

Manual PrimCAM

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1 License Agreement

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8. Governing law

This agreement shall be construed, interpreted and governed by the laws of Switzerland.

2 Important Information for 3.x

2.1 What's new?

General

- Drawing and manufacturing are now saved in one file *.CAM (*.JOB doesn't exist anymore). Data transfer from PrimCAM 2.x ⇒ 3.x is possible by simply loading old files. Thereby the JOB file is integrated into the CAM file automatically.
- Installing into an existing PrimCAM directory (update) automatically generates a subdirectory containing the old version.
- PrimCAM starts automatically by doubleclicking a CAM file.
- Part library elements are now normal CAM files with special names defining the position in the library (menu, flyout or window).
- 3D data basis (will allow simple 3D functions in the future)

CAD

- Mouse wheel support for zooming and moving the drawing section:
 - rotating mouse wheel ⇒ zoom dynamically (keeps the current mouse position still and zoomes "from there")
 - pressing mouse wheel and moving mouse ⇒ moves drawing section
- Intelligent, configurable snap mode that accelerates the drawing process and shows optically when and where elements are snapped
- Layer deactivation by right click on layer icon in the top bar (left click activates layer)
- Different views of a part, also 3D views

Manufacturing

- helix milling: optional additional circle on z depth
- contour parallel pockets (any shape)
- multiplying parts when generating NC code on up to 100 positions (xy offset and angle)

Software tools

- Tables (jobmanager, tool library etc.) scrollable by mouse, scroll bar or keyboard
- Simulation zoomable

2.2 Installation

- **Windows 7/Vista:** Do not install PrimCAM to the directory \Program Files (x86), \Program Files or \Programme, but instead to a directory such as c:\primcam. The reason for this is that PrimCAM is "Portable Software", meaning the PrimCAM directory contains all settings and config files. This allows to copy the software to another computer or directory, also an USB stick and run it from there after installing the dongle drivers. New windows operating systems prevent, depending on the setting of UAC (user access control), writing to program directories. Saving of config files etc. is impossible this way.
- Make sure you have **administrator rights** during install. This allows installation of the dongle drivers.
- If you don't have administrator rights, you can install the dongle drivers later by executing
 - CBUSETUP.EXE (for the USB dongle) or

- HLDINST.EXE (for the parallel dongle) at the command prompt: *hldinst -install*

2.3 Reprogramming Dongles for 3.x

- Dongles running only old versions of PrimCAM (PrimCAM 1.x DOS, PrimCAM 2.x Windows), can be reprogrammed for PrimCAM 3.x (with costs).
- Old parallel and serial dongles can be changed to USB dongles.

3 Foreword

The demand for flexible, in due time, high-quality, and at the same time **profitable manufacturing** is permanently increasing. With the introduction of new C technologies a significant higher productivity is perceptible not only in construction, but also in manufacturing. Medium and particularly small sized companies are challenged, 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.

3.1 Why 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 nearby machine construction enterprise. The following requirements were essential to this system:

- Simple user interface quickly learned by a mechanic
- acceptable software costs
- It should run on a PC
- Significant time saving for programming

During evaluation it turned out that no system available on the market satisfied our requirements. Therefore we decided to develop a system according to the requirements enumerated above by ourselves.

We profited from various marginal conditions. We were ourselves NC machine users, thus familiar with the practical aspects. In addition we could profit from our former extensive experiences in software development for related projects.

3.2 Using the Manual




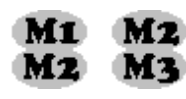
This paragraph briefly shows the structure and gives hints for optimal use of this manual.


- The **system overview** gives a short technical description of PrimCAM and should show the possibilities and capabilities of the system. It is very useful during the evaluation phase or simply helps you to get a rough survey of the features.
- 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.
- The **introduction** briefly explains the most important basics of PrimCAM. Help system, coordinates input, mouse control, and selection of the elements are mentioned here.
- 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 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 unit by unit.
- **PrimCAM intern** explains internal system settings, used formulas and tables for

feed rate calculations and so forth.

- The **glossary** (dictionary) explains words of the manual probably not familiar to everybody.
- Most accesses to a manual are made via **index**. We try to have the index as complete as possible.

PrimCAM's development continues. Function descriptions in the manual are not up to date all the time. Because of that the **reference** to the individual functions of PrimCAM has been moved to the help system, which can be accessed by pressing F1.

    These symbols 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.

3.3 Improvement suggestions / errors

If you have **suggestions for the improvement** of the software or find **errors**, tell us by email to **support (at) primusdata.com**. For errors a test file showing the malfunction is helpful. We try our best to implement new ideas and correct errors in upcoming versions of the software.

4 System overview PrimCAM

The system overview provides a brief abstract of the possibilities and capabilities of PrimCAM. It is useful for people who already have some experience with other CAD/CAM-systems and need a quick survey of the features PrimCAM has to offer, for instance during the evaluation phase.

4.1 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 than 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
- easy and fast to use
- Large capabilities are hierarchically organized and therefore clearly arranged
- Cost effective hard- and software
- Short training time: 1-2 days, not months

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

4.2 User interface

During the development of the user interface we have always considered how the mechanic approaches his task, how we can 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 numerous functions
- Icons for selecting the functions
- 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

4.3 CAD part for creation of the drawing

The CAD part of PrimCAM has got all the functionality necessary for 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; mouse wheel support for zooming and panning
- 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 drawings incompatible to CAM (with overlapping lines or little gaps between the lines which 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)
- Basic dimensioning possibilities for drawings
- Advanced options to change characteristics like color, layer, text height for several objects together (e.g. change diameter of all circles to 10mm)

4.4 CAM part for programming manufacturing

The CAM part features efficient functions necessary for the creation of (2½ D) NC-programs driving 3 axes. 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 one another. 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 into a library together with the manufacturing and can be recalled again in later projects, increasing efficiency. The following CAM features are included:

- Contour and path milling
- Pocket milling with islands

- Pockets either axis or contour parallel
- Different approach and depart methods (tangential, rectangular, along the contour)
- Automatical creation of the approach and depart paths for milling with manual fitting methods
- 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
- Modifications in geometry automatically change tool paths. Objects already programmed can be changed in geometry without losing their manufacturing process
- Various technologies, 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 (cycles defined in the control, for example different drill-cycles like high speed peck drilling)

4.5 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 break-outs for 9, 15 and 25 Pol DSub connectors. With PrimCAM you can insert break-outs for these connector types in the part library together with the manufacturing, including the program for drilling, milling and so forth. If you need these break-outs, they can simply be 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

4.6 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.

4.7 Data transfer

PrimCAM integrates a simple data transfer possibility to CNC machines using the serial port for sending and receiving NC programs.

- 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,...

4.8 NC editor

PrimCAM contains an editor using window technique. It can change or create NC-programs or post-processors.

- ASCII editor for NC-programs and post processors
- Selection of commands through icons
- Delete, insert, move, block and search functions

4.9 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 stage. The calculation of the manufacturing time for a part permits a scheduling of the present CNC-machines.

- 2D and 3D views of the part
- Continuous display of the coordinates, time, feed rate, speed, tool etc.
- Estimation of the total machining time
- Speed for the simulator can be adjusted

4.10 Text engraving / Fonts / Font editor

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

- Possibility to import fonts from external formats and save them in the PrimCAM format (for example AutoCAD SHP/SHX or Windows TrueType)
- 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

4.11 Import / Export functions

The import and export functions of PrimCAM enable data transfer to and from various other systems. For instance, construction drawings, fonts etc. 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 exchange file format
- HPGL Hewlett Packard Graphics Language
- Excellon Format of the company Excellon for their manufacturing machines
- NC-Drill Drilling positions from PCB programs

In development / planned:

- DIN/ISO DIN 66025 standard format for nc machines
- Gerber Gerber Plot Data Format
- Sieb+Meyer

Various export formats are supported

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

4.12 Postprocessors

PrimCAM can create NC code for different machines. Defining the manufacturing steps is independent of 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 (M3)

- 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 available post processors is extended continuously.
- 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 such as "DRILLING WITH D10")

4.13 Hardware requirements

PrimCAM has no special hardware requirements. Today's computers fulfill the requirements normally:

- PC with operating system Windows XP or higher
- Graphics card / monitor with minimal resolution 1280x1024 and minimal 256 colors
For optimal speed, the graphics card should support OpenGL hardware acceleration (most computers have this feature today).
- optionally RS232 port for transfer of programs to NC machine

5 Installation

5.1 Software installation

If you use an installation CD, the installation starts automatically. You can also manually call the file install30.exe.

If you download the program from Internet, execute the file install30.exe

- Make sure that you are logged in as a system **administrator** when installing PrimCAM, because that's the only way the hardlock drivers can be installed.
- Select the language for the installer. The language of the PrimCAM user interface can be switched between any available language later.
- **Windows 7/Vista:** Do not install PrimCAM into the directory \Program Files (x86), \Program Files oder \Programme, but instead to c:\primcam for instance. The reason for this is that PrimCAM is "Portable Software". The PrimCAM directory contains all settings and configuration and can easily be transferred to another computer or USB stick and be run there without any changes (maybe after installation of the dongle drivers on the new system).
Newer windows operating systems prevent, according to the settings in UAC (user access control), writing to the program directories. This makes saving of config files into the program directory impossible.
- Enter the target directory. 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. Old tables, tool- and material libraries and so forth are overwritten with the factory defaults.

- Select the machine controls you use in the following window.
- Now PrimCAM is copied to the computer and decompressed.

5.2 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.

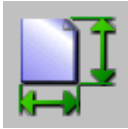
Start PrimCAM from the windows start menu.



Change to the **menu settings**. Put the mouse cursor on the icon for settings, which is at the top of the screen's right border, and click the left mouse button.



Choose the function **settings** by putting the mouse cursor on the icon and clicking the left mouse button.



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 contain a frame or 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 time gaps in which the system automatically saves your work. Time gaps from 1 to 60 minutes are available. The automatic saving is activated by hooking the field behind active. 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 lines**. You can choose the extension in mm for lines and the extension factor for circles.



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.



PrimCAM supports several **languages** for the user interface and help system. You can choose them here.



Finally you can make settings for the **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. Leave the parameter window with **OK**.



If you got PrimCAM M3 with open postprocessor system, you have to install the postprocessors for your machine controls.



This is done in **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 help system 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 inverted color. 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 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.

6 Introduction

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

6.1 Help System

PrimCAM's help system is used by pressing F1 on the keyboard wherever you need help. PrimCAM checks which icon the mouse is on and shows a help text according to the theme.

6.2 Coordinate input

PrimCAM uses a two-dimensional, cartesian coordinate system. 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 relative by entering @ (at symbol) 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 right bottom of the window. 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 any width. Thereby the cursor moves in fixed steps.

If the system prompts for 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**:

Type	Meaning	Sample
A	1st distance	A10
B	2nd distance	B20
D	diameter or distance	D24.95
F	factor (scaling)	F1.5
H	height (text)	H15

L	length of lines etc.	L22
R	radius	R5
W	angle in ° (Grad)	W60
X	X coordinate	X10.5
Y	Y coordinate	Y12
Z	Z coordinate	Z-3.5

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

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

6.3 Using the mouse

When using the mouse the following principle is always applied:

- The **left mouse button** is equal to the **ENTER** key on the keyboard, therefore it's used to confirm a function, to select etc.
- The **right mouse button** is equal to the **ESC** key on the keyboard and is used to abort, finish selection, quit function etc.

6.4 Select / Deselect

A function often requires the selection of various objects for the manufacturing. This can be accomplished by

- clicking each individual object with the left mouse button or
- setting a window over a group of objects.

If each object is selected individually, you have to put the mouse cursor on the object's periphery and click the left mouse button. If there's no object in the mouse cursor rectangle when clicking, PrimCAM opens a window you can drag over a group of objects and confirm the selection with the left mouse button.

- If the window is pulled from the left top side to the right bottom side, all the objects located completely inside the window are selected.
- If the window is set on the contrary from the right bottom side to the left top side, PrimCAM also selects objects which are partially inside the window. If the required objects are selected, you can finish the selection with the right mouse button.



