

## 1 INTRODUCTION

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The *FMSsoft Manual* provides you with information about using a FMSsoft package.

It also includes a section on the basics of AutoCAD.

The chapter entitled Tips & Tricks will hopefully provide the answers to most of your questions.

The Command reference explains the general FMSsoft commands.

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## 2.1 FMSsoft documentation

The FMSsoft documentation is composed of various different parts:

- FMSsoft Installation Manual
- Package-specific manual, e.g. MASsoft special
- Online help
- Readme.doc

**Note:** you will find the most recent alterations to the FMSsoft information and instructions in the *readme.doc* file.

## 2.2 Typographical conventions

Text elements which the computer indicates for you or which can be entered by you are displayed in different fonts and styles so that they stand out from the rest of the text.

### Formatting conventions

<i>Text element</i>	<i>Example</i>
The Return key	↵
Menu options, prompts and other text displayed on the screen are shown in Times New Roman	Command: CHOOSE Module name ?/<default>: .
Text which you enter is shown in bold face Times New Roman	Enter <b>fms_module</b> in the command line
File names, directory names and comments are shown in italics	<i>c:\mas50\drawing.dwg</i>

### **3 USING FMSSOFT**

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This section explains how to work with FMSSoft. The most important commands are described in the context in which they are used.

For more detailed information about each command, please see the Command reference section.

### **3.1 Managing your files**

It is a good idea to organise each project and its drawing files in its own directory, to ensure that your projects do not become confused.

You can either use a commercially available drawing manager (see Autodesk application catalogue) or organise your projects yourself. You will find information about creating and editing directories in your operating system manual.

### **3.2 Use online help**

The FMSsoft online help is the simple and easy way to find information about your current task, a function you are not familiar with or a command you would like to use.

### **3.3 Generating modules**

#### **3.3.1 *Selecting modules***

All modules must be specified in detail before they are inserted in a drawing, using the so-called physical features such as length, width, delivery unit etc.

For this reason, there are dialogue boxes for each module, in which the physical features of the module can be specified. The dialogue box informs you about all the possible structural shapes, and also provides you with the order number and the price of each part.

These dialogue boxes are opened with the CHOOSE command, either manually or by clicking the module name on the menu tablet or in one of the pull-down menus. If your menu tablet is not yet configured, you can find out how to configure it in the FMSsoft installation manual

#### **3.3.2 *Inserting modules***

##### **3.3.2.1 FMS\_MODULE**

You insert modules with the FMS\_MODULE command. You can, however, only insert modules if their physical features have been specified, or if they do not require specification of their physical features.

Once you have entered the module name, you may be able to digitise the length, breadth and/or thickness.

The new value can be entered by digitising or entering a value on the keyboard. This value does NOT alter the entry in the Choose mask. The new value is only valid temporarily for the current insertion process.

### 3.4 Editing modules

At this stage you can already adapt existing modules to suit your own requirements or generate your own modules and allocate parts list information to them.

If you want to:

- Change the physical features of an existing module, for example select a different structural shape:  
Command: **CHMOD**
- Change the length and/or width of one or more existing modules:  
Command: **MODSTRETCH**
- Edit an existing module, make constructional alterations, e.g., inserting a crossbar on a table:

Command: **DISASS** to disassemble a block and save the parts list information

Now make your changes, then use

Command: **JOIN** to reassemble the elements and add any parts list information which was previously stored, possibly altering the parts list information with: **DDATTE**

Caution: You can only ever disassemble one module with the DISASS command, otherwise conflicts arise with the parts list reference. Further modules must be broken down with EXPLODE, but the parts list information will be lost when you use this command.

- Generate your own module and add parts list information to it:

Draw your construction element

Command: **MOBLOCK** to form a block and enter the parts list information

Command: **WBLOCK** is optional. You should use this command if you want your block to be available in other drawings. It is saved on hard disk with this command.

- Change the parts list information of an existing module:  
Command: **DDATTE**

#### 3.4.1 Changing the parts list information: CHMOD

Once the object you wish to change has been digitised, the choose dialogue box for this module is activated. You can make any changes you like in this dialogue box. As soon as you close the dialogue box with "OK", a new module replaces the old one. If you close the dialogue box using the Cancel button, the module remains unchanged.

### 3.4.2 **Changing the module size: MODSTRETCH**

Modules can be stretched with the MODSTRETCH command, although only modules which are actually available in various lengths and widths can be modified. The modules should not be in mirror image.

The modules should only be stretched orthogonally if possible, otherwise you may get surprising results. The best way to do this is to activate the Ortho mode in the drawing tools dialogue box.

### 3.4.3 **Changing parts lists language: LANGCHG**

The command **LANGCHG** realises the translation of parts list information from one language into another.

### 3.4.4 **Your own blocks: MODBLOCK**

If you have made your own construction elements, it is advisable to add parts list information to them, so that one of the great advantages of 3D construction -a consistent parts list - is maintained.

The **MODBLOCK** command in FMSsoft is designed for this purpose. It adds parts list attributes to the graphics and forms a new block.

Once the special module has been created, it can be inserted in your drawing at any point you wish using the **INSERT** command.

### 3.4.5 **Saving your own blocks on hard disk: WBLOCK**

You have created a new FMS module yourself with the MODBLOCK command and now wish to store this on hard disk for use in other drawings in future. For this purpose you should use the AutoCAD WBLOCK command.

### 3.4.6 **Editing parts list information: DDATTE**

You wish to edit the parts list information linked to the blocks.

Caution: If you enter incorrect information here, the parts list may not be drawn up correctly.

### 3.4.7 **Breaking down modules: DISASS**

The **DISASS** command breaks a module down into individual parts (or “disassembles” it) and stores the parts list information for this module.

Once the module has been selected the individual elements can be edited, i.e., copied, moved, erased, etc. or new ones added. The command is useful if you wish to disassemble something like a table frame which has been inserted as a module.

Caution: You can only ever disassemble one module with the DISASS command, otherwise conflicts arise with the parts list reference. The following message is displayed:

Module perhaps already disassembled! Continue? <Yes>/No:

Further modules can be disassembled with the EXPLODE command, but their parts list information will be lost.

### 3.4.8 *Assembling modules: JOIN*

A new block is allocated the parts list information of the module which was previously disassembled with the **DISASS** command.

Modules which have been disassembled should be reassembled before you exit the drawing editor, otherwise their parts list information is lost.

## 3.5 Working with layers

Several layers are set by FMSsoft. Parts of elements are therefore found on certain layers, for example there are special layers for the main graphics, the connection nodes and for dimensioning. If you insert modules with the FMS\_MODULE command, the elements will be automatically allocated to the correct layers.

### 3.5.1 *Altering the module layer: MODLAY*

The layer structure of the FMSsoft module can be adapted for any layer environment with the **MODLAY** command. The layer names may be up to 19 characters long. If the layers you wish to use are not yet available before the **MODLAY** command is selected they are generated. Any colour can be used for each layer; these colours must then be selected.

The FMSsoft layer names can be changed at any time, for example, entire project parts can be generated on other layers.

### 3.5.2 *Moving individual modules to other layers: CHANGELAYER*

The **CHGLAY** command is used to change the layer of a set of selected elements.

The main graphics for all FMSsoft elements are indicated in the '0' layer, i.e., the layer into which a block of this kind is inserted determines the colour of the element. If you insert your own blocks in another layer using this command, they may remain the same colour.

### 3.5.3 *Setting the current layer: SET LAYER*

With **SETLAY** the current layer can be changed simply by clicking on an element that is in the layer you want to go into.

## 3.6 Texts and dimensioning

Dimensions and texts can be inserted in three-dimensional drawings. To do so, a suitable UCS must be selected before elements of this kind are generated. So that the individual dimensioning and text layers can be faded in and out separately from one another, the texts and dimensions must lie on different layers. Separate layers in a viewport can be frozen and thawed with the MPLAYER command, so that texts and dimensions can only be seen in the viewports you wish them to appear in.

This is important for automatic preplotting with the PREPLOT command, or with the FORM command.

### 3.6.1 ***Automatic setting of UCS and layers: SWITCH***

The SWITCH command is used to set a UCS and a layer for dimensioning or labelling in three-dimensional drawings. You must use this command when you want an automatic printout with PREPLOT or FORM.

### 3.6.2 ***Changing the layer manually: SETLAY***

With this command, you click on an element with the pointer to move to the layer the element is in.

If you want to link a text directly to an object so that the label is visible from all directions, set it on the same layer as the object.

## 3.7 **Parts list and calculation of prices**

FMSsoft automatically draws up a parts list which includes all the parts you have inserted in a drawing. Even your own parts will appear in the list, provided you have given them parts list information using the MODBLOCK command.

### 3.7.1 ***Changing price information's: PRICEEDIT***

Inside the price editor the user can adapt the prices of FMSsoft modules according to his needs.

### 3.7.2 ***Actualise the prices of existing projects: PRICEUPDATE***

With the command **PRICEUPD** you can actualise the prices of existing projects after a price update.

### 3.7.3 ***Allocating item numbers: ITEMNO***

You can allocate item numbers to different parts, which will also appear in the parts list.

In the parts list, the parts are arranged by item number in ascending order. The parts without an item number are arranged at the end of the list in alphabetical order.

### 3.7.4 ***Module ID for the parts list printout: ASSEMBLY***

The ASSEMBLY command lets you alter the ID character of a group of modules. The modules in an assembly are filtered for the parts list printout by the ID character.

If this ID character is given for creating the parts list, only the parts list for this assembly is drawn up.

### 3.7.5 *Editing parts lists: PLIST*

The typical method of creating a parts list is as follows:

- decide which default file you wish to use
- insert the accessories (if necessary): **ACSE...**
- fill in the parts list heading: **HEADING**
- generate the list: **CREATE**
- print the file (in file and CDF format) and insert it as a block

## 3.8 Layout and plotting

With FMSsoft you are not limited to plotting your current view, as is the case with AutoCAD. FMSsoft also provides you with DIN drawing forms that you can use to arrange your elements quickly and easily and print them out from various viewpoints next to each other. These commands can save you a great deal of time.

There are two FMS commands for this purpose: FORM and PREPLOT:

- FORM inserts a specified drawing form in the model area or in the paper space, depending where you are in the program. You can and must complete the rest of the process, in other words the arrangement of your viewports, yourself.
- PREPLOT is more simple to use. This command automatically generates various different views in the paper space and leaves all the possible changes open for you to make yourself.

### 3.8.1 *Inserting a drawing form: PREPLOT*

The FORM command sets a form drawing in the X/Y level of the current UCS and lets you plot and print the layout in scale. It is mainly used in the paper space, since there is not much you can do with a form drawing in the model space.

### 3.8.2 *Preparing a drawing for printing: PREPLOT*

This is the more comfortable of the two commands. Four viewports are created with all the forms except the DIN A4 form. FMSsoft independently fades the dimensions and texts you have created in and out as required, with the SWITCH command.

