for the management of tools, materials, cutting feed rate tables, machine controls and so forth. These permit the automatic calculation of speed and feed rates from the material and tools chosen.

- Tool library with saved values for tool material; diameter and length correction for automatic calculation of the compensated tool paths
- Material library with user definable cutting feed and speed rates for the different material classes
- Cutting feed rate tables for milling, drilling and reaming. These indicate for each diameter and each material class the feed rate per round or per tooth. Those tables are delivered with PrimCAM, already filled in. But they can easily be changed by the user at any time.
- Feed and speed rate tables with empiric values gained by everyday use of the machine. These values can be used instead of the mathematically calculated values when the conditions (cutting depth, material, diameter...) are similar to a value stored in the table.
- Machine library with machine specific data like postprocessor, communication interface parameters, maximal feed and speed rates, tool change point and so forth.

Data transfer

You are able to communicate with CNC-machines through serial interfaces from inside PrimCAM.

- Transmits NC-programs from the PC to the control and vice versa
- Data transfer through RS232 interface COM 1-4
- user defined transmission parameters: word length, stop bits, parity, baudrate,...

* DNC-run for long programs (blockwise transmission)

NC Editor

PrimCAM contains an editor using window technique, with which NC-programs or post-processors can be changed or created.

- ASCII editor for NC-programs and post processors
- Selection of commands through icons

* Syntax check for post processors

- Renummeration of NC-programs
- Delete, insert, move, block and search functions

Simulation

PrimCAM contains a graphic simulator, where the created NC-programs can be simulated. Errors in programs will be discovered and eliminated at an early state. The calculation of the manufacturing time for a part permits a disposition of the present CNC-machines.

- Two views of the part
- Current indication of the coordinates, time, feed rate, speed, tool and so forth
- Estimation of the total machining time
- Speed for the simulator can be adjusted

Text engraving / fonts / font editor

It is often necessary to provide an article number or serial number on a work piece. This can be managed with the engraving features of PrimCAM. PrimCAM itself is already delivered with a number of different fonts, which you can engrave in any size and direction. Because of the vectorial structure of these fonts, their quality is guaranteed during scaling.

- Font editor for creating or modifying fonts yourself
- Font editor can make use of the functions of the CAD part (construction lines, curves, arcs, ...)
- Automatic minimization of the manufacturing time in a font
- Possibility to import fonts from external formats and save them in the PrimCAM format (for example AutoCAD SHP/SHX or *Windows TrueType)

Import /export functions

The import and export functions of PrimCAM enable data transfer to and from various other systems. For instance, construction drawings, fonts and so forth can be imported from other CAD systems. PrimCAM also supports various export formats to communicate with other systems and printers. The following import possibilities are available:

- DXF AutoCAD drawing interchange file format
- HPGL Hewlett Packard Graphics Language

*DIN 66025/ISO
*Gerber Gerber Plot Data Format
*Sieb+Meier

- Excellon Format of the company Excellon for their manufacturing machines
- NC-Drill Drilling positions from PCB programs

Also various export formats are supported

- DXF AutoCAD drawing interchange file format
- HPGL Hewlett Packard Graphics Language
- BMP Windows Bitmap

Postprocessors

PrimCAM can create NC code for different machines. Defining the manufacturing steps is independent from the machine, meaning that you can choose the particular machine for the manufacturing process after the definition of the manufacturing steps. For that purpose you simply select the postprocessor for the appropriate control and recreate the machine specific NC code automatically within seconds.

- The postprocessors are open and can be created or changed by the customer himself
- Post processors for the usual control types are already existing, for example GE Fanuc, Heidenhain (DIN/ISO and Dialog), Deckel, Maho, Excellon, Datron, Isel, etc. The list of disposable post processors is extended currently.
- When possible, higher machine commands are supported (for instance drilling cycles, ...)
- Possibility of automatically documenting the machine code (for example with the names for the manufacturing steps as "DRILLING WITH D10")

Hardware requirements

- IBM compatible PC with processor 486 or higher
- 8 MB RAM
- 10 MB harddisk space
- Operating system Windows 95 / 98 / NT or 2000
- free RS232 port (only necessary for serial transmission of NC programs to a machine)
- Graphics card with a resolution of 1024x768 points, 256 colors (at least 1MB memory)

2. Installation

2.1 Hard- and software requirements

To run PrimCAM on your computer, the following hard- and software requirements have to be fulfilled:

- IBM-compatible PC with processor 486 or higher
- 16 MB RAM memory
- 20 MB free harddisk space
- Operating system Windows 95 / 98 / NT / 2000 / XP
- free RS232-interface (only for the serial transmission of programs to machine)
- Graphics card with resolution 1024x768 pixels in 256 colors (at least 1MB display memory)

2.2 Hardware key (hardlock, dongle)

The black hardlock can be connected to each parallel interface of the computer. It is totally transparent. This transparency allows you to line up several different modules (for example to run programs of different manufacturers). Pay attention that the arrow on the key directs to the computer. The other end of the key can simply be connected to the printer cable.
The green-gray hardlock can be ordered optionally and can be used both on the parallel as well as on the serial port of the computer. The ends of the dongle are marked as PARALLEL or SERIAL. Simply take out the screws and put them inside on the other side.
The hardlock is found automatically on each parallel port of the computer. If you want to use the serial port or change the search order for the hardlock, look at chapter 5.3.

2.3 Software installation

If you use an installation CD, the installation starts automatically. You can also manually call the file setup.exe.
If you download the program from Internet, execute the file install.exe. There are files setup.exe, setup.w02, … extracted. Then you can start setup.exe.
After selecting the language for the installation you are prompted for the destination drive.
If your operating system is Windows **NT** and you choose the installation directory for PrimCAM, don't put any spaces into the installation path (like c:\program files\primcam). This will cause an error message „Hardlock not found" because the installation program cannot install the hardlock driver into a path containing spaces. Also make sure that you are logged in as a system administrator when installing PrimCAM, because that's the only way the hardlock driver can be installed.
If PrimCAM has already been installed on the indicated directory, you are asked whether you would like to make a new installation or an update.

- **Update**: The new version of PrimCAM will be installed, but all your old configuration files, tables, tool-, material- and part libraries will be kept.
- **New installation**: PrimCAM will be installed completely new. Thereby old tables, tool- and material libraries and so forth are overwritten with the factory settings.

Now PrimCAM is copied to the computer and decompressed. At the end of the process, the file README.TXT is presented, which contains the most actual information.

2.4 Software configuration

PrimCAM is installed on your computer now. If you made a new installation and not an update, you have to set the necessary parameters for machine control, postprocessors and so forth inside PrimCAM.

Start PrimCAM from the windows start menu or from the program group it is installed in.

Change to the **menu settings**, which serves to configure the system. Put the mousecursor on the icon for settings, which is at the top of the screen's right border, and click the left mousebutton.

Choose the function for setting the **system parameters** by putting the mousecursor on the icon PAR and click the left mousebutton.

Set the **initial drawing area** and **prototype**. The initial drawing area sets the size of the drawing when you start a new drawing. The optional prototype is always copied into the new drawing and could consist of a company header for example. You can also set the size of the construction lines here.

Set time for **automatic saving** allows you to set the timegaps in which the system automatically saves your work. Timegaps from 1 to 60 minutes are available. The red point on the field next to a number stands for the selected timegap. The automatical saving is activated by clicking the field behind `active:` which gets a green hook thereupon. Automatic saving doesn't save the file under the given name, but under the filename *AUTO1.CAM*, *AUTO2.CAM* and *AUTO3.CAM*. *AUTO1.CAM* is at any time the most actual version, *AUTO3.CAM* the oldest. In case of a power failure for instance, you can load *AUTO1.CAM* , the most actual available version of the manufactured drawing.

Set the parameters for the **center line**. You can choose the prolongation in mm for lines and the extension factor for circles.

Here you make the settings for ToolTips, small information flyers that appear when you rest with the mouse on an icon that contains information. You can switch them on and off and set the delay for their appearance.

For certain operations like slanting a text, PrimCAM has to approximate arcs and circles by lines. Here you can set the **resolution** for this approximation. Set the maximum bulge error in drawing units (mm).

If your PrimCAM version supports several **languages**, you can switch here the language for the user guidance and the help system. A number of flags are presented, which should

represent the various languages.

Finally you can make settings for the **Windows printer**.

Save the selected **parameters** to the harddisk now. PrimCAM also saves these parameters when it is left, but not when you just turn off the computer without leaving PrimCAM.

If you got PrimCAM-M3 (with open postprocessor system), you have to install the postprocessors for your machine controls. This happens in the **utilities** in the

machine library. All your machines are managed here, together with the postprocessor belonging to it's control. You have to create a new machine entry for every machine which needs a different NC code. This function is only available for PrimCAM-M3.

Insert machine creates a new machine entry in the machine library. Use the helpfunction by pressing F1 to get to know what you have to enter in the individual fields.

Use **select machine** to set the actual machine from the machine library. The actual machine is always represented in red. If you create a NC-program, PrimCAM automatically generates the correct code for the actually chosen machine.

Quit the machine library by clicking the quit icon. Confirm the saving of the machine library by clicking YES.

The software installation for your PrimCAM is finished now, and you can continue with the tutorial or quit the system, changing to the **menu file functions** and

clicking **quit** to exit PrimCAM.

3. Introduction

This introduction explains you briefly the most important basics of the user interface of PrimCAM. The helpsystem, the coordinates input, the mouse control and the selection of the elements are mentioned here.

3.1 Help system

PrimCAM has got a very sophisticated helpsystem consisting of hundreds of pages helptext and saves you in most cases the time to look up subjects in the manual. The use is really simple. You just press F1 on the keyboard, wherever you need help. PrimCAM realizes on which symbol the mouse is placed, and shows a helptext according to the theme.

3.2 Coordinate input

PrimCAM uses a two-dimensional, cartesian coordinatesystem. For the representation of the **origin** the following sign is used:

 sign for the origin

Coordinates can be entered as X- and Y-values or polar. Coordinates can also be entered relatively by entering @ (ampersand) before the coordinates. The number indication will then be relative to the last entered point. The following inputs are valid for example for the starting point of a line:

Input	**Meaning**
X10.2Y50.0	X=10.2 Y=50.0
X10Y50	X=10.0 Y=50.0
10.0,50	X=10.0 Y=50.0
50<45	Distance from the origin = 50, angle = 45 degrees
@10,20	From the last point in X+10, Y+20
@50<90	From the last point in Y+50 (angle 90°)

When drawing, the coordinates are shown at the top of the screen's border,on the right side. If exact input should be made not using the keyboard but with the mouse, it can be helpful to switch on the **grid**. The grid can be set to a any width. Thereby the cursor moves in fixed steps.

If the system prompts for some coordinates, which can be entered not only on the keyboard but also with the mouse, the specification and default type (in parenthesis) of the coordinates appear in the command line at the left bottom of the screen. The system accepts the following coordinate types:

A	1. Distance	e.g.:	A10
B	2. Distance	e.g.:	B20

D	D for diameter or distance	e.g.:	D24.95
FFactor (scaling)		e.g.:	F1.5
H	Height (text)	e.g.:	H15
L	Length of lines ...	e.g.:	L22
R	R for radius	e.g.:	R5
W	W for angle	e.g.:	W60
X	X for the X-coordinate	e.g.:	X10.5
Y	Y for the Y-coordinate	e.g.:	Y12
Z	Z for the Z-coordinate	e.g.:	Z-3.5

During the input of coordinates you can also use the built in calculator. For this purpose the following functions can be used: + - * / cos() sin() tan() acos() asin() atan() exp() ln() sqrt() ** (for power of). The following inputs are possible for example:

Input	**Meaning**	
X12Y33/2	X=12	Y=16.5
X(3*4+16)Y0	X=28	Y=0
X(10**2)Y0	X=100	Y=0
XSQRT(64)Y20	X=8	Y=20
X15*sin(30)Y40	X=15*0.5	Y=40

3.3 Using the Mouse

When using the mouse the following principle is always applied: The left mousebutton is equal to the ENTER key on the keyboard, therefore it's used to confirm a function, to select and so forth.
The right mousebutton is equal to the ESC key on the keyboard and is used to abort, finish selection, quit function and so forth.

3.4 Select / Deselect

A function often requires the selection of various objects for the manufacturing. This happens either by clicking each individual object with the left mousebutton or by setting a window over a group of objects. If each object is selected individually, you have to put the mousecursor on the object's periphery and click the left mousebutton.
If there's no object in the mousecursor rectangle when clicking, PrimCAM opens a window, which you pull over the group of objects, and which you can finish again with the left mousebutton. If the window is pulled from the left side to the right side, all the objects located completely inside the window are selected.
If the window is set on the contrary from the right side to the left side, PrimCAM also selects objects which are touched by the border of the window. If the required objects are selected, you can finish the selection with the right mousebutton.

If you have selected too many or the wrong objects, you can switch form select to deselect, by pressing the icon

 to choose the **select mode**. Now you can deselect already selected objects again by clicking the left mousebutton on the selected objects. Another clicking of the function switches back to selecting.

4. Tutorial

This chapter consists of exercises, where you can get to know the functions of PrimCAM step by step. We tried to take practical examples, in which you draw pieces, work on them, and at the end create an NC program. At the beginning the exercises are rather simple and become more and more complex. Because of that it is important that you work through them in the given order. This way you get to know the extensive possibilities of PrimCAM. If you later wish information concerning a specific topic, look for the unit which deals with this specific topic in the index. If you don't understand something, you can get help by pointing to the according icon with the mouse cursor and pressing F1 on the keyboard. This is the way to enter the comprehensive helpsystem of PrimCAM.

Unit 1: The first drawing

Contents:

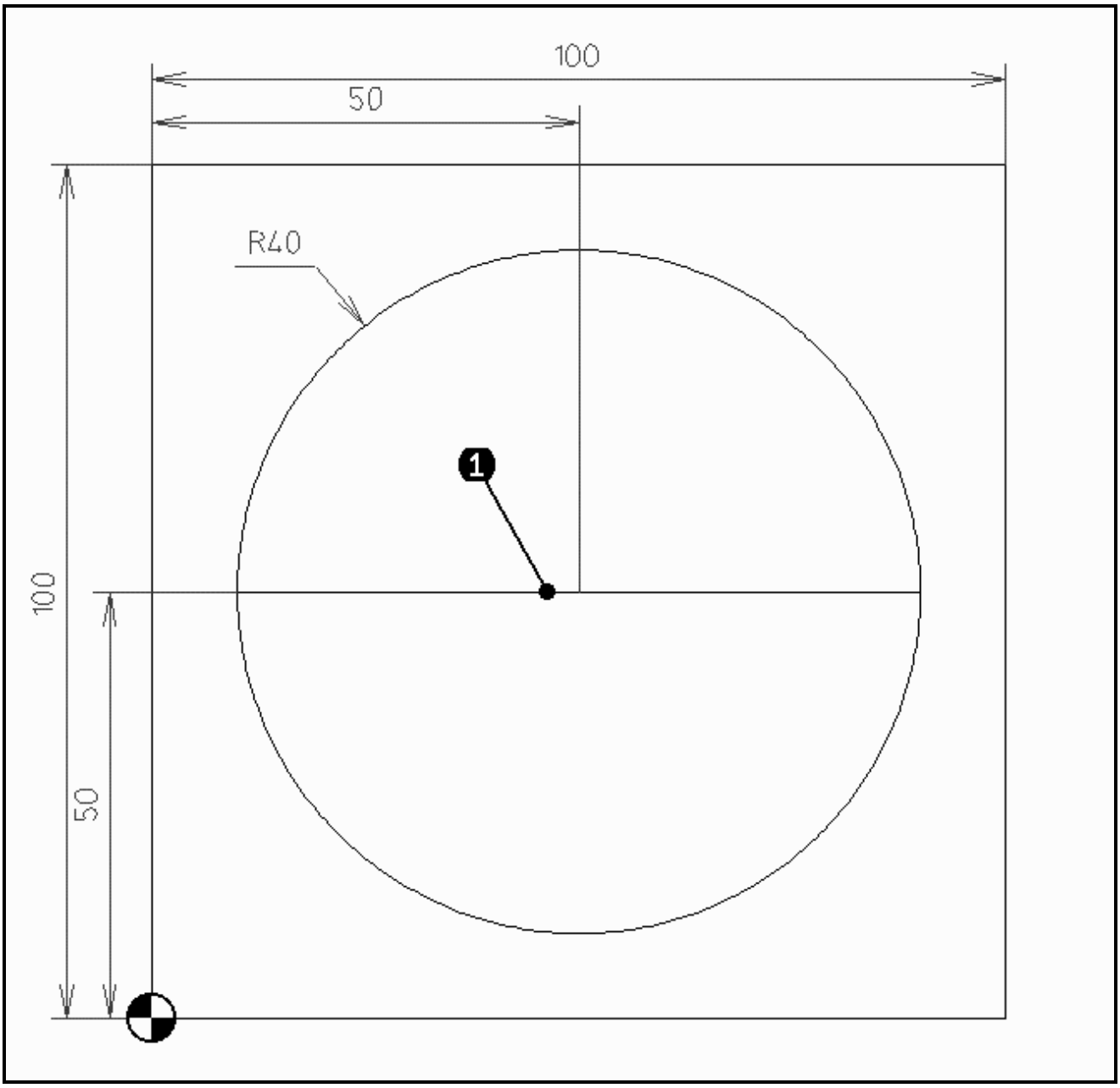
- To get to know the user interface of PrimCAM
- To create your first drawing and save it
- To work with mouse and keyboard
- Use of the helpsystem

Task:

The following drawing consists of

- A square with side length 100 mm, drawn from the origin
- A circle with centerpoint (X=50mm,Y=50mm) and radius 40 mm
- A line from point(X=10mm,Y=50mm) to point(X=90mm,Y=50mm).

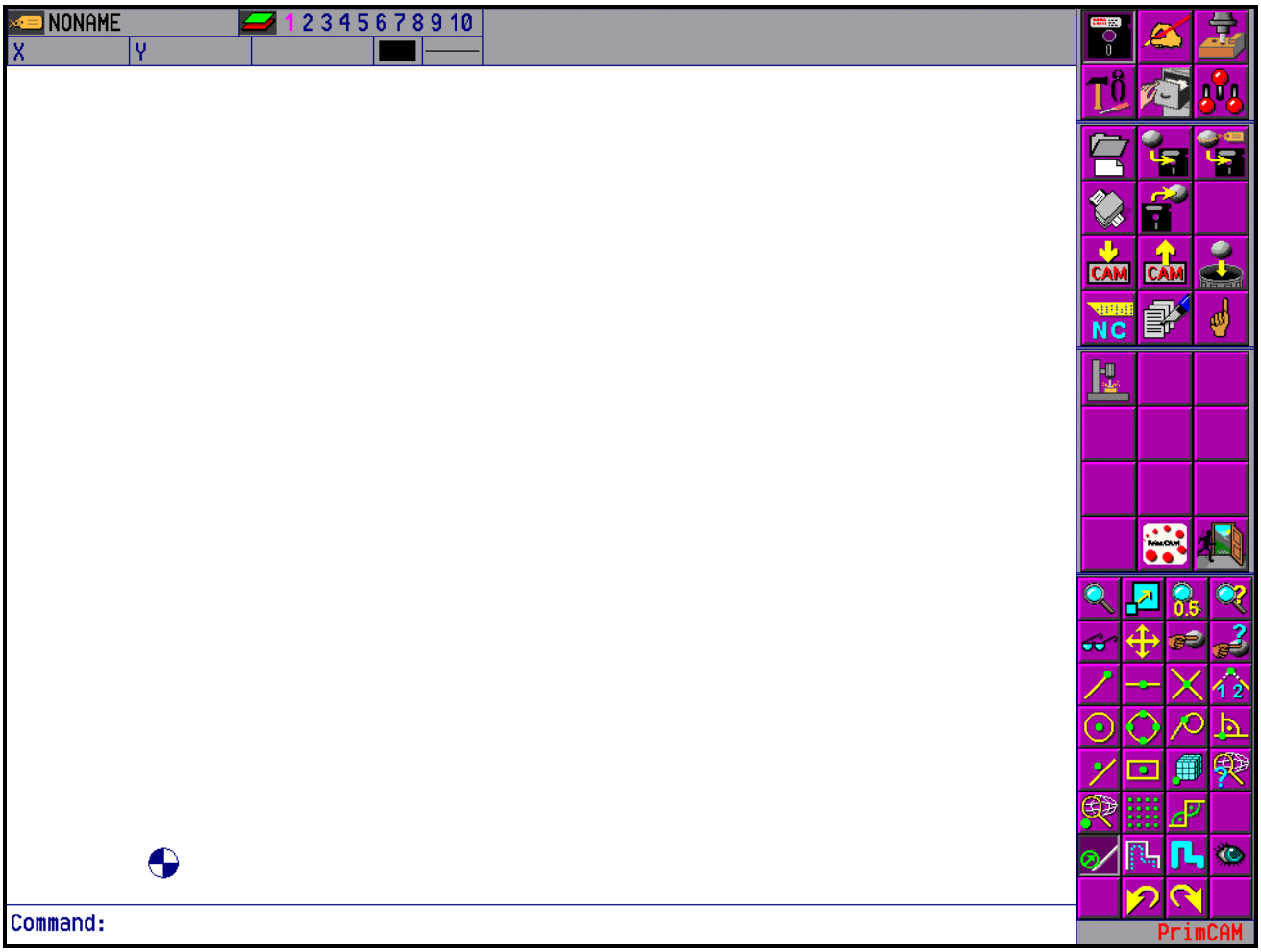
Draw these elements. Then delete the line and draw instead a vertical line from (X=50mm,Y=10mm) to (X=50mm,Y=90mm). Save the drawing to the harddisk.



