1 MGEsoft

BOSCH REXROTH`s MGEsoft is a powerful software package for planning, design and construction of supporting frames using the aluminium profile system MGE (Basic mechanical elements) from BOSCH REXROTH.

On the one hand MGEsoft provides the components of the BOSCH REXROTH MGE modular assembly system and on the other hand, functions which assist in the planning and construction process.

This assistance ranges from the calculation of profiles and linear guides, to tools for drawing protective barriers and doors to automatic linking of MGE elements which makes construction work sensationally easy and fast.

The whole package is supplemented by many functions which simplify the handling of AutoCAD.

This manual explains the menus and special functions of MGEsoft.

2 Table of content

1 MGEsoft 1
2 Table of content 2
2.1 FMSsoft documentation 5
2.2 Typographical conventions 6
3 overview 7
3.1 3D-construction with MGEsoft 9
1.2 This section provides information about how to find the components for frame design in your menus and how the pull-down-menus are organised. MGEsoft menus 12
1.2.1 Pull-down-Menu 12
1.2.2 *MODULES* 12
1.2.3 *TOOLS* 12
1.3 Where can I find the components in the menu? 14
4 TIPS 18
4.1 Several viewports for greater clarity! 19
4.2 Editing in the top view? 20
4.3 Specified user coordinate systems 21
4.4 Macros for selecting and inserting modules! 22
4.5 Different layers structure the drawing 23
4.6 Changing between the TSsoft, MAS and MGEsoft packages 24
5 MGEsoft PLANING functions 25
5.1 CONNECT 26
5.1.1 Notes 26
5.1.2 Connection point 27
5.1.3 Rotate 27
5.1.4 Move and Copy 27
5.1.5 Groove/Butt/Lcorner/Flip 27
5.1.6 Previous or Next position 28
5.1.7 Exit 28
5.2 POSITION 29
5.2.1 Options: 29
5.3 SPECFIN 31
5.3.1 Add 31
5.3.2 Erase 32
5.3.3 Detail 32
5.3.4 Exit 32
5.3.5 Notes on the parts list 32
5.4 ELINE 34
5.4.1 Dimensions 35
5.4.2 Profile 35
5.4.3 Profile length 35
5.4.4 Insertion position 35
5.4.5 Insert profile 35
5.4.6 Load type 36
5.4.7 Insert bearing 36
5.4.8 Force 36
5.4.9 Insert force 36
5.4.10 Erase 37
5.4.11 Move 37
5.4.12 Calculate 37
5.4.13 Representation 37
5.5 BCALC 38
5.5.1 Procedure: 38
5.5.2 Levelling foot data 39
5.5.3 Basis for the calculation: 39
5.6 LG 40
5.6.1 Procedure 41
5.6.2 Type 41
5.6.3 Dimensions 41

5.6.4 Load 41
5.6.5 Bearing distance Fix 42
5.6.6 Bearing distance List 43
5.6.7 Status 43
5.6.8 Generate 43
5.6.9 eXit 43
5.6.10 Determine loads 44
5.6.11 Depiction and parts list 44
5.6.12 What conditions must be fulfilled? 44
5.6.13 Tips 45
5.6.13.1 Insert bearing correctly: trolley vertical! 45
5.6.13.2 Use bearing distance LIST 45
5.6.14 Tasks in the LG calculation and the relevant program options 46
5.6.15 Interpretation of the calculated safety factor 47
5.7 CANGLE 48
5.8 FRAMETRAY 50
5.9 DOOR 51
5.10 TABLE 52
5.11 PROTECTIVE BARRRIER 53
5.12 New Technology starts with 40/50 Profiles 54
5.13 Technical Support 55

2.1 FMSsoft documentation

The FMSsoft documentation is composed of various different parts:

- FMSsoft Installation Manual 5.9b
- Package-specific manual, e.g. MGEsoft 5.9b
- 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

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

3 overview

Over the next few pages you will find a brief introduction to MGEsoft and what it has to offer you.

3.1 3D-construction with MGEsoft

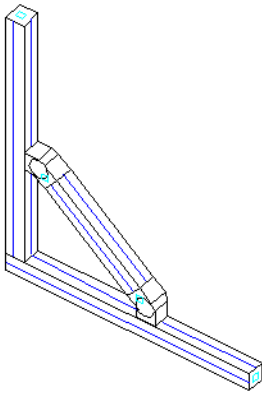
MGEsoft is the optimum design and construction tool for anyone who designs and builds supporting frames with BOSCH REXROTH basic mechanical elements. With more than ten years of experience in the development and use of 3D applications we don't just talk about 3D construction, we do it!

Optimum handling during construction

You are working in the isometric view. You thus see the frame "grow" as though you were screwing profiles together yourself.

Since the FMSIntelligent functions make your construction work easier:

- Frame reinforcements with joints or 45° connectors can be created in seconds without any calculations
- "Connecting" profiles and their connecting elements (e.g. brackets) is greatly simplified as the elements know their possible connections and are even placed automatically at the profile grooves. This requires a construction technique a little different from "normal" 3D construction, but can be learned quickly and its speed is unmatched. A time saving of about 70% is possible.
- Aside from these connection functions there is also a positioning command which makes rotation, moving, copying and aligning so easy, that you will never again want to draw in 2D!



That is virtual prototyping on your PC!

Convenient changing of components

FMSsoft modules are not "dumb" lines or areas, but "intelligent parametric objects. That is why it is so easy for example to change the cross-section, length or type of profile. Simply "click"!

This even allows you to change the dimensions of entire frames by "stretching" using a single command. The parts list information is automatically updated.

Calculating the accessories