## 4 AUTOCAD BASICS

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We make absolutely no claims that this section is fully comprehensive. For complete documentation of all the basics and commands in AutoCAD, please refer to the AutoCAD user manual. This section is intended as a brief explanation of all the most important information you need in order to work with FMSsoft:

- Entering commands
- Co-ordinate systems
- Selecting objects
- Changing (editing) objects
- Reference to certain points in an object (object snap)
- Viewing the drawing (zoom)
- 3D basics
- Printing a drawing (plotting)

## 4.1 Entering commands

Commands can be given in several different ways. You can either type commands directly on the keyboard or select them from a menu.

If you enter a command directly when prompted, a set of options of a dialogue field will then appear. To select one of the options, either enter the capital letter in the option you wish to select (*for Previous: PR, for Center: C*). Then press the return key (↵). You must always press the return key after entering information on the keyboard, whether it is a command, a point or an option. All you need to do for the text or value shown in < > to be accepted is to press the ↵ key.

### Example:

Befehl: **CIRCLE**

3P/2P/TTR/<Center point>: **2P**

First point on diameter: **0,0**

Second point of diameter: **100,100**

As well as the input method just described, you can also use the menus provided by AutoCAD, such as the menu bar at the top of the screen, your tablet menu or various "Tool boxes" faded in Windows.

## 4.2 Co-ordinate systems

### 4.2.1 Entering co-ordinates

Co-ordinates can be entered either two or three-dimensionally. An entry for two-dimensional X/Y co-ordinates can also be made as a three-dimensional entry, but the Z value is set as zero.

Co-ordinates are defined by entering XYZ, cylinder or sphere co-ordinates. A differentiation is made between absolute and relative co-ordinates. Absolute co-ordinates always refer to the co-ordinate system currently in use and relative co-ordinates to the last point entered.

#### 4.2.1.1 Absolute co-ordinates

are entered in X,Y or X,Y,Z form (Z is optional). If no entry is made for Z, AutoCAD sets Z as 0. These co-ordinates always refer to the current user co-ordinate system (UCS). See below for an example.

#### 4.2.1.2 Relative co-ordinates

relate to the last point entered or, in other words, the place where you are at the moment. You can enquire where this point is or change it using the LASTPOINT command. If you have drawn a line, for example, the last point is at the end of the line. Relative co-ordinates are entered with an @ in front of them to make them recognisable as such.

**Example:**

*Draws a line from (1,1) to (6,6)*

Command: **LINE**

From point: **1,1** *this is an absolute co-ordinate*

To point: **@5,5** *this is a relative co-ordinate, relative to (1,1)*

To point:

To enquire where the last reference point was, enter:

Command: **LASTPOINT**

New value for LASTPOINT <5,5,0>:

Entering their co-ordinates is not, however, the only way to set points. They can be digitised directly with the pointer or inserted in certain points of an object with the object snap command (see later).

**4.2.1.3 World co-ordinate system**

AutoCAD uses a fixed Cartesian co-ordinate system, the so-called world co-ordinate system (WCS). A Cartesian co-ordinate system has three axes positioned vertical to one another, the X, Y and Z axes. When you start AutoCAD you will automatically be in the WCS. This is indicated with a “W” in the co-ordinates cross.

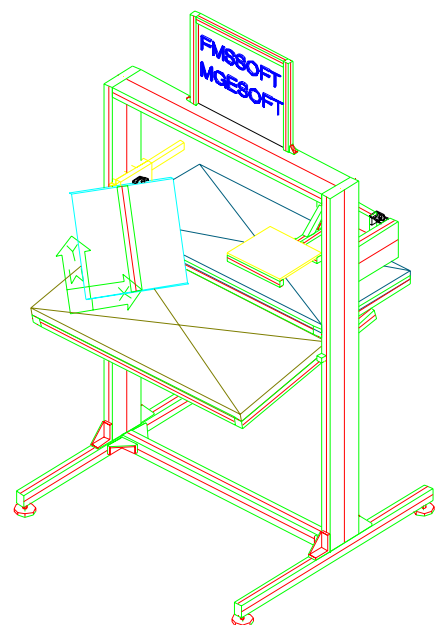
First you see a top view of the XY level. The third axis, the Z axis, points directly out of your screen.

**4.2.1.4 User co-ordinate system**

You can set your own co-ordinate system at any point you wish and adjust it as required. This co-ordinate system defined by you is known as the user co-ordinate system (UCS).

A UCS is mainly advantageous for three-dimensional drawing, since you set it in relation to objects which have already been generated and can therefore define a point in relation to the object more easily.

In the example to the right, the copy holder is standing at a “crooked” angle on the table. As you can see, the UCS symbol is at the bottom left-hand corner of the holder. This means that we are drawing in 2D on the surface and can consider the corner as the origin of the co-ordinate system (0,0,0).

**4.2.1.5 UCSICON**

To work with the UCS, you must activate the UCS icon. To do so, enter:

Command: **UCSICON**

ON/OFF/All/Noorigin/Origin<current>: **ON**

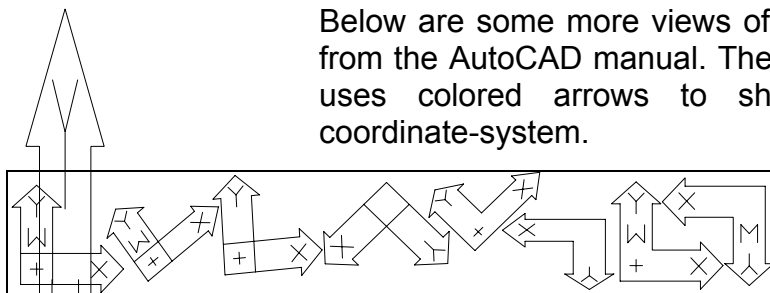
The icon is normally shown at the bottom left-hand corner of the screen, irrespective of where the origin actually is. You will now want to know where this point is. To move the UCS symbol to the point of origin, enter:

Command: **UCSICON**

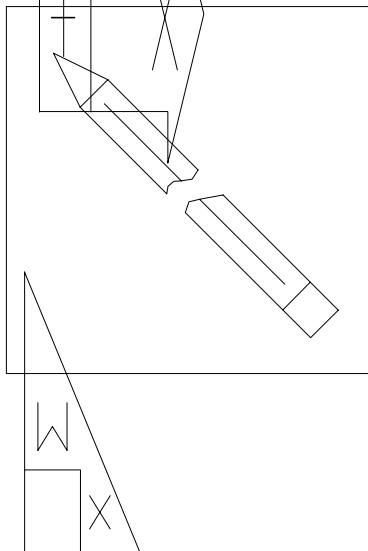
ON/OFF/All/Noorigin/Origin<Current>: **O**

The symbol itself contains important information. The “W” in the Y axis means that the user coordinate system is set to the world co-ordinate system; they therefore have the same origin and their axes are identical. The “plus” sign indicates that the symbol is at its origin. If there is a square around the “plus” sign, you are looking at the UCS from the positive Z axis.

Below are some more views of the icon we have taken from the AutoCAD manual. The 2000-Series of Autocad uses colored arrows to show the 3D cartesian coordinate-system.



Instead of its usual appearance, the icon may also be shown in two other forms. If the broken pencil is shown, you have laid out the 3D view so that you are looking onto the X/Y level directly from the side.



**Caution:** when you see this symbol it is risky to digitise points with your pointer or to insert elements, since the size of one co-ordinate will always be infinite. This is why the parts disappear!

The triangle indicates that you are in the paper space. In the paper space, several views can be shown next to each other.

**See also:** PSPACE, MSPACE, TILEMODE, MVIEW (in the AutoCAD manual)

#### 4.2.1.6 UCS

To move the user co-ordinate system to point (100,100,100) of the current UCS, enter the following information:

Command: **UCS**

Origin/ZAxis/3point/Object/View/X/Y/Z/Prev/Restore/Save/Del/?/ <World>: **O**

Origin <0,0,0>: **100,100,100**

*Other important options here are 3P, X, Y, Z (see AutoCAD manual)*

### 4.3 Layer

'Layer' is an important term in AutoCAD. A layer is a drawing level to which you can allocate a colour and type of line. Every object lies on a layer. Layers are controlled with the LAYER and DDLMODES commands. Freezing and thawing are important options when working with layers. They are used to fade layers in or out. You can use layers to give your drawing a clear structure by allocating certain elements to certain layers, for example dimensions are on the DIM layer or transfer systems on the TS layer.

**See:** LAYER, MPLAYER, DDLMODES (*in the AutoCAD manual*)

### 4.4 Selecting objects

Before you can edit objects in any form you must select them. The most simple method is to select the one you want to edit directly with your pointer, i.e. by moving onto the object with your cursor and selecting the pick button.

You can also select several objects, a so-called selection set, by surrounding them with a selection window. When you move the selection window from left to right, objects are only included if they are entirely enclosed in the window ("Window"). If, however, the window is dragged from right to left (window as line of dashes), all objects are included in the selection which touch the window in any way or are inside it ("Crossing").

**See also:** REMOVE, ADD, PREVIOUS

### 4.5 Editing

#### 4.5.1 Move

You can use the MOVE command to move objects around. As well as selecting the set you wish to move, you must also select the base point and end point of the movement. The movement therefore takes place along the displacement vector between the base and end points. Neither the base point nor the end point have to be situated on an object.

**Example:**

*Moves the object by 100 in the X direction*

Command: **MOVE**

Select object: *Select object*

Base point or displacement: *select a point on the object*

Second point of displacement: **@100,0,0**

#### 4.5.2 Rotate

Three rotation commands are available. The ROTATE command rotates objects on the XY level around the Z axis. To rotate an object around the axis

of your choice in the space, you should use the ROTATE3D command. The 2D ROTATE command is the same as a 3D rotation around the Z axis of the current co-ordinate system.

In general, the base point around which the object is to be rotated is first specified. In three-dimensional cases you can determine the axis of rotation by entering two points or use one of the co-ordinate axes directly. If you position the UCS in the object first, you will see quite clearly how you need to rotate. The direction of rotation is determined by the right hand rule, and the positive axis direction is from the base point to the end point. Finally, you must enter the angle of rotation.

**Example:**

Command: **ROTATE**

Select objects: *select an object*

Base point: *select the point of rotation on the object*

<Rotation angle>/Reference: **90** *you are turning the object 90° around the Z axis*

### 4.5.3 Copy

It is important to differentiate here between the AutoCAD copy command and the Windows copy command.

When you use the AutoCAD command, you must first create a selection set and then determine the start and end point of the distance by which the copy should be moved away from the original.

To duplicate an object and move the copy by 100 in the X direction:

Command: **COPY**

Select objects: *select an object*

<Base point or displacement>/Multiple: **0,0**

Second point of displacement: **100,0**

If you do not work with AutoCAD for Windows the following paragraph does not apply for you.

The COPY command in Windows in the EDIT menu copies an individual or several selected objects onto the Windows clipboard. These can now be used in other Windows applications, so that you can copy an AutoCAD object or a whole drawing directly into another application, such as WinWord. All you need to do is copy the contents of the clipboard into any application you wish with the EDIT and INSERT commands.

### 4.5.4 Erase

The ERASE command removes individual objects or whole selection sets from your drawing. The deleted objects are not, however, lost immediately. You can reverse the last erase command with the WHOOPS command or use the U or UNDO command.