Procedure:

The user interface

Start PrimCAM from the windows start menu or from the program group it is installed in. The following graphic user interface appears on the screen.



On the right border of the screen you see the **menubar** with the icons where you select the functions. Click now the 2x3 icons at the top one after another with the mouse (move the mousecursor on the icon and click once with the left mousebutton). You see how the icons (two blocks of twelve icons) below do change. The six icons at the top represent the **main menu**. With the main menu you can change to file input/output, drawing, manufacturing, tools, part libraries, and parameter settings. If you change to the drawing functions in the main menu for instance, the **submenu** changes to functions for drawing lines, circles and so forth.
The static 8x4 icons at the bottom, which have a smaller size and are presented in groups of four, symbolize functions which you will need very often. For this reason these icons are always present, even if you switch over the six areas of the main menu. These static Icons represent the **zoom-**, **snap-** and **switchfunctions**.
At the top of the screen's border you see general information. These are the **name** of the actual drawing you are working on (next to the yellow label), the selected **layer** (number in violet), the **coordinates** of the mousecursor (X, Y and angle or radius), the selected **line- or fillstyle** and the **drawing color**.

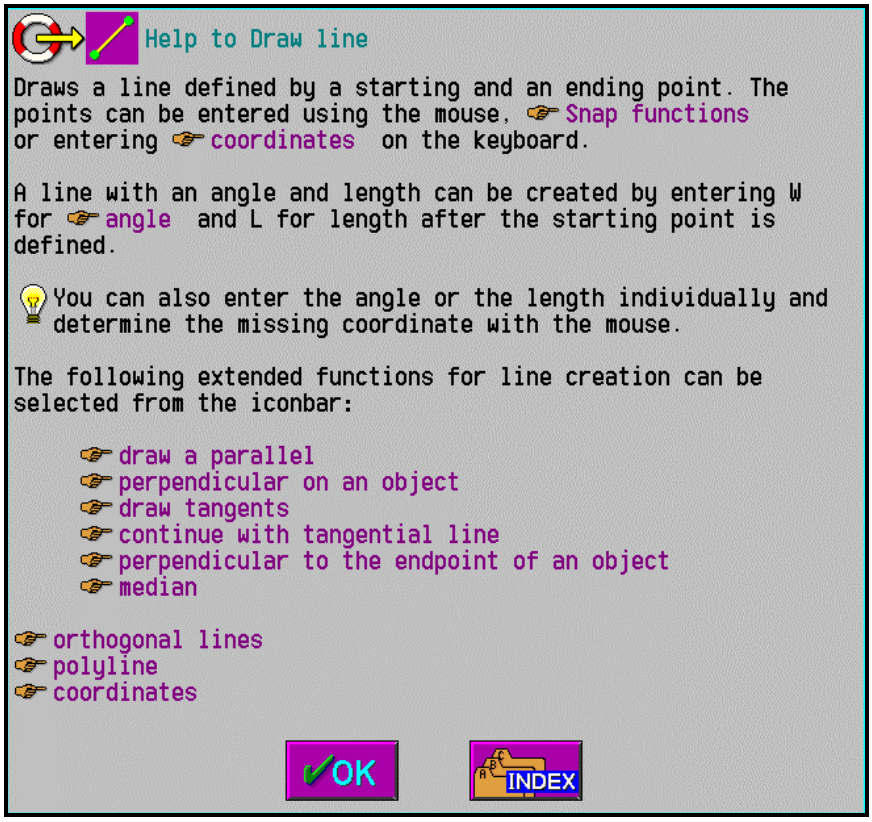
Now we would like to start with our first drawing.

Click with the left mousebutton the disk at the top of the icons (put the mousecursor on the icon and press on the left mousebutton). This switches to the **menu files**, where you can start a new drawing.

If you click the icon **new drawing,** the screen will be deleted, and PrimCAM is ready for a new drawing. At the top left border of the screen, next to the yellow label, NONAME appears as a temporary name of the new drawing, until you give it its own name.

Click the icon **menu draw** (hand with the pencil), to activate the drawing functions. You see how the two blocks of twelve icons (each 4x3 icons) placed below the main menu change to drawing function icons.

In the third line you should see now the icon for the function **draw line** (if you don't see it, look at the explanation of the icon after the next one; it tells how you make it visible). Move the mousecursor to the icon and press F1 on the keyboard. The helpwindow for drawing a line opens on your screen.



In the helpwindow you see several forefingers pointing at a word in violet. These are **references** to other helptexts. Now put the yellow mousecursor on the word `coordinates,` written in violet, and press the left mousebutton.
Immediately the helpwindow concerning coordinates opens. If needed, you also can ask here for more help, clicking the word `examples.`
You can quit the help by clicking the left mousebutton while the mousecursor is placed on the icon `OK` or pressing *ESC* on the keyboard. There is an icon `Index` that allows searching the help system for

keywords. If you select a helptext by clicking a reference in another helptext, there is also a symbol that allows returning to to previous helptext.

Test the use of the functions of the different icons, which are visible on the screen, moving the mouse to them and pressing F1.

Some drawing functions for rectangles and lines

In the third line you see the icon for the function **draw rectangle**. The two green points on the icon mean that the rectangle will be created by two points, which are placed opposite diagonally. The yellow lines are lines that will be created by this function. So the function rectangle creates four new lines by entering two points.

If the rectangle icon is not visible, then you are on the second page of the drawing functions, and you'll get back to the first page by *clicking* the icon **select page.**

Click the rectangle icon. The message `Rectangle from point` appears at the bottom of the screen, which is equal to the demand for entering the rectangle's first edge. The line at the bottom of the screen is called command line. It always indicates what you should do next during the execution of a function or makes suggestions for enterings and so forth. If you enter coordinates on the keyboard for instance, they appear in the box at the bottom of the screen's border. If you don't know what to do next in a function, take a look at the command line's text.
Now enter on the keyboard:
0,0
At the bottom of the screen the box for input will be opened behind the command text `Rectangle from point`, where you see the numbers you just entered. You can also change these numbers (with the common keys, for instance the key backspace), till you finish the input with the return key (ENTER). Pay attention to enter a comma between the two zeros and not a point. In PrimCAM a point is used as a decimal sign, a comma for the separation of two different coordinates. The 0,0 means that the X-coordinate and the Y-coordinate of the rectangle starting point both are zero.

Then the outline of a rectangle appears (coloured in violet), which starts at the **origin**, represented by the origin sign, and ends at the actual cursor point. At the bottom of the screen PrimCAM asks for the second edge of the rectangle `Rectangle to point.` By moving the mouse you can see how the size of the rectangle changes. But instead of entering the second rectangle edge with the left mousebutton, you can again enter its exact coordinates with the keyboard:
100,100<ENTER>
The rectangle changes its color to white. This means that the rectangle is now created. It starts at the absolute coordinates X=0, Y=0 and ends at point X=100, Y=100.

It's possible that only a part of the rectangle is placed inside the visible screen area. By *clicking* the icon **zoom all** (move the mousecursor over this icon and press the left mousebutton) the drawing is zoomed so that all objects of the drawing do fit into the drawing area.

Then we create the **circle** inside the rectangle by defining centerpoint and radius. Move the mousecursor to the icon for the circle and click the left mousebutton. The command line `Circle in center` asks for the centerpoint of the circle, which you enter again with the

keyboard:
50,50<ENTER>
If you move the mouse now, you can see the violet circle that can be dragged around its centerpoint. At the top of the screen's border you see the circle's radius behind the light blue R. It changes currently while you move the mouse. The command line expects the `Radius (R)`. You could now determine the radius by clicking the left mousebutton. But for reasons of precision - you need a radius of 40mm - you should enter:
40<ENTER>

Now the line which runs from X=10, Y=50 to X=90, Y=50 is still missing. Click the function for **creating a line** with a known start- and endpoint. Enter at the commandline `Line from point`
X10Y50<ENTER>.
It's not relevant whether you use upper or lower case here. A violet line that runs from the point just entered to the actual cursor position will be shown immediately. If you move the mouse, the endpoint of the line moves analogously on the screen. Pay attention again to the indication of the coordinates at the top of the screen's border during this motion. While the coordinates of the endpoint are shown in the X- and Y-box, the letter W with the actual angle appears behind it. So you are always informed about the line's angle. For the second endpoint of the line, which is requested now in the command line with `Line to point` , we enter
X90,Y50<ENTER>

As you realize, we did specify the X and the Y explicitly during the two last inputs of coordinates. If we enter only 10,20 without an indication of letters, PrimCAM knows that it should interprete the first coordinate as X, and the second as Y. Instead we could also write Y20X10, which means Y=20 and X=10, leading of course to the same result.

Deleting objects

We suppose that the horizontal line through the circle is wrong and should be vertical instead. Now we have to delete this line and draw a new one.

The icon for **delete object** is in the second of the two blocks of twelve icons. Put the mouse on this icon and inform yourself about the delete function by pressing F1.

If the icon delete is not visible, then you are on the second page of the functions for editing, and you'll get back to the first page by *clicking* the icon **select page**.

If you have activated the function delete, you are asked to `Select objects to delete.` Now you have to choose the object you would like to delete. Move the violet, rectangular mouse cursor approximately over the middle of the line you want to delete, near point **(1)**. Then press the left mousebutton. The line changes its colour to violet, as a sign for being selected.
You can confirm the question `Delete selected objects?` by pressing the left mousebutton, whereupon the line will be deleted. If you pressed instead the right mousebutton, which means abort, deletion will be stopped, and the object isn't selected anymore (white color).
You are still in the function delete and could select other objects to delete. To leave delete, press the right mousebutton.

Finally we still have to renew the **line** inside the circle. This line has to be vertical. Enter the starting and ending point of the new line in the following way:
50,10<ENTER>
50,90<ENTER>

The geometry is created now. Before we continue, we would like to save the drawing. Change to the **menu file functions**

and click the function **save as ...** . A window appears where you can determine path and file name. Enter as a name for the file *LEARN1<ENTER>* . Click the left mousebutton on the icon OK, which saves your file.

If you would like to quit PrimCAM, click the icon **exit**. Otherwise you can continue with the next unit.

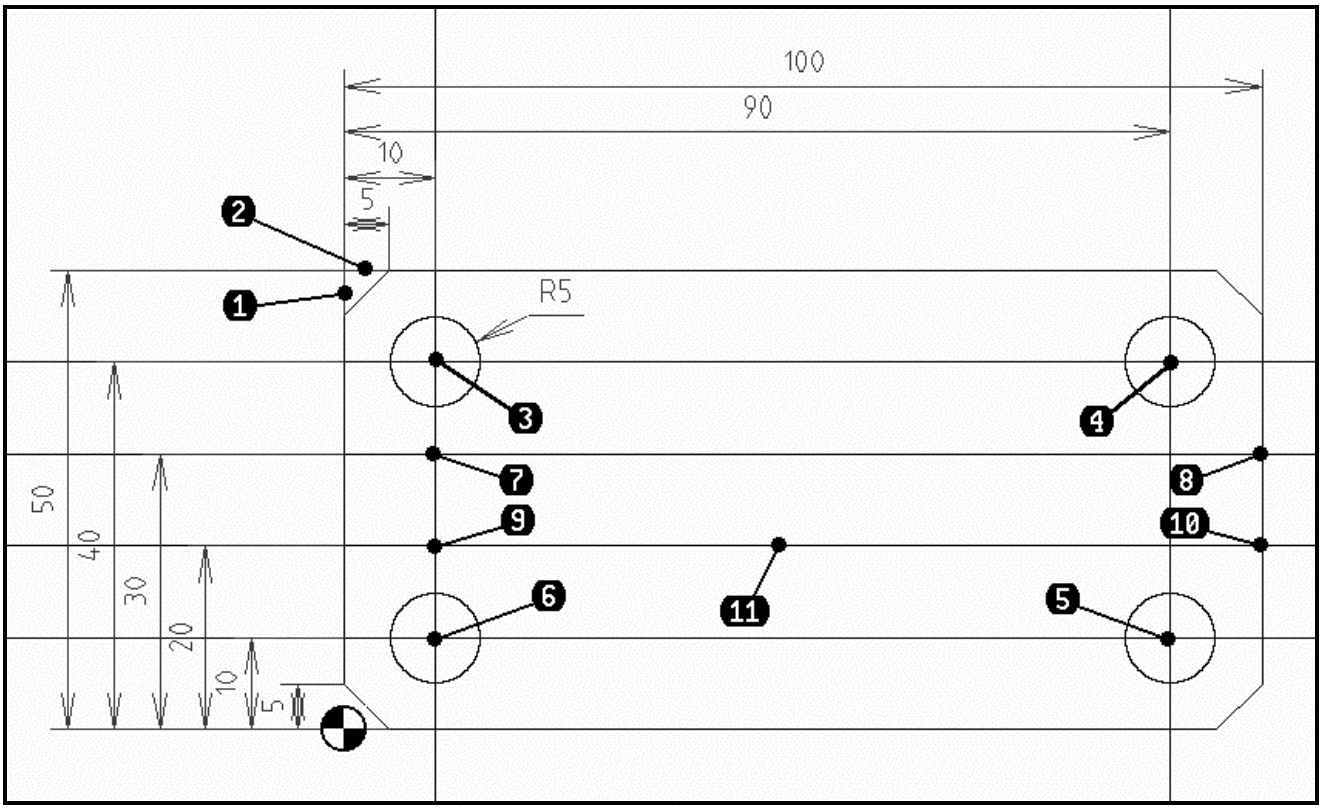
Unit 2: The first NC program

Contents:

- Create the first NC program
- Program a piece with four holes and a groove
- Get to know the user interface with drawing functions

Task:

Create a NC-program which centers the four holes with the spot-drill and drills them afterwards. The groove in the middle of the plate should also be milled with a 10mm mill.



Procedure:

Creating the geometry

When we work on a piece, our usual procedure is to create the drawing with PrimCAM first and afterwards the individual manufcturing steps like drilling, milling, engraving and so forth are applied. Another possibility would be to import an already existing drawing, which for instance comes from another CAD system, using the import facilities of PrimCAM. But we'll start with the usual case and draw the geometry in PrimCAM.

Click the left mousebutton when the cursor is placed on the disk in the line of icons at the top. This changes to the **menu file functions,** where you can begin a new drawing.

If you click the icon **new drawing**, the screen will be deleted, and PrimCAM is ready for a new drawing. On the top left border of the screen, next to the yellow label, `NONAME` appears as a temporary name for the new drawing, until you give it its own name.

Click the icon **drawing** for activating the drawing functions. You can see how the two blocks of twelve icons (each 4x3 icons), which are placed below the main menu, switch over to the drawing functions. The first of the two blocks contains mainly functions for drawing objects like line, circle, text, and so forth. In the second block there are functions to edit objects like rotating, mirroring, copying, and so forth. Use the helpsystem for examining functions you would like to get to know. Just move the mousecursor on the according icon and press F1 on the keyboard.

In the third line, among the drawing functions, you should see now the icon for the function **rectangle**.

If the icon rectangle is not visible, you are on the second page of drawing functions and you'll get back to the first page by *clicking* the icon **select page**.

Click the icon rectangle and enter at the command `Rectangle from point` the first edge of the rectangle:
0,0<ENTER>

The outline of a rectangle appears in purple. It runs from the **origin**, represented by the sign on the left, to the actual cursorpoint. At the bottom of the screen the command line asks for the second edge of the rectangle: `Rectangle to point.` Moving the mouse you can see how the size of the rectangle changes. Instead of defining the second edge of the rectangle with the mouse, you enter again the exact values on the keyboard:
100,50<ENTER>
The rectangle changes its color to white. It runs from the absolute coordinates X=0, Y=0 to the point X=100, Y=50.

Use **zoom all** if the whole rectangle isn't visible on the screen.

Now chamfers of 5mm x 45° should be placed on all four edges of the rectangle. The function for champfers is selected by choosing **fillet**. When you click with the left mouse button, a bar with special functions flips out.

There you can select the function **chamfer**. The program will ask you for the first and the second object. For the first edge you click the first line next to **(1)** using the mouse and then the second line next to **(2)**. After that the two selected lines will be connected with a chamfer. For the chamfer distance you enter
5<ENTER>
which produces a symmetric chamfer.

To place the circles for the holes and the lines for the groove we will use **construction lines**. Construction lines are lines for orientation. They are located on a separate layer and can be faded out or deleted at the end of the drawing. Choose the **construction line horizontal**. You see a horizontal, purple line that can be moved with the mouse. By entering a Y coordinate or clicking the left mousebutton you can place the construction line. As soon as it is placed, it changes it's color. Enter now

10<ENTER>
20<ENTER>
30<ENTER>
40<ENTER>

This way four construction lines parallel to the X axis at the coordinates Y=10, Y=20, Y=30 and Y=40 will be created. The function can be ended with the right mousebutton or with ESC on the keyboard.

Now you create two **vertical construction lines** at X=10 and X=90

10<ENTER>
90<ENTER>

Choose the function **circle with centerpoint and radius** to create the four holes in the angles of the rectangle. They should be placed exactly on intersections where the construction lines do cross.

For placing the centerpoint of the circle exactly on the intersections, choose the snap mode **snap intersection**, located in the below half of the screen. The mouse cursor takes the figure of a rectangle with a cross in its middle. If you click with the mouse on a intersection of two lines, PrimCAM acts as if you had hit exactly this intersection with the mousecursor. Thereby the intersection only has to be somewhere inside the small rectangle of the mousecursor. With snap modes you can exactly "catch" certain points in your drawing, as if you had calculated and entered the exact coordinates.

Now you put the cursor-rectangle to the point **(3)** where two construction lines intersect and click the left mousebutton. The centerpoint of the first circle is now determined and placed exactly on the intersection of the construction lines. For the `Radius(R)` you enter *5*, which creates a circle with 5 mm radius or 10 mm diameter.

Instead of drawing the circle for the three other angles the same way, we can multiply it with **copy**. Pay attention to put the cursor-rectangle on the periphery of the circle at `Select objects to copy,` so that a part of the periphery of the circle is inside the mouse cursor, and then click the left mousebutton. If the circle has changed its color to purple, abort the selection of the objects with the right mousebutton.

Now you have to choose the `Copy-point`. The copy-point is the reference point of an object. With its help the object can be set to an exactly determined place afterwards. We would like to place the centerpoints for the three other circles on intersections of the construction lines. For this reason choose the snap mode **snap center** and click the periphery of the circle. You see now a violet copy of the circle that you can move on the drawing area using the mouse. In the middle of this circle there is a little cross. It symbolizes the copypoint and is at the same time also the position of the mousecursor.

PrimCAM asks for the `Insert-point` now.The insertpoint is the point the copypoint will be placed to. We would like to set it on intersections of the construction lines. Now you may have noticed that the chosen snapmode was only valid for the next operation and then was left automatically. If you click the icon for **snap intersection** twice, the function gets permanent and stays active until you click the icon a third time.
If you click a snap function, it switches it's state from on(once) to on(permanent) to off.

Click now one after another the three intersections of the construction lines next to **(4)**, **(5)** and **(6)** with the left mousebutton. After the circles are placed in the four corners, abort the copy function with the right mousebutton.