If you have selected too many or the wrong objects, you can switch form **select**



to **deselect**. Now you can deselect already selected objects again by clicking the left mouse button on the selected objects. Another click on the icon switches back to selecting.

7 Tutorial

This chapter contains exercises to get to know the functions of PrimCAM step by step. 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 way you enter the context sensitive help system of PrimCAM.

7.1 Unit 1: The first drawing

Contents:

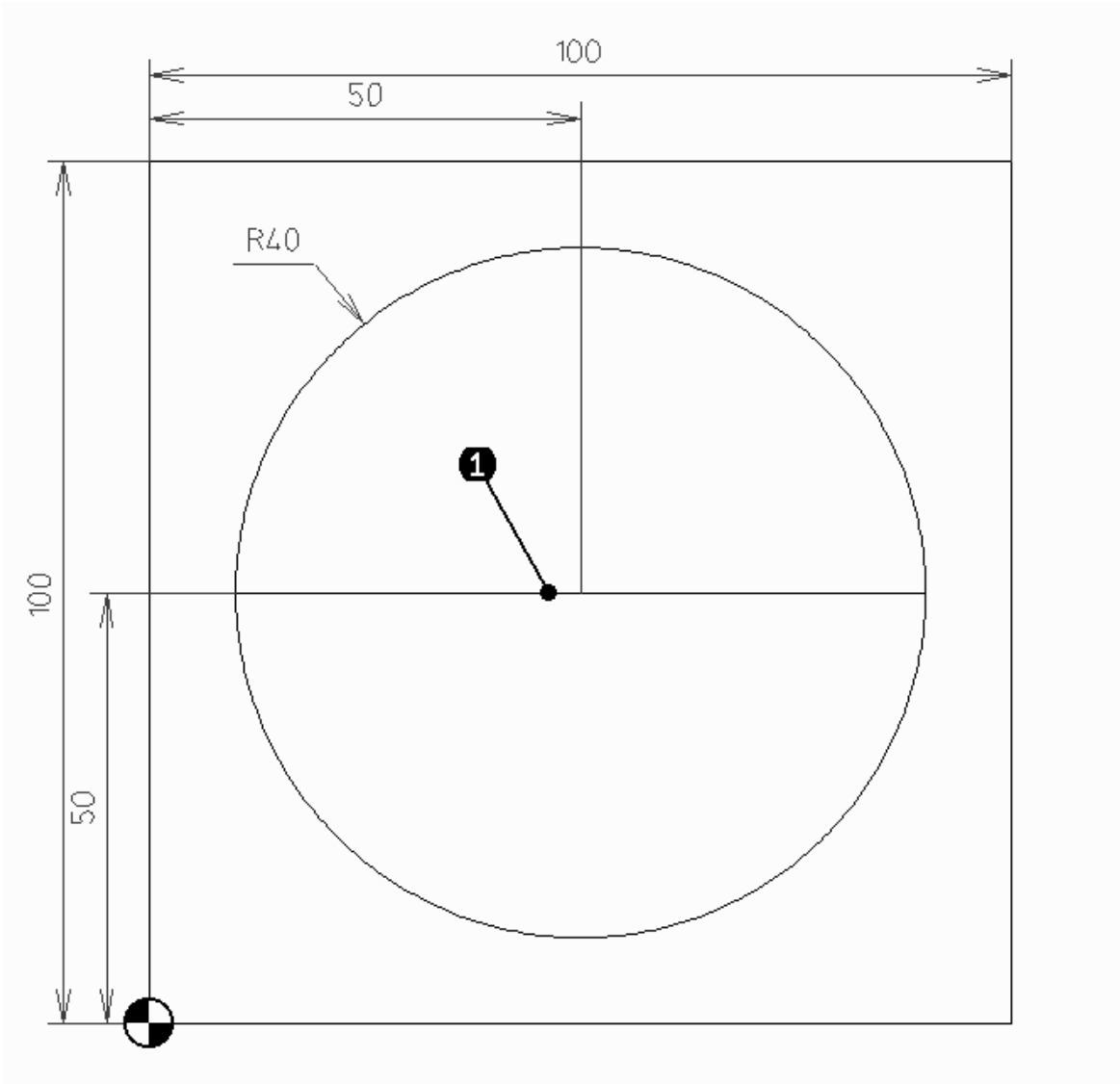
- Get to know the user interface of PrimCAM
- Create your first drawing and save it
- Work with mouse and keyboard
- Use of the helpsystem

Task:

The following drawing consists of

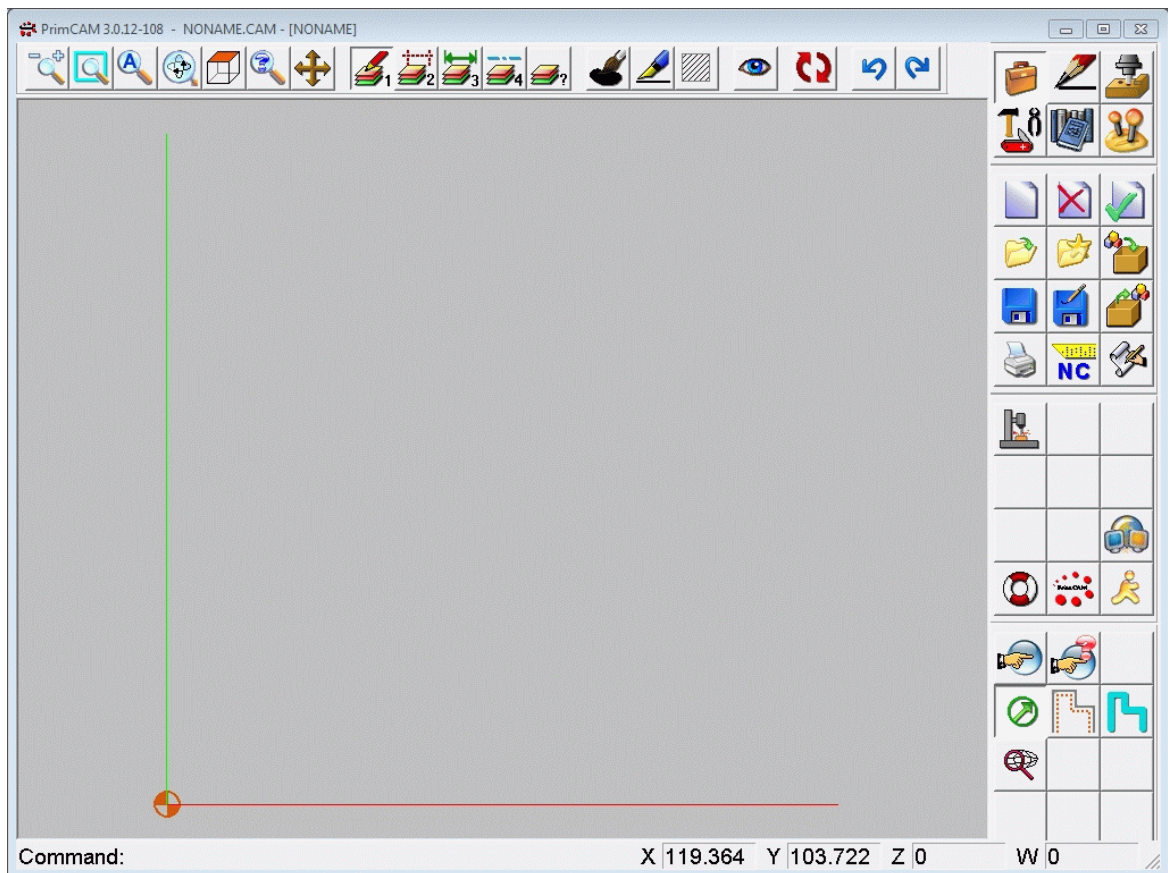
- A square with side length 100 mm, drawn from the origin
- A circle with centerpoint (X=50mm,Y=50mm) and radius 40 mm
- A line from point(X=10mm,Y=50mm) to point(X=90mm,Y=50mm)

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.



7.1.1 The user interface

Start PrimCAM from the windows start menu. The following graphical user interface appears on the screen.



On the right border of the screen you see the **menu bar** with the icons to select the functions. Click the 2x3 icons at the top one after another with the mouse (move the mouse cursor over the icon and click once with the left mouse button). You see how the icons (two blocks of twelve icons) below do change. The six icons at the top represent the **main menu**. Here 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 etc.

The static 4x3 icons at the bottom symbolize functions you will need quite often. For this reason these icons are always present, even if you switch the six areas of the main menu. These static icons represent the **snap** and **switch functions**.

At the top border you see **program version** and **drawing name** of the file you have currently loaded.

The menu bar at the top contains **zoom functions**, **layer switching**, selected **color**, **line and fillstyle** as well as **undo/redo**.

Let's start with our first drawing now.



Click the workbook icon at the left top with the left mouse button (put the mouse cursor over the icon and press on the left mouse button). This switches to the **menu files** where you can start a new drawing.



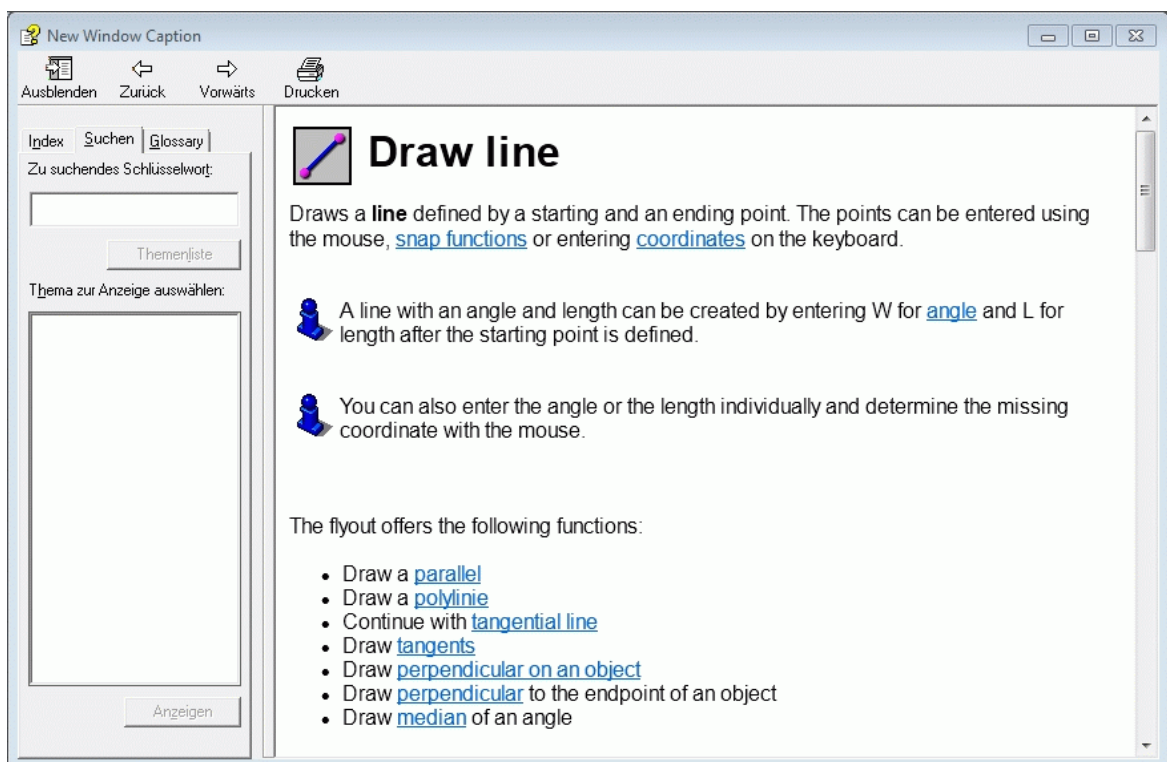
If you click the icon **new drawing**, the drawing area will be cleared, and PrimCAM is ready for a new drawing. At the top border of the screen NONAME appears as a temporary name of the new drawing until you give it its own name.



Click the icon **menu draw** 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 the icon for the function **draw line** now (if you don't see it, look at the explanation of the icon after the next one: **flip page** - it tells how you make it visible). Move the mouse cursor over the icon and press F1 on the keyboard. The help window for drawing a line opens on your screen.