Command: **ERASE**

Select objects: *select the objects you want to delete*

### 4.5.5 *Undo, U*

You can use the UNDO or U commands to reverse commands you have just carried out.

U only undoes the last command carried out.

REDO undoes an accidental deletion (i.e. deletes an Undo or U).

## 4.6 Viewing the drawings

### 4.6.1 *Zoom*

The ZOOM can be used to increase and decrease the size of sections of a drawing in the current drawing viewport like the zoom lens on a camera. The most important zoom commands are:

- Window (standard setting)
- Dynamic
- Extents
- Previous and
- Scale X or XP

Try out these options to get to know them.

**Tip:** after entering ZOOM EXTENTS also enter ZOOM 0.8x. The drawing will then have a little free space at the edge, which is important, for example, before dimensioning.

### 4.6.2 *Plan*

Sets a plan view to the current co-ordinate system or the world co-ordinate system.

## 4.7 Object snap

The object snap is used to find certain points on an object. These may be, for example, the end point of a line or the centre of a circle. When you are drawing in AutoCAD, it is absolutely ESSENTIAL for you to use the object snap, i.e. the reference to end points, base points etc. This is the only way to ensure that your 3D drawings are correct, as this is the only way you have access to the Z co-ordinates of the points selected.

### **Example:**

*You draw a line and want to move the UCS to the end point of the line.*

Command: **LINE**

From point: **0,0,0**

To point: **100,100,0**

To point:

Command: **UCS**

Origin/ZAxis/3point/Object/View/X/Y/Z/Prev/Restore/Save/Del/?! <World>: **O**

Origin <0,0,0>: **END** of select the line with the snap window

Every time AutoCAD prompts you to enter a point, you can and must enter an object snap!

## 4.8 3D Basics

### 4.8.1 3D Viewpoint

This command determines your viewpoint in the space. To view a drawing in the isometric view used in FMSsoft enter the following information:

Command: **VPOINT**

Rotate/<Viewpoint>: **1,-1.3,1**

*You are now viewing the drawing from point (1,-1.3,1) towards the origin.*

DDVPOINT is the same command when you are using a dialogue box.

The DVIEW command gives a very useful viewpoint for use in presentations. You can change the view of an object interactively and also create views in perspective.

### 4.8.2 Hide

To gain a better spatial impression of your drawing, enter the following information:

Command: **HIDE**

All the lines which are not visible from your point of view are hidden.

To continue editing, enter REGEN and the drawing is reconstructed.

**See also:** SHADE, SHADEIF, SHADEDGE and RENDER.

Caution: with larger drawings the commands HIDE, SHADE and RENDER mentioned above may be time-consuming!

## 4.9 Printing your drawings

Using the PLOT command, you can print your drawing on printers and plotters. You can also choose whether the hidden lines should be removed for the plot or not.

Command: PLOT

## **5 TIPS & TRICKS FROM OUR USER FORUM**

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This section sums up some of the handy tips & tricks from our FMSSoft user forum.

## 5.1 Overview:

<i>Problem</i>	<i>see</i>
Drawing file is growing very large.	Handling "large projects" with XREFs
Circles are not round.	Drawing circles and lines
Dashed lines are shown as solid lines.	Drawing circles and lines
Dimensioning too small, too large or not visible.	Dimensioning
How do I import a parts list into EXCEL?	Importing a parts list in EXCEL
How do I find my insertion point?	Tips on insertion points
Too many objects on the screen cause confusion.	Fading out objects
How can I export graphics to another application?	Exporting drawings
How can I alter the length of profiles?	Changing the length of profiles

### 5.1.1 Handling “large projects” with XREFs

Editing large projects with FMSSoft and AutoCAD may be very slow and confusing due to the large amount of data involved. Large projects are, for example, projects which include several automated stations and manual workplaces and are also linked with a transfer system. We will now show you how you can use external references (XREFs) to

- organise projects like this more clearly
- allow several design engineers to work on the project simultaneously
- divide planning and design more clearly.

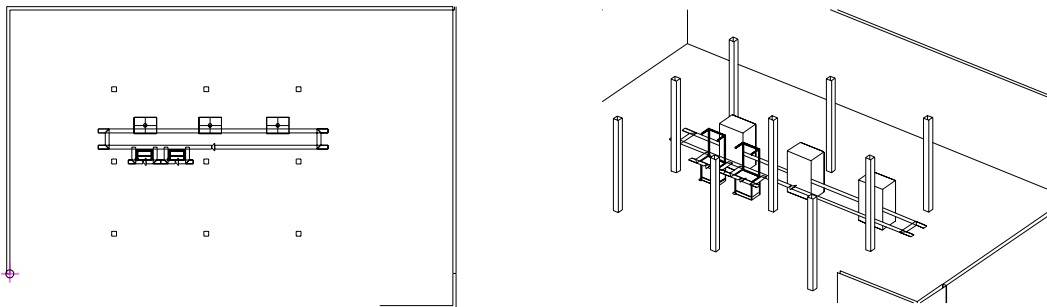
#### 5.1.1.1 External references, XREF

External references let you work with other AutoCAD drawings without these having to be permanently integrated in the current drawing and without their content changing.

When opening and plotting, AutoCAD automatically loads every external reference; this means that every XREF corresponds with the current status of the referenced drawing file. Another user can edit the externally referenced drawing at any time.

The AutoCAD command for creating and managing external references is XREF.

#### Example:



In the example shown, the plan is to arrange three identical automated stations (*STATION1*) and two manual workplaces (*MAN1*) in two shunt places in an existing plan of a hall (*FACTORY*). Details will be determined later.

**Procedure:**

*All the drawings for this projects are in C:\PROIP1*

Open the *TOTAL* document

Inserting the hall:

Command: **XREF**

?/Bind/Detach/Path/Reload/<Attach>: **Attach**

Xref to attach: **c:\pro\p1\factory**

FACTORY loaded

Insertion point: **0,0**

XScale <1>/Corner/XYZ: **1**

YScale (Vorgabe=X): **1**

Rotation angle <0>: **0**

We use the 3D objects (drawing, 3D surface, 3D objects...) to sketch the automated stations and the manual workplaces in this drawing. You should also draw connection points and centre lines where required and useful.

You now save these sketch elements as a drawing on hard disk in the project directory.

Command: **UCS**

Origin/ZAxis/.....: **Origin**

Origin <0,0,0>: *select the base point, e.g. of the workplace*

Command: **WBLOCK**

File name: **c:\pro\p1\man1**

Block name: *enter the name of the block (or \*)*

You now insert this drawing in the *TOTAL* drawing again but as an external reference.

Command: **XREF**

?/Bind/Detach/Path/Reload/<Attach>: **Attach**

Xref to attach: **c:\pro\p1\man1**

MAN1 loaded

Insertion point: **0,0**

XScale <1>/Corner/XYZ: **1**

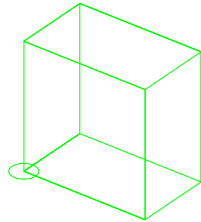
YScale (Default=X): **1**

Angle of rotation <0>: **0**

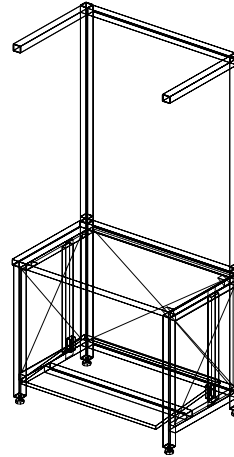
The AutoCAD version 12 cannot yet recognise circular references, i.e. you cannot, for example, insert the *TOTAL* drawing in the *MAN1* drawing again. For this reason it is advisable to make the link with the transfer system directly in the *TOTAL* drawing.

Details are now added to the stations and their design perfected directly in the station drawings *MAN1* and *STATION1*.

The example shows two different degrees of detail in *MAN1*.



sketched



detailed

### 5.1.2 *Drawing circles and lines*

You want to draw broken lines (line type dotted, hidden,...). The type of line is set correctly but the lines on the monitor are always solid.

You now have two options:

- Alter the line type scale as follows:

Command: **LTSCALE**

New scale factor <Current>:      *try out various scale factors, e.g. 10, 100*

- Reset the view resolution with

Command: **VIEWRES**

Do you want fast zoom <Y>: **Y**

Enter circle zoom percent (1-2000) <Entry>: *try approx. 30-100*

The circle zoom percent option controls the precision with which lines, circles and curves are drawn. It only controls the picture on the graphics monitor and does not influence a printout of the drawing.

**TIP:** you can also alter the AutoCAD prototype drawing so that you do not have to make these settings again for every new drawing.

### 5.1.3 Dimensioning

The dimensions are not as you had imagined or the dimension text or the arrows are not visible.

AutoCAD has numerous dimensioning variables which you can use to alter the appearance of the dimensions in your drawings.

There are two ways of altering and looking at these variables:

- You can open a dialogue box with  
Command: **DDIM**
- You can change to dimensioning mode with  
Command: **DIM**  
and enter the following information to show the variables  
DIM: **status** *the values are now shown*  
(you quit this mode again with DIM: **EXIT**)  
In dimensioning mode, the value of a variable is changed by entering the variable name and then a value.

The changed variables apply for all dimensions you make from now on, not those which already exist. To give all these existing dimensions current values, enter the following in dimensioning mode:

DIM: **UPD** *for update*

now select the dimensions to which you wish to allocate new values.

#### 5.1.3.1 Setting the dimensioning variables

- First set DIMSCALE to 1.0 as follows:  
Command **DIM** *is only necessary if you are not in dimensioning mode*  
DIM: **DIMSCALE**  
Current value <Value> New value: **1.0**
- Set the other dimensioning variables so that the proportions are right when the drawing is plotted at a scale of 1:1.

Check that your current text style is at level 0 as follows:

Command: **STYLE** Name of the text style (or ?) <Standard>: ?

List text styles <\*>: ↵

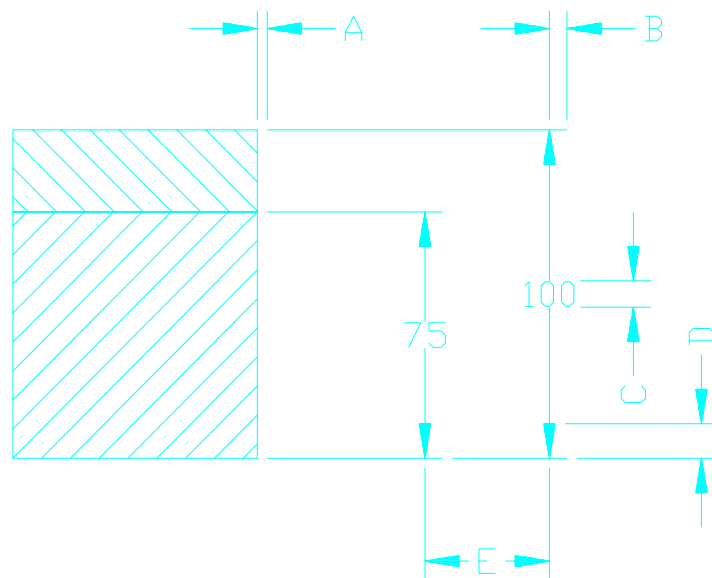
if it is not, you must set the height at 0!