Finally we have to place the two **lines** in the middle of the piece, where a groove should be milled in between them. The correct snapmode (snap intersection) is still active from the last operation. Just click the two endpoints of the lines to be created next to **(7)** and **(8)** for the first and next to **(9)** and **(10)** for the second line.

The geometry of this piece is now finished. Before we add the manufacturing, we would like to save the drawing. Change to **file functions**

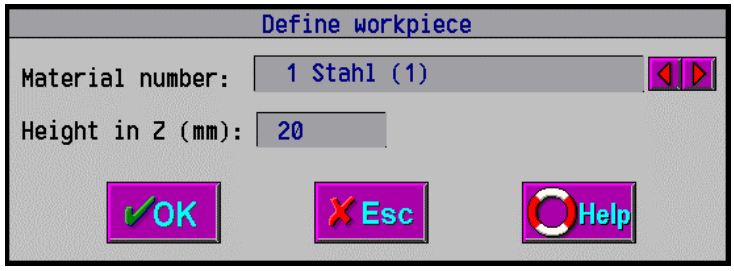
and click the function **save as ...** . Now enter the name of the file in the field `Name`, for example *LEARN2<ENTER>*.

Manufacturing the piece

Change to the menu **manufacturing** now for drilling the holes and milling the groove.

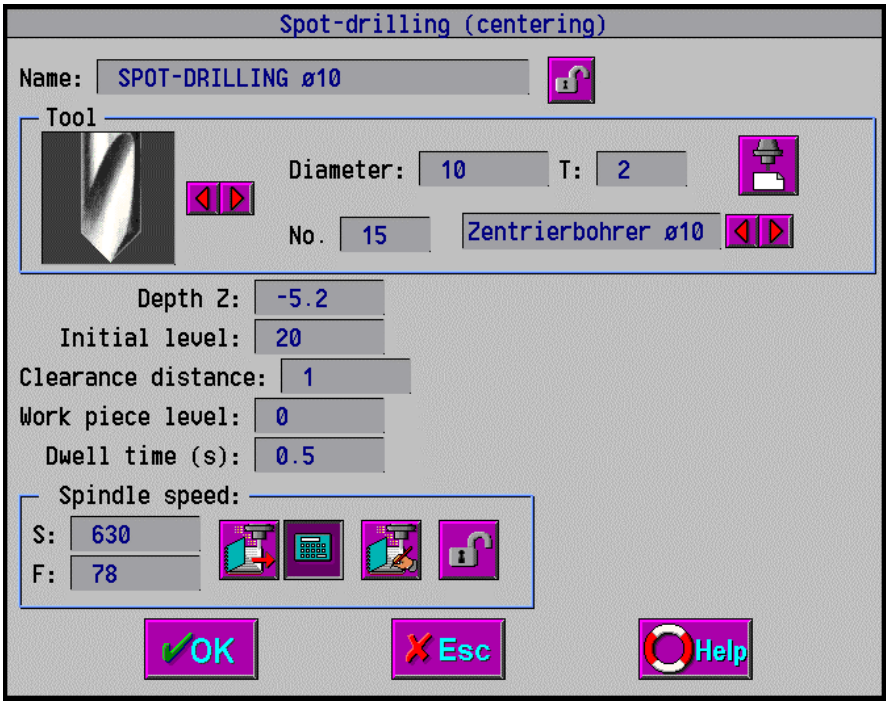
First the holes should be **spot-drilled (centered)**. At the instruction `Select objects to spot-drill` click the four circles by moving the rectangular mousecursor over the circle's periphery and pressing the left mousebutton. If all four circles are coloured purple (sign for "selected"), abort the selection of objects with the right mousebutton.

Because you haven't defined a work piece yet, the following window opens:



Determine the material of your workpiece here by scrolling back- and forward with the two arrows. All the materials defined in the material library will be enumerated. It is necessary for PrimCAM to know the specific material you are using for this piece to calculate the correct feed and speed rate automatically. Click the right arrow once, so the window will show `2 Al Mg Si0.5 (6)`for instance, which stands for aluminium.

The `Height in Z (mm)` is needed for simulation only. Our piece is 10 mm thick, so you enter 10 in this box. If the work piece is defined, the following window for spot-drilling appears:

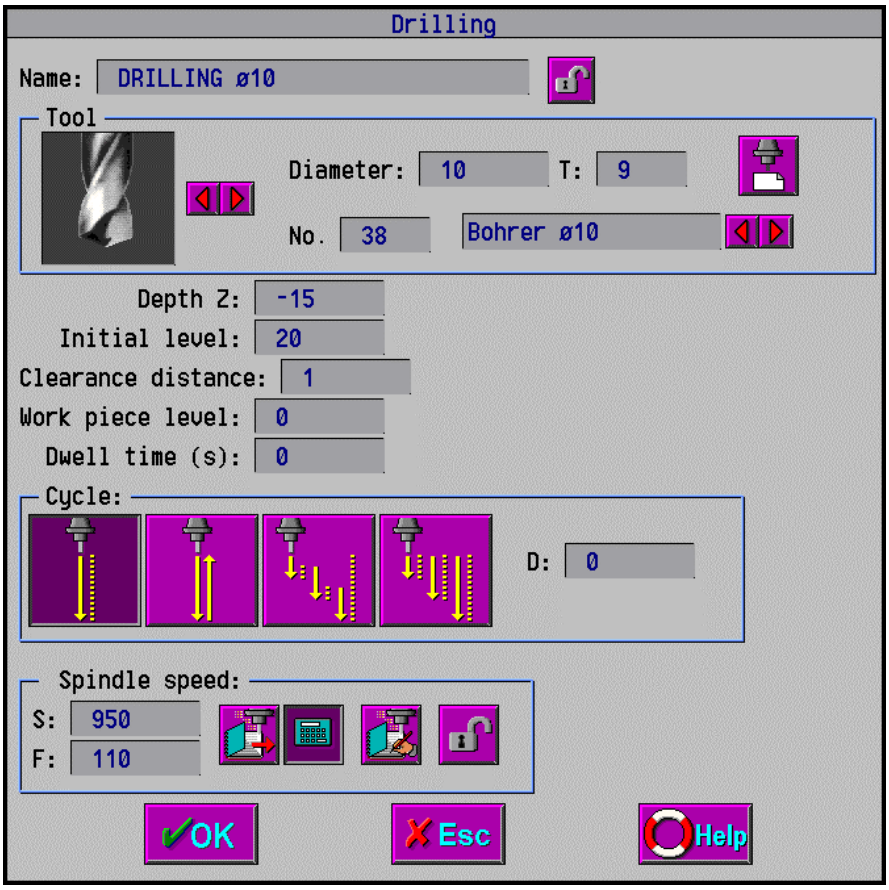


Fill in the window according to the above example. If you don't understand an entry in the window, move the mouse to it and press F1. A help window concerning the entry will be opened. Short explanations of the entries will follow:

The `Name` of the manufacturing will be given automatically by the system, but it can be changed here at will. It contains information concerning the manufacturing process and the used tool. For example NC-programs can be documentated automatically with this feature.
The `Tool` type will be set to spot drill automatically.
After having chosen the `diameter`, the corresponding spot drill will be searched in the tool library. Supposing that there is none with the required diameter, the next bigger will be taken.
Enter for `Depth Z` *-5.2 mm*. This means that, measured from the workpiece level, it will be spot drilled to 5.2 mm deep. This way, with a 90° spot-drill, there is a little chamfer cut at the border of the hole.
Above the `Initial level` ZA the tool will move in rapid traverse with all 3 axis. Below the initial level the tool moves only in Z-direction in rapid traverse. For this reason you should set the initial level higher than all existing obstacles like closers and so forth, therefore no collision can happen with an obstacle in rapid traverse.
From the initial level the tool moves down to the `Clearance distance` ZC in rapid traverse and then begins to cut with feed rate.
The `Workpiece level` ZW is the level the workpiece surface is placed. ZW usually is 0. If, for instance, a drilling should be made in a groove of the workpiece, then set ZW to the negative depth of the groove. The drilling then starts at the depth of the groove.
The `Dwell time` indicates how many seconds the drill should stay at the bottom of the hole, for example for cutting free.
`Speed` and `Feed rate` are indicated in rpm respectively mm/min and automatically calculated by PrimCAM. The values can be changed manually at any time. But you have to notice that the window

calculates the further down placed values new after any change. For this reason a change of the speed will also recalculate the feed rate.
Click the icon OK if the values are correct.

After spot-drilling the four holes will be drilled with a 10mm drill. Choose **drill**. Precede analogous to the spot-drill for the selection of the holes. After selecting the four holes the following window appears:

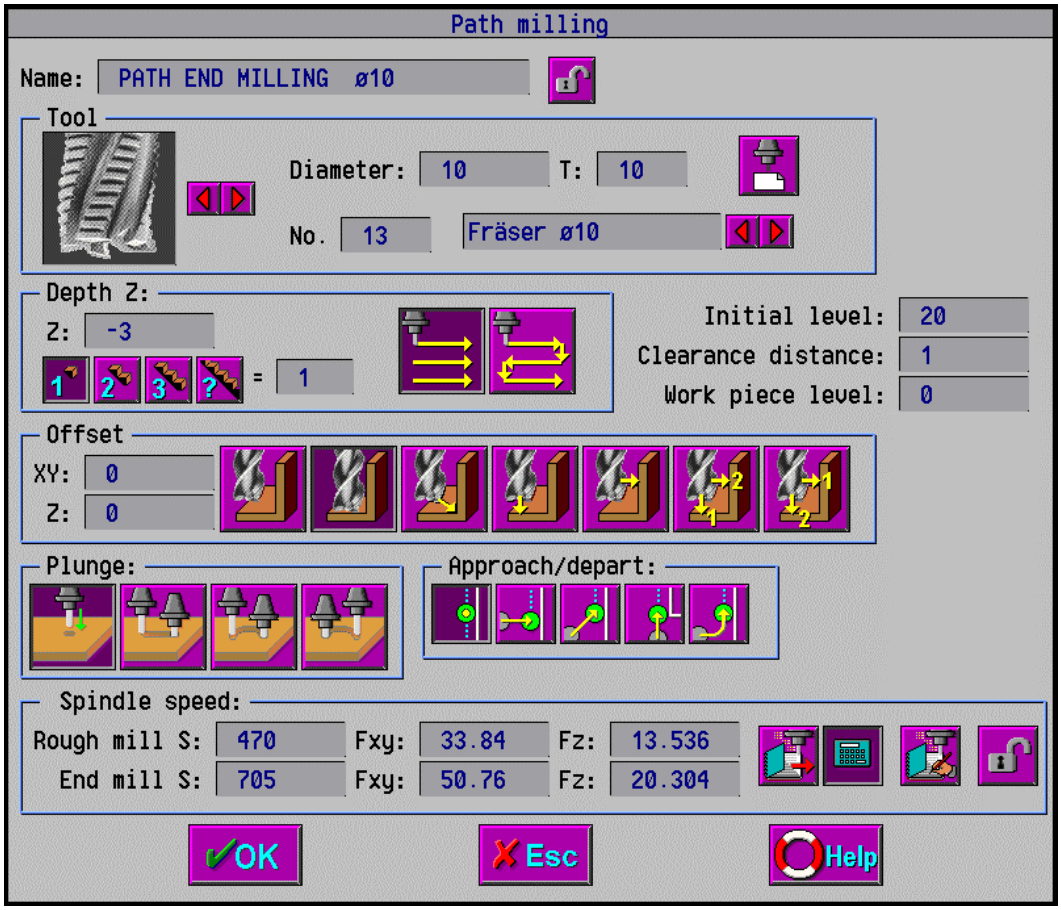


Fill in the window according to the above example. The correct drill is searched automatically according to the diameter of the circle. As Cycle you choose drill feed/rapid, which means that the tool cuts into the work piece in feed rate and then leaves in rapid traverse. Thereupon the holes are programed.

Finally the groove should be milled in the center of the work piece. This is accomplished by the function **mill path**. At the prompt Select objects to path milling you click the lower line next to **(11)** and abort the selection of objects with the right mousebutton. After having selectioned the line, you are asked to select direction of the path. If you now move the mousecursor around the start or endpoint of the chosen line, you can see that the direction pointer (little circle with enclosed arrow) jumps from one side to the other side of the line. The enclosed arrow also changes the direction by doing so.

With the **direction pointer** you determine whether the mill moves on the path itself, on the right or left side. Besides you can indicate in which direction the manufacturing should proceed. Pay attention

that the indicated direction pointer is placed above the line, and that the direction pointer points to the right side, then you confirm with the left mousebutton. The following window appears:



Fill in the window according to the above example.

Pay attention to choose the **end mill process**. With end milling an indicated offset in XY is not noticed.

The `Plunging` in Z can be straight down, because we anyway plunge outside the material.

`Approaching` should be in the direction of the path. Thereupon the tool plunges down outside the workpiece and then approaches to it in direction of the line.

The `Offset` is set to zero in XY- and in Z-direction. The XY Offset anyway has no consequences for the end mill process.

Now you should play a bit with functions that activate different viewing modes of the drawing. The views are activated in each case by clicking the corresponding icon and disactivated by clicking it again.

Click the function **direction pointer visible**. The direction pointers for the drills (circle with cross) and for the path milling (circle with arrow) are now shown on the drawing in red. If we

also had engraved text, the direction pointer for that would be drawn as a circle with an enclosed T. The direction pointers are standing for symbols of manufacturing processes.
By a second click on this icon the direction pointers are switched off again.

Switch on the **tool paths**. The tool path consists of a green line, along which the tool is led, and red circles with the diameter of the tool, indicating where a motion starts or ends. Rapid traverses are represented in dashed yellow lines. Drills are shown as green circles with the drill diameter.
By clicking this icon again you can switch on and off the indication of tool paths.

This switch function makes visible the **material removement** of the manufacturing. You see a light blue path with the thickness of the tool's diameter wherever the tool passes through. You see clearly that the mill drives down outside the plate and then approaches to the workpiece in direction of the line. The function can also be switched off by a second click.

Now the manufacturing for our piece is programmed. We have to start the postprocessor to create the NC-code for our machines. The NC-code icon is in the **menu file functions**.

With **NC-program** the postprocessor is started. It now translates the programmed manufacturing to the finished NC-code for the installed machine. Before you confirm with *OK*, you can check whether the correct machine and the correct postprocessor are selected. You are also able to change the filename of the NC-file, but you usually don't need this feature since the NC-editor expects the default name.

With the **NC-editor** you can look at the created NC-program. The NC-editor is a common ASCII-editor for editing any text files. With the cursor keys and the mouse you can move around in the NC-code.

The first NC-program, LEARN2.NC, is now created. The NC code for an DIN/ISO-compatible machine could look something like this:

```
(LEARN2)
N0010 T1
N0020 (SPOT DRILL WITH D16:)
N0030 M09
N0040 M98 P1000
N0050 T2
N0060 G54 G90 G40 G80 G00 G43 H3 X10 Y40 Z20 S750 M13
N0070 G81 G98 X10 Y40 Z-5.2 R1 F120
N0080 X90
N0090 Y10
N0100 X10
N0110 G80
N0120 (DRILL WITH D10:)
N0130 M09
N0140 M98 P1000
N0150 T3
N0160 G54 G90 G40 G80 G00 G43 H51 X10 Y40 Z20 S1200 M13
```

```
N0170 G81 G98 X10 Y40 Z-15 R1 F150
N0180 X90
N0190 X10 Y10
N0200 X90
N0210 G80
N0220 (PATH END MILLING WITH D10:)
N0230 M09
N0240 M98 P1000
N0250 T1
N0260 G54 G90 G40 G80 G00 G43 H73 X-7 Y20 Z20 S470 M13
N0270 G00 Y25
N0280 Z1
N0290 G01 Z-3 F20
N0300 G41 G01 X0 Y20 Z-3 D85 F50
N0310 X100 F50
N0320 G40 G01 X107 Y25
N0330 G54 G90 G40 G80 G00 G43 H73 X107 Y25 Z20 S470 M13
N0340 M09
N0350 G40 G80
N0360 M30
%
```

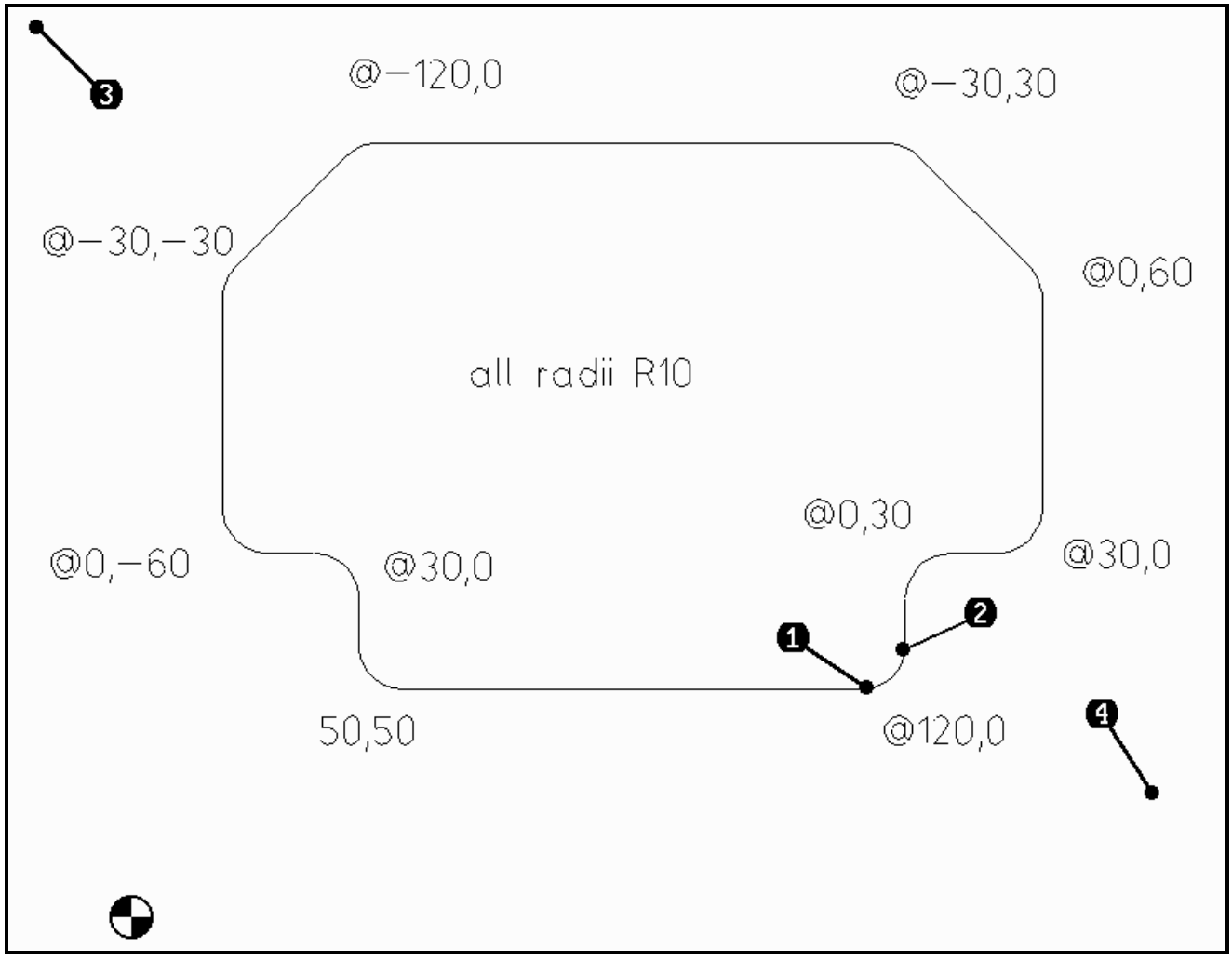
Unit 3: Coordinates, contour milling

Contents:

- Relative coordinate input
- Contour milling
- Simulation of tool paths

Task:

Draw the represented contour and create a NC-program which leads the cutter along this contour.



Procedure:

Draw contour

For creating this contour, the simplest method is to work with the function **polyline** and to fillet the contour's edges afterwards. If you click the left mouse button on the function line, the special functions for **line** appear.

Choose **polyline** here. A polyline is a sequence of lines, where the endpoint of the previous line is always the start point of the next line. This way you can create a sequence of connected lines. Enter at the prompt `polyline from point` the following coordinates:

50,50<ENTER> polyline starts at point X=50,Y=50
@120,0<ENTER> relative coordinate input, from the last point in X120, Y0

For relative coordinates the sign @ has to be entered. The numbers X=120 and Y=0 are then calculated relatively from the last entered point. So the line goes from point X=50, Y=50 to X=170, Y=50.