In the help window you see several blue underlined texts. These are **references** to other help texts. Put the mouse cursor over the word [coordinates](#) and press the left mouse button. Immediately the help window for [coordinates](#) opens. Pressing back gets you back to the help for lines.

Have a look at the meaning of different icons on the screen by by moving the mouse cursor over them and pressing F1.

7.1.2 Some drawing functions



In the third line you see the icon for the function **rectangle**. The two magenta points on the icon mean that the rectangle will be created by two points, which are placed diagonally opposite. The blue 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 **flip page**.

Click the rectangle icon and move the mouse into the drawing area. 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 inputs etc. If you enter coordinates on the keyboard for instance, they appear in the box at the bottom of the screen's border.

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', showing you the numbers just entered. You can change these numbers (with the common keys, for instance the key backspace) until 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. 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 the actual drawing color), 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 mouse button, you can again enter its exact coordinates with the keyboard:

`100,100<ENTER>`

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 visible inside the drawing area. By clicking the icon **zoom all** (move the mouse cursor over this icon and press the left mouse button) the drawing is zoomed so that all objects of the drawing do fit into the visible area.



Next we create the **circle** inside the rectangle by defining center point and radius. Move the mouse cursor to the icon for the circle and click the left mouse button. The command line 'Circle in center' asks for the center point of the circle which you enter with the keyboard:

`50,50<ENTER>`

If you move the mouse now, you can see the circle that can be dragged around its center point. At the right bottom border of the screen you see the circle's radius behind the letter R. It changes its value permanently while you move the mouse. The command line expects the Radius (R). You could now determine the radius by clicking the left mouse button. 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 command line 'Line from point'

```
X10Y50<ENTER>
```

It's not relevant whether you use upper or lower case here. A 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 bottom of the screen 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 interpret 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.

7.1.3 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 located 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 **flip page**.

If you have activated the function delete, you are asked to 'Select objects for delete'. Now you have to choose the object you would like to delete. Move the rectangular mouse cursor over the line you want to delete, near point **(1)**. Then press the left mouse button. The line changes its color as a sign for being selected.

You can confirm the question 'Delete selected objects?' by pressing the left mouse button, whereupon the line will be deleted. If you pressed instead the right mouse button, which means abort, deletion will be stopped, and the object isn't selected anymore.

You are still in the function delete and could select other objects to delete. To leave delete, press the right mouse button.



Finally we 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**



and click the function 'Save as'. A dialog box appears where you can determine path and file name. Enter as a name for the file `LEARN1<ENTER>`.



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

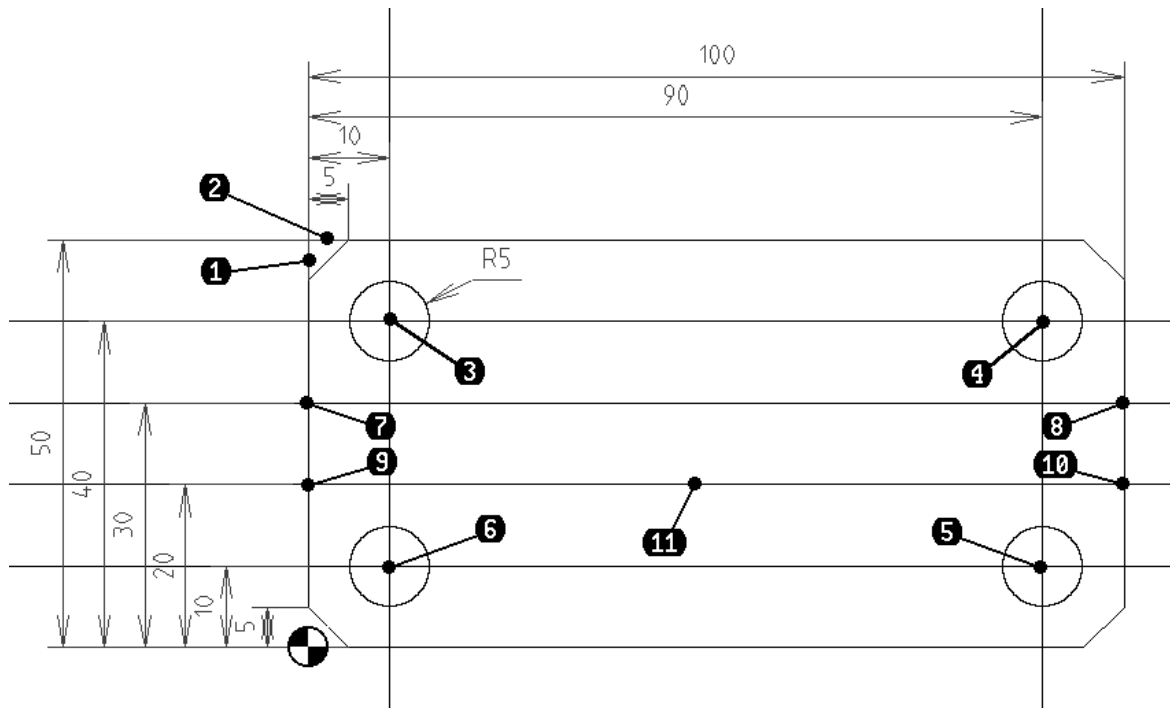
7.2 Unit 2: The first NC program

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.



7.2.1 Creating the geometry

When we work on a piece, our usual procedure is to create the drawing with PrimCAM first. Afterwards the individual manufacturing steps like drilling, milling, engraving etc. 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 mouse button when the cursor is placed on the workbook in the top line of icons. 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 cleared, and PrimCAM is ready for a new drawing. On the top border of the screen 'NONAME' appears as a temporary name for the new drawing, until you give it its own name.



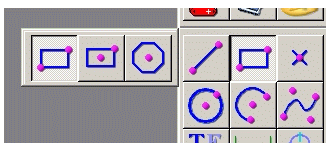
Click the icon **draw** to activate the drawing functions. You can see how the two blocks of twelve icons (each 4x3 icons), which are placed below the main menu, switch over to the drawing functions. The first of the two blocks contains mainly functions for drawing objects like line, circle, text etc.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



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



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 **flip page**.



Click the icon **rectangle**. There appears a flyout that shows special functions for the rectangle function. If you just move the mouse cursor to the drawing area, the special function flyout disappears and the normal rectangle function is active. But you could also activate a special function like 'Rectangle with center' or 'Polygon' by clicking the according function in the flyout.

Enter at the command 'Rectangle from point' the first edge of the rectangle:
0,0<ENTER>

The command line 'Rectangle to point' requests the second edge of the rectangle:
100,50<ENTER>

The rectangle stretches from absolute coordinates X=0, Y=0 to X=100, Y=50.



Use **zoom all** if the rectangle is only partly visible on the screen.



Now chamfers of 5mm x 45° should be placed on all four edges of the rectangle. The function for chamfers is selected by choosing **fillet**. When you click with the left mouse button, a flyout with special functions for fillet 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. Do the same with the other 4 edges of the rectangle.



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 red line that can be moved with the mouse. By entering a Y coordinate or clicking the left mouse button you can place the construction line. As soon as it is placed, it changes its 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 mouse button 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 center point 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. To achieve this, we work with snap functions. PrimCAM features an automatic snap mode that makes special snap functions largely redundant.

As soon as you move the mouse to the point **(3)**, the mouse pointer gets snapped at the point X10Y40, shown by a red magenta mouse circle. Click with the left mouse button now and the center point of the circle is exactly placed to the coordinates X10Y40.

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 mouse button. If the circle has changed its color to magenta, abort the selection of the objects with the right mouse button.



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 center points for the three other circles on intersections of the construction lines. Move the mouse to position **(3)** till the little magenta circle for successful snap is shown. Then confirm with the left mouse button.

PrimCAM asks for the 'Insert point' now. The insert point is the point the copy point will be placed to. We would like to place it on intersections of the construction lines. Click now one after another the three intersections of the construction lines next to **(4)**, **(5)** and **(6)** with the left mouse button. After the circles are placed in the four corners, abort the copy function with the right mouse button.



Finally we have to place the two **lines** in the middle of the piece, where a groove should be milled. 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 the 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, for example LEARN2<ENTER>.

7.2.2 Manufacturing of the piece



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



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

As you haven't defined a work piece yet, the following dialog box opens:

Define workpiece

Material: 4 Aluminium (6)

Raw part

X: 100
Y: 100
Z: 10

D: 100
L: 100

Rotation axis: A B

Surface in Z: 0

Origins
1: 2: 3: 4: 5: 6:

OK Esc

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. Select the block as raw part, not the cylinder. The raw part dimensions are necessary for correct simulation. Our piece is 10 mm thick, so you enter 10 for the Z value. If the work piece is defined, the following dialog box for spot-drilling appears:

Spot-drilling (centering)

Name: SPOT-DRILLING ø10

Tool

Diameter: 10 T: 0

No. 51 Spot drill ø10

Depth Z: -5.2

Initial level: 20

Clearance distance: 1

Work piece level: 0

Dwell time (s): 0.5

Spindle speed:

S: 630 F: 78

OK Esc

Fill in the dialog box according to the above example. If you don't understand an entry in the dialog box, move the mouse to it and press F1. A help window belonging to the entry will appear. 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. NC-programs can be documented automatically with this feature. After choosing **Tool** type and **Diameter**, the corresponding spot drill will be searched in the tool library.

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 in 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** ZS 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 to cut 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 dialog box recalculates the further down placed values 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 **drilling**. Precede analogous to the spot-drill for the selection of the holes. After selecting the four holes the following dialog box appears:

The 'Drilling' dialog box contains the following fields and options:

- Name:** DRILLING ø10
- Tool:**
 - Image of a drill bit
 - Diameter: 10
 - T: 1
 - No. 43
 - Drill ø10 (dropdown menu)
- Depth Z:** -15
- Initial level:** 20
- Clearance distance:** 1
- Work piece level:** 0
- Dwell time (s):** 0
- Cycle:**
 - Five icons showing different drilling cycles: 1. Drill down, 2. Drill down then up, 3. Drill down then up then down, 4. Drill down then up then down then up, 5. Drill down then up then down then up then down.
 - Parameters for the selected cycle: F: 2, F2: 500, D: 2
- Spindle speed:**
 - S: 950
 - F: 110
 - Icons for tool selection, calculator, manual entry, and lock.
- Buttons:** OK (with green checkmark), Esc (with red X), and a circular refresh icon.

Fill in the dialog box according to the above example. 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. Now the holes are programed.

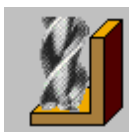


Finally the groove should be milled in the center of the work piece. This is accomplished by the function **mill path**. At the prompt 'Select objects for path milling' you click the lower line next to (11) and abort the selection of objects with the right mouse button. After selecting the line, you are asked to 'Select direction' of the path. If you now move the mouse cursor 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.

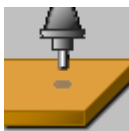
With the **direction pointer** you determine whether the mill moves on the path itself, on the right or left side. Besides you can indicate in which direction the manufacturing should proceed. Pay attention that the indicated direction pointer is placed above the line, and that the direction pointer points to the right side, then you confirm with the left mouse button.

The following dialog box appears:

Fill in the dialog box according to the example above.



Pay attention to choose the **end milling** process. With end milling an offset entered in XY is ignored.



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



Approaching should be in the direction of the path, so that the tool plunges down outside the workpiece and then approaches to it in direction of the line. Confirm the entries in the dialog box with **OK**.

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 deactivated 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 symbols of manufacturing processes. A second click on this icon switches off the direction pointers.