**See AutoCAD manual.**

### 5.1.3.2 The most important dimensioning variables and what they mean:

DIMSCALE	general size factor which is used for all dimensioning variables which specify sizes, distances and inserts ( <b>SCALE</b> )
DIMTXT:	Height of the dimension text ( <b>TeXT</b> )
DIMASZ:	<b>Arrow size</b> at the end of the dimension lines.
DIMEXE:	specifies how far the dimension guide line extends beyond the dimension line
DIMEXO:	Dimension guide lines are moved by this value from the selected starting point
DIMDLI:	Controls the movement of dimension lines to continue linear dimensioning with the commands CONTINUE and BASE LINE. The following dimensions are then altered by this value, if necessary, to overwrite the existing dimensions.
DIMZIN	Places displayed after the comma. If 11 is set, all zeros after this are hidden
DIMASO:	Controls the generation of dimensioning objects. If DIMASO is activated, all the dimensioning objects (text, lines, arrows, curves) are grouped together as unnamed blocks. If the variable is deactivated these objects are individual objects. It should be set to On, so that the dimensions are followed correctly when stretching modules ( <b>ASsOciative</b> )

The following illustration shows the effect of the dimensioning variables:



	<i>Variable</i>	<i>Meaning</i>
A	DIMEXO	Distance between guide lines
B	DIMEXE	Extension of guide lines
C	DIMTXT	Size of text
D	DIMASZ	Size of arrow
E	DIMDLI	Dimension line increment

Example:

<i>DIMTXT</i>	2.5	3.5	5	7	10
DIMASZ		3.5			
DIMDLI		7			
DIMEXO		0			
DIMEXE		1.5			

#### 5.1.4 *Importing a parts list in EXCEL*

EXCEL does not recognise the CDF format used by FMSsoft. You can import the files as text files and set the comma as a separator.

Don't forget: The parts list created with FMSsoft is called *Drawingname.DSx*, where x is the ID for the program package.

#### 5.1.5 *Tips on insertion points*

##### 5.1.5.1 **Fading out objects**

You can fade out objects which you do not currently need and which are in the way when you want to digitise a point. The FADE command is used for this purpose.

See also: [FADE](#)

##### 5.1.5.2 **Construction by moving**

It is often useful to construct a model by moving the modules. This way you have good visual control and the modules can be positioned precisely.

By this methods, the module is first inserted at any point you wish in the drawing. The module can then be moved into the correct position with object snap modes using the MOVE command.

##### 5.1.5.3 **Object snap modes**

Point specifications can often be made with objects which are already included in the drawing. Object snap modes can be given as an option during point entry or pre-set with the OSNAP command. When they are pre-set, OSNAP nodes are first searched for with every digitising action (e.g. OSNAP node, end point, intersection...)

##### 5.1.5.4 **co-ordinate filter**

If not all the co-ordinates are required for digitisation, individual co-ordinates or pairs of co-ordinates can be filtered out, for example, entering .XY before digitisation lets you filter out the X and Z co-ordinates of the digitised point only. The Y co-ordinate can then be entered on the keyboard or by digitising again.

##### 5.1.5.5 **Entering relative co-ordinates**

If you wish to enter relative co-ordinates to specify a point, use the "rel" option. This option can be entered during every FMSsoft point specification and is used to save a starting point for the next relative co-ordinate entry with "@".

Relative co-ordinate entries begin with an "@" character, e.g., @100,200,300 means that the co-ordinate selected is positioned 100 units in the X direction, 200 in Y and 300 in Z from the last point.

### 5.1.5.6 Connection nodes

After insertion of a CAD module, FMSsoft sets one or more Cnodes at points where the next module can be inserted. It is advisable to use Cnodes as reference points for further modules. This ensures that the modules are positioned precisely in relation to one another.

Several Cnodes can be inserted in the drawing with the **CNODE** command

## 5.1.6 *Tips for building up your drawing*

### 5.1.6.1 Using layers to maintain clarity in a drawing

Before inserting a module, you can use the MODLAY command to specify on which layer the block should be laid. You can set the colour of a layer or layers with the DDLMODES command.

You can use the CHGLAY command at any time to change the layer on which objects lie.

This means you can, for example, lay assemblies on their own layers and freeze layers which are not required for part of the drawing you are currently working on. As a result, the drawing construction speed is increased and the drawing is less complicated.

### 5.1.6.2 Freezing layers:

If layers which are not currently required are faded out with the LAYER Freeze, the objects on this layer are not included in construction of the drawing. Make sure you lay parts of a drawing which belong together (e.g., drawing heading and drawing frame) on layers with names beginning with the same letter. You can then freeze whole sections of a drawing at once.

The layer structure of the FMSsoft modules can be adapted for any layer environment with the MODLAY command.

### 5.1.6.3 Copying modules

We recommend you copy modules which are the same rather than inserting new ones, to keep the amount of drawing data in your drawing file to a minimum. Copied modules are also inserted in the drawing faster than newly generated modules.

If the module is made up of assemblies of the same type, one of these assemblies should be constructed first. The others can then be generated by copying and rotating. Modules must not, however, be copied in mirror image, since mirror image blocks are returned to their original position when they explode command is used.

#### 5.1.6.4 Clearing blocks

When blocks are erased from the drawing, the graphics are removed but the block definition remains for future block insertion. To delete the block definition, the UCS world” and “WBLOCK DName \*” commands must be entered. Then quit Drawing Dname with QUIT. Drawing files can be kept much smaller if unneeded blocks are deleted in this way.

#### 5.1.6.5 Using viewports for greater clarity

The commands VPORTS and MVIEW can be used to display several views or sections of a layout, making your work much simpler.

#### 5.1.7 *Exporting drawings*

You can export drawings into other applications in one of the following ways:

- You create a DXF file from your drawing and import this into the other application.
- You print in a plot file. Here you must select a HP plotter which responds to HPGL as the printer. You can import this plot file into other applications.

The last method has the advantage that you can select any section of the drawing.

Drawings can, of course, only be imported into other applications if these have the necessary import filters.

#### 5.1.8 *Changing the length of profiles*

There are two ways of changing profile lengths:

- Use the CHMOD command: you can now enter the Choose mask and change all the physical features there, including the length. The element you have changed will still have its base point in the same position.
- Use the MODSTRETCH command: Stretch the profile as you would with the AutoCAD STRETCH command (Crossing window etc.).

Important: MODSTRETCH does not function with mirror image modules or those altered with the **SPECFIN** or **MODBLOCK** commands.

## 6 COMMAND REFERENCE

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The FMSsoft Command Reference, lists all general FMSsoft commands in alphabetical order.

The FMSsoft Command Reference is also available on line from the \*Tools\* menu. Help contains all FMSsoft commands - general commands described in the FMSsoft Command Reference and also the specific commands (e.g. MASsoft special commands).

After the command listing the glossary follows with explanations of FMSsoft terms.

## 6.1 FMSSoft command reference

2DCOPY

2DMOVE

ASSEMBLY

FADE

BLKALIGN

BOSCH

CHGLAY

CHMOD

CHOOSE

CHOOSE DIALOGUE BOX

CLIP

CNODE

DISASS

FMS\_MODULE

FORM

ITEMNO

JOIN

LANGCHG

MAGNIFIER

MOBLOCK

MODLAY

MODNAME

MODSTRETCH

PLIST

POSITION

PREPLOT

PRICEEDIT

PRICEUPD

SETLAY

TSSTART

SWITCH

## 6.2 FMSsoft commands, arranged in categories

In this index you will find all the FMSsoft commands arranged in categories. This index is particularly useful if you have already planned your next step but cannot remember exactly which commands you can use for carrying out certain tasks. The index is designed as a reference aid, and to help you get to know commands that are related to each other. The command names are arranged alphabetically in each category; you will find some names in more than one category.

### Editing

[2DCOPY](#)  
[2DMOVE](#)  
[BLKALIGN](#)  
[CHGLAY](#)  
[CHMOD](#)  
[CNODE](#)  
[DISASS](#)  
[JOIN](#)  
[LANGCHG](#)  
[MODSTRETCH](#)  
[POSITION](#)  
[SETLAY](#)

### Selecting and inserting modules

[BOSCH](#)  
[CHOOSE](#)  
[CHOOSE DIALOGUE BOX](#)  
[FMS\\_MODULE](#)  
[ISSTART](#)

### Special modules

[DISASS](#)  
[JOIN](#)  
[MODBLOCK](#)  
[MODNAME](#)  
[PLIST](#)

### Texts and dimensioning

[ITEMNO](#)  
[PREPLOT](#)  
[SWITCH](#)

### Parts list

[ASSEMBLY](#)  
[ITEMNO](#)  
[MAGNIFIER](#)  
[LANGCHG](#)  
[PLIST](#)  
[PRICEEDIT](#)  
[PRICEUPD](#)

### Plotting

[FORM](#)  
[PREPLOT](#)

### Display

[FADE](#)  
[CLIP](#)  
[MODLAY](#)  
[PREPLOT](#)  
[SWITCH](#)

## 6.3 2DCOPY

### Copying parallel to the X/Y axes

The 2DCOPY command lets you copy elements. The Z co-ordinate of the selected objects remains unchanged, so that they are always copied parallel to the X/Y axes.

Format:

Command: **2DCOPY**

Select objects: *select the objects*

Select base point: *choose a point*

Second point of displacement: *choose the destination point*

With this command you can, for example, copy modules easily, and are able to work in the top view and use all the object snap modes such as Endpoint, Intersection, etc.

Example:

To copy a grab container on a work table in MASsoft:

Command: **2DCOPY**

Select objects: *select the grab container*

Select the base point: **bas** ↵ from *pick a container*

Second point of displacement: *digitise a point on the table*

**See also:**    **COPY (AutoCAD)**

## 6.4 2DMOVE

### Moving along the X/Y axes

The 2DMOVE command lets you move elements in the drawing. The Z-axis co-ordinates of the selected objects remain unchanged, so that they are always moved parallel to the X/Y axes.

Format:

Command: **2DMOVE**

Select objects: *select the objects*

Select base point: *choose a point*

Second point of displacement: *choose the destination point*

With this command you can, for example, move modules easily, and are able to work in the top view and use all the object snap modes such as Endpoint, Intersection, etc.

### Example:

To move a grab container on a work table in MASsoft:

Command: **2DMOVE**

Select objects: *select the grab container*

Select base point: **ins** ↵ from *pick a container*

Second point of displacement: *digitise a point on the table*

**See also:**    **MOVE (AutoCAD)**

## 6.5 ASSEMBLY

### Forming assemblies

The **ASSEMBLY** command lets you alter the ID character of a group of modules. The modules in an assembly are filtered for the parts list printout by the ID character.