Click **zoom all** to place the entire drawing on the screen area. You see that the starting point for the next line is already set at the endpoint of the previous line. Continue the input with the following coordinates:

@0,30<ENTER>
@30,0<ENTER>
@0,60<ENTER>
@-30,30<ENTER>
@-120,0<ENTER>
@-30,-30<ENTER>
@0,-60<ENTER>
@30,0<ENTER>

To close the contour, you choose the snapmode **snap endpoint** and click with the rectangular cursor on the starting point of the first line. Pressing the right mousebutton finishes the function polyline.

Now all the edges of the contour should be **filleted** with a radius of 10 mm. We can select here the function for

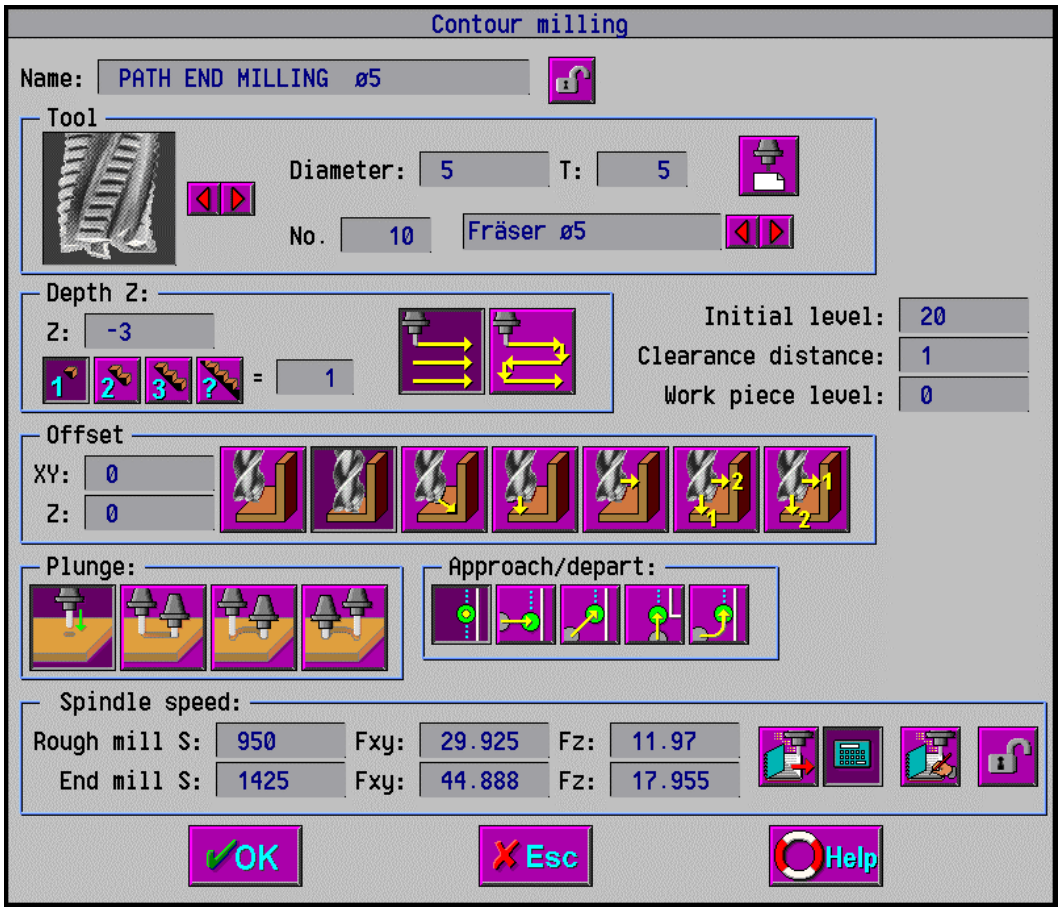
Filleting several edges together. Select the whole contour by placing a window from **(3)** to **(4)**. After entering the radius all the edges of the contour are filleted.

If you had to fillet only individual edges, we could choose the function for **filleting** an edge. At the prompt `1st object to fillet` we would click the line next to point **(1)**. For the `2nd object to fillet` you would select the line next to point **(2)**. On the keyboard you now enter the `radius` *10*. An arc is created between the two lines with a radius of 10mm, whereby the two lines are shortened analogous. If you fillet more edges, you only have to click the 2 required objects for each fillet. The radius is adopted from the first entry and remains modal until it's changed with the command R.
If all edges are filleted, you can save the drawing under the name LEARN3. For saving you first have to change to the **menu filefunctions**.

Manufacturing the contour

We switch to manufacturing functions by opening the **menu manufacturing**. This time we won't work on an individual line like in the last unit, but on a closed contour. Therefore we choose from the manufacturing menu the

Contour milling. Instead of selecting all lines and arcs individually, you set a window around the entire contour. Move the mouse to the left side at the top, next to point **(3)**, click with the left mousebutton, and then pull the window down to the right side, next to point **(4)**, so that the violet frame encloses the whole contour. Press the left mousebutton next to point **(4)**. If all lines of the contour have changed their color to violet, quit the selection clicking the right mousebutton.
Then you can place the `Startpoint` for the cutter, best above the center of the bottom line, so that you have enough space to approach and depart the contour.
Then the `direction for contour` is asked. Move the mouse around the direction pointer. You see how the direction pointer changes its direction, and the pointer jumps from inside the contour, on the contour itself or to the outside of the contour. Thereupon you can determine on which side to mill and specify up-cut or down-cut.
Set the arrow inside the contour and choose up- or down-cut. Confirm by pressing the return-key or by clicking the left mousebutton. Then the following window opens:



Notice that the `cut depth` is milled in 2 steps. This means that the mill passes through the path twice, for rough cutting all the material. During the simulation of the tool path, you will see that the mill drives more than once along the contour.

For **plunging** you choose the first icon, which means a cutting down in Z-direction.

For **approach / depart** you also chose cut down straight in Z-direction at the starting point.

After you have completed the menu analogous, you finish it with OK.

To see the **material removement**, you switch on the according icon. This function is specially designed for making visible collisions of the milling tool with any contours or whether it leaves out some material which actually should be removed.

Simulation of tool path

Finally the mill paths should be simulated for a check. This can be accomplished in the **menu utilities**.

The camera is the symbol for **simulation**. The simulation helps you check the manufacturing steps you have defined. The tool paths will be simulated in two views of the workpiece. As soon as you click the icon, the simulation window with the two views will be opened immediately and the simulation begins to calculate and to run. During the simulation the actual tool coordinates, the machining time, used tools, speed and feed rate are shown. The following functions are available in the simulation:

Reduce simulation speed decrements the time factor. With a time factor of 10 for instance, the simulation will run 10 times faster then in reality. The minimal time factor is 1.

Increase simulation speed increments the time factor for the simulation. The maximum time factor is 100.

Minimal simulation speed sets the minimal simulation speed. The time factor is set to 1, what is equal to the effective manufacturing speed.

Maximal simulation speed sets the maximum simulation speed with a time factor of 100. The simulation runs 100 times faster than the real manufacturing is going to run.

Stop simulation stops the simulation at the actual position. It can be started again with the green traffic light or with the step icon.

Start the simulation starts the simulation again after a stop.

Simulation step by step executes a step of the simulation with every click on the icon. Thus you can watch what happens step by step.

Reset simulation starts the simulation again from the beginning. You can press this icon any time, at the end of a simulation to restart it or even in the middle.

Quit simulation finishes the simulation of the tool paths.

Finally you can change to the menu with the file functions and create the **NC-program**,

which you can view with the **NC-editor**.

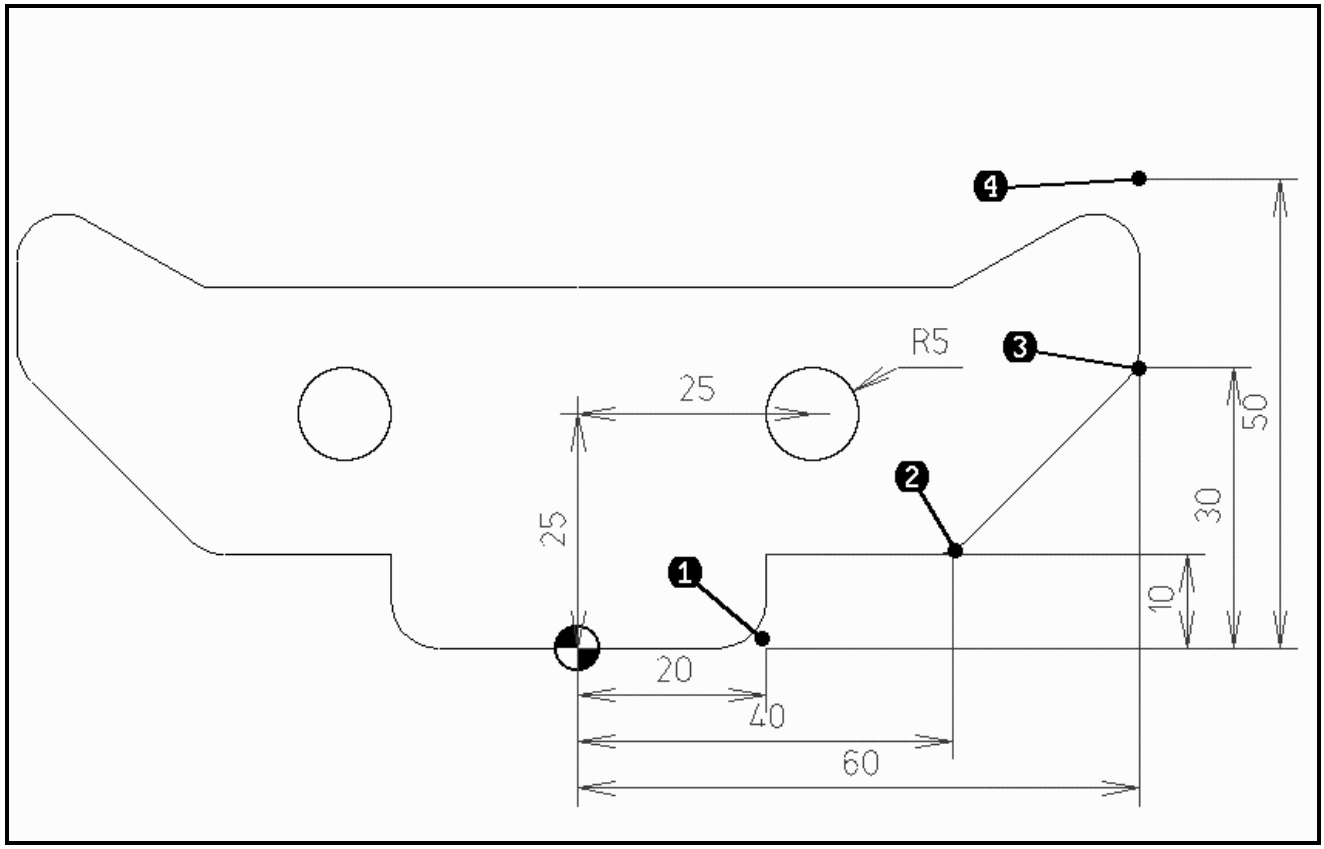
Unit 4: Relative polar coordinates, mirroring

Contents:

- Input of relative polarcoordinates
- Mirroring

Task:

Create the following drawing, using the relative polarcoordinate input. Save the drawing, because it is needed for unit 5 and 6.



Procedure:

Creating the geometry

Coordinates can not only be entered as X,Y, but also polar. Therefore the length is entered first and the direction (angle) afterwards. Examples are:

10<90	Point at X=0, Y=10 (length 10 mm, angle 90°)
W90L10	Point at X=0, Y=10 (length 10 mm, angle 90°)

Polar coordinates can also be entered relative, that means starting from the last entered point. You have to put the sign @ before the coordinates then:

@50<45	Point with distance 50 from the last point in direction 45°
@100<90	Point X=0, Y=100; distance 100 from the last point, direction 90°
@100<0	Point X=100, Y=0; distance 100 from the last point, direction 0°

The drawing can be created the easiest way by starting with a **polyline**, afterwards the necessary edges are simply filleted. To access the icon polyline, we select line and choose the according line in the special functions bar. We will use here a combination of cartesian coordinates (X,Y), relative cartesian coordinates (@X,Y), polarcoordinates (L<W) and relative polarcoordinates (@L<W). Enter the following coordinates in the given order and complete each one by pressing *<ENTER>*. Only one half of the contour is created, the other half will be generated afterwards by mirroring.

0,0	We start with point 0,0
20<0	Point X=20, Y=0; distance from origin 20, angle 0°
@10<90	From the last point distance 10 in Y-direction (90°)
@X20Y0	From the last point 20 in X, 0 in Y
@28.284<45	From the last point 28.284 in direction 45°
W90	Continue in the 90° angle
L20	Line of the length 20 (in the previously determined angle of 90°)
@23.094<210	From the last point in direction 210°, length 23.094
W0	Angle 0° (horizontal straight line)
X0	Pull to the point X=0

As you see, we don't need to enter both coordinates at the same time, but we can also determine first the angle (W90) and afterwards the length (L20) or first only the Y-coordinate and afterwards the X-coordinate. Abort the polyline by pressing the right mousebutton.

Fillet the four edges next to **(1)**, **(2)**, **(3)** and **(4)** according to the above drawing with radius 5.

For the drilling you create a **circle** at the position X=25, Y=25 with radius R=5. You can also just enter the radius at the question `circle in centerpoint`, as represented in the following:

X25Y25R5<ENTER>

One half of the drawing is created now. To get the second half, you **mirror** the object at the Y-axis.

Choose the mode **horizontal with copy** (second icon form the left side). This means that a copy of your piece is created, which is mirrored at the Y-axis. You could also delete the original half by choosing another option. You `select objects to mirror` by setting a window over the entire contour. Abort the selection of the objects with the left mousebutton. For the `axis of symmetry` you enter *0* or *X0*.

With **zoom all** you can get the whole drawing into the visible area.

Change to the **file functions**

and **save** the drawing under the name *LEARN4*.

With **load file** you can get again the saved drawing *LEARN4* at any time.

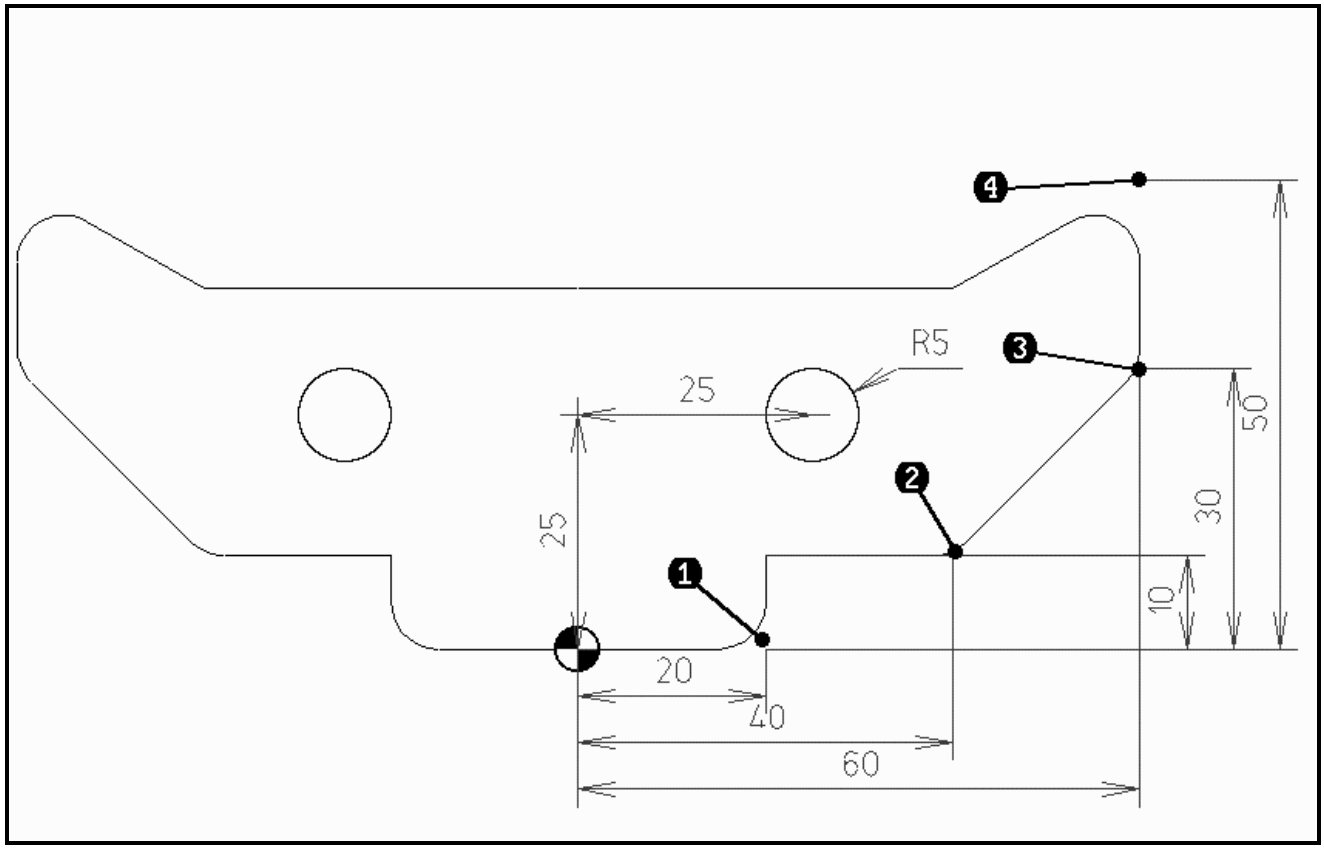
Unit 5: Pocket milling, zoom functions

Contents:

- Pocket milling
- Zoom functions

Task:

Load the drawing from unit 4, mill the inside of the contour as a pocket and drill the two holes.



Procedure:

With **load file** you can get the drawing *LEARN4* from the last unit.

Manufacturing: Drill and mill pocket

Activate the **menu manufacturing functions**.

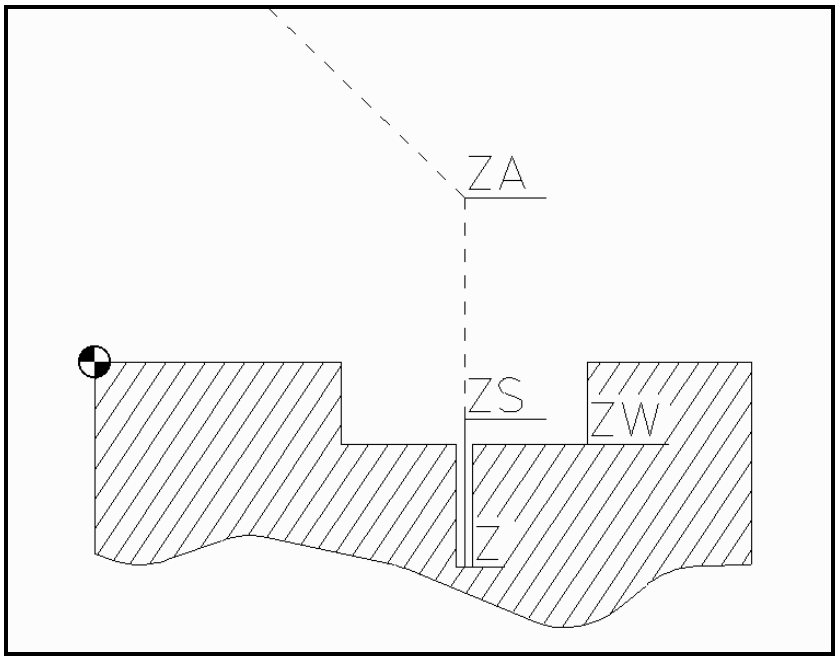
Choose the function **spot drill** and select the two circles by setting the mousecursor on the circle's periphery and pressing the left mousebutton. After aborting the selection you enter the

following data into the menu:

- Diameter: 16 mm
- Depth Z: -5.2 mm
- Initial level: 20 mm
- Clearance distance:1 mm
- Work piece level: 0 mm
- Dwell time: 0 s

After entering the diameter of 16 mm, PrimCAM searches the tool library for an appropriate spot-drill. It will be one that is at least as big as the given diameter. The diameter input field is automatically overwritten by the actual diameter of the found tool and its parameters are inserted in the field `tool no`. The name for the manufacturing is automatically filled out by PrimCAM.
If you haven't defined the workpiece yet, the according dialog box will open. You can get explanations to the individual items in the window by pointing at them with the mouse and pressing the key F1.