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 removal** 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. After **Saving** 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 translates the programmed manufacturing to the finished NC code for the selected 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 examine 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 a DIN/ISO-compatible machine could look something like this:

```
N0010 (LEARN2)
N0020 (SPOT DRILL WITH ø10)
N0030 T2 M6 (SPOT DRILL ø10)
N0040 G90 G00 G40 G43 G54 H0 D0 X10 Y10 Z20 S630 M13
N0050 G82 G98 X10 Y10 Z-5.2 R1 P500 F78
N0060 Y40
N0070 X90
N0080 Y10
N0090 G80
N0100 (DRILL ø10)
N0110 T9 M6 (DRILL ø10)
N0120 G90 G00 G40 G43 G54 H0 D0 X10 Y10 Z20 S950 M13
N0130 G81 G98 X10 Y10 Z-15 R1 F110
N0140 Y40
N0150 X90
N0160 Y10
N0170 G80
N0180 (PATH END MILLING ø10)
N0190 T10 M6 (MILL ø10)
N0200 G90 G00 G40 G43 G54 H0 D0 X-7 Y25 Z20 S750 M13
N0210 G00 Z1
N0220 X0 Y20
```

```
N0230 G01 Z20 F50  
N0240 X100 F100  
N0250 X107 Y25 Z-3 F61.393  
N0260 G00 Z20  
N0270 M09  
N0280 G40 G80  
N0290 M30
```

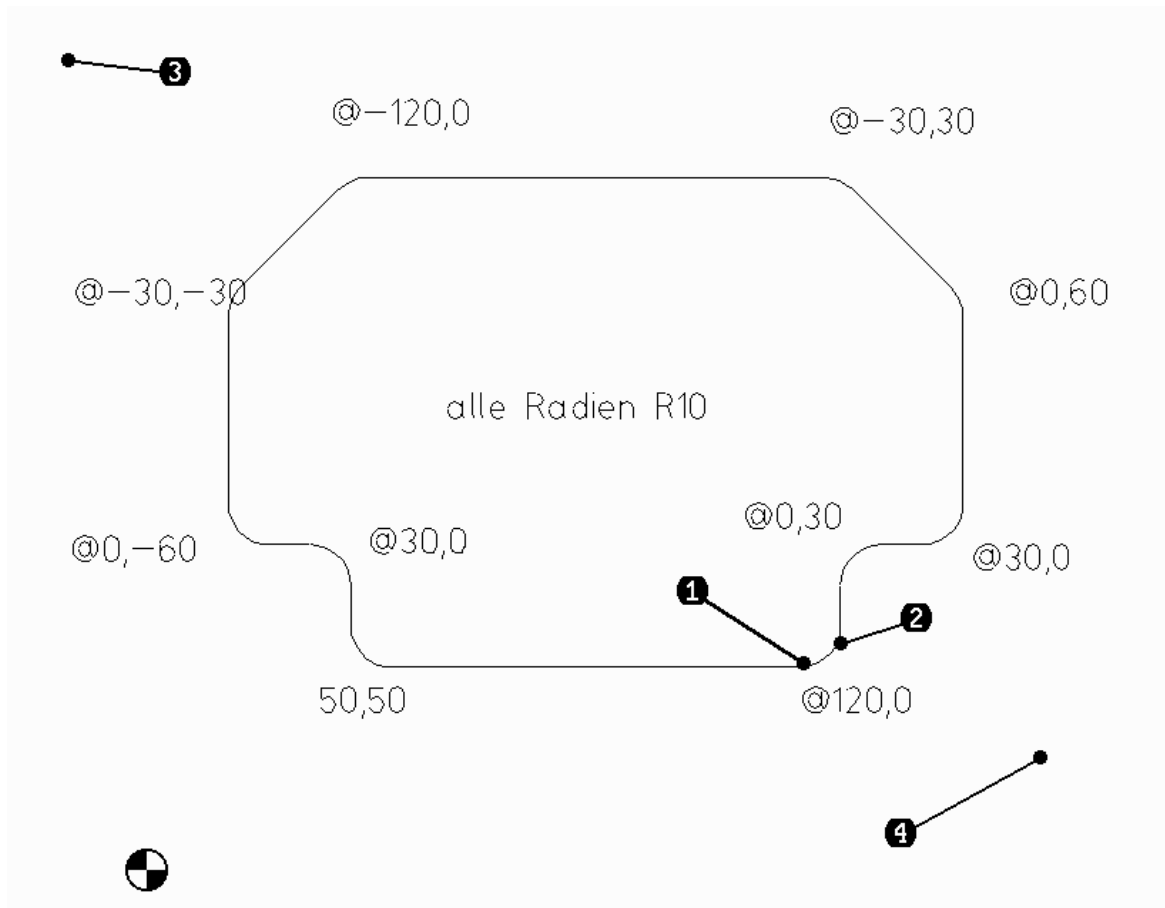
7.3 Unit 3: Coordinate input, contour milling

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.



7.3.1 Draw contour



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



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 view the entire drawing on the screen area or use the mouse wheel to zoom accordingly. 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 move the mouse to the starting point of the contour at X50, Y50 and click the left mouse button as soon as the magenta circle (as a sign that the point was snapped) appears. The right mouse button ends the function **polyline**.



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



Fillet multiple 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' you 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 accordingly. 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 **file functions**.

7.3.2 Manufacturing the contour



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



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 mouse button, and then pull the window down to the right side, next to point **(4)**, so that the frame encloses the whole contour. Press the left mouse button next to point **(4)**. If all lines of the contour have changed their color to magenta, quit the selection by clicking the right mouse button. 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 to and depart from the contour. 'Select direction' for the contour is asked then. 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 to the outside of the contour. Thereupon you can determine on which side to mill and specify up-cut milling or climb milling. Set the arrow inside the contour and choose up-cut or climb. Confirm by pressing the return key or by clicking the left mouse button. The following dialog box opens:

Contour milling ? X

Name: 🔒

Tool

Cutting depth

Offset

Plunge:

Lead In/Out:

Spindle speed:

Rough mill S:	<input type="text" value="3000"/>	Fxy:	<input type="text" value="200"/>	Fz:	<input type="text" value="100"/>	
End mill S:	<input type="text" value="5000"/>	Fxy:	<input type="text" value="500"/>	Fz:	<input type="text" value="200"/>	

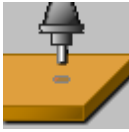
✓ OK

✗ Esc

🔄

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 in only', omitting an approach and depart path.

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



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

7.3.3 Simulation of the tool path



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



The film reel is the symbol for **simulation**. The simulation helps you check the manufacturing steps you have defined.



As soon as you click **RESET**, the simulation begins to run. During the simulation the actual tool coordinates and the machining time are shown. The following functions are available in the simulation:

The simulation speed can be set in the time factor. Time factor 10 means a 10 times faster simulation than real time machining. The minimum for the time factor is 1.



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 simulation continues the simulation after a stop.



Simulation step by step executes a step of the simulation with every click on the icon.



Reset simulation starts the simulation again from the beginning.



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**.

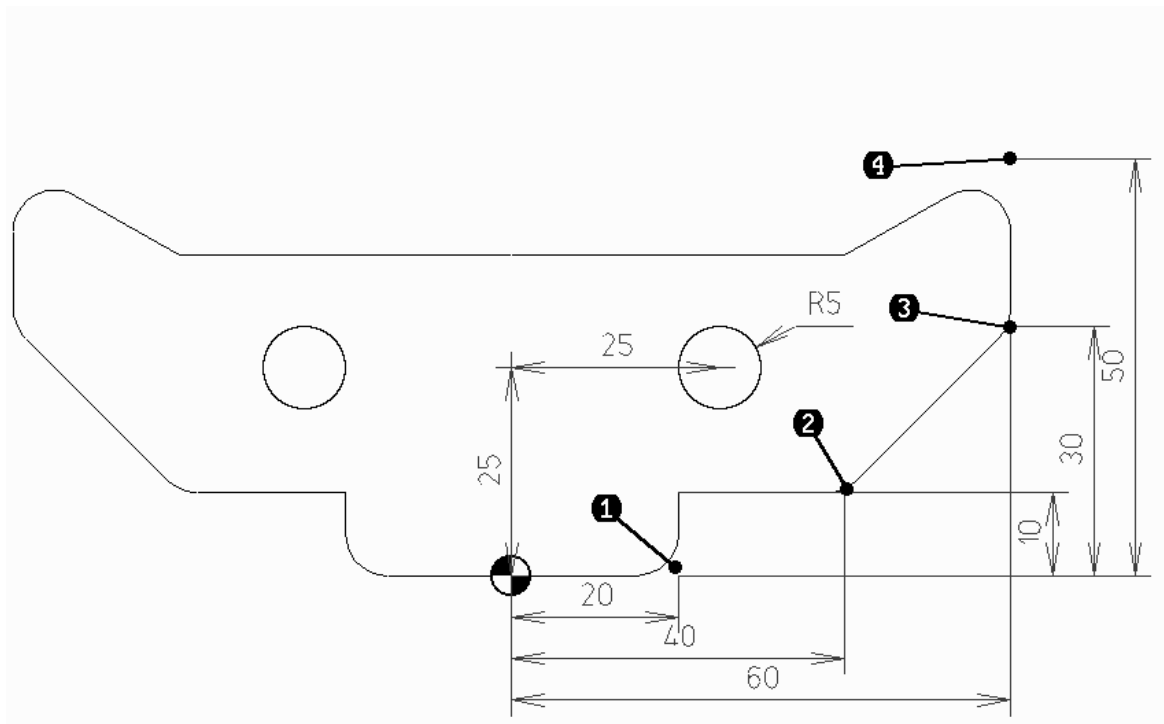
7.4 Unit 4: Relative polar coordinates, mirroring

Contents:

- Input of relative polarcoordinates
- Mirroring

Task:

Create the following drawing, using the relative polar coordinate input. Save the drawing as it is needed for unit 5 and 6



7.4.1 Creating the geometry

Coordinates can not only be entered as X,Y, but also in polar format: 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 polyline icon, we select line and choose polyline from 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. Complete the polyline by pressing the right mouse button.



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 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**. 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 (click at the left top and then at the right bottom). Abort the selection of the objects with the left mouse button. 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 the saved drawing *LEARN4* at any time.

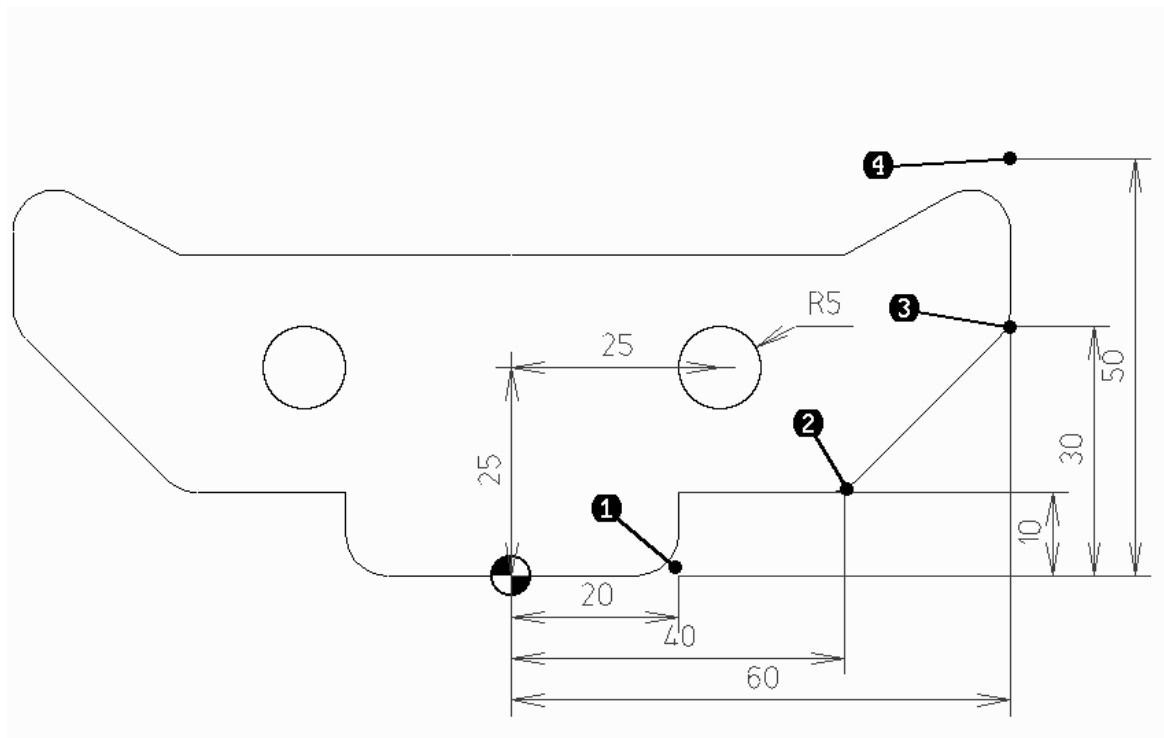
7.5 Unit 5: Pocket milling, zoom functions

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.