Format:

Command: **ASSEMBLY**

ID character <G>/C/M/T/A/1/2/3/4/5/6/X: *select an ID character*

Select objects: *select the objects you wish to allocate to this ID character*

Possible ID characters:

- G: stands for Basic Mechanical Elements (MGEsoft)
- C: stands for Transfer System (CTSsoft)
- M: stands for Manual Work Systems (MASsoft)
- T: stands for Transfer System (Tssoft)
- 1-6: stands for user's own modules available
- X: stands for superior modules - in the parts list printout this means all modules, whatever their ID character

If this ID character is given for creating the parts list, only the parts list for this assembly is drawn up.

**See also:** [PLIST](#)  
[MODBLOCK](#)

## 6.6 FADE

### Temporarily fades out objects which are not currently required

The **FADE** function is a support function for digitising. It can be used to fade elements in and out.

Format:

Command: **FADE**

<In>/Out:

Layer or <Object>:

Select objects:

Options:

**Objects:** The selected objects are either faded **out** (invisible) or faded **in** (visible). The objects which are faded out can be brought back into view at any time with the **REDRAW** command.

**Layer:** **Out:** The selected objects are set on the layer specified by the user and the layer frozen. They then remain invisible until the layer is thawed again.

**Caution:** this means that the objects which were already on this layer also disappear.

**Layer:** **In:** The specified layer is thawed, i.e., the objects on this layer appear again.

### Example:

makes all objects disappear except the selected object

Command: **FADE**

<In>/Out: **Out**

Layer or <Objects>: **Objects**

Select objects: *select an object*

**See also:** **REDRAW (AutoCAD)**

**LAYER (AutoCAD)**

## 6.7 BLKALIGN

### Aligning blocks

The command BLKALIGN moves a selection set from the base point of a chosen base element to a new insertion point, and rotates all the elements so that the angle of the base element (Block!) is the same as the angle of the block on which the insertion point lies.

Format :

Command: **BLKALIGN**                      *Select objects:*

If you have selected more than one:

Select base element:

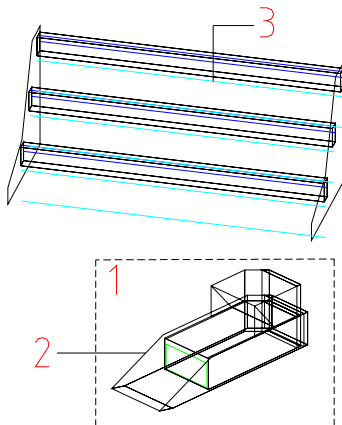
Insertion point: *Object snap prompt "next" !*

1. This might seem complicated, but it is really very simple!

For example: several system grab containers inserted into each other (angle 0°) and a grab container rack (any angle!) are positioned on the table. If you want to suspend the containers in the grab container rack, you can use the BLKALIGN command.

When you are asked to "Select objects: ", select the entire pile of grab containers.

2. The "base element" is then the container at the bottom of the pile, which is the one which will actually be suspended.
3. The "insertion point" is a point on the yellow guide line in the rack. The entire pile is first moved to this point, and then aligned to match the angle of the rack. This is why the command is called BLockALIGN.



## 6.8 BOSCH

### Searching for construction elements in the database

If you want to search for a construction element directly in the database, according to your own criteria, enter:

Format:

Command: **BOSCH**

The following dialogue box is displayed:

Order nr.	Designation	Price	Physical features
0 842 010 004	FORK LIFTER	3000.00	H=750 MM;PNEUMATIC
0 842 010 005	FORK LIFTER	3842.00	H=1000 MM;PNEUMATIC
0 842 010 006	FORK LIFTER	4200.00	H=1300 MM;PNEUMATIC
3 842 518 160	FORK LIFTER	-	H=500 MM;ELECTRICAL
3 842 517 580	FORK LIFTER	-	H=900 MM;ELECTRICAL

### Searching for a construction element:

1. Select the product line.  
The **Choose...** and **Module<** buttons can only be selected if the product line selected is the same as the FMSsoft package currently loaded. Otherwise both are inactive.
2. Either enter the order number you are looking for in the Order no. editing field, or enter the ordering name in the Name editing field. You can use wildcard characters such as an asterisk (\*) in both cases.
3. Select the **Search...** button. The result of the search is displayed.
4. Mark the construction component you need in the list box with your pointer.
5. Select the **Module<** button to insert the element directly in your drawing.
6. Select the **Choose...** button to define the physical features of the element exactly.

Please see also: **CHOOSE**

## 6.9 CHGLAY

### Changing the insertion layer and the colour of modules

The **CHGLAY** command is used to change the layer of a selection set.

Format:

Command: **CHGLAY**

Select objects: *select the objects concerned*

New layer for these objects: *Enter layer name*

Enter whichever layer name you require. If this layer does not yet exist the following message appears:

Layer <Layer name> has been created

Please choose a colour...

Red/Yellow/Green/Blue/Cyan/Magenta..<White>: *Enter a colour*

The entire selection set is then inserted on this layer.

**Note:** the main graphics for all FMSsoft elements are indicated on the '0' layer in the "BYLAYER" colour, i.e., the layer into which a block of this kind is inserted determines the colour of the element. If you insert your own blocks on another layer using this command, they may remain the same colour.

## 6.10 CHMOD

### Changing the physical features of a module

Format:

Command: **CHMOD**

Select object: *select an object*

Once you have digitised the object you wish to modify, the choose dialogue box for this module is activated. You can make all the necessary changes in this box.

As soon as you close the dialogue box with OK (button), the old module is replaced with a new one.

If you cancel (button) no alterations are made to the module.

## 6.11 CHOOSE

### Selecting modules

Every module has different physical features, such as length, width, packing unit etc., which must be specified before the graphics element is added to the drawing.

For this reason, there are dialogue boxes for each module, in which the module characteristics can be specified.

These dialogue boxes can be called with the **CHOOSE** command. This can be done either manually, by typing the module name in the menu tablet or in one of the pull-down menus.

Format:

Command: **CHOOSE**

Module name ?/<Default>: *Enter the module name here*

In <Default> is the name of the module last selected.

When a question mark or a module name which is not available is entered, the following prompt appears:

Search text <\*>: *Enter ↵ for a list of all the modules which can be selected. You can also use an asterisk here (\*).*

Examples:

Search text <\*>: \* *lists all modules*

Search text <\*>: **SEL\*** *displays all modules that begin with "SEL"*

Search text <\*>: **S\*L\*** *Modules that begin with "S" and contain an "L"*

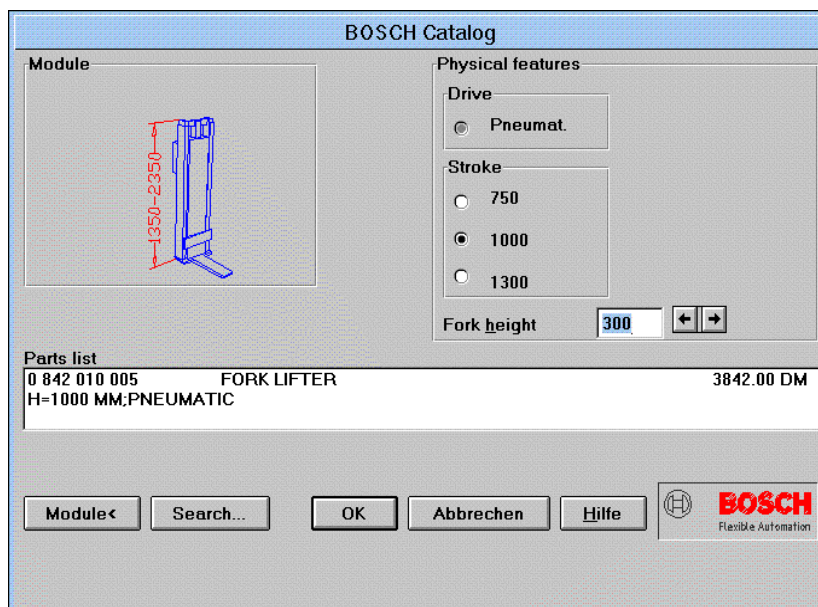
See also: [FMS\\_MODULE](#)

[BOSCH](#)

### 6.11.1 *The CHOOSE dialogue box*

appears if you have selected a module using **CHOOSE**. This is the central dialogue for selecting FMS construction components.

The example shown here is the dialogue box for the case lifter in MASsoft.



- In the **physical features** field you can specify the part needed in greater detail. The order number and the price will then be displayed automatically in the parts list field.

#### Rules for entering physical features

- In the **Parts list** field, the parts list information for each separate module is displayed.

#### Rules for the display of parts list information

- **MODULE<** for direct insertion of the part which has just been selected. The same function is called as when you enter **FMS\_MODULE** in the command line.
- **SEARCH...** helps you to find specific parts in the database. You can search for order numbers or names. When you select this button, the Search dialogue box appears. You can also enter this dialogue box by selecting **BOSCH** from the command line.

See also: [FMS\\_MODULE](#)  
[BOSCH](#)

**6.11.1.1 Rules for entering physical features:**

1. The value entered is checked for acceptability.
2. If the value is incorrect, the range of acceptable values is displayed in the status line.
3. The information should be entered in sequence from left to right and from top to bottom.

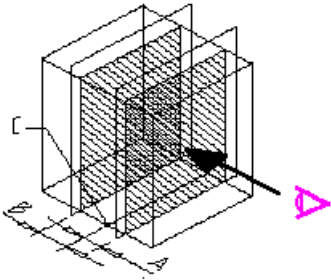
**6.11.1.2 Rules for the display of parts list information:**

1. The parts list information is only displayed for individual modules, NOT for complex modules.
2. The price displayed does NOT include any accessories selected.
3. Special elements are displayed without a price.
4. The total price for complex modules is not calculated until the parts list is generated.

## 6.12 CLIP

### Creates cross-section views of 3D models

The **CLIP** command lets you create cross-section views of a 3D model. No elements outside the section are depicted. To create the clip, the clip direction is specified, then a point is set on the clip level "C". The clip depth can then be determined towards the front "A" (towards the viewer) and towards the back "B" (away from the viewer). A strip of thickness A+B is then cut out of your model. Drawing elements inside this strip are depicted in full. If 3D surfaces are cut, you only see dots at the penetration point of 3D surface and clip level.



Format:

Command: **CLIP**

View from Top/Right/Left/Behind/Front or Off<Off>: *you are looking at the clip surface from here*

Select point on clip: *Select point C*

Clip depth in front direction: *Enter measurement A*

Clip depth in rear direction: *Enter measurement B*

### Settings in the clip view:

- The UCS is set with the view.
- The UCS originates in the selected point "C".

**Caution:** As soon as a clip has been created for a certain view, you should not change the view direction again, since you are in clip mode in the AutoCAD command DVIEW.

Before you change the view direction it is important to deactivate the clip mode!!

**TIP:** Insert the drawings of the profile cross-section in the clip level!

## 6.13 CNODE

### Inserting a connection node

A connection node (Cnode) is a guidance point defined by FMSsoft.

After insertion of a CAD module, FMSsoft sets one or more Cnodes at points where the next module can be inserted. It is advisable to use Cnodes as reference points for further modules. This ensures that the modules are positioned precisely in relation to one another.

Several Cnodes can be inserted in the drawing with the **CNODE** command.