The meaning of the different levels is illustrated in the following picture :



The **depth Z** indicates how far down from the workpiece level ZW the drilling should go. Z is normally negative and indicates the depth of a hole or a pocket.
The **initial level ZA** is this level above which the tool can move freely in rapid traverse. This level is placed above clampings and so forth. Below the initial level the tool moves in rapid traverse only in Z-direction, in XY-direction only with feed rate.
The **clearance distance ZS** is the distance from the workpiece level to which the tool moves down in rapid traverse in Z. Below ZS the tool moves only with feed. ZS is generally chosen approximately 1 mm. The clearance distance prevents that the tool hits the workpiece surface in rapid traverse because of the accuracy of measuring the tool.

The **workpiece level ZW** is equal to the height of the workpiece's surface, measured from the workpiece origin. ZW is usually equal to zero. If you have to make some drillings in a deepening of the workpiece (10mm) for example, you set ZW= -10. The drill then moves 10 mm still in rapid traverse and then starts to cut. Pay attention to the fact that the deepening has to exist before the drilling, otherwise the drill will collide with the workpiece at Z=0.

Choose the function **drill** and select the two holes analogous to the last step. Make the following inputs for the drilling:

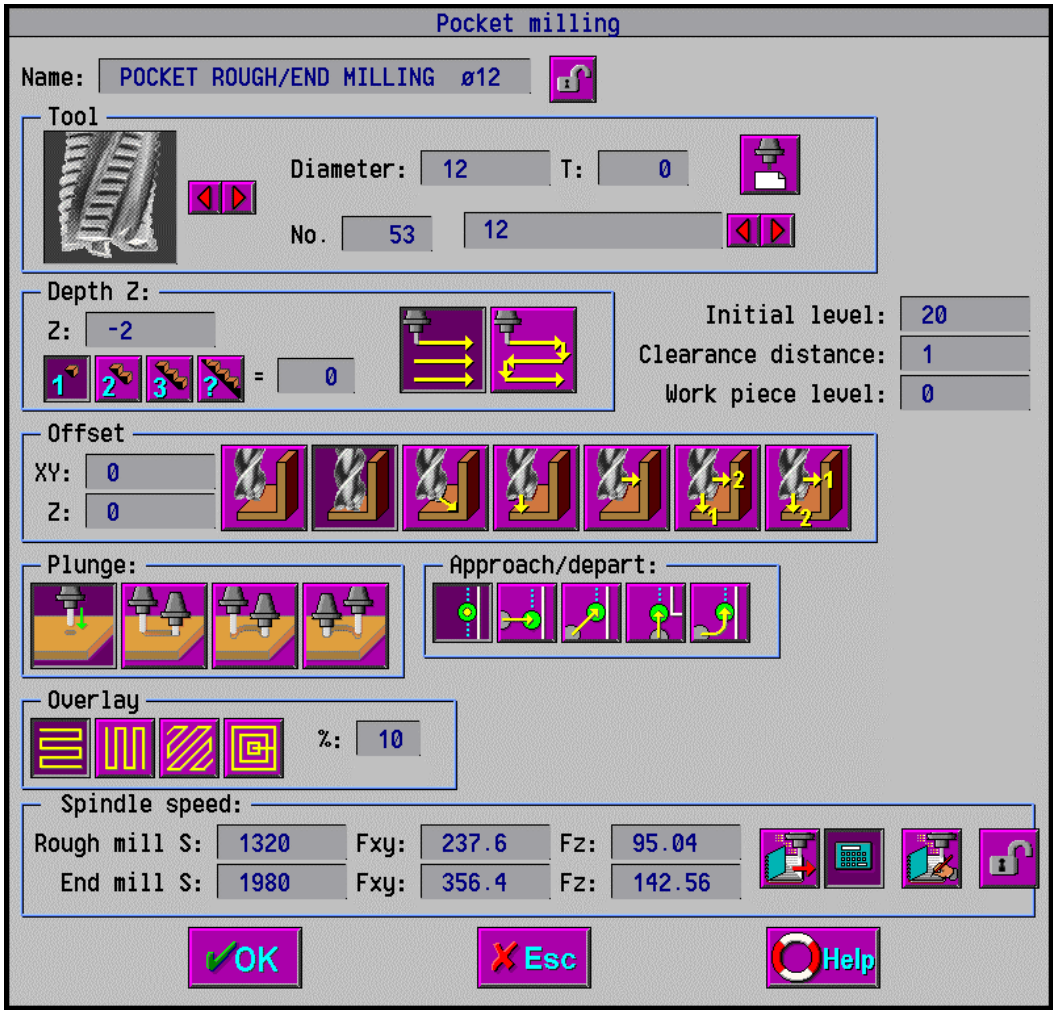
- Diameter: 10 mm
- Depth Z: -22 mm
- Initial level: 20 mm
- Clearance distance:1mm
- Work piece level: 0 mm
- Dwell time: 0 s
- Cycle: high speed peck

Select the function **mill pocket**. At the prompt to `select object to mill pocket` you set a window over the whole object. All parts of the drawing, also the two circles, change their color to violet, which means that they are all selected. If you now would generate a pocket, the two holes would stay inside the pocket as islands, because contours inside a contour milled as a pocket are interpreted as islands.

Because the two circles are drillings and not islands, they have to be deselected first. The icon switches from **select** (ball not crossed out red) to

deselect (grey ball is crossed out red). Now click the two circles inside the contour. They change their color to white and aren't selected anymore. With this switch function you can toggle optionally between select and deselect, until you have selected the correct parts of your drawing. With the right mousebutton you abort the deselecting.

Set the starting point for the pocket inside one of the two drilling holes using the snapfunction center. Set the direction point so that the pocket is manufactured inside. Afterwards the window appears, where you have to set the parameters for the pocket milling. Fill it in according to the example below:



The **overlay** determines in which direction the tool is led. We choose the axis parallel manufacturing in X-direction. Klick *OK* if you have finished entering.

The function **material removement** makes visible where exactly the mill passes and cuts material. You can switch on or off this function by repeated clicking.

Now choose the function **zoom window** to show a detail of the drawing enlarged. Set the selection window to the left edge at the top of the contour. As you can see now, the 12 mm mill doesn't reach the edge completely. We have to take a smaller 10 mm mill to manufacture the 5 mm radii completely. But this doesn't mean the whole pocket has to be programmed again. In the next unit we will see how you get access again to the dialog window for the pocket in the job manager, where we can change the diameter easily.

First we want to switch back to the view of the entire contour again. That can be accomplished by **zoom previous,** a function that appears in the flip-out bar when you choose zoom window. This zoom function activates the last view of the drawing. By repeated clicking on this icon we can toggle between the two last views.

In this context another zoomfunction, **zoom half,** should also be mentioned. If you click this function, the represented view is scaled down, so that you'll get a better survey of the drawing.

Switch back to the view of the whole picture again with **zoom all**.

Save the drawing with the inserted manufacturing. It is necessary for the next unit, where you get to know the job manager.

Unit 6: Jobmanager

Contents:

- Jobmanager

Task:

Load the drawing from unit 4/5. Change the diameter of the mill for the pocket from 12mm to 10mm. Create, using the jobmanager, the NC-code first without the drillings, afterwards with the drillings.

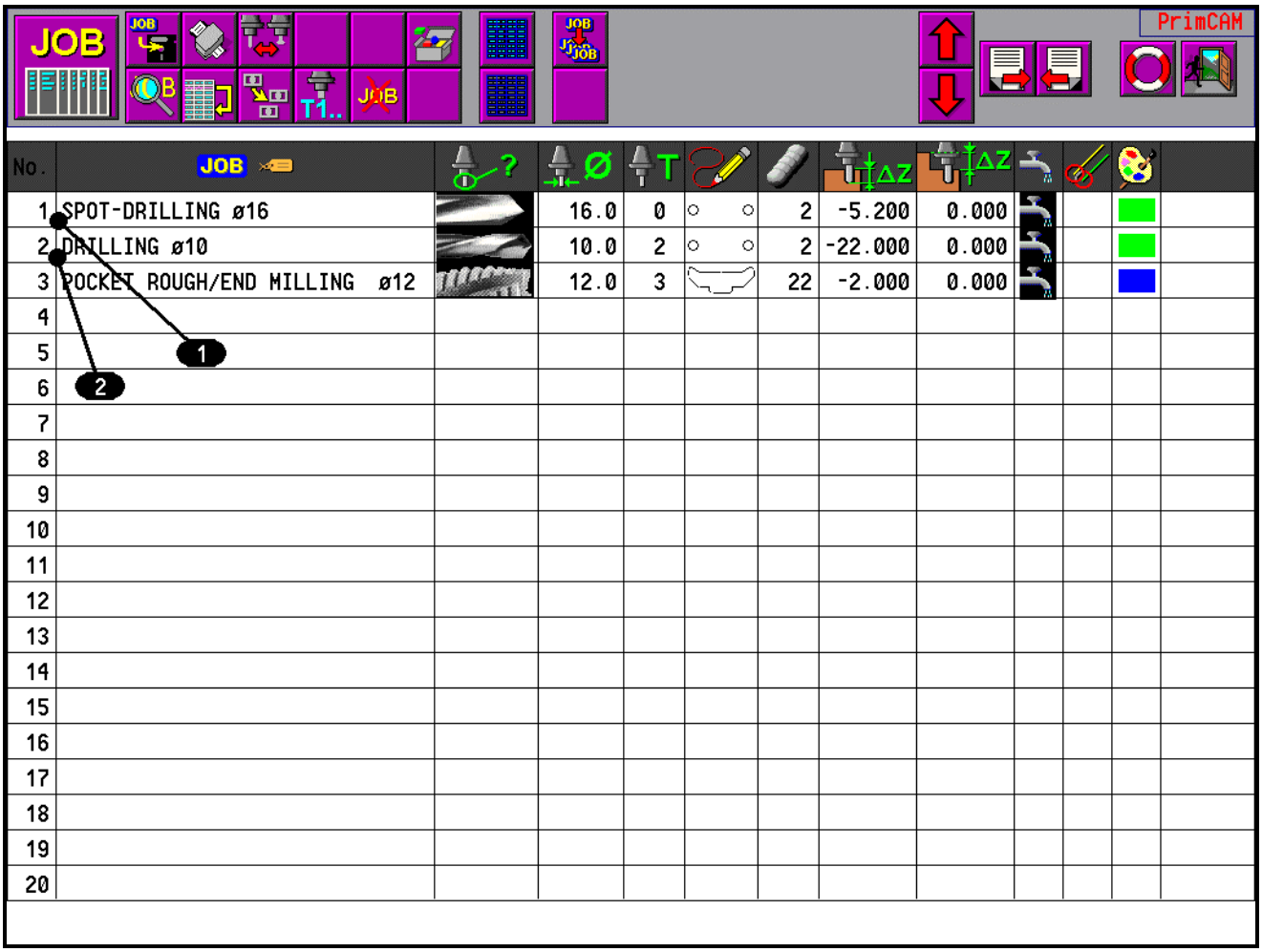
Procedure:

With **load file** you can get again the saved drawing *LEARN4* from the last unit.

The jobmanager

For calling the jobmanager, you change to the **menu utilities** . The jobmanager is nothing else than a scedule that lists all the steps of a manufacturing in a table.

In the **jobmanager** you see the three lines with the manufacturing processes spot-drill, drill and mill pocket. Click now the third line (pocket milling) beyond the term Job. This way you get again the window you already have seen during the definition of the pocket. Change here the diameter from 12 mm to 10 mm and confirm the entering by OK.



At the top you see the various functions of the jobmanager. Moving the mouse to an icon and pressing the key F1 explains you the functions.
The following data about the individual manufacturing processes are listed in the jobmanager:

No.: Number of the manufacturing. All manufacturings are enumerated. They are also written to the NC-program and executed according to this numerical order. By clicking a number you can make the manufacturing inactive (grey), by reclicking you make it active (light blue) again. Inactive manufacturings aren't written to the NC-program.

Description of the manufacturing process with the diameter of the tool (D10). A click on the field with the manufacturing opens the window, which was already filled in during the definition of the process. It can be changed here.

Tool type, for example mill, end mill, drill.

Diameter of the tool used.

Tool number (magazine) of the tool used. This is the number written to the NC program (and not the library number of the tool).

Here you see an **image** of the manufactured contours. So you can see which job processes which outline.

Shows the **count of objects** that are milled by this job.

Depth Z of the manufacturing measured from the workpiece level

Workpiece level ZW of the job

Coolant on/off. If the coolant is of, the water-tap is crossed out red. Clicking on the water-tap allows to switch on/off the coolant for each job individually.

Radius compensation
P: Radius compensation by PrimCAM always
X: Radius compensation according to the setting in the machine library

Color for path simulation allows you to set different colors for the tool path representation in the simulator and drawing (when you activate the tool path display.

Click now on the field `No.` of the manufacturing spot-drill **(1)** and drill **(2)**. The two lines change their color to grey and become inactive. They are neither simulated anymore nor inserted into the NC-program.

Quit the jobmanager by clicking the icon exit.

Now change in the **menu utilities**

to the **simulation**. As you see, the holes are not simulated anymore.

Go to the **menu file functions**

for generating the **NC-code**, and view it in the

NC-editor. You can notice that there is no NC-code for the drillings. Go now back to the jobmanager and fade in again the manufacturing steps 1 (spot-drill) and 2 (drill) by clicking the two fields next to **(1)** and **(2)**. If you would like to activate a lot of jobs at the same time, you can also move over the corresponding numbers with the left mousebutton pressed down. You can inactivate the other way round as well.

In the simulator you can see that the according jobs are now active again.

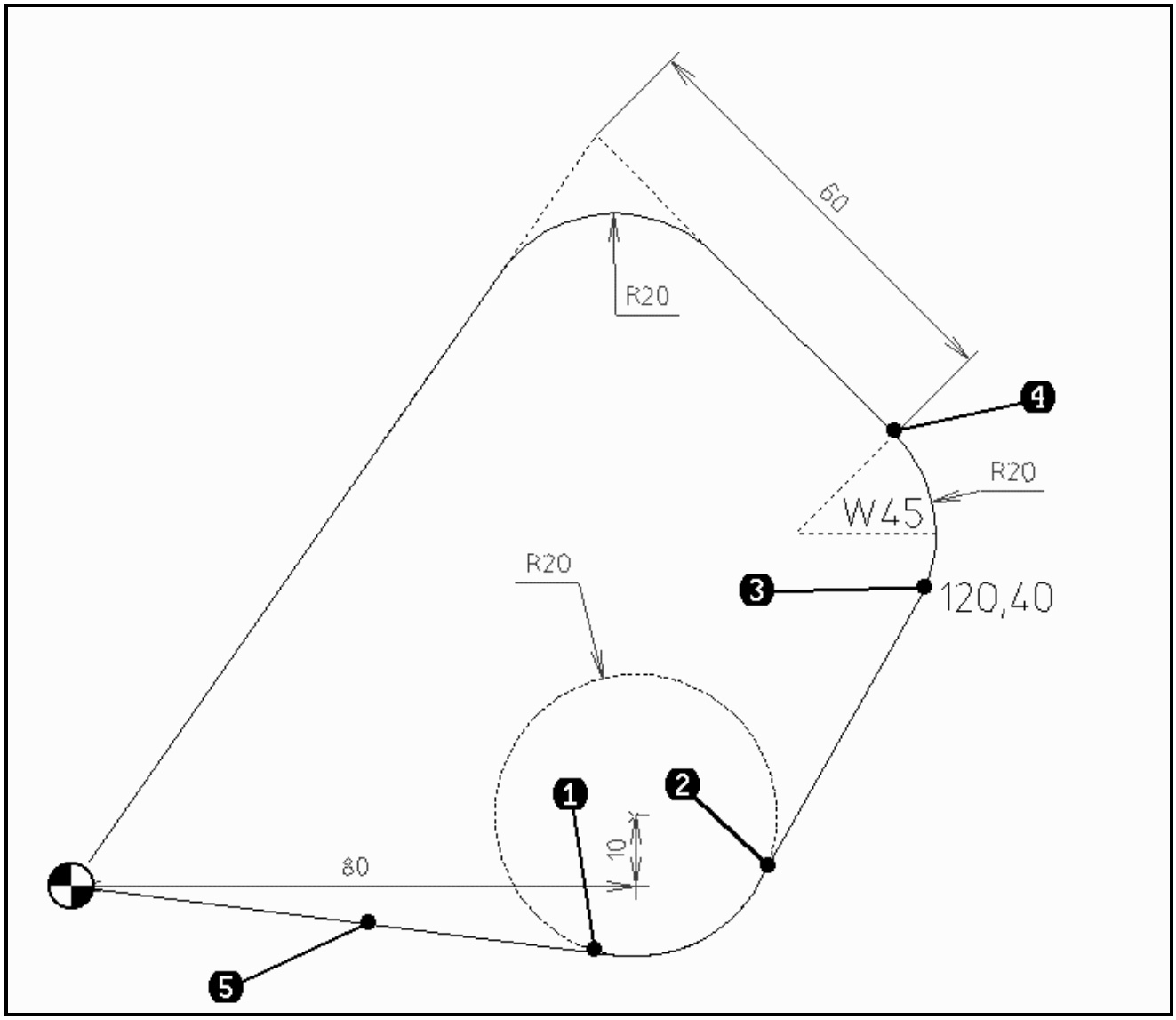
Unit 7: Rough cutting, end milling, offset

Contents:

- Creating complex geometries
- Manufacturing: rough cutting and end milling
- Approach and depart methods

Task:

Create the contour shown below, rough cut it inside with an offset of 2 mm and end mill it afterwards with the same tool without offset.



Procedure:

First we create the geometry. You'll notice that it is dimensioned in an unusual way. No technical designer will dimension his drawings like this. But you get to know some special possibilities of PrimCAM with this drawing. Try to create the contour first on your own. Consult the instructions below if you have any troubles.

First enter a **circle** with the coordinates X=80, Y=10, R=20
80,10R20<ENTER>

With the function **line tangent to circles** you create a tangent from the one circle (click the circle next to position **(1)**) to the origin X=0,Y=0.
0,0<ENTER>

Create a second **tangent** from the same circle (click circle next to position **(2)**) to the point X=120,Y=40
120,40<ENTER>

An iconbar opens when you select the function draw **arc**. Click there the function

continue tangential with arc . At the prompt `endpoint of tangential object` you click the end of the previously created line next to point **(3)**. If you now move the mouse, you see a violet arc streching from the starting point to the mousecursor. The arc will be connected with the line so that a tangential connection results. To get an arc radius of 20 mm, you enter:
R20<ENTER>
If you now move the mouse, you see that the radius stays fixed at 20mm. After entering the angle
W45<ENTER>
the arc is completely specified.

The function **continue tangential with line** is presented if you activate first the function draw line. To continue with a tangential line you have to determine only the length of the new line, because the direction is already determined by the condition that the line has to be tangential to the selected arc. For connecting the 60 mm long line, you click for `select endpoint of tangential object` next to **(4)** and then enter L60 or
60<ENTER>.

Close the contour with a **line**. For this purpose use

Snap end point, for snaping exactly the endpoints of the lines.

Fillet the above edge with a radius of 20 mm.

For deleting the part of the circle you don't need anymore, you have to **break** the circle first. This means the circle will be split into individual arcs, one of them can be deleted afterwards. After clicking the function **break**, the prompt `select object to break` appears.

Select the the cirlce and at the prompt `intersection points to break` click the two points next to **(1)** and **(2)**, where the lines touch the circle. The intersections are marked by green crosses. With the right mousebutton you abort the selection of intersection points.