7.5.1 Manufacturing: Drill and mill pocket



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



Activate the **manufacture functions**.

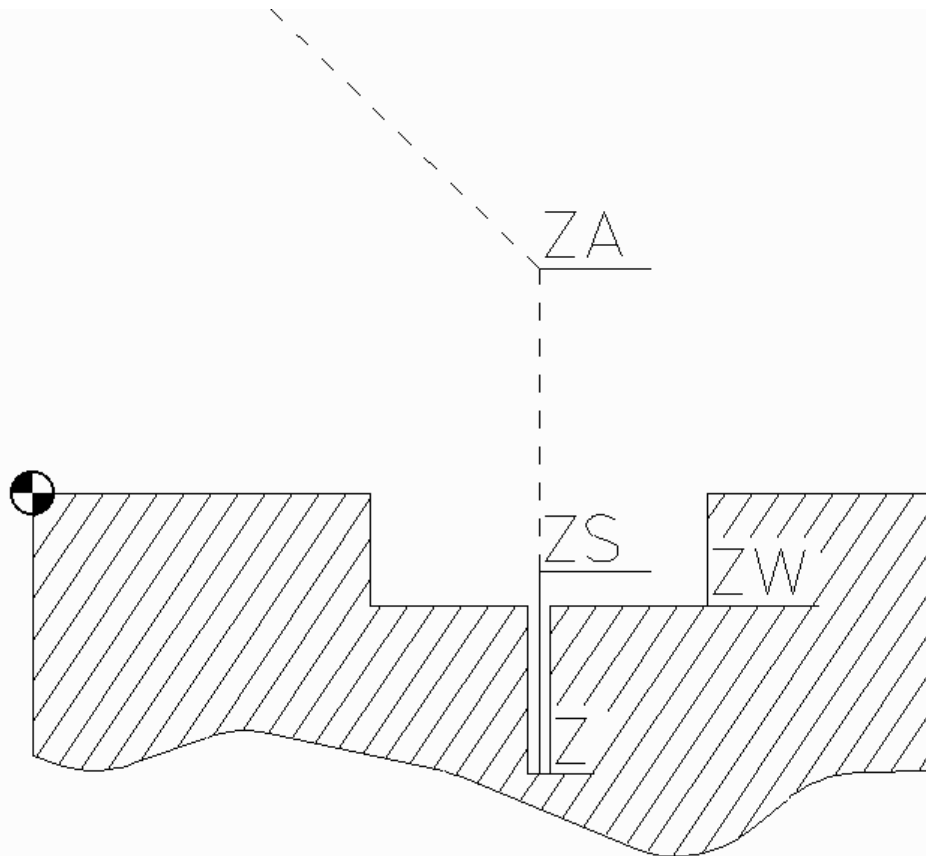


Choose the function **spot drilling** and select the two circles by setting the mouse cursor on the circle's periphery and pressing the left mouse button. After aborting the selection you enter the following data into the menu. If you haven't defined the workpiece yet, the according dialog box will open.

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
Spindle speed S:	3000 rpm
Tool feed F:	200 mm/min

After entering the diameter of 16 mm, PrimCAM searches the tool library for an appropriate spot drill. The name for the manufacturing is automatically suggested by PrimCAM, but can be changed here. 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 the level above which the tool can move freely in rapid traverse.

This level is placed above clampings etc. 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 at about 1 mm. The clearance distance prevents that the tool hits the workpiece surface in rapid traverse because of the inaccuracy of measuring the tool length.

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 down in rapid traverse and starts to cut there. 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:	1 mm
Work piece level:	0 mm

Dwell time:	0 s
Cycle:	high speed peck
Spindle speed S:	5000 rpm
Tool feed F:	300 mm/min



Select the function **mill pocket**. At the prompt '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 magenta, 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.

Click to **select mode** and the mode switches from **select** to




deselect. 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 between select and deselect, until you have selected the correct parts of your drawing. With the right mouse button you abort the deselecting.

Set the starting point for the pocket inside one of the two drilling holes using the snap function. Set the direction pointer so that the pocket is manufactured inside (manufacturing arrow must be inside the contour, not on it or outside). Afterwards the pocket milling dialog box appears where you can enter the parameters as shown below:

Pocket milling

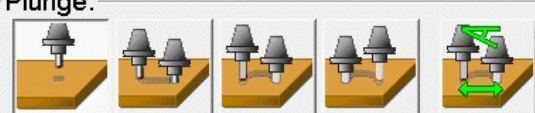
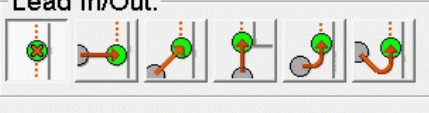
Name: POCKET END MILLING \varnothing 12



Tool

 Diameter: 12 T: 0
No. 50 Finemilling Tool \varnothing 12




Cutting depth
Z: -2 Initial level: 20
Clearance distance: 1
Work piece level: 0


Offset
XY: 0 Z: 0


Plunge:  Lead In/Out: 


Overlay  %: 50 Direction 


Spindle speed:
Rough mill S: 1320 Fxy: 240 Fz: 100
End mill S: 1980 Fxy: 360 Fz: 150

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

 The function **material removal** 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 box 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 flyout bar when you choose zoom various. This zoom function activates the last view of the drawing. By repeated clicking on this icon we can toggle between the two last views.

Remember that you can zoom very easily with the mouse. Move the mouse cursor to the drawing area you are interested in and turn the mouse wheel. Moving the mouse with the mouse wheel pressed moves the visible view 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.

7.6 Unit 6: Jobmanager

Contents:

- Get to know the 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.

7.6.1 The jobmanager



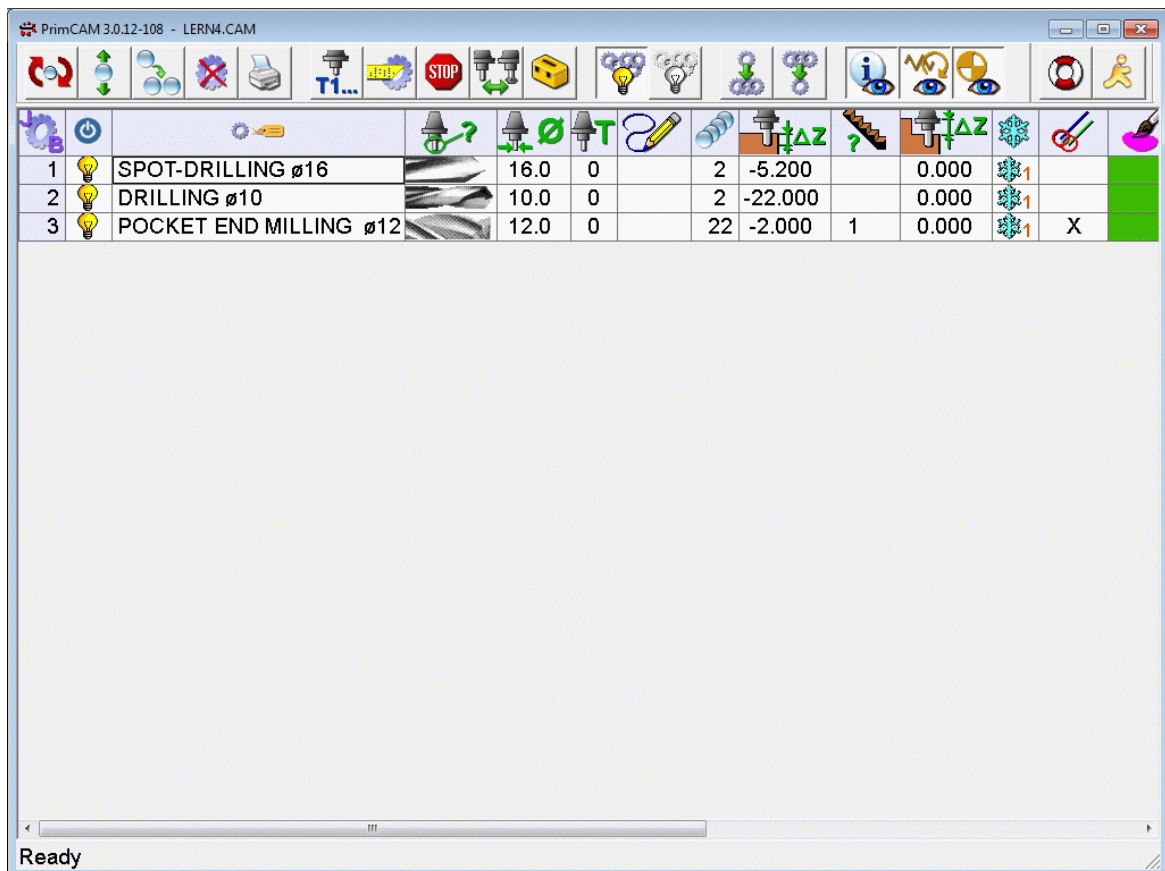
With **load file** you can open the saved drawing LEARN4 from the last unit.



Change to the **menu utilities** and call the **jobmanager**. The jobmanager is a schedule that lists all the steps of a manufacturing process 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 at 'POCKET END MILLING \varnothing 12'. This opens the dialog you already have used during the definition of the pocket. Change the diameter here from 12 mm to 10 mm and confirm it 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 the functions.

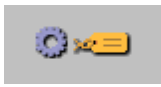
The following data about the individual manufacturing processes are listed in the jobmanager (if all the 'Show' icons with the eye are switched on):



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 relicking you make it active black) again. Inactive manufacturings aren't written to the NC-program.



The light bulb signals active/inactive manufacturing steps. They can also be switched on/off here.



Description of the manufacturing process with the diameter of the tool. A click on the field with the manufacturing opens the dialog box, 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



Count of passes of the manufacturing process



Workpiece level ZW of the job



Coolant on/off. If the coolant is off, the snowflake is gray. Clicking on the icon 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



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).



Spindle speed



Feed XY



Feed Z



Machine origins, at which the job is executed



Manufacturing axes XYZ or 4. Axis

Click now on the field No. of the manufacturing spot-drilling **(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 to the **menu utilities**



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



Go to the **menu file functions**



to generate the NC code, and review 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-drilling) and 2 (drill). 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 mouse button pressed down.
In the simulator you can see that the according jobs are now active again.