Format:

Command: **CNODE**

X/Y/Z-direction or point:

### Options:

**Point:** Enter a point directly or digitise

**X/Y/Z.** The distance here is only related to the co-ordinates. The sequence is as follows:

First point: enter the first point

Distance or second point: When you enter a point, FMSsoft calculates the distance between the 1st and 2nd points and sets the Cnode at the distance calculated in the direction of the selected axis. If you select Distance, FMSsoft asks you to insert a distance.

Rel: You are then asked for a reference point:

Base point: select a point  
X/Y/Z-direction: @X,Y,Z draws the point at distance X,Y,Z from the reference point

**Use:** The Cnode is activated with the object snap "Node".

**Depiction:** Depiction of the Cnodes can be controlled with the AutoCAD system variables **PDMODE** and **PDSIZE**. If only Cnodes are to be used as insertion points, the object snap "Node" must be pre-set with the **OSNAP** command.

### Examples:

to lay a connection node in the origin of the UCS:

Command: **CNODE**

X/Y/Z-direction or point: **0,0,0**

to define a connection node at a distance of 100 in the X direction starting at the origin:

Command: **CNODE**

X/Y/Z-direction or point: **X**

First point: **0,0,0**

Distance or second point: **100**

## 6.14 DISASS

### **Explodes modules and stores the parts list information**

The **DISASS** command breaks a module down into individual parts (or “**disassembles**” it) and stores the parts list information for this module internally.

Format:

Command: **DISASS**

Select object: *select an object*

Once you have disassembled the module you can alter the graphics of the module.

**Caution:** you can only ever disassemble one module at a time with the **DISASS** command. Before you can use the **DISASS** command again you first have to select the **JOIN** command.

Modules can also be broken down into parts with the **EXPLODE** command, but if you use this method you will lose the parts list information for the module!

**See also:** [JOIN](#)

## 6.15 FMS\_MODULE

### Inserting modules

The **FMS\_MODULE** is used to insert the modules. This is usually carried out on the basis of the physical features determined with the **CHOOSE** command.

Format:

Command: **FMS\_MODULE**

Module name ?/<Default>: *Enter the module name here*

In <Default> is the name of the last module selected.

When a question mark or a module name which is not available is entered, the following prompt appears:

Search text <\*>: *Enter ↵ for a list of all the modules which can be selected. You can also use an asterisk here (\*).*

Examples:

Search text <\*>: \* *lists all modules*

Search text <\*>: **SEL\*** *displays all modules that begin with "SEL"*

Search text <\*>: **S\*L\*** *Modules that begin with "S" and contain an "L"*

If you enter a module name from this list, when you are asked for a module name, it may still be possible to digitise the length, width and/or thickness.

Length<Default>: *enter new value or press ↵ to keep the same value.*

The new value can be entered by digitisation or entering a value using the keyboard. This value does NOT alter the entry in the selection field. The new value only applies temporarily for the current insertion process.

Finally, insert the module. It will be displayed on the screen:

Insertion point: *enter the point or digitise a point with the pointer*

Other options are also available in MGEsoft:

**Example:**

determine the physical features of a module with

Command: **CHOOSE**

to insert this module.

Command: **FMS\_MODULE**

Module name ?/<module selected above>: ↵

Insertion point: *enter an insertion point, e.g. 0,0,0 or digitise a point*

See also: [\*\*CHOOSE\*\*](#)

## 6.16 FORM

### Inserting form drawings

The **FORM** command inserts form drawings in the X/Y levels of the current UCS. Forms are mainly used in the paper space.

Format:

Command: **FORM**

Form name: *name of the form you want to use, e.g. DIN\_A4.DWG*

Scale factor <1>: *1 means the drawing will be inserted 1:1, 2 that it will be twice the size, etc.*

Angle <0>: *angle to the X axis of the current UCS*

Insertion point: *point on the X/Y level of the current UCS*

4 DIN form drawings are supplied with an FMSsoft package in DWG format. They consist of a DIN drawing header and a drawing frame.

The drawings are called:

DIN\_A4.DWG

DIN\_A3.DWG

DIN\_A2.DWG

DIN\_A1.DWG

When an FMSsoft package is installed, these drawings are placed in the relevant package directory. You can also use your own forms if you copy the DWG form file to the appropriate package directory under the DWG name DIN\_Ax.DWG (x = 1-4).

**Note:** If you want to use the same form drawings for different FMSsoft packages, you can install these in a joint directory. Add this directory to the path to the support files. As long as the drawings are in these files (ACAD and FMSsoft directories), you only have to give the drawing name when inserting the form without having to specify the directory. The drawing names can be entered either with the keyboard or the screen menu.

The forms supplied are in layers called "\$P\$KOPF...". This means that after insertion, the graphics can be frozen and thawed again as often as required.

**See also:** [PREPLOT](#)

## 6.17 ITEMNO

### Allocating item numbers

The **ITEMNO** command can be used to allocate and change item numbers and insert them in the drawing on the current UCS.

Format:

Command: **ITEMNO**

Select objects:

If you have selected a complex module (e.g. NTS or KTS from the TSoft range), a selection menu appears in which you can specify in greater detail the part to which you wish to allocate an item number:

```
<Part 1>  -> 1
<Part 2>  -> 2
<Part 3>  -> 3
```

Parts list level or <eXit>: *enter the ID number of the part you require*  
from this point the prompts are the same for all objects:

Item no. <1>:

Erase old guide line? <Yes>/No:

From point:

To point:

The start number can be selected. Every time a number is allocated, it increases by 1. If the module already has a number, it is possible to overwrite the number or to keep the old number. If you want to overwrite the number, the old guide lines can be erased. The new guide line is then drawn on the current UCS.

Copied modules (= the same block name) have the same item numbers as the original. If you wish a copied module to have a different number, the item number of the module concerned must be specifically allocated with the **ITEMNO** command. A new parts list set is then created for the module.

The parts are arranged by item number in ascending order. The parts without an item number are arranged at the end of the list in alphabetical order.

**Note:** because the item numbers are drawn at the z=0 level of the current UCS, the following procedure is recommended.

1. First set the layer and the UCS with the SWITCH command
2. Then select the **ITEMNO** command.

**See also:**   **PLIST**  
                  **SWITCH**

## 6.18 JOIN

### Joining modules which have previously been disassembled with **DISASS**

A new block is allocated the parts list information of the module which was previously disassembled with the **DISASS** command. In this way you can alter modules in graphics without losing the relevant parts list information.

Before you call **JOIN** you should set the UCS so that the block can be inserted in the correct position in future. You should usually set UCS world.

Format:

Command: **JOIN**

Block name: *Enter a unique block name*

Basepoint of insertion:

Select objects:

See also: [\*\*DISASS\*\*](#)

[\*\*MODBLOCK\*\*](#)

**BLOCK (AutoCAD)**

## 6.19 LANGCHG

### Changing parts lists from one language into another

The command **LANGCHG** realises the translation of parts list information from one language into another. You can choose between 5 different languages. The designations of the modules are translated. Afterwards the parts list has to be generated once again.

Format:

Command: **LANGCHG**

from German/English/Italian/French/Spanish: *choose the source language*

to German/English/Italian/French/Spanish: *choose the target language*

See also: [PLIST](#)

## 6.20 MAGNIFIER

### Displaying the parts list information

When you sweep the cursor across the components in **MAGNIFIER**, the parts list information for the component under the cursor is shown, such as order number, name and price.

Format:

Command: **MAGNIFIER**

Position cursor at modules: *move across the module required with the pointer*

## 6.21 MODBLOCK

### Adding parts list information to drawing objects

If you have made your own construction elements, it is advisable to add parts list information to them, so that one of the great advantages of 3D construction - a consistent parts list - is maintained.

The **MODBLOCK** command in FMSsoft is designed for this purpose. It adds parts list attributes to the graphics and forms a new block.

Caution: any elements you have drawn yourself should be on layer 0 and the UCS must be set as it will be for insertion later.

Format:

Command: **MODBLOCK**

Block name : *enter a clear name, e.g. CONTAINER*

Basepoint of insertion: *select a point, ideally on the object*

Select objects: *select all the elements you want to include in the block*

Item no.: *Entry or ↵*

Quantity: *Entry or ↵*

Name: *Entry must be made!*

Order no.: *Entry must be made!*

Remark: *Entry or ↵*

Physical feature 1: *Entry or ↵*

Physical feature 2: *Entry or ↵*

ID character <G>/C/M/T/A/1/2/3/4/5/6/X: *Entry must be made!*

**Caution:** The name, order number and the ID character are essential pieces of information for creating the parts list.

Once the special module has been created, it can be inserted in your drawing at any point you wish using the **INSERT** or **DDINSERT** commands.

**See also:** **WBLOCK (AutoCAD)**

## 6.22 MODLAY

### Altering the module layer

The Layer structure of the FMSsoft modules can be adapted for any layer environment with the **MODLAY** command. The FMSsoft layer names may be up to 19 characters long. If the layers you wish to use are not yet available before the **MODLAY** command is selected they are generated. Any colour can be used for each layer; these colours must then be selected.

The FMSsoft layer names can be changed at any time, for example, entire project parts can be generated on other layers.

Format:

Command: **MODLAY**

In the dialogue box which then appears, you can define 10 layers. The first four layers are of a general nature, that is, they are available in all FMSsoft packages.

- **Layer 1** is the **insertion layer** for modules. The main graphics for the modules is drawn on layer 0.
- **Layer 2** is used for drawing **connection nodes**
- **Layer 3** contains the elements of **additional graphics** (e.g. profile grooves)
- **Layer 4** depicts interference limits or **system limits**.

See also: **LAYER (AutoCAD)**

**DDLMODES (AutoCAD)**

**CHGLAY**

## 6.23 MODNAME

### Renaming FMSsoft blocks

When existing drawings are inserted, you must give their name and directory. You must be careful not to insert any drawings with the same name into each other, since this causes a name conflict in the block definitions. If this cannot be avoided, you can rename all FMSsoft blocks in the current drawing with the **MODNAME** command.

Format:

Command: **MODNAME**

New DWG name: *enter a new name*

## 6.24 MODSTRETCH

### Stretching modules

The **MODSTRETCH** command lets you extend modules, although only modules available in various lengths and widths can be modified.

The modules should only be stretched orthogonally if possible, otherwise you may get surprising results. For this reason, activate **ORTHO**.

Format:

Command: **MODSTRETCH**

Crossing window to select modules to be stretched...

First corner point: *digitise other corner point("crossing")*

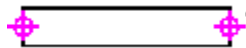
Basepoint: *select the basepoint for the extension*

New point: *select the end point for the extension*

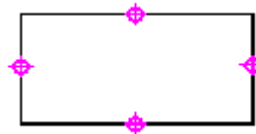
All the parts list information for the modules are correctly adjusted to correspond with the drawing during this process, since the stretched modules are regenerated. Objects which are entirely in the window are moved, as with the **MOVE** command.

The relevant points for stretching are:

for variable length



for variable length and width:



Only one of these points may be in the crossing window at any one time.

You can, of course, stretch several modules at a time.