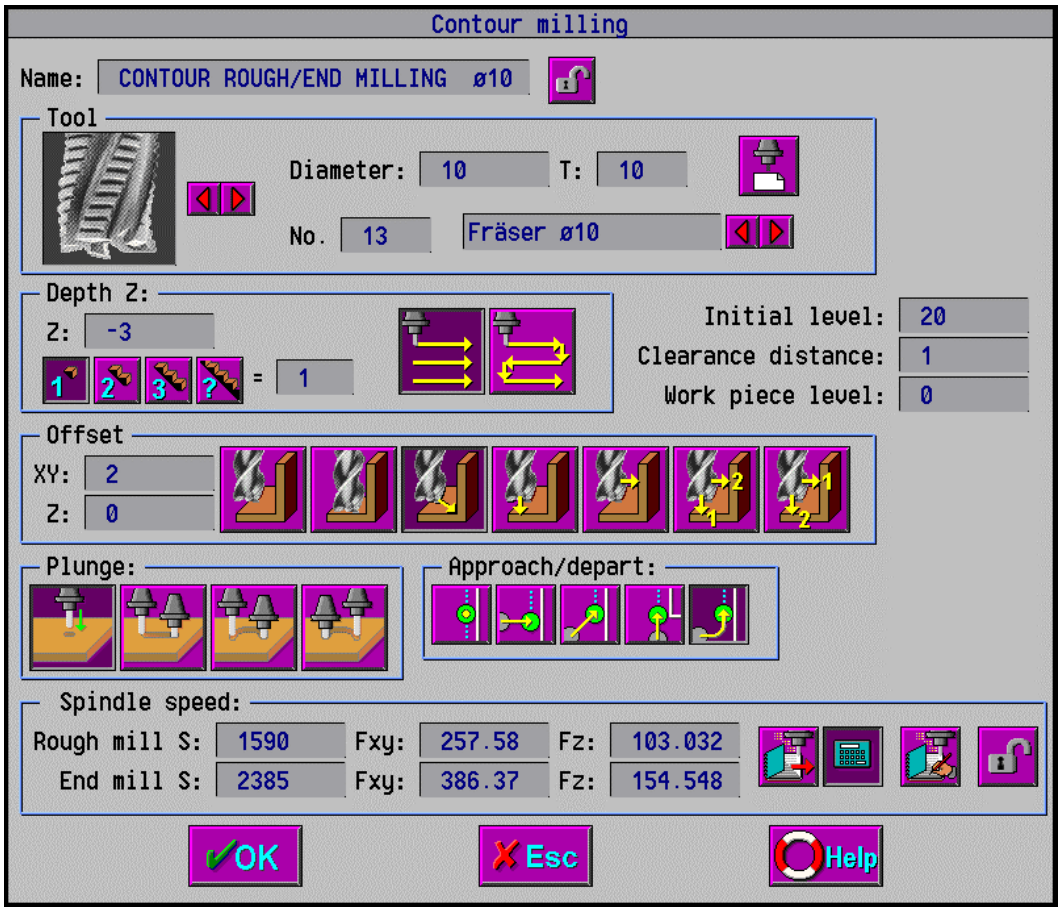
Delete now the part of the circle you don't need anymore. For that purpose click the part of the circle you would like to delete and confirm afterwards the question `delete selected objects?` with the left mousebutton.

After deleting objects it can be necessary to **refresh the screen** . The deleting may leave little gaps in other objects, which are visible on the screen. When drawings become larger, they need more time to refresh, for this reason the screen is not automatically refreshed after every action.

We recommend to **save** the file now, for example with the name *LEARN7*.

Manufacturing: Rough cutting and end milling

Now the created contour should be milled inside with the function **mill path** , and with the same tool first in one rough cut cycle with 2 mm offset, then in end mill without offset. You should use a 10 mm mill for this.
The simplest way to select the contour is by setting a window over the whole drawing with the cursor. Pay attention to place the direction pointer inside the contour, approximately next to point **(5)**. The direction of the manufacturing arrow should point to the right side. Enter the following data into the appearing window:



For `manufacturing` you have to choose **rough cutting+end milling**. The piece gets rough cut and end milled in two passes with the same tool: First rough cut with the chosen offset and afterwards end milled with no offset.

The `plunging` in **Z-direction should** be **straight down**. Other possibilities would be **line in Z angular** or **helical line left/right**.

The **approach/depart** to/from the contour should be **tangential**.

With **switch on/off tool paths** you can make the paths visible, then you see clearly the 2 parallel paths. The first one runs with 2mm offset, the second one follows exactly the contour. Normaly you choose, depending on the mill diameter, a smaller offset for the rough cut cycle. Usually it would be 0.2 mm for example. The offset is chosen so big here that you can better distinguish between the 2 paths.

Finally create the NC-program for the manufacturing.

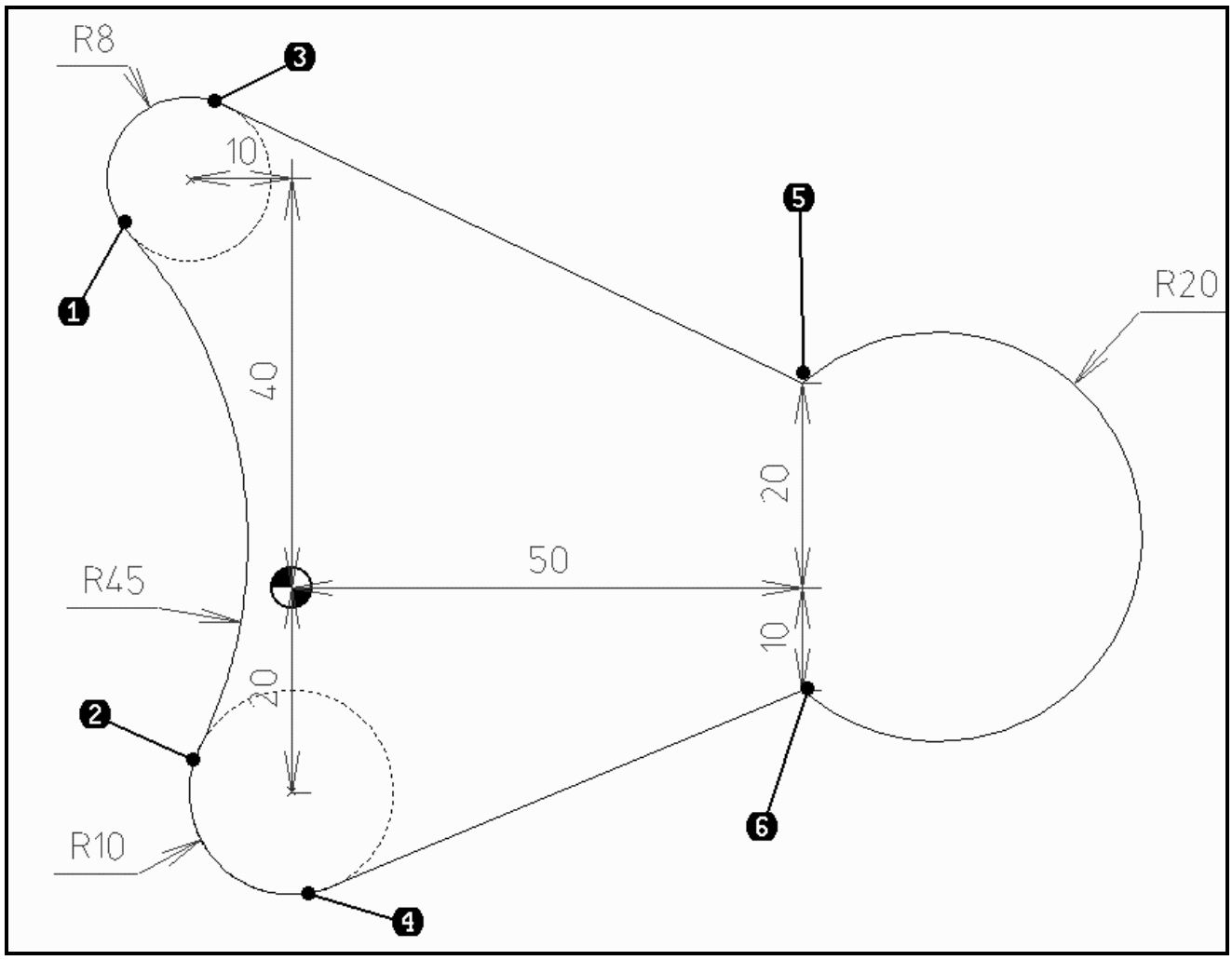
Unit 8: Advanced drafting functions

Contents:

- Advanced drafting functions
- A circle tangential to two other circles
- Arc with two points and radius

Task:

Create the drawing below.



Procedure:

Create the following two **circles with centerpoint and radius**
X-10Y40R8<ENTER>
X0Y-20R10<ENTER>

Create a **circle tangenting 2 objects and point**, which touches the other two circles. Select the first circle at the prompt `select 1st tangent object for circle` next to **(1)**. Click the second circle at point **(2)**. You can now drag around the circle. It touches the 2 selected objects near the 2 points you picked and goes trough the cursorpoint. You now have two possibilities to determine the circle. You can either enter a point or the radius, depending on your

specific situation. If you intend to enter the radius you have to drag the circle approximately to the final position. You can check hereby the size of the radius in the third coordinate box in the upper left corner of the screen. It is necessary to drag the circle before entering the radius to determine the proper solution out of the several ones. If the circle is close to the required one you enter *45* and the circle is defined.

Create with **tangent** the two lines to the circles with R8, R10. At `tangent from object` click the above circle approximately at point **(3)**, where the line should be tangent, and enter *50,20* as endpoint. Proceed in the same way for the second tangent, where you choose point **(4)** as the starting point and create a line to X50Y-10.

The function **arc through 3 points** also allows input of two points and the radius. Choose as `1st point for arc` the point next to **(5)** (with snapfunction snap endpoint) and as `2nd point for arc` the point next to **(6)** (also with snapfunction snap endpoint).
Instead of entering a point at the prompt `3rd point for arc` you enter now the radius of the arc with *R20*.
Four possible arcs appear in violet, which all fulfill the required conditions. Select the proper arc according to the drawing at the prompt `select required arc`.

Now you still have to **break** the circles and delete the unnecessary parts. During **breaking** you have to pay attention that you always select the object first, afterwards all points where to break. The points get marked with green crosses if the breaking was successful. Quit the function with the left mousebutton. After that you can select the next object.
First select the circle with 8mm diameter and break it next to **(1)** and **(3)**.
Break the circle with 10mm diameter next to **(2)** and **(4)**.
The circle with 45 mm diamater is broken next to **(1)** and **(2)**.

Delete the pieces of the circle not needed anymore.

Unit 9: Part libraries / iconeditor

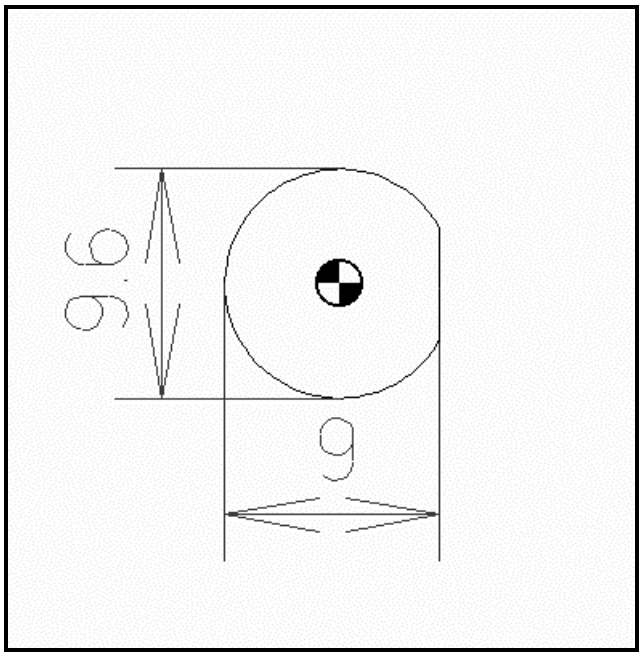
Contents:

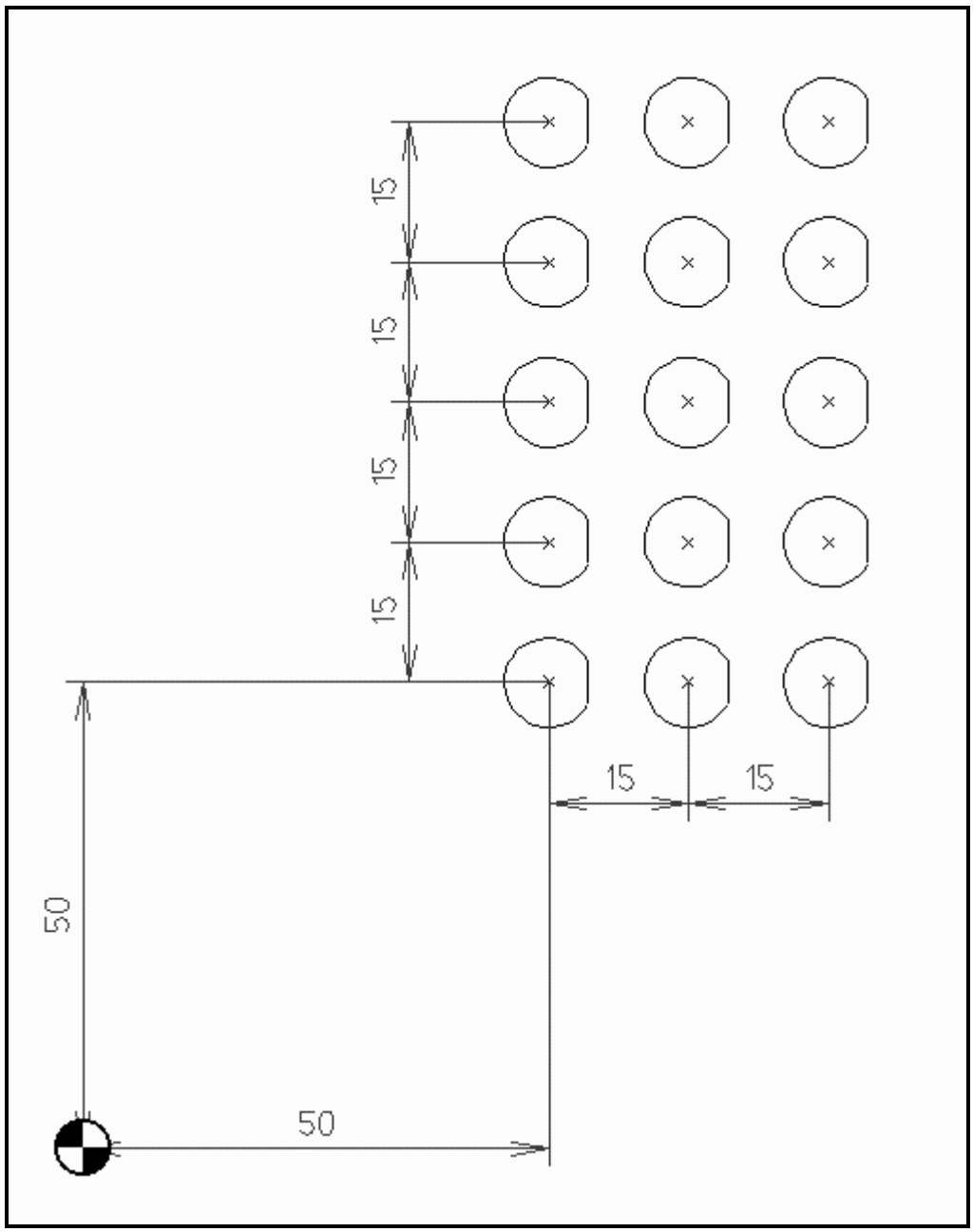
- Insert a simple part with manufacturing into the library

Task:

In the following drawing you see the sparing for a BNC-connector. Draw this connector and program the manufacturing. Create an icon for the part library that shows the connector and will be „photographed“. Then you insert the drawing into the part library, so that it’s available for other manufacturings. Create the program for a front panel with 3x5 BNC-connector sparings.
For the manufacturing the following steps are necessary:

- Rough cut the contour to 0.2 mm offset with the 3 mm mill, plunge helical and approach tangential
- End mill the contour with the 1 mm mill, so that the edge radii aren’t too large





Procedure:

We start with the geometry of the connector's sparing. We choose the function **circle with centerpoint and radius**. The centerpoint is the origin, the radius the half of the diameter (9.6/2 mm).

0,0R(9.6/2) <ENTER>

Pay attention that here we calculate the radius from the diameter by dividing it by 2. We could also have entered the diameter directly in the following way:

0,0D9.6 <ENTER>

For the line on the right side of the circle we create first two vertical construction lines, which will make the construction easier. Select the function **construction line vertical**.

To use the dimension of 9.0, which reaches from the circle's left side to the line on the right side, you select the snapfunction **snap quadrant** . This snap function gives us the points on a circle or an arc positioned on the periphery at the angles of 0, 90, 180 or 270 degrees. By the way, all the snapfunctions are always available during geometry input in the below screen part, as are the zoomfunctions. Now we create the construction line by clicking the circle on the very left. The construction line will be placed tangential to the circle.

Now you create the second construction line, which runs parallel to the first one in a distance of 9 mm. The function construction line parallel gets visible in the special functions bar when you click construction line horizontal or vertical. At the prompt `select object for parallel` click the previously created construction line. PrimCAM now asks for a distance D, with which a parallel to the selected line should be generated. Pay attention, that the mouse is placed on that side of the original line where you would like to create the parallel. Now enter *9 <ENTER>* for the distance and abort the function for creating other parallels with the right mousebutton or with ESC.

For cutting off the circle on the right side, we simply have to create a **line**, whose starting and ending points are the intersection points of the circle with the parallel construction line created just a moment ago.

Switch on the **automatic snapfunction**. Now click with the mouse the two intersection points of the construction line with the circle, which creates the line.

To cut off the right part of the circle, which we don't need, we have to **break** the circle first. This function serves to split the circle into arcs.

At the prompt `select object to break` you click intentionally on the intersections of the new created line and the circle. That means, that actually two objects are in the rectangle of the cursor, and PrimCAM doesn't know which one of the two you actually would like to break. PrimCAM opens a window with a scrollbar in the top left corner of the screen, in which the individual objects that can be selected are listed. You can move the scrollbar by mouse, and the selected object is represented in the meantime in violet. Confirm with the left mousebutton, when the circle is represented in violet. At the question about the intersection points for breaking, you set the mouse on the two endpoints of the line and confirm with the left mousebutton. A successful breaking is marked with a green cross. If the circle is broken at the two points, you finish the function by clicking the right mousebutton.

Now the smaller, unnecessary arc of the circle is deleted. Activate the function **delete**, select the arc with the left mousebutton, and confirm with the same button again.

The two construction lines, which aren't necessary anymore for the further procedure, should be also deleted. If you have an extensive construction with many construction lines, it would be exhausting to delete them all individually. But you can delete them all at the same time, because the construction lines lie on a different layer than the rest of the construction. Change to the menu **settings** , where you find

the function **set layer**. Now all the used layers are represented in a table. This serves to manage the layers. Here you can simply make invisible the construction lines by clicking the state column on the second line of the construction layer. The entry changes to `0`, meaning that the layer isn't visible on the drawing anymore.

Instead of only making invisible the objects on the layer, we would like to delete them definitely from the drawing. Click again the 0 in the state column to make visible the construction lines again. With the function **delete objects in layer** all the construction lines are removed now. At the question `layer to delete objects (1-5)` you can either enter the layer number on the keyboard or even more simple click somewhere on the line of the construction line layer. Confirme `should objects in layer construction line layer really be deleted` with `OK` and quit the layer management with the exit icon.

Manufacturing

Now you get back again to the drawing by clicking the icon **Quit** on the right top of the screen. We recommend to save the drawing now, for example under the name *LEARN9*. Now we would like to start with the **manufacturing.**

For the sparing it's sufficient to mill the connector's contour with the tool along the periphery, what can be done with the function **mill contour**. Select the two objects of the contour and enter as `starting point`
0,0 <ENTER>.
Thereupon the tool plunges in the middle of the circle. Select the mill direction upwards, whereby the direction pointer has to be placed inside the contour (on the left side of the vertical line; we would like to approach to the contour from inside with the mill). In the dialogbox you enter the following:

- Diameter: 3 mm
- Depth: -2.5 mm
- Steps: 1
- Initial level: 20 mm
- Clearance distance: 1 mm
- Workpiece level: 0 mm
- Offset XY: 0.2 mm
- Offset Z: 0 mm
- Manufacturing: Rouch milling (1st icon from the left)
- Plunge: Helical motion (third icon from the left)
- Approach/depart: Tangential (last icon)

Speed and feed are automatically calculated by PrimCAM, and therefore you don't have to change them.

Make the **toolpath visible.** The toolpath is represented by green lines. Mill start and endpoints are represented as red circles with the diameter of the tool.