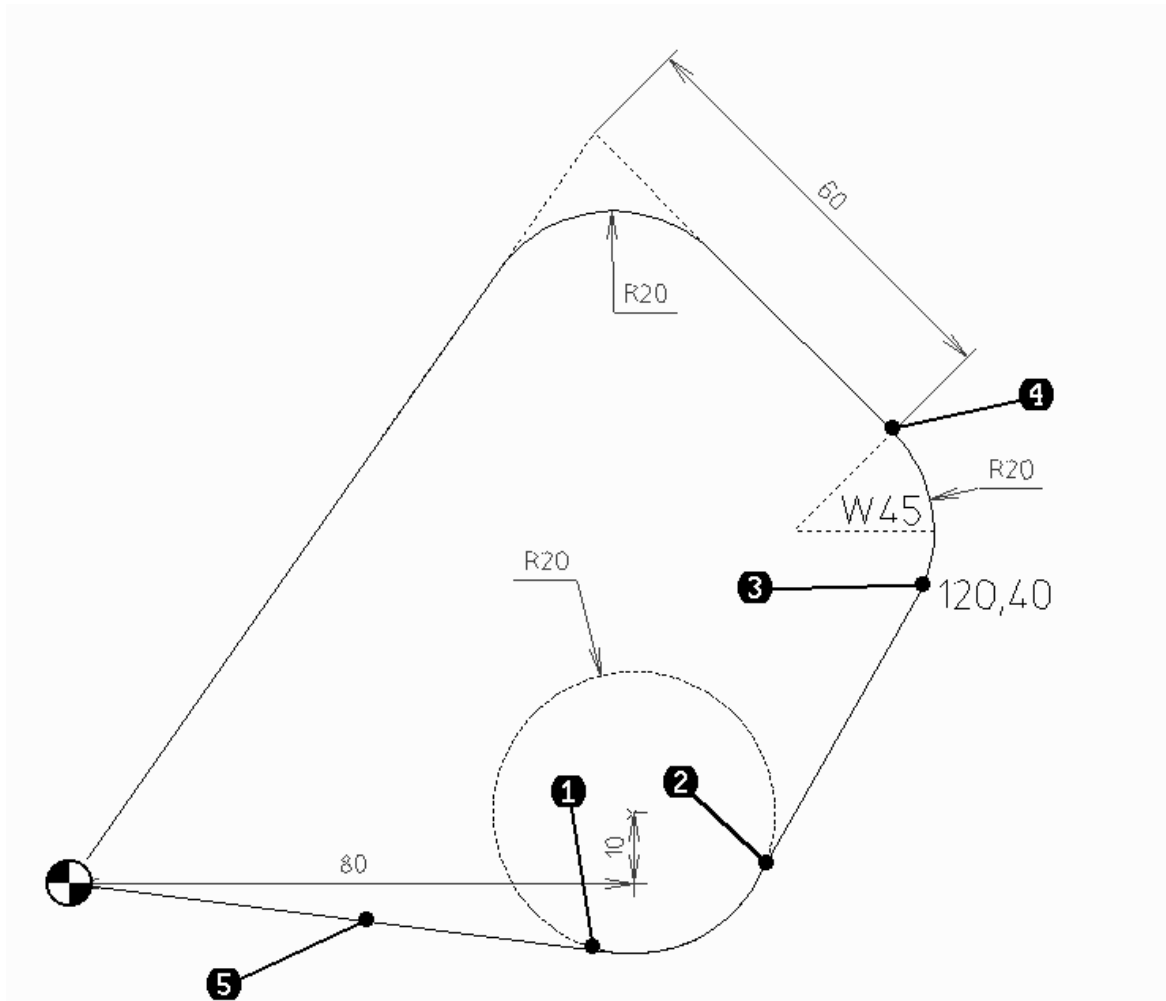
7.7 Unit 7: Rough cutting, end milling, offset

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.



7.7.1 Draw contour

First we create the geometry. You'll notice that it is dimensioned in an unusual way. 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>`



The function **tangent** is in the flyout of line. 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>



Clicking on **bow** opens a flyout with special functions to draw bows. Choose



join tangential. At the prompt 'endpoint from tangential object' you click the end of the previously created line next to point **(3)**. If you now move the mouse, you see a magenta arc stretching from the starting point to the mouse cursor. The arc will be joined to 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 **join tangential line** is visible if you activate draw line. To continue with a tangential line you have to set 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**.



Fillet the upper edge with a radius of 20 mm.



To delete the part of the circle you don't need anymore, you have to **split** the circle first. This means the circle will be broken into individual arcs, one of them can be deleted afterwards. After clicking the function split, the prompt 'select object to break' appears. Select the the circle and at the prompt 'intersection to break' click the two points next to **(1)** and **(2)**, where the lines touch the circle. With the right mouse button 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 the question 'delete selected objects?' with the left mouse button.



Save the file as *LERN7*.

7.7.2 Manufacturing: Rough cutting and end milling



Now the created contour should be milled inside with the function **mill path** using the same tool first for a rough cut cycle with 2 mm offset, then for end milling 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 mouse. 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 dialog box:

Contour milling ? X

Name:

Tool

Diameter: T:

No.

Cutting depth

Z:

=

Initial level:

Clearance distance:

Work piece level:

Offset

XY:

Z:

Plunge:

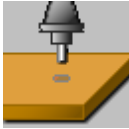
Lead In/Out:

Spindle speed:

Rough mill S:	<input type="text" value="1590"/>	Fxy:	<input type="text" value="260"/>	Fz:	<input type="text" value="100"/>				
End mill S:	<input type="text" value="2385"/>	Fxy:	<input type="text" value="380"/>	Fz:	<input type="text" value="150"/>				



For manufacturing you 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. You see clearly the 2 parallel paths. The first one runs with 2mm offset, the second one follows the contour exactly. Normally 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 this big here that you can better distinguish between the 2 paths.

Finally create the NC-program for the manufacturing.

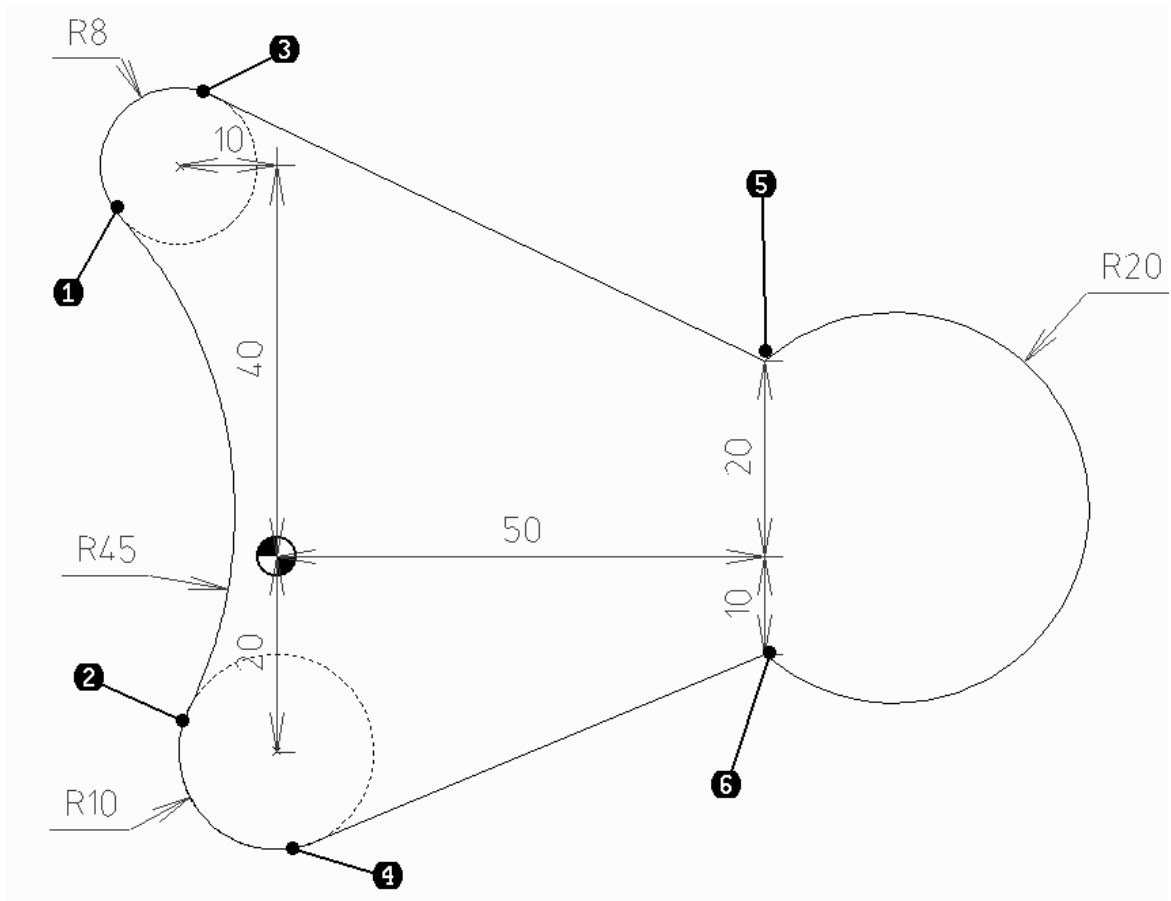
7.8 Unit 8: Advanced drafting functions

Contents:

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

Task:

Create the drawing below.



7.8.1 Advanced circle functions



Create the following two **circles with centerpoint and radius**

$X-10Y40R8<ENTER>$

$X0Y-20R10<ENTER>$



Create a **circle tangent 2 objects and a point**, which touches the other two circles. This special function is located in the flyout of the circle icon. 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 the circle with the mouse. It touches the 2 selected objects near the 2 points you picked and goes through the cursor point. You now have two possibilities to determine the circle. You can either enter a third point of the circle periphery or the radius, depending on your specific needs. 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 possible 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. This is a flyout function of line. 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**, a flyout function of arc, also allows input of two points and the radius. Choose as '1st point for arc' the point next to **(5)** and as '2nd point for arc' the point next to **(6)**. 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 magenta, which all fulfill the required conditions. Select the proper arc according to the drawing at the prompt 'select required arc'.



Now you have to break the circles and delete the unnecessary parts. During **split** you have to pay attention that you always select the object first, afterwards all points where to break. Quit the function with the left mouse button. 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 diameter is broken next to **(1)** and **(2)**.



Delete the pieces of the circle not needed anymore and save the drawing.

7.9 Unit 9: Part libraries / icon editor

Contents:

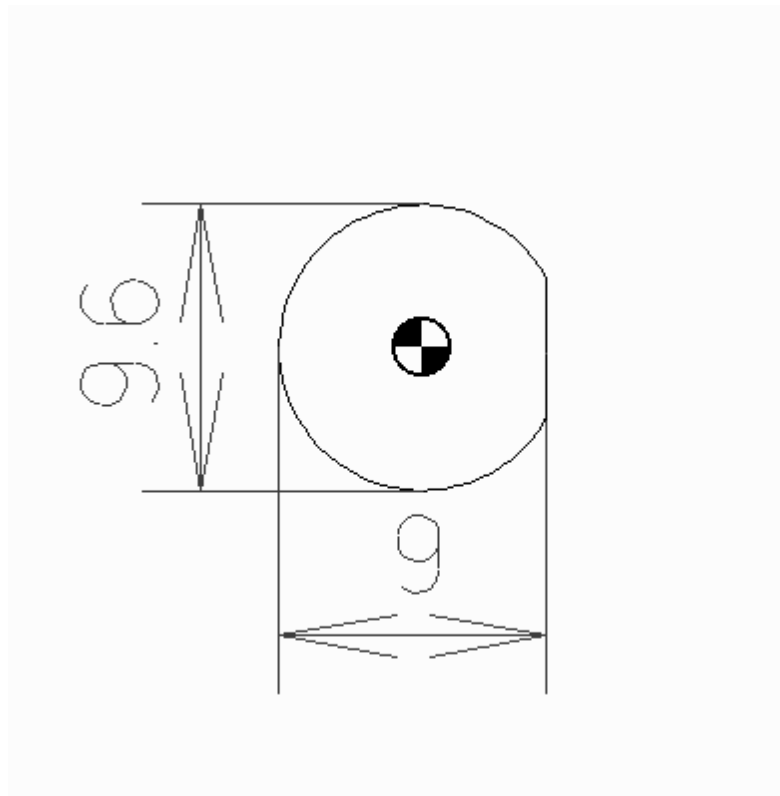
- Insert a simple part with manufacturing into the library