**Important:** this command does not function with mirror image modules or those altered with the **SPECFIN** or **MODBLOCK** commands.

**See also:** **STRETCH (AutoCAD)**

**SPECFIN**

**MODBLOCK**

## 6.25 PLIST

### Creating and editing a parts list

Format:

Command: **PLIST**

A dialogue box is displayed:

The options in the dialogue box are:

[ID character](#)

[Edit](#)

[Accessory](#)

[Format](#)

[Type](#)

[Form file](#)

[Create](#)

[Print](#)

[Calculation](#)

[Edit parts list file](#)

### 6.25.1 *ID character*

The parts list is only drawn up for modules with a certain ID character. With "X", all modules are included. It is standard for the identification of the software package used to be given.

### 6.25.2 *Edit*

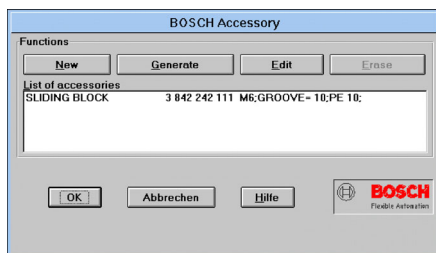
Module<: You can digitise one or more modules whose attributes are then displayed in a dialogue box and can also be altered. You can access the same dialogue box with the DDATTE command.

Heading: Here you are asked about the variables for the parts list heading which are used in your parts list file (.VRL or .DWG).

Accessory....: Elements which are not displayed as graphics are inserted in the drawing as symbols and also cause an entry to be made in the parts list.  
This includes, for example, section links, foundation brackets, foundation sets, brackets, connecting kits, etc.

### 6.25.3 Accessory

The following dialogue box is displayed:



**New:** Inserts user-specific accessories in the drawing.

Please note that name, order no. and identification must be entered.

**Generate:** Calculates accessories if you have defined modules including accessories, e.g., profile including connecting elements or conveyor units including belt.

The accessory blocks are inserted in the <package>ACCESSY layer (e.g., MASACCESSY).

The accessory elements calculated are marked in the comments field with an asterisk (\*).

When it is called again, all the marked accessory blocks are first deleted.

**Edit:** Lets you alter the entry marked in the parts list.

**Erase:** Erases the entry marked in the parts list.

**Please note:** For the calculated or altered accessories to appear in the parts list, you must recreate the parts list (select button).

### 6.25.4 Format

Here you define the format of your parts list.

**Block:** an AutoCAD drawing file is used as a model, and the parts list which has been drawn up is then inserted in the drawing as a block

**File:** a file with the extension .VRL is used as a model

**CDF:** the parts list is drawn up in CDF format, which can be imported into a spread-sheet program or database. When this format is used, all pieces of information in the parts list are separated by commas (comma delimited format).

### 6.25.5 Type

In what form do you wish to see the modules in the parts list? All individual modules (including those in blocks) or as a structured parts list (Module A

consists of ...) or perhaps just the top level, i.e. without any subsidiary modules which may be inserted in modules?

INDIVIDUAL	shows parts list information for all individual parts in the modules
STRUCTURE	shows the module structure, that is, the primary module and the subsidiary modules or individual elements it is divided into
ASSEMBLY	shows only the top level

### 6.25.6 **Form file**

Here you can select three files which FMSsoft uses as a model for the parts list. Depending on the format, this is a .DWG or VRL file. It is only impossible to name a form when the file is produced in CDF format.

### 6.25.7 **Create**

Choose modules/<All>: [↩ to create the parts list for all modules](#)

Modules in externally referenced drawings (XRefs) are also acceptable here.

The parts list is drawn up in accordance with the information you provide. If you have selected File or CDF for the format, a file called Drawingname.DSX (X = identification of the FMS package) will be saved on the hard disk. At the same time, the parts list created will be shown in the dialogue box.

If there is a **module with no price** in the parts list, the **total price** is entered **in brackets**.

### 6.25.8 **Print**

The last parts list file created is printed. This is the file which can also be seen in the window of the dialogue box.

### 6.25.9 **Calculation**

For calculation, enter your purchase discount and the sales discount and the foreign-currency exchange rate if required.

### 6.25.10 **Parts list file:**

The parts list created above is shown in this window. There are scroll bars at the right for turning through the pages. The marked line can be altered in the edit box underneath.

#### **Important!**

1. When you alter lines in the parts list file, these alterations do not have any influence on the parts list information for each individual module. Please note also that the total price does not change automatically when price alterations are made in this way.
2. The next time the [Create](#) command is used, the parts list file is overwritten!

## 6.26 PRICEEDIT

### Changing price information's

Inside the price editor the user can adapt the prices of FMSsoft modules according to his needs.

Format:

Command: **PRICEEDIT**

**BOSCH Price editor**

Parts list file

```
60200|1|PROFILE 20x20|3 842 992 888||L=%6.1f MM;-/:;G|
60204|1|PROFILE 20x20 2N VS|3 842 992 924||L=%6.1f MM;-/:;G|
60201|1|PROFILE 20x20 3N|3 842 992 889||L=%6.1f MM;-/:;G|
60202|1|PROFILE 20x20 R|3 842 992 890||L=%6.1f MM;-/:;G|
60203|1|PROFILE 20x40|3 842 992 891||L=%6.1f MM;-/:;G|
60001|1|PROFILE 30x30|3 842 990 720||L=%6.1f MM;-/:;G|
60002|1|PROFILE 30x30|3 842 990 721||L=%6.1f MM;M8/-;G|
60003|1|PROFILE 30x30|3 842 990 722||L=%6.1f MM;D11/-;G|
60004|1|PROFILE 30x30|3 842 990 723||L=%6.1f MM;M8/M8;G|
60005|1|PROFILE 30x30|3 842 990 724||L=%6.1f MM;M8/D11;G|
60006|1|PROFILE 30x30|3 842 990 725||L=%6.1f MM;D11/D11;G|
```

Search

Price informations

Basic price	Additional price 1	Additional price 2	Additional price 3	Additional price 4
<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="0.00"/>
Additional price 5	Additional price 6	Factor	Formula no	Phys. features
<input type="text" value="0.00"/>	<input type="text" value="0.00"/>	<input type="text" value="1.00"/>	<input type="text" value="-1"/>	<input type="text"/>

Formula

User price

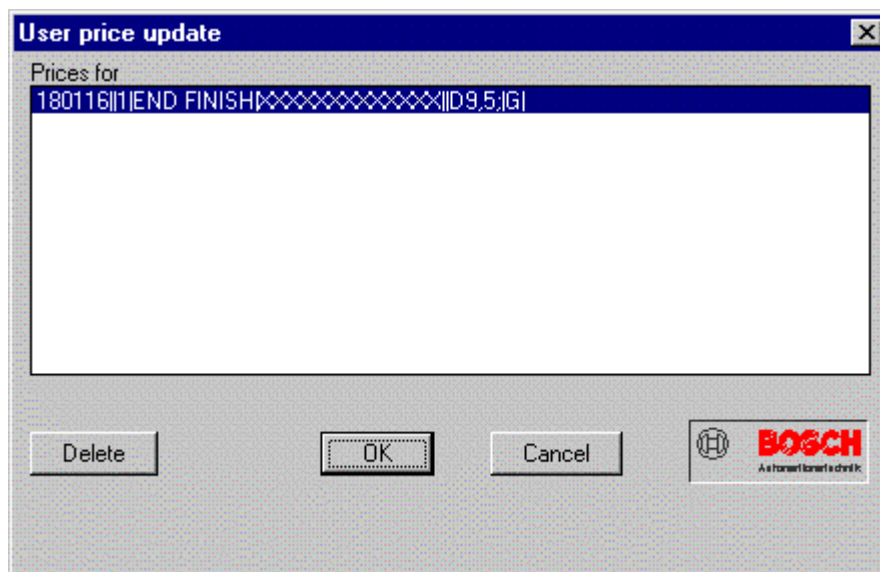
**BOSCH**  
A functional standard

**PRICEEDIT** reads the parts list information and the price file. The prices are build of a basic price and up to 6 additional prices, as well as a calculation formula. If you want to search for specific parts list entries, type in the desired designation and press „Search“. You can also use placeholders like \* or ? . (To find for example a bracket, type in „\*bracket\*“).

Formulas are used for the price calculations of components with price relevant physical features. The formulas are assigned to the parts list entries via a formula number. Price relevant physical features can be assigned to variables SM1-SM7. The variable GP is substituted by the basic price.

Formula number 0 indicates that this parts list record has only a fix basic price.

**CAUTION:** You should alter prices only in compelling cases. If you have made a modification, you can save it via „Save“. The last modifications are written additionally into the file PREIS.SAV. This file can be taken for the update of the user prices, after a new software installation or a price update. Before starting the update, you have the possibility to choose which records from PREIS.SAV will be taken into account, and which will be erased from the file.



You should save the PREIS.SAV file, to avoid data loss at a release upgrade.

## 6.27 PRICEUPD

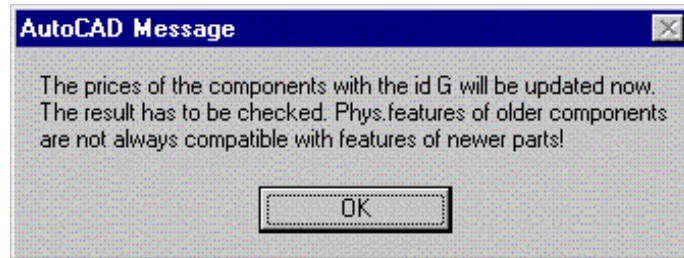
### Actualise the prices of existing projects

With the command **PRICEUPD** you can actualise the prices of existing projects after a price update. All order numbers are checked and their prices actualised, if there are corresponding price file entries. The result of components whose price has been calculated with a formula should be checked afterwards. Missing physical features of older components can lead

to wrong prices and have to be corrected. Use therefor the command **CHMOD**.

Format:

Command: **PRICEUPD**



## 6.28 POSITION

### Rotating, moving and copying in 3D

The **POSITION** command is used to change the position of objects in the surrounding area. It is a combination of the standard commands **UCS**, **ROTATE**, **MOVE**, **COPY** and **UNDO**.

The **POSITION** command makes it easy to position objects in their surroundings and considerably reduces the number of entries you have to make.

Format:

Command: **POSITION**

Base element: *select a point on the object to which all the following operations will refer.*

Select objects: *Select additional objects which should be moved with the base element.*

<Exit>/Align/Xrotate/Yrotate/Zrotate/Move/Copy/Undo:

This menu lets you position a module in the surrounding area without having to use the UCS with standard commands. Neither do you need to set a UCS beforehand!

Options:

**Exit:** Quits this menu

**Align:** Realigns the element. The new X-direction is taken from the current zero point and the alignment point (3D!)

**X/Y/Zrotate:** Rotates the module along one of these three axes

**Move:** Moves the module

**Copy:** Copies the module

**Undo:** Reverses "operations" to the beginning of the command.

## 6.29 PREPLOT

### Preparing a 3D drawing for plotting

[Example p. 68](#)

**PREPLOT** is the command used to prepare a 3D drawing for plotting. **PREPLOT** inserts the form, opens the viewport, sets scales, updates the dimensioning factor and handles the layers for texts and dimensioning in the individual viewports.