Make visible the **material removement.** To minimize the edge radii a bit, the contour should now

be end milled with the 1mm mill. Instead of repeating now the whole programming process for the 1mm mill, we just copy the manufacturing process of the 3mm mill and change the radius of the cutter in the copy. For that purpose we open the **menu utilities**, where we select the

jobmanager. In line no. 1 the just programmed the manufacturing step for the contour rough cutting with the 3mm mill is listed.

With the function **copy job** we create a second manufacturing process, identical to the first one and change it afterwards. At the question `job no. to copy: (1-1)` you can either confirm the already suggested number 1 with the return key or click directly with the mouse somewhere on the first line. Abort the selection with the right mouse button. You would like to copy the first job to the place number 2, so you enter 2 and confirm with return. Here you can also just click the second line.

Click on the second line, which indicates the second manufacturing process. A window for **mill contour** opens, where you can make the analogous changes.

- Diameter: 1 mm
- Manufacturing: end milling only

Notice that you dont have to change the offset to zero, because it's not considered for end milling anyway.
For checking the path, we switch the first path temporarly to inactive, so that it isn't visible anymore. Click on the 1 under no., and the entry changes its color from light blue to grey (by another clicking the entry can be activated and desactivated again). Then quit the jobmanager by pressing the exit icon. Now you see clearly the mill path and the material removement of the 1 mm mill.

For activating all the defined manufacturing proceesses again, you go to the **menu manufacturing** and activate the two manufacturing processes
using the function **fade in all jobs**. If the job isn't active, it's represented only in grey in the jobmanager. Then it also isn't considered during the generation of the NC-programs. You can take advantage of this, if ,for example, part of a piece is already finished, and you only have to execute some specific manufacturings. For that purpose you just switch off the manufacturing steps not needed and generate the NC-program.

Creating icons

Now the icons will be created using the iconeditor. The programmed connector can later be selected from the part library by just clicking those icons. The icon editor is located in the **menu utilities**.

If you start the **icon editor**, you see three rectangular areas in the drawing area. The small violet rectangle represents the icon, which is just created, in its original size. On the left side you see a large violet rectangle, in which you can draw. Here you see the individual points of the icon. On the left side of the screen there is a color table. By clicking with the mouse on a color you can change the current drawing color.
The menubar at the right border of the screen provides the drawing functions, similar to those used to draw the geometry. In the iconbar at the top of the screen's border you find the functions for loading, creating, saving icons and so forth. You can always use the helpsystem to get more information on how each of them does work.

Begin a **new icon** . The violet area is deleted now, and you can begin with the creation of a new icon.

Instead of drawing the icon for the BNC-connector, we also have the possibility to **photograph** it and afterwards change it according to our requirements. If you call the function photo, the icons for drawing and the colour setting dissappear. Instead of those the functions for moving the picture and for zooming do appear.

With **autozoom** you can bring the interesting objects into the view-finder. After clicking the function you select the arc and the line. As soon as you abort the selecting, it is „zoomed“ automatically, until the objects are in the view-finder.

With the various **scrollfunctions** you move the connector symbol to the middle of the icon and

flash the **photography**. Then the editing possibilities for the icon are faded in again.

The icon’s inside should be filled with blue colour now. We need a closed contour for the filling function. If the contour has some gaps, select the colour white and close them using the function that sets individual points.
Now fill the inside of the connector’s periphery by activating the function **fill**. Click the colour blue in the colour table and then click inside the icon drawing area.

Save the icon to the harddisk.

Quit the icon editor.

Inserting a part into the library

Now the drawn connector’s sparing should be inserted together with the milling into the part library. First you have to select the library the new part should be inserted into. Click **library** and select the library with D-SUB connectors that flips out in the bar on the left.

To insert the part, go to **Software tools** and

select **insert part into library.**

Activate the option for **inserting geometry and manufacturing**. The two other options would insert either only the geometry or only the manufacturing into the library. Choose the left icon in the next line to indicate that you want to use an icon as the symbol for the part and not a scaled down picture. You can also optionally give the name *BNC* to the part. This name will then be displayed as a tooltip when you move the mouse over the icon.

Select the connector and enter as `reference point` the center of the arc:
0,0 <ENTER>.
The reference point is the base point of the library object. If the object is inserted from the library later, PrimCAM asks where the reference point should be placed to.
Now the window for the selection of the icon place is opened. Choose the first free place, the middle icon of the 2nd row.
As icon for loading you select the photographically created one, which symbolically represents the BNC-connector. Thereupon the programmed part is inserted into the library.


Now we start a new drawing, in which a BNC connector should be inserted from the library and afterwards be copied in a matrix. Open the menu **file functions** and create a **new drawing**. If you haven't saved your old drawing LEARN9 yet, you will now be asked whether you would like to save the changings. Confirm this with OK.

By clicking the **menu part library** the icon for the BNC connector appears. Click the BNC icon now and move the cursor into the drawing area. You see now the outline of the connector moving with the mousecursor.
You could rotate the connector for example 90° by entering *W90* or scale the connector with *F3.5* by a factor of 3.5.
Enter as `insert point` *50,50 <ENTER>*.

If you don't see the entire connector just inserted on your screen, you can make it visible by calling the function **zoom all**.

Now you could insert other connectors in the same way. But abort the inserting of other connectors now with the right mousebutton and go to the **menu draw**.

With the function **matrix** you create copies of the sparing, which are ordered in figure of a matrix in lines and columns. You can also determine the distance between the individual copies. If you select the object by clicking only the arc now, you see that the line is also selected. That's because parts inserted from the library are inserted as blocks. For most of the functions blocks are considered as one object during selecting.

If you would like to decompose a block into its individual parts, this happens with the function **split**.

Abort the `select objects to generate matrix` with the right mousebutton. Fill in the appearing window with the following data:

- Number of columns: 3
- Number of rows: 5
- Distance columns: 15
- Distance rows: 15

Save the drawing under the name LEARN9_2.

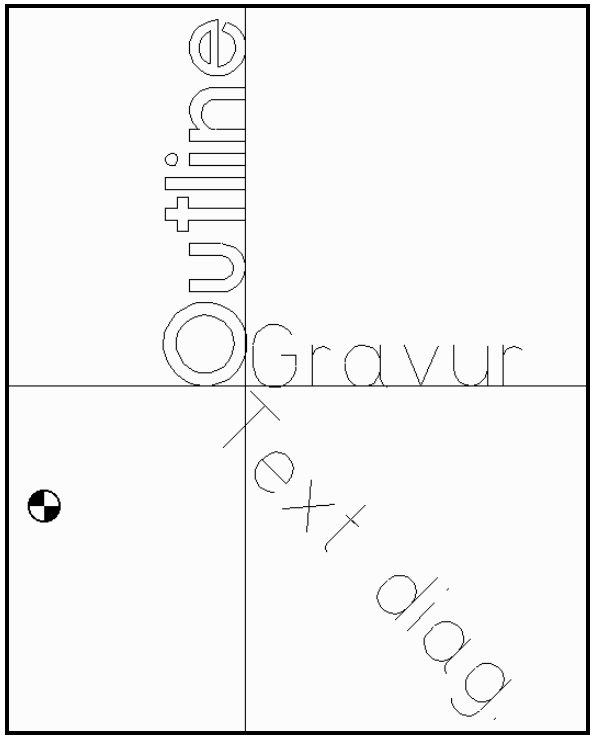
Unit 10: Fonts and Engravings

Contents:

- Draw text in different sizes and fonts
- Create engravings
- Mill outline-fonts

Task:

Draw the text using the fonts visible in the picture below. Engrave the stroked fonts and mill the outline-font.



Procedure:

To align the texts exactly, we first create a cross of construction lines. We find this function among the **construction lines**.

Create a **cross of construction lines** at X=50, Y=30. The two lines are drawn as soon as you enter the point.

If you activate the function to **draw text**, the actual fontname (next to the oblique F) and the actual height (H) are shown in the menubar at the top. H represents the height of an uppercase letter. There are also functions for text alignment, changing, enumeration as well as font placement along arcs.

Now click the field with the fontname and choose `ORION`. Pay attention that there is a oblique F (for PrimCAM font) in front of the fontname and not a TT (for TrueType fonts). You can, if necessary, click on the TT, changing back to PrimCAM fonts. Enter now
H15 W0
to set the textheight and the angle.

If the **text alignment** isn't already set correctly, click the according field and choose it so that the text is aligned at the left and at the baseline (bottom). Set the `insert point` for the text on the intersection of the construction lines using snap intersection point. Then you see a blinking textcursor. As you enter the text *Gravur* you see that the text is written according to the text alignment.

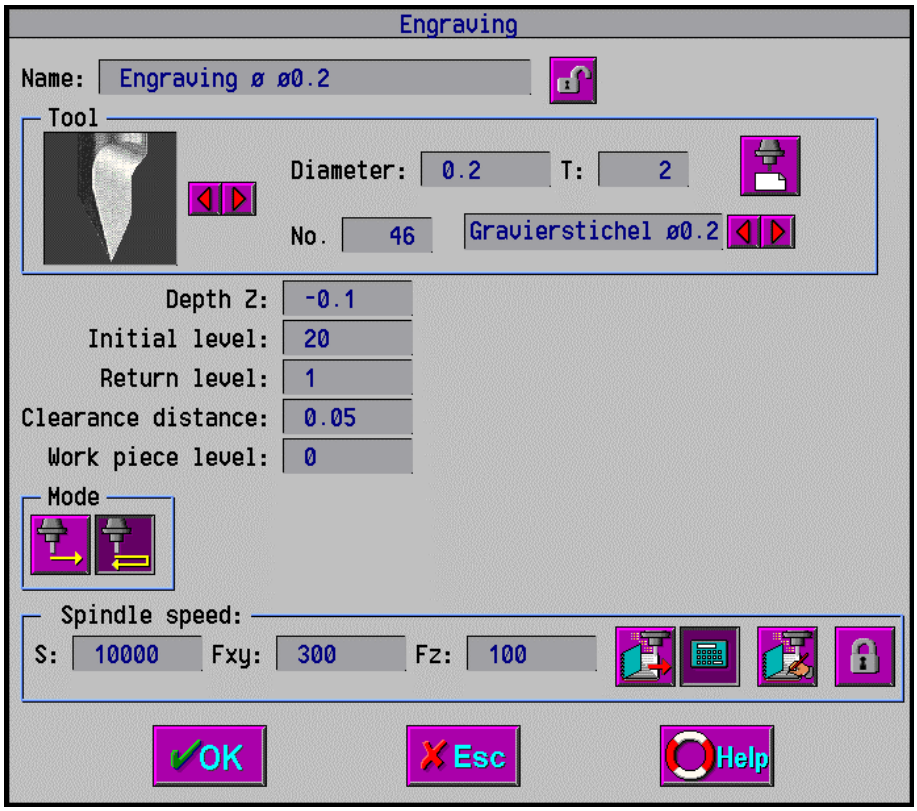
Now change the **text alignment** according to the shown icon and set the writing direction (angle of text) to -45° by entering *W-45*. Snap again the intersection of the construction lines as the `insert point` of the text. Enter
text diag.

Now switch back to the previous text alignment left and baseline and select the font `Futuros`. Enter *W90* as writing direction, which writes the text in the positive Y-direction. The textheight is set to 20mm with *H20*. As starting point select again the intersection point and enter the text '*Outline*'. FUTUROS is an outline-font. This means the text doesn't consist of strokes, but is defined by a contour.

Save the drawing under the name *LEARN10*.

Manufacturing

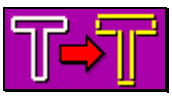
Change to the menu manufacturing and click the icon **engraving**. Select the text `'Gravur` and `'text diag'`. Fill in the appearing window according to the following:



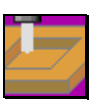
Mode: During the engraving, it often happens, that chips still remain in the path. With the option **double pass** the tool moves once more through the path in the opposite direction. The chips are removed, which results in a clearer engraving. But this option needs almost twice as much time and doubles the NC code size.

Return level: During the engraving the tool isn't moved back to the initial level ZA between the individual letters, but only to the return level. We suppose, that there are no obstacles between two letters of the same text. But between two different texts the tool is moved to ZA.

The text `'outline'` shouldn't be engraved on the lines or arcs itselves, but the outline should be milled inside with the function mill pocket. But the text still doesn't consist of individual lines and arcs, which are needed to serve as borders for the pockets and islands. Therefore the text object has to be split first. The function split text is accessible in **split**.

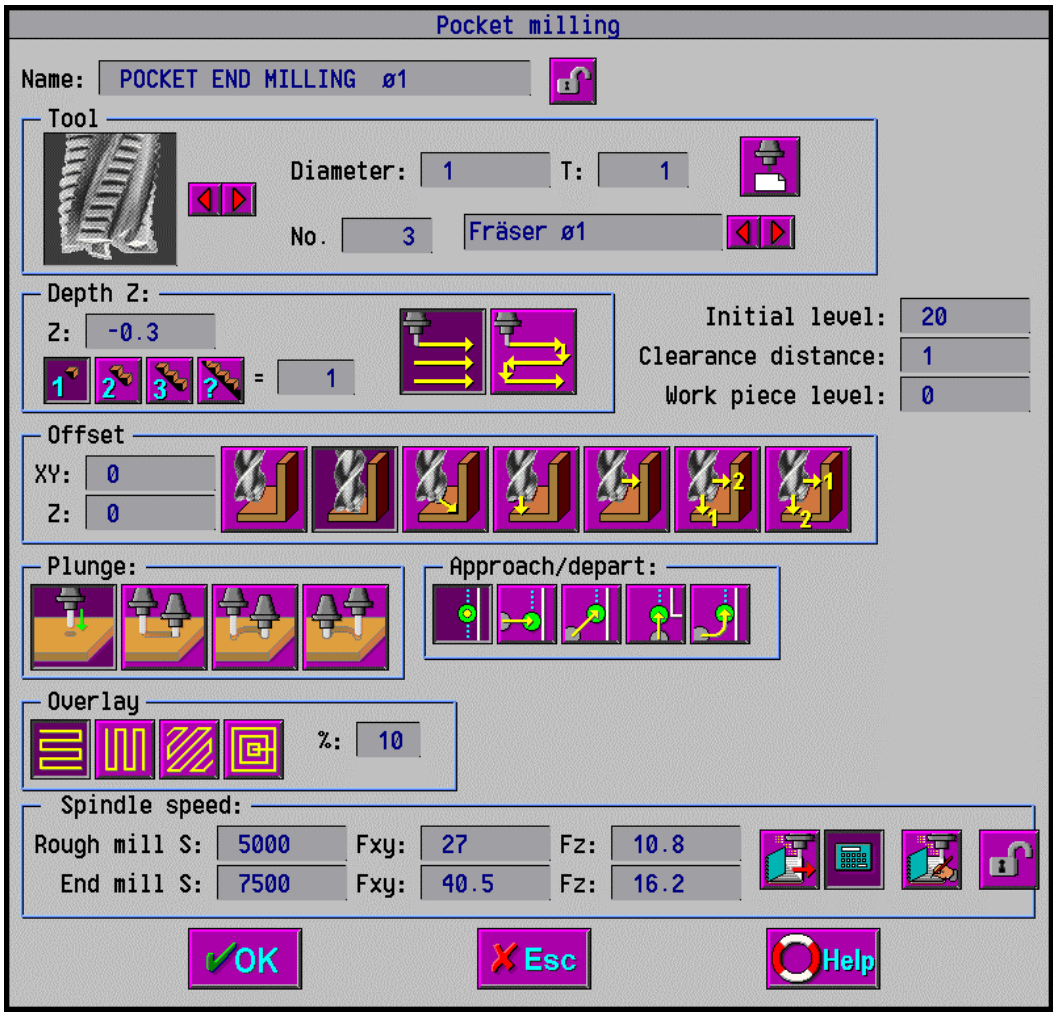


Click the function **split text** and select the text `'Outline'`.



If the text is split into lines and arcs, you can **mill** it in the usual way as a **pocket**. You can just click the whole text by clicking one individual object of the text. The text has been split (into arcs and lines), but it has been saved as a block. Fill in the window for **pocket milling** in the

following way:



The text `engravings` should be modified now. Click the function **change**, which allows you access to all the characteristics of an object.

In case **change object** isn't active yet, you click the according icon in the menubar. Select the textobject '`Gravur`.

In the dialog box you can change all the characteristics of the text. Set the **text alignment** so that the text appears in the left quadrant below. Also change the textheight from 15mm to 10mm and the used font to *SIMPLE*.

Click the function **change text**, wherewith you can change the content of a textobject. Select now the text '`text diag`' and change it to '`text 45 degrees`'.

5. PrimCAM intern

5.1 Feed / speed calculation

The calculation of optimal feed and speed rates is done automatically in PrimCAM. The following formulas are used to do so.

5.1.1 Feed / speed for drilling tools

$$n = \frac{1000 * vc}{d * Pi} * nk$$

n	[1/min]	Speed
vc	[m/min]	Cutting speed from material library
d	[mm]	Tool diameter
Pi		3.1415...
nk		Correction factor for tool rotation number

$$v = t * n * vk$$

v	[mm/min]	Feed
t	[mm]	Feed per rotation: value from drilling table
n	[1/min]	Rotation number
vk		Correction factor for tool feed

5.1.2 Feed / Speed for milling cutter

$$n = \frac{1000 * vc}{d * Pi} * nk * mk$$

n	[1/min]	Rotation number
vc	[m/min]	Cutting speed from material library
d	[mm]	tool diameter
Pi		3.1415...
nk		rotation number correction factor of tool
mk		material correction factor of tool (HSS=1 normed)

$$a = dz * d$$

a	[mm²]	Cutting area
dz	[mm]	Cutting depth
d	[mm]	Tool diameter

$$f = \begin{cases} a<0.25: f = 2, \\ 0.25<a<1: f = 0.5/a, \\ 1<a<2: f = 0.3/a \\ 2<a: f = 0.1/a \end{cases}$$

f		Correction factor for cutting depth
a	[mm²]	Cutting area

$$v = t * z * n * vk * f$$

$$vz = 0.4 * v$$

v	[mm/min]	Feed
t	[mm]	Feed per tooth per rotation: table value for milling cutter
z		Teeth count
n	[1/min]	Rotation number
vk		Feed correction factor of tool
f		Correction factor for cutting depth
vz	[mm/min]	Feed in Z direction for milling

5.1.3 Special tools

Rough cutter: For a rough cutter, the feed is multiplied by a factor of 2.

Graver:	The feed is multiplied by a factor of 1.5.
	The cutting depth is not considered (f=1)
Screw tap:	The ratio n/v for the pitch of the screw is guaranteed
Digitizer:	Fixed feed from the tool database

5.2 PrimCAM M3 with open postprocessor system

PrimCAM M3 includes an open postprocessor system that allows you to generate code for different machines. You can purchase postprocessors for new machines from your dealer or write them yourself. Postprozessors are stored in your PrimCAM subdirectory \PP and come in two differend kinds: *.PP and *.DLL.
PP's are postprocessors in source form. They are slower than DLL's, but you can edit them with a normal text editor like the one included in PrimCAM. PP's are only supported in PrimCAM M3.
DLL's are precompiled postprocessors. They generate your code much faster than *.PP's, but you can't edit them. They can only be built by PRIMUS DATA. If you have a PP that you think is final, you can send (disk or e-mail) it to PRIMUS DATA. It will then be compiled to a DLL and sent back to you.

5.2.1 Upgrade from PrimCAM M1/M2 to PrimCAM M3

If you've been working with PrimCAM M1/M2 and want to update to M3, you got a password that reprograms your dongle to an M3 version plus optional postprocessors on disk. After copying the postprocessors from disk to your \PP directory and entering the password in PrimCAM, you have access to the machine library that allows you to set postprocessors and parameters for different machines.

5.2.2 Machine library

Now you have to define entries in the machine library for your controls. This is done in the menu **utilities,**

Machine library. Here you have a table containing a line for each of your machines with according postprocessors and settings for serial communication etc. You have to define a new entry for each machine that needs individual nc code.

Insert machine generates a new entry in your machine library. Use the help function F1 to learn what to enter in the individual fields of the dialog box.

If you plan to transfer NC programs to the machine using the serial port of your PC, you have to set the **interface parameters** accordingly. Set the senddelay to 1ms.

Use **select machine**, to tell the system which machine you are working with at the moment. The machine currently selected is displayed in red. If you generate NC code, PrimCAM automatically uses the proper postprocessor for the currently selected machine.