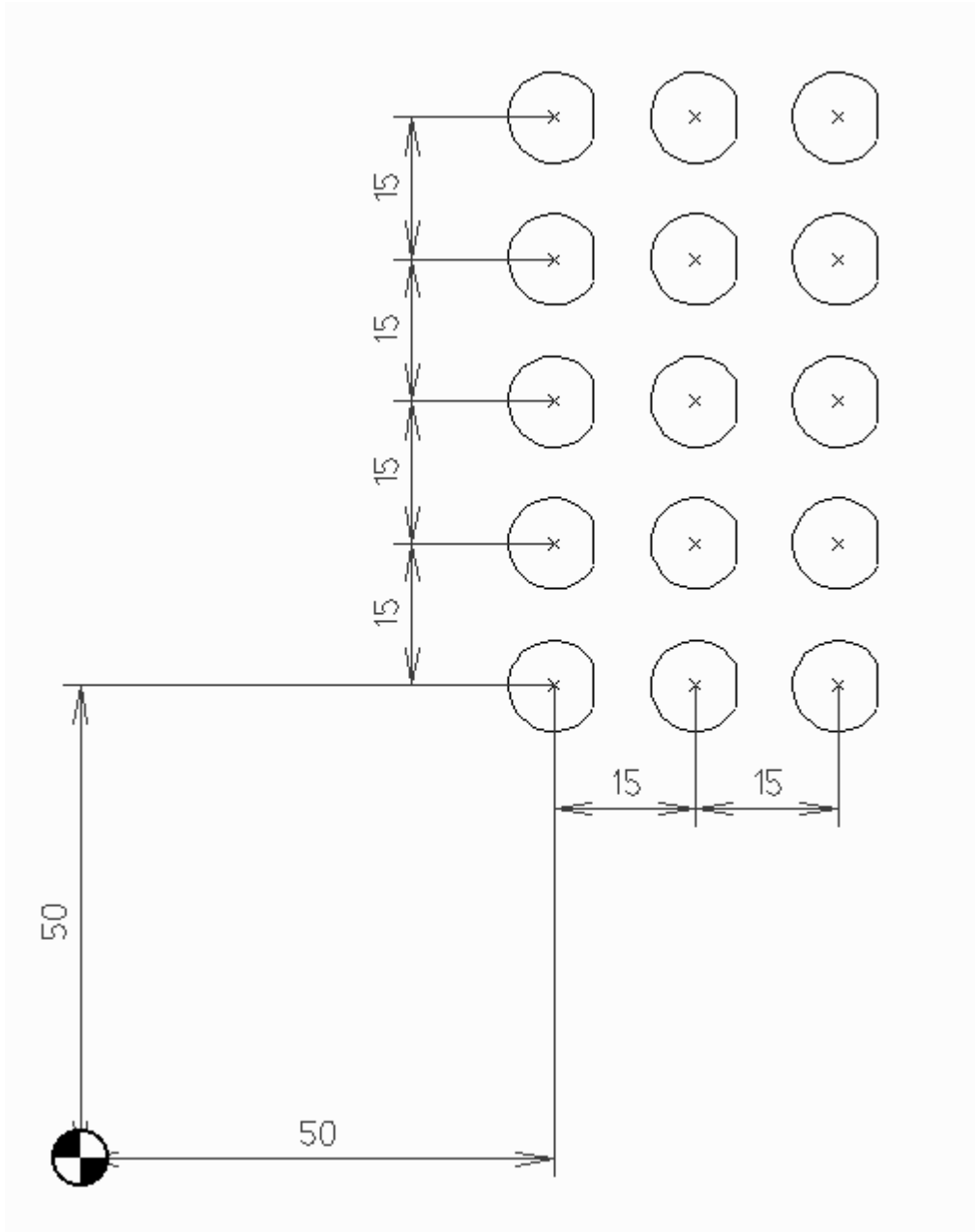
Task:

In the following drawing you see the break-out 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 break-outs.

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





7.9.1 Creating the geometry



We start with the geometry of the connector's break-out. We choose the function circle with center point 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 we calculate here the radius from the diameter by dividing it by 2. We could also have entered the diameter directly 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**.

The first construction line is created at the left periphery of the circle. Move the mouse to the left side at X-4.8 Y0 and click with the left mouse button as soon as the little magenta mouse cursor shows that the quadrant point of the circle has been snapped. The construction line will be placed tangential to the circle.



Now create the second construction line, which runs parallel to the first one in a distance of 9 mm. The function **construction line parallel** appears in the flyout when clicking 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, by 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 mouse button 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.






To cut off the right part of the circle, which we don't need anymore, 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 magenta. Confirm with the left mouse button when the circle is represented in magenta. At the prompt 'intersection to breake', you set the mouse on the two endpoints of the line and confirm with the left mouse button. 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 that aren't necessary anymore for the further procedure should now be hidden first and then also deleted. Click with the right mouse button the icon **Layer construction**. This hides the construction lines. Clicking the left or right mouse button switches between the following states:

	left mouse button ⇒ grey	layer invisible
	right mouse button ⇒ colored	layer visible
	right mouse button ⇒ colored & pressed	layer visible and active



Now the construction lines should be deleted permanently. 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 once, because the construction lines lie on a different layer than the rest of the construction. Change to the menu settings where you find



Table of layers. All the used layers are represented in a table. Here you can simply make invisible the construction lines by clicking the state (on/off) column on the second line 'construction layer'. The lamp switches off, 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 permanently from the drawing. Click the lamp in the state column to make visible the construction lines again. With the function **delete objects of 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 simply click somewhere on the line of the construction layer. Confirm 'should objects in layer construction line layer really be deleted' with OK and quit the layer management with the exit icon.

Save the file, for example as *LEARN9*.

7.9.2 Manufacturing



Let's start with the manufacturing now.



For the break-out it's sufficient to mill the connector's contour with the tool along the periphery, using the function **mill contour**. Select the two objects of the contour and enter as starting point

0,0 <ENTER>.

Thereby the tool plunges in the middle of the circle. Select the mill direction upwards, 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 dialog box enter the following:

Diameter:	3 mm
Depth Z:	-2.5 mm
Steps	1
Initial level:	20 mm
Clearance distance:	1 mm
Workpiece level:	0 mm
Offset XY:	0.2 mm
Offset Z:	0mm
Manufacturing:	Rough milling (1st icon from the left)
Plunge:	Helical motion counterclockwise (4th icon from the left)
Approach/depart:	Tangential line & bow (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 the **material removal visible**. 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 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 below the 1st line.

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

Diameter: 1 mm
Manufacturing: end milling only (2nd icon from the left)

Notice that you don't 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 temporarily off, so that it isn't visible anymore. Click on the 1 under no., and the entry changes its color to grey (by another clicking the entry can be activated again). Then quit the jobmanager by pressing the exit icon. Now you see clearly the mill path and the material removal of the 1 mm tool.



For activating all the defined manufacturing processes again, 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.

7.9.3 Inserting a part into the library



Now the drawn connector's break-out 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.



Choose the function **insert part into library**, located in the bottom block of 12 icons. At 'select place for icon" choose a free place in the 2 blocks of 12 icons above.



Activate the option for **insert geometry and manufacturing**. The two other options would insert either only the geometry or only the manufacturing into the library.



Choose **Icon of drawing from part**. This generates a scaled down picture of the drawing for the icon. 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 by placing a window over it, cancel the selection with the right mouse button 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 we start a new drawing, in which a BNC connector should be inserted from the library and then 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 mouse cursor.

You could rotate the connector for example 90° by entering W90 or scale the connector with F3.5 by a factor of 3.5. With *W0* and *F1* you have the original part back. Enter as insert point *50,50* <ENTER>.



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



With the function **matrix** you create copies of the break-out. They are ordered in figure of a matrix with lines and columns. The function matrix can be found in the flyout of **copy**. 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 mouse button. Fill in the appearing dialog box with the following data:

Number of columns (X):	3
Number of rows (Y):	5
Distance columns (X):	15
Distance rows (Y):	15

Buttons:

Save the drawing under the name LEARN9_2.

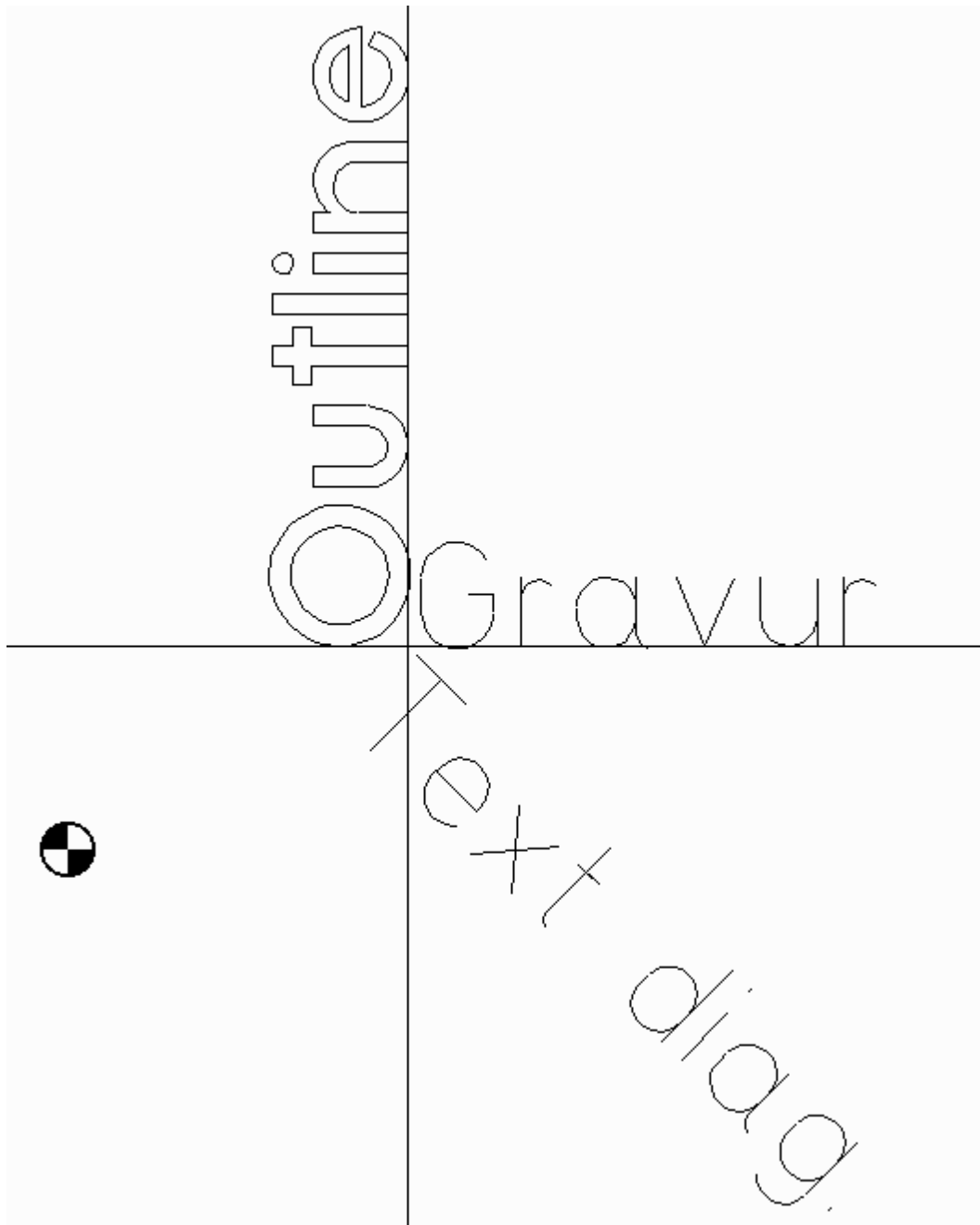
7.10 Unit 10: Fonts and Engravings

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.



7.10.1 Text creation



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 font type (TrueType or PrimCAM font), font name and the actual height (H) are shown in the toolbar 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 and special functions for texts.



Now click the font format field to set the PrimCAM font format. Choose ORION for font name. The oblique F in front of the font name means a PrimCAM font is chosen and not a TT (for TrueType fonts). Enter now

H15 W0

to set the text height 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. 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 text height 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*.

7.10.2 Manufacturing



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

Mode When engraving, it often happens that chips 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. PrimCAM expects 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 itself, but the outline should be milled inside with the function mill pocket. But the text 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.



Go to **draw**. The second page of the lower 12 block of icons shows the function **split**. Use **split text** and select '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. Set the starting point for the pocket into the 'O'. You may have to use the mousewheel to zoom. Fill in the dialog box for pocket milling in the following way:

Pocket milling

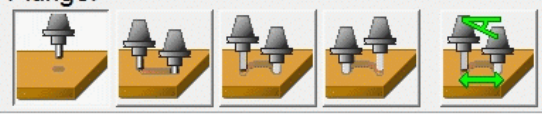
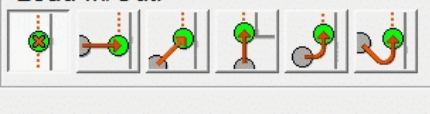
Name: POCKET END MILLING ø1



Tool

 Diameter: 1 T: 2
No. 3 Mill ø1

Cutting depth
Z: -0.3 Initial level: 20
Clearance distance: 1
Work piece level: 0

Offset
XY: 0 Z: 0

Plunge:  Lead In/Out: 

Overlay:  %: 25 Direction: 

Spindle speed:
Rough mill S: 5000 Fxy: 240 Fz: 216
End mill S: 7500 Fxy: 410 Fz: 324

OK Esc



The text 'Gravur' should be modified now. Click the function **change**, which allows you access to all the characteristics of an object. Then select the text object '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 bottom quadrant. Also change the text height from 15mm to 10mm and the used font to *SIMPLE*.