**Tips:** For automatic calculation of the scale, drawing elements which are not part of the model must not be visible. You should therefore lay these elements in a layer (e.g. with the FMSsoft command **CHGLAY**) which you will freeze before you start **PREPLOT**.

Adjusting the layer to the appropriate view only functions correctly if you have previously used SWITCH to create the layer names. For this reason, you should use the **SWITCH** command before dimensioning a view!

Format:

Command: **PREPLOT**

Select objects:

Views frame(ON)/Scale/Modify/Automatic/<eXit>:

Options:

**Automatic:** The views are created fully automatically in a DIN form selected by you. The following prompt will appear:

Format DIN A1/A2/A3/<A4>: *select the form you wish to use*

Four Forms are available: DIN A1, A2, A3, A4. In A4 format, it is standard for an isometric view to be created. All other formats have 4 views: Top, front, side and isometric views. Except for the isometric view, all views are to scale.

The optimum scale is now calculated and offered to you as a standard value. If you increase the value, your objects will be shown smaller.

min. scale 1:<calculated value>: *enter a different value or ↵*

The paper space is depicted and you are asked to give an insertion point:

Insertion point: *Enter a point or digitise*

**Modify:** The modify option allows you to vary scales and view directions for views which have already been created. To do so, select the viewport you want to change. The frame must be activated! When you have entered the view direction and the scale, your viewport will be regenerated.

Select viewport: *click on the frame of the viewport*

View direction <Front>/Behind/Left/Right/Top/Bottom/Iso:  
*select the new view direction*

min. scale 1:<calculated value>: *enter a different value or ↵*

- Frame:** The frame option activates and deactivates the viewport frame. The current status is shown in brackets behind the option frame. The viewports are all on one layer, "FMS\_VPORTS".
- Scale:** You can enquire what the scale of a viewport is.
- Select object: *click on the frame of the viewport you wish to know about.*

### 6.29.1 **PREPLOT, example**

To show your current drawing on a DIN A3 form with the **PREPLOT** command, and insert it at point (0,0), enter the following:

Command: **PREPLOT**

Select objects: **Select the work table**

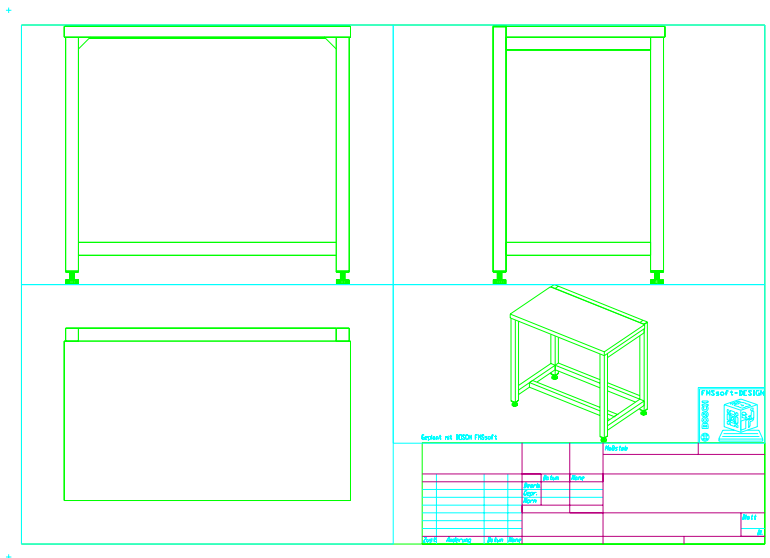
Views frame(ON)/Scale/Modify/Automatic/<eXit>: **A**

Format DIN A1/A2/A3/<A4>: **A3**

min. scale 1:<calculated value>: ↵

Insertion point: **0,0**

Printout using an work table from MASsoft as an example:



## 6.30 SETLAY

### Setting the current layer by clicking

With **SETLAY** the current layer can be changed simply by clicking on an element that is on the layer you want to go into.

Format:

Command: **SETLAY**

Select object: *Click element*

Example:

At the start of an FMSsoft application you will be on the drawing layer for that application. Insert any module you like.

Now change to another layer, for example layer 0 as follows:

Command: **LAYER**

LAYER ?/Set/...: **SET**

New current layer<current layer>: 0

Layer 0 is now the current layer

To change back to the main graphics layer (e.g. MGE), enter:

Command: **SETLAY**

Select object: *pick an FMS module*

## 6.31 TSSTART

### TS Definition of standard settings for modules

When you start to work on a new drawing, a dialogue box is displayed and you are asked to provide information about the most important project-related physical features of the modules.

Once these physical features, which apply for the entire system, have been entered, the modules are initialised with these standard values.

TSSTART can be used to modify the standard settings for a drawing while it is being worked on.

Format:

Command: **TSSTART**

The TSSTART command is not included in every FMSsoft package.

## 6.32 SWITCH

### Setting a UCS and a layer

The **SWITCH** command can be used to assist in the dimensioning and labelling of 3-dimensional models. With the **SWITCH** command, layers, UCS and viewpoints are modified so that dimensioning can take place immediately in each view.

Format:

Command: **SWITCH**

Graphic/Text/<Dim>:

<Top>/Front/Left/Right/ View: *select the view direction you require*

Origin of the UCS <0,0,0>: *digitise a point of the object*

### Options:

**Graphic:** activates the main graphics layer of the current package and resets the UCS. This is the same as the previous editing status.

**Dim:** sets layer for dimensioning.

**Text:** sets layer for texts.

The query which follows determines the view direction. It is not displayed if you have previously selected Graphic:

The layer names which are then set are formed so that they describe the current mode:

e.g. with dimension "M" and view direction "TOP", the layer is called MTOP

The origin of the UCS should be as close as possible to the objects which you want to dimension. Using the object snap mode (Endpoint, Base ...), pick the element you want to dimension. If you wish to dimension the same object in another view, confirm the entry with ↵ when you are asked for the origin of the UCS.

**See also:** [PREPLOT](#)

## 7 FMSSOFT DEFINITIONS

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[Lisp](#)

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## 7.1 Module

Modules are objects which can be used as basic graphics elements for interactive assembly on screen. They are simplified three-dimensional drawings of **BOSCH FMS products**.

Modules can be arranged in groups of modules. The module group determines the module name, which means, for example, that you can find all types of profile under the module name PROFILE and all types of gripper pan under the module name GRIPPER PAN.

All the variations of shape and design are included, and the modules supplied can be supplemented with special constructions and also combined to form new user-defined modules.

Precise connection to existing modules is ensured with defined [connection points](#) and special FMSsoft functions.

FMSsoft differentiates between three kinds of module in its depiction of them:

- Graphic modules are visible in the drawing and are entered in the [parts list](#)
- Non-graphic modules are not depicted in the illustration. They are only indicated by an entry in the [parts list](#) (e.g. [accessories](#)). Non-graphic modules are indicated by the letter Z in the illustration.
- Complex modules are composed of graphic and non-graphic modules. They are standard [macros](#) generated by the program.

## 7.2 Macro

A macro is a [LISP](#) program which defines and uses FMSsoft [modules](#).

## 7.3 Physical feature

Physical features define a [module](#) in greater detail. They may be, for example, width, length or the number of packing units.

The physical features for an assembly can be determined individually for each module. The Physical features are requested in a dialogue window, tested for correctness and stored in the drawing. The stored Physical features can be displayed at any time.

## 7.4 Parts list

During the interactive construction process, a parts list is created simultaneously by the system, i.e. the [parts list information](#) for each [module](#) positioned in the drawing can be accessed immediately and is available for creating temporary parts lists and at the end for drawing up the final parts list. You can then order the parts required using the list which is then issued. Since everything is drawn up automatically, there is no risk of any misunderstandings arising about your order.

Of course you can make additional entries in the parts list and change the given information.

### 7.4.1 *Parts list information*

Parts list information consists of:

- Item number
- Number
- Name
- Order number
- Comment
- Physical feature
- Identification
- Price

This data is “appended” to all modules, both graphic and non-graphic, as an attribute.

## 7.5 Identification

The identification is used primarily to differentiate between products from different product lines (MGE, MAS and TS). You can, however, also give your own identification with the command ASSEMBLY, to develop the structure of the parts list still further.

## 7.6 Connection point

Connection points are points which are depicted in addition to the graphics of the module to simplify the addition of further construction elements. Points can be accessed with the object snap mode NODE. Connection points are positioned in their own layer.

## 7.7 DWG

DWG (drawing) is the file name extension for AutoCAD drawings.

## 7.8 Layer structure

In AutoCAD you can arrange various drawing elements on different layers (levels of a drawing). Each of these layers has its own colour and type of line. This enables you to structure drawing information, for example, dimensions, text and axes can be positioned in separate layers, as can buildings, conveying technology or telephone lines. This type of structure is known as layer structure.

## 7.9 Element layer

If blocks are inserted in a drawing, the layer which is currently selected at the time of insertion is the element layer of the block reference. The same applies for all other elements (e.g. circles, lines, etc.). In other words, the current layer when the element is created is the element layer. The command **LIST** displays the element layer as one of the first pieces of information.

## 7.10 Block definition

If several elements are combined to form a block, a so-called block definition is created, which can be accessed with the name of the block. When a block is inserted this definition is only indicated and a block reference created.

## 7.11 UCS

UCS (**U**ser **C**o-ordinate **S**ystem).

You can place your own system of co-ordinates at any point in an area and position it as required. This system of co-ordinates you have defined is called the user co-ordinate system (UCS).

There is also a permanent system of co-ordinates, the world co-ordinate system (WCS).

## 7.12 Accessories



Accessory parts are only registered as parts list information in FMSsoft and not depicted as graphics. Accessories are denoted in drawings by the symbol shown here to the left. It may indicate, for example, conveyor section connectors, foundation sets, brackets and connecting kits.

## 7.13 LISP

LISP or AutoLISP is the first language used for programming AutoCAD. You can use it to write macros and functions, thus extending the range of uses for AutoCAD. Lisp can be learned quickly and easily and is extremely flexible.

As well as using AutoLISP, you can program AutoCAD in other languages using the ADS interface.

## 7.14 Assembly

An assembly is a group of modules with the same identification.

## 7.15 Wildcard characters

Wildcard characters are especially useful in the search for the right name for modules and components. The most important wildcard character is the \* (asterisk). It stands for any character string and can be used anywhere in the search character chain.

### Examples:

Search command <\*>: \* *lists all modules*

Search command <\*>: **SCH\*** *displays all modules that begin with the letters "SCH"*

Search command <\*>: **S\*H\*** *modules which begin with "S" and have an "H" somewhere in their name*

Place holders are also suitable for use in BOSCH catalogue dialogue.

## 7.16 View

The viewing direction is based on the world co-ordinate system, WCS.

<b>View from</b>	<b>You are looking towards</b>
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Above	-Z
Right	-X
Left	+X
Rear	-Y
Front	+Y

## 8 TECHNICAL SUPPORT:

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