Leave the machine library by pressing end. Confirm the question to save the machine library.

5.2.3 Sending / receiving NC programs

Transfer of NC programs to a CNC machine is accomplished in the **text editor**.

If you choose **send file**, the actually opened file is sent to the machine. When receiving files, you are asked for the name of the file to be saved to disk

5.2.4 Writing postprocessors

Postprocessors are text files ending with the extension *.PP or precompiled files ending with *.DLL. You can only edit *.PP's. To edit a postprocessor, go to the text editor, choose file open and set the file type to Postprocessor (*.PP). The postprocessors follow a C syntax.
If you want to write a new postprocessor, the best thing to do is open an existing postprocessor and save it under a new name and edit it then. After saving the file, it can be selected in the machine library as postprocessor for a machine.

Predefinded system variables

There are predefined system variables for a postprocessor. They are stored in a STRUCT called nc. So if you want to access the actual value for the drill cycle, you write nc.cycle. The following table lists the variable names with their types and gives a short description:

type	name	description
double	a,b,c;	end coordinates for axis A,B,C
int	a_ch,b_ch,c_ch;	flag 0: a,b,c have changed 1: have not changed
int	cool;	flag 0: coolant off 1: coolant on
int	cycle;	drilling cycle 0 = drill F/R 1 = drill F/F 2 = drill high speed peck 3 = drill peck 4 = tap 5 = fine drilling 6 = surface measurement 7 = digitizing
double	d;	distance (used for certain cycles)
double	d1,d2,d3,d4,d5, d6,d7,d8,d9,d10;	floating point values with variable meaning
int	dcode,hcode;	correction index on machine: diameter, length
double	dcorr,hcorr;	correction values on machine: diameter, length
double	delay;	[ms]delay time (drilling cycles)
double	dia;	tool diameter
double	f;	[mm/min] feed
int	f_ch;	flag 0: f has not changed 1: f has changed
int	f_feedmode;	Calculation of feed rate for movements in 3 axis (for example when plunging) Flag 0: XYZ feed rate is calculated from XY and Z components (default) 1: Z feed rate is used for XYZ movements

int	f_plus;	flag 1: output of "+" for real numbers (X+5.000) 0: don't output "+" for doubles (X5.000)
int	i1,i2,i3,i4,i5, i6,i7,i8,i9,i10;	integer values with variable meaning
double	i,j;	center of arc (absolute): i=xcoord, j=ycoord
char	*infname;	input file name(*.PP)
char	*jobname;	name of job
int	jobcount;	count of jobs
int	jobtyp;	job type 1: hole 2: path 3: pocket 4: stop 5: insert 8: overmill 9: circlepocket 10: engrave 11: helix
int	jobtyp;	type of job
double	lead;	[mm/round] lead (Tapping, Helix)
int	n;	line number
int	numbering;	flag 0: don’t output line numbers 1: output line numbers
int	objcount;	Count of objects for the actual job (for example holes to drill,...)
int	origin;	origin index (G54...)
char	*outfname;	output-file(*.NC)
char	*progname;	program name
double	px, py, pz;	starting point; actual coordinates
double	r, sa, ea;	radius, starting angle, ending angle
char	*s1,*s2,*s3, *s4,*s5;	string with variable meaning
double	s;	spindle speed
int	s_ch;	flag 0: spindle speed has not changed 1: has changed
int	side;	Radius compensation 1: center 2: right 3: left
char	*str;	string for inline code
int	subtyp;	job subtype 0: path 2: contour 3: pocket 4: circle 5: helix drill 6: helix tap 7: circle pocket
int	tool,nexttool;	actual tool nr, next tool nr

double	toolchx,toolchy, toolchz;	tool change point
int	toolmat;	tool material 0: HSS 1: HM 2: TIN 3: TICN
char	*toolname;	tool name
int	tooltyp;	tool type 0: roughing mill 1: end mill 2: engraving tool 3: bezel mill 4: radial mill 5: concave mill 6: special mill 7: spot drill 8: drilling tool 9: screw-tap 10: reamer 11: fine drilling steel 12: spot facer 13: core drill 14: digitizer 15: any tool type
double	x,y,z;	end coordinates of axis X,Y,Z
int	x_ch, y_ch, z_ch;	flag 0: x,y,z have not changed 1: have changed
double	xmin,ymin,zmin;	workpiece extension: min. coordinates
double	xmax,ymax,zmax;	workpiece extension: max. coordinates
double	za, zd, zs, zw;	initial level, manufacturing level clearance distance workpiece level

Functions of the postprocessor

A postprocessor description consists of function definitions. PrimCAM calls those functions when generating NC code for the machine. If PrimCAM for example has to output code for a counterclockwise arc, it calls the user defined function arcto_ccw and processes all the steps defined in that function. The variables can be output directly or used to do some necessary calculations. The following part enumerates the functions to be defined for a postprocessor:

function	*variables valid when called*
arcto_ccw	arc counterclockwise: *x,y,z,y_ch,y_ch,z_ch,i,j,r,sa,ea,f,f_ch*
arcto_cw	arc clockwise: *x,y,z,x_ch,y_ch,z_ch,i,j,r,sa,ea,f,f_ch*
begin	program start: *progname,tool, xmin, ymin, zmin, xmax, ymax, zmax,* *toolchx, toolchy, toolchz*
chorigin	change origin *origin*
comment	comment *str*
cooloff	coolant off
coolon	coolant on
cutcompoff	radius compensation off and move to x,y,z *x,y,z,x_ch,y_ch,z_ch*
cutcompon	radius compensation on and moveto x,y,z *x,y,z,x_ch,y_ch,z_ch,side* side=1: center side=2: right side=3: left
cutlineto	move to x,y,z with feed *x,y,z,a,b,c,x_ch,y_ch,z_ch,a_ch,b_ch,c_ch,f,f_ch*
drill	call drilling cycle *x,y,z,x_ch,y_ch,z_ch*
drilloff	end of drilling cycle
drillon	define drilling cycle *x,y,z,x_ch,y_ch,z_ch,cycle,delay,d,za,zr,f* cycle = 0: normal (feed /rapid) 1: ream (feed/feed) 2: high speed peck 3: peck 4: tap 5: fine drilling
end	program end *x,y,z,x_ch,y_ch,z_ch*
feed	feed change *f*
insert	insert string "str" into nc code *str*
moveto	rapid traverse to x,y,z

x,y,z,a,b,c,x_ch,y_ch,z_ch,a_ch,b_ch,c_ch,f,f_ch

speed spindle speed change
s,tool

special special functions (for modules…)

stop machine stop

sethcorr length correction on
tool,hcode,hcorr

setdcorr diameter correction on
tool,dcode,dcorr

toolch tool change with move to x,y,z
tool,nexttool,dcode,hcode,dcorr,hcorr,
dia,toolmat,tooltyp,cool,s,origin,x,y,z,za,zr

tooldef tool definition: called after begin()
tool,nexttool,dcode,hcode,dcorr,hcorr,
dia,toolmat,tooltyp,cool,s,origin,x,y,z,za,zr

wait delay time
delay [s]

Special functions for technology modules

Modules like dispensing (open/close dispensing valves) use the function special() of the postprozessor. Following an overview of the parameters:

Modul	Function	Variables
Dispensing general DOSIEREN	special()	i1=30: valve off i1=31: valve on
	wait()	delay: waiting time in [s]
Dispensing Datron DISPENS	special()	i1=20: DPBASE: called before every contour d9=dispensing quantity i1=21: DPON i1=mode d1=angle of dispensing d2=dock length d3=Z offset dock d4=feed dock i1=22: DPOFF i1=mode d1=angle d2=dock length d3=Z offset dock d4=feed dock d5=length leave d6=Z offset leave d7=feed leave d8=Z plane end i1=23: PONOFF switch off prepressure

Examples of function definitions

```
void begin(void)
 progname,tool: valid variables
{
        nc.f_plus = 1;                          format flag to output "+" for real numbers
        print("\n %s G71", nc.progname);        print program name, followed by „G71„
        nout();                         output line feed and line number
        print("G54");                          select workpiece coordinate system
        nout();                         output line feed and line number
        print(" G00 X100 Y100 Z50");            make rapid move to X=100, Y=100, Z=50
}



void arcto_cw(void)
 x,y,z,x_ch,y_ch,z_ch,i,j,r,sa,ea,f,f_ch
{
        nout();
        print(" G02");
        gcode = 2;

        print(" I");
        printd(nc.i-nc.px);
        print(" J");
        printd(nc.j-nc.py);
        print(" X");
        printd(nc.x);
        print(" Y");
        printd(nc.y);
        if(nc.z_ch){
                print(" Z");
                printd(nc.z);
        }
        if(nc.f_ch){
                print(" F");
                printd(nc.f);
        }
}
```

5.3 Hardware key (hardlock, dongle)

PrimCAM is delivered with different dongles:

- Metal USB dongle for local and network use
- Black dongle for parallel port
- Green-grey dongle for parallel and serial port
- Green-lila dongle for network use

5.3.1 USB-Dongle

The drivers for the USB dongle are installed with PrimCAM automatically. If the installation fails ((PrimCAM starts in demo mode or error message „MPIWIN32.DLL not found“ appears), you can make the installation manually:

- Make sure you are logged in with administrator rights (Win2000, NT, XP)
- Call CBSetup.exe in the primcam directory and choose CRYPTO-BOX USB

If you purchased a network license, do the following:

- Call CBSetup.exe on the server
- copy \network\usb on the installation disk to a server directory (for example c:\cbnetsrv)
- Put cbnetsrv.exe in the autostart or
- Start cbnetsrv.exe manually, activate the dialog by double-clicking it’s icon in the status bar, stop the server and check „Run as service“. Then restart the computer.

5.3.2 Parallel/Serial-Dongle

The black hardlock can only be used with the parallel port. It is found automatically by PrimCAM, searching the ports Hex 378, 278, 3BC in this order. This search order can be changed as described below.
The green-gray hardlock can be used on the parallel as well as on the serial ports. It is found automatically on parallel ports. To use them on a serial port, you have to set the address the dongle should be searched for by setting an environment variable in the file AUTOEXEC.BAT:

SET HL_SEARCH=*[Port], ...*

In Windows NT, you can set an environment variable in Settings/Control/System/Environment:
Variable: HL_SEARCH
Wert: *[Port]*

[Port] is composed of an I/O address in hexadecimal and a port identifier:

I/O address	Meaning
378	Parallel port
278	Parallel port
3BC	Parallel port
3f8	Serial port COM1
2f8	Serial port COM2
3e8	Serial port COM3
2e8	Serial port COM4

I/O address	Meaning
p=parallel	normal parallel port
s=seriell	normal serial port
e=ECP	parallel port in ECP mode

Examples:

SET HL_SEARCH=3f8s
The hardlock is only searched for on the parallel port COM1.

SET HL_SEARCH=378p
The hardlock is only searched for on parallel port 0x378.

5.3.3 Parallel/Serial Dongle Network Version

Using the network dongle (green-lila), you can use one or more PrimCAM licenses on different computers on the network, but only on as many computers as there are licenses.
All the files for the installation of the network dongle are located on the CD in the subdirectory \HLServer or can be downloaded from www.primusdata.com.

Which HL-Server should be used on which operating system ?

- NetWare 3.1x, 4.xx — HL-Server NLM (HLSERVER.NLM) (IPX, TCP/IP) for installation see subdirectory \NLM
- Windows NT/2000 — HL-Server service (HLS32SVC.EXE) (IPX, TCP/IP, NetBios)
- Windows 95/98/ME — HL-Server 32bit application (HLS32.EXE) (IPX, TCP/IP, NetBios)

1. Introduction

HL-Server 32 consists of three parts: A service running under Windows NT, an application running under Windows 95 and an administration tool (HLSADMIN.EXE) to be used on NT and 95. If you want to run the administration tool from then Control panel, copy HLSADMIN.CPL into your system directory and make sure that HLSADMIN.EXE can be found in the PATH. A Network Hardlock icon will appear in the Control Panel; double-clicking it will execute HLSADMIN.EXE.

HLS32 supports IP, IPX and NetBios. IPX can be used by all IPX clients, IP support is provided by the Win16 and Win32 clients. IP must be available though Windows Sockets (WINSOCK.DLL rsp. WSOCK32.DLL). These are included in Windows NT, Windows 95 and MS-TCP/IP for WfW 3.11. Other WinSock implementations should run as well, we have tested CompuServe Internet Dialer and Trumpet WinSock successfully.

2. Installation

Installing HLS 32 service (Windows NT)

1. Start HLSADMIN.EXE
   It runs on WinNT 3.51 with Service Pack #3 or higher, Win95, WinNT 4.0
   It does not run on Win32s !
2. In the tree view, select the machine marked as "local"
3. Click then green traffic light in the toolbar; the HL-Server service will be installed and will start to run
4. Install your Hardlock using the “Add Hardlock” Button in the toolbar

It is not necessary any more to add the Hardlocks to be served manually. Whenever an unknown Hardlock is requested by an application, HL-Server looks for it and adds its automatically, if available.

Alternatively, the service can be installed and started from the command line or a batch file. The HLS32SVC.EXE should reside in the Windows system directory.

```
hls32svc -install
```

hls32svc -start

The HL-Server service is now running and can be used. It will be started at every system boot until it is removed by “Remove” from then “Service” menu.

To find HL-Server via IPX from within other network segments, the SAP agent must be active on the HL-Server machine (Control Panel -> Network -> Add Software -> SAP Agent).

Installing HLS 32 application (Windows 95)

Syntax:

HLS32 <options>

This call may be issued from a command line prompt, or included in the Startup (or Autostart) group.

HL-Server automatically adds up to ten requested Hardlocks. You may also specify module addresses at the command line, if you want to check whether they are properly connected:

HLS32 -m:29809 -m:12345

Please look at

HLS32 -?

for a complete command line help.

Hardlocks can be added to and removed from a running server using HLSADMIN.EXE or HLS32CMD.

Protocols

All available protocols are activated for HLS32 and HLS32SVC version 3.20, including NetBIOS.
They can be switched off and back on by HLSAdmin or by the following methods:

HLS32SVC:

Start once from the NT prompt:

hls32cmd -enable <protokoll> to switch on, or
hls32cmd -disable <protokoll> to switch off

Examples:

hls32cmd -disable netbios
hls32cmd -disable ipx
hls32cmd -enable ip

After this sequence, NetBIOS and IPX are deactivated, only TCP/IP is active. This settings will remain even if the machine is rebooted.

HLS32.EXE

Additional command line paramater:

-comm:<protocol1,protocol2>

Examples:

hls32 -comm:ipx,ip

HLS32 will be started with IPX and TCP/IP activated, but

without NetBIOS.

If the -comm: parameter is omitted, all available protocols will be enabled (IPX, TCP/IP, and NetBIOS)

NetBios support works for all active bindings, so you can use NWLINK-NetBios, NetBEUI, NetBios over TCP/IP etc. concurrently.

Timeout

To avoid the blocking of licences by not correctly terminated or chrased applications or machines, login entries in the HL-Server will be deleted after a defined time without any access. The default value for this time is 15 minutes, as it is for the TSR and the NLM.

For the HL-Server service, this value can be changed by

hls32cmd -timeout <minutes>

The value will be kept in the NT registry an reused after a system restart.

Using the HLS32.EXE, the timeout can be change by the command line switch -t:<minutes>

hls32 -t:60

Valid values range from 1 to 9999 minutes.

3. HL-Server Client for TCP/IP

Search order of protocols:

32 Bit: IP, IPX

IP is searched first because it is usually searched faster than IPX or NetBios.

The search order can be changed using the environment variable HL_SEARCH. HL-Server access may be speeded up by excluding unused protocols from the search.

Example:
set HL_SEARCH=IP,IPX
IP is searched first, if unsuccessful IPX is searched,
never NetBios or local ports

Search order for IP-addresses:

1. environment variable HLS_IP (see below)
2. if no environment variable is defined: DNS or HOSTS is searched for host HLSERVER
3. if no address found: search via broadcast (255.255.255.255) on the local network segment

Environment variables:

set HLS_IPADDR=

one or more IP addresses or names may be specified; if more than one is specified, all of them will be searched; it is not predictable which HL-Server will be used

Example:

set hls_ipaddr=192.9.209.17,luzie.aladdin.de,192.9.209.2,192.9.201.7

Broadcast addresses may be specified also:

set hls_ipaddr=192.9.209.255,192.9.201.255,192.9.207.3

Because IP networks usually show greater differences in packet delivery time than IPX networks, timeout and retry parameters for the clients can be changed in a wide range. Default values are set to be sure of communication using an already running 64kbit route.

set HLS_WAIT=

delay time between two retries (milliseconds)

default	TCP/IP: 1000, IPX: 200 (*)
min	200
max	30000

set HLS_RETRIES=

number of retries before DONGLE_NOT_FOUND is returned

default	5
min	2
max	30

(*) the defaults are different, but SET HLS_WAIT sets then values for IPX *and* TCP/IP !

5.3.4 Problems with the dongle

If the hardlock is not found (PrimCAM runs as demo version), the following information could help:

- USB-Dongle: there is a diagnosis tool in \network\usb\marx probe
- parallel connected:
  - Set the parallel port to SPP or NORMAL (not EPP, ECP)
  - Plug out external devices (like printer) connected to the hardlock
- serial connected: is environment variable set?
- network dongle:
  - actual drivers on client installed?
    - DOS input box, HLDINST -INFO in the primcam directory shows if actual drivers are installed
    - if not: run HLDINST -INSTALL
  - Does network dongle run on the client?
    - set PrimCAM/settings/parameter/dongle to „local" for testing
  - Is HL-Server installed on the server
    - start HLSADMIN.EXE and see if dongle appears
  - Is TCP/IP installed and running?

5.4 Send/receive nc files in the nc editor

The nc editor allows sending/receiving nc files via the serial port of the PC. Baud rate and interface settings can be configured in the machine library for every machine.
For the serial connection of the machine with the PC you need a so called „serial nullmodem„ that connects the according lines. It looks the following:

	Dsub9	**Dsub25**		**Dsub25**	**Dsub9**		
TD	3	2		2	3	TD	Transmit Data
RD	2	3		3	2	RD	Receive Data
RTS	7	4		4	7	RTS	Request To Send
CTS	8	5		5	8	CTS	Clear To Send
GND	5	7		7	5	GND	Ground
DSR	6	6		6	6	DSR	Data Set Ready
DTR	4	20		20	4	DTR	Data Terminal Ready

If you use no handshake or software handshake (XON/XOFF), then only TD-RD, RD-TD, and GND-GND are necessary
For RTS/CTS hardware handshake RTS-CTS and CTS-RTS are also connected.
For DSR/DTR hardware handshake DSR-DTR and DTR-DSR are also connected.

Software handshake uses the following characters for syncronization:

	Dezimal	**Hexadezimal**
XON	17	0x11
XOFF	19	0x13

6. Appendix

6.1 Forms for suggestions / error reports

Suggestions serve as hints to enhance future versions of PrimCAM. Use the following forms and fax them directly to the manufacturer (Fax-No. 0041 55 418 49 50).

Error reports should allow a systematic collecting and forwarding of errors you find in PrimCAM. This is the only way for the manufacturer to prevent known errors in future versions.
If you find a behaviour of PrimCAM that seems to be erroreous and doesn't seem to be caused by a user input error, write the error down to the error list. Fax this list as soon as you have collected some errors.

For debugging, it is often necessary to have the file that produced an error. The error can be reproduced easily this way. If you find an error, you may save the following files to a floppy disk:

?.CAMdrawing file
?.JOB job file
?.NC NC file (The extension may be different, according to your settings)

The question mark stands for the name of the program you are working with at the moment. If you have some files together, also copy the following files to the floppy disk:

*.PP postprocessors (in directory \pp)
*.TAB Feed tables (for drilling, milling, reaming..., in directory \tables)
*.TOL tool database

Send this floppydisk together with the suggestion/error report list to PRIMUS DATA. If you have e-mail, you can also send them to **support@primusdata.com**

PrimCAM suggestion / error report form

PrimCAM version:	
serial number:	
company:	

please copy and fax:	PRIMUS DATA CH: 0041 418 49 50

date	description	OK

7. Glossary

Postprocessor In PrimCAM, the whole drawing and defining of manufacturing steps is independent of the CNC machine finally used to mill and drill. When you finished defining the manufacturing process, you choose a machine with the according postprocessor. The independent machine code is then translated by the postprocessor to a machine specific code.

8. Index