The flyout for change shows the function change text with cursor. Choose the 'Text diag' and change it to 'Text 45 degree'. Then save the file.

8 PrimCAM internals

8.1 Feed / speed calculation

The calculation of optimal feed and speed rates is done automatically in PrimCAM. The following formulas are used in this process.

8.1.1 Feed / speed for drilling tools

$$n = \frac{1000 \cdot vc}{d \cdot \pi} \cdot nk$$

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

$$v = t \cdot n \cdot vk$$

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

8.1.2 Feed / speed for milling tools

$$n = \frac{1000 \cdot vc}{d \cdot \pi} \cdot nk \cdot mk$$

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

$$a = dz \cdot d$$

a	[mm ²] Cutting area
dz	[mm] Cutting depth
d	Tool diameter

$$f = \left\{ \begin{array}{l} a < 0.25 \Rightarrow f = 2 \\ 0.25 < a < 1 \Rightarrow f = 0.5/a \\ 1 < a < 2 \Rightarrow f = 0.3/a \\ 2 < a \Rightarrow f = 0.1/a \end{array} \right\}$$

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

$$v = t \cdot z \cdot n \cdot v_k \cdot f$$

$$v_z = 0.4 \cdot v$$

v [mm/min] **Feed**
t [mm] Feed per tooth per rotation: table value for milling tool
z Teeth count
n [1/min] Spindle speed
v_k Feed correction factor of tool
f Correction factor for cutting depth
v_z [mm/min] **Feed in Z direction** for milling

8.1.3 Special tools

Rough cutter	For a rough cutter, the feed is multiplied by a factor of 2
Engraver	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
Measuring sensor	is moved with a fixed feed from the tool library

8.2 PrimCAM M3 with open postprocessor system

PrimCAM M3 includes an open postprocessor system that allows you to generate code for different machines. The machines are managed in a table (machine library). You can purchase postprocessors for new machines from your dealer or write them yourself.

Postprocessors are stored in your PrimCAM subdirectory \PP and come in two different 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 faster than *.PP's, but you can not edit them. PP's can be compiled into DLL's by PRIMUS DATA.

8.2.1 Upgrade from PrimCAM M1/M2 to PrimCAM M3

If you've been working with PrimCAM M1/M2 and want to upgrade to M3, you got a password that reprograms your dongle to an M3 version plus optional postprocessors.

- Copy the postprocessors to the \PP subdirectory of PrimCAM.
- Enter the command *upgrade. Now you can enter the password and have access to the enhanced functions of PrimCAM M3.

8.2.2 Machine library / settings



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



machine library. Here you have a table containing a line for each of your machines with according postprocessors and settings. 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. Choose a machine name and postprocessor for the machine.



PrimCAM includes a simple machine transfer program for the serial port that runs with a lot of machines. Here you can **set the serial parameters**.



Use **select machine** to tell the system which machine you are working with at the moment. The machine currently selected is highlighted. 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.

8.2.3 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.

8.2.4 Writing postprocessors



Postprocessors are text files ending with the extension *.PP or precompiled files ending with *.DLL. You can only **edit** *.PP's. To **open** a postprocessor, change to the subdirectory \PP and set the file type to Postprocessor (*.PP).

If you want to write a new postprocessor, the best thing to do is open an existing postprocessor, save it under a new name and edit it then. Changing postprocessors requires knowledge of the programming language C. Postprocessors follow a C syntax.

8.2.4.1 Predefined system variables

A postprocessor can use predefined system variables. They are stored in a STRUCT called nc. So if you want to access for example 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;	target coordinates for axes A,B,C
int	a_ch,b_ch,c_ch;	0: a,b,c have not changed 1: a,b,c have changed
int	cool;	0: coolant off 1: coolant 1 on 2: coolant 2 on 3: coolant 3 on
int	cycle;	drilling cycle 0: drill feed/rapid 1: drill feed/feed 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;	[mm] correction values on machine: diameter, length
double	delay;	[ms] dwell time (drilling cycles)
double	dia;	tool diameter
double	f;	[mm/min] feed
int	f_ch;	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) 0: XYZ feed rate is calculated from XY and Z components (default) 1: Z feed rate is used for XYZ movements

type	name	description
		This value can be set for example in the function begin()
int	f_plus;	1: output of "+" for floating point numbers (X+5.000) 0: don't output "+" for floating point numbers (X5.000) This value can be set for example in the function begin()
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	*infile;	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: overmil 9: circlepocket 10: engrave 11: helix
double	lead;	[mm/round] lead (Tapping, Helix)
int	n;	line number
int	numbering;	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 (54 = G54, ..., 59=G59)
char	*outfile;	output-file(*.NC)
char	*programe;	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;	0: spindle speed has not changed 1: spindle speed 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

type	name	description
		4: circle 5: helix drill 6: helix tap 7: circle pocket
int	tool,nexttool;	actual tool no, next tool no
double	toolchx, toolchy, toolchz;	tool change point
int	toolmat;	tool material 0: HSS 1: HM 2: TIN 3: TICN
char	*toolname;	tool name
int	tootyp;	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;	target coordinates for axes X,Y,Z
int	x_ch, y_ch, z_ch;	0: x,y,z have not changed 1: x,y,z have changed
double	xmin,ymin,zmin;	workpiece extension: min. coordinates
double	xmax,ymax,zmax;	workpiece extension: max. coordinates
double	za;	initial level
double	zd;	manufacturing level
double	zs;	clearance distance
double	zw;	workpiece level

8.2.4.2 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 `arco_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. If a function is not defined in the postprocessor, it is not called.

Function	Description / <i>variables valid when called</i>
arcto_ccw	arc counterclockwise <i>x,y,z,y_ch,y_ch,z_ch,i,j,r,sa,ea,f,f_ch</i>
arcto_cw	arc clockwise <i>x,y,z,x_ch,y_ch,z_ch,i,j,r,sa,ea,f,f_ch</i>
begin	program start (for initializations etc.) <i>programe,tool, xmin, ymin, zmin, xmax, ymax, zmax,toolchx, toolchy, toolchz</i> <i>nc.s1 (string: used postprocessor name)</i>
chorigin	change origin <i>origin</i>
comment	comment <i>str</i>
cooloff	coolant off
coolon	coolant on
cutcompoff	radius compensation off and move to x,y,z in a line <i>x,y,z,x_ch,y_ch,z_ch</i>
cutcompon	radius compensation on and moveto x,y,z in a line <i>x,y,z,x_ch,y_ch,z_ch,side</i>
cutlineto	move linear to x,y,z with feed <i>x,y,z,a,b,c,x_ch,y_ch,z_ch,a_ch,b_ch,c_ch,f,f_ch</i>
drill	call drilling cycle <i>x,y,z,x_ch,y_ch,z_ch</i>
drilloff	end of drilling cycle
drillon	define drilling cycle <i>x,y,z,x_ch,y_ch,z_ch,cycle,delay,d,za,zr,f</i>
end	program end
feed	feed change <i>f</i>
insert	insert string "str" into nc code <i>str</i>
moveto	rapid linear traverse to x,y,z <i>x,y,z,a,b,c,x_ch,y_ch,z_ch,a_ch,b_ch,c_ch,f,f_ch</i>
special	special functions (for modules etc.)
speed	spindle speed change <i>s,tool</i>
stop	machine stop
sethcorr	length correction on <i>tool,hcode,hcorr</i>
setdcorr	eter correction on <i>tool,dcode,dcorr</i>
toolch	tool change with move to x,y,z: <ul style="list-style-type: none"> • is called for every tool change • after the tool change the new position (x,y,z) must be approached <i>tool,nexttool,dcode,hcode,dcorr,hcorr,dia,toolmat,tooltyp,cool,s,</i>

Function	Description / <i>variables valid when called</i>
	<i>origin,x,y,z,za,zr</i>
tooldef	tool definition: called after begin() <i>tool,nexttool,dcode,hcode,dcorr,hcorr,dia,toolmat,tooltyp,cool,s,origin,x,y,z,za,zr</i>
wait	delay time <i>delay [s]</i>

8.2.4.3 Special functions for technology modules

Modules like dispensing (open/close dispensing valves) use the function **special()** of the postprocessor. Here is an overview of the parameters for this function:

Modul	Funktion	Variablen / Beschreibung
DOSIEREN(Dispensing general)	special()	i1=30: valve off i1=31: valve on
DISPENS (Dispensing Datron)	special()	i1=20: DPBASE: called before every contour d9=dispensing quantity i1=21: DPON i2=mode d1=angle of dispensing d2=dock length d3=Z offset dock d4=feed dock i1=22: DPOFF i2=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

8.2.4.4 Examples of function definitions

```
void begin(void)
// progname,tool: valid variables
{
  f_plus = 1;           //format flag to output "+" for real numbers
  print("\n %s G71", progname); //print program name, followed by 'G71'
  nout();              //output line feed and line number
  print("G54");       //select origin G54
  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);
  }
}
```

8.3 Hardware key (hardlock, dongle)

PrimCAM was delivered with different dongles, currently only with a USB dongle,

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

8.3.1 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
- Call CBUSetup.exe in the primcam directory and choose CRYPTO-BOX USB

If you purchased a **network license**, do the following:

- Call CBUSetup.exe on the server
- copy \networkusb 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.
- Make sure that an installed firewall doesn't block access to the server. The default TCP port used is 8765. Make sure access to this port on the server is possible.

8.3.2 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:

- In AUTOEXEC.BAT as follows:
SET HL_SEARCH=[Port], ...
- Start - right click Computer - Properties - Advanced system settings - environment variables:
Create a new variable *HL_SEARCH* with value for example *3f8s*

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

I/O-Adresse	Bedeutung
378	Parallel port
278	Parallel port
3BC	Parallel port
E050	Example for a parallel port on a PCI expansion card (Adress can be obtained under: Start - System control - System - Device manager - LPT cards)
3f8	Serial port COM1
2f8	Serial port COM2
3e8	Serial port COM3
2e8	Serial port COM4

Portkennung	Bedeutung
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.

8.3.3 Parallel/Serial Dongle Network Version

Using the network dongle (green-magenta), 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 ?

- Windows NT/2000: HL-Server Service (HLS32SVC.EXE)
- Windows 95/98/ME: HL-Server 32bit application (HLS32.EXE)

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 resp. 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 (or alternatively: "hls32svc

-stop" followed by "hls32svc -remove").

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 <protocoll>
```

to switch on, or

```
hls32cmd -disable <protocoll>
```

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 parameter:
-comm:<protocoll1,protocoll2>

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 crashed 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 and reused after a system restart.

Using the HLS32.EXE, the timeout can be changed 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.fast.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

8.3.4 Problems with the dongle

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

- **USB dongle:** : there is a diagnosis tool in `\network\usb\marx probe`
- **USB network dongle:** is the service `cbnetrv.exe` running?
- **parallel connected:**
 - Set the parallel port to SPP or NORMAL (not EPP, ECP)
 - Plug out external devices (like printer) connected to the hardlock for testing
- **seriell connected:** is the environment variable set?

- **parallel network dongle:**
 - actual drivers on client installed?
 - open input prompt, call *HLDINST -INFO* in the PrimCAM directory and check if actual drivers are installed
 - if the drivers are not actual: call *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(actual version)?
 - start HLSADMIN.EXE and see if dongle appears
 - Is TCP/IP installed and running?

8.4 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 'serial nullmodem' that connects the according lines. It looks the following:

	Dsub9	Dsub2 5		Dsub2 5	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 synchronization:

	Dezimal	Hexadezimal
XON	17	0x11
XOFF	19	0x13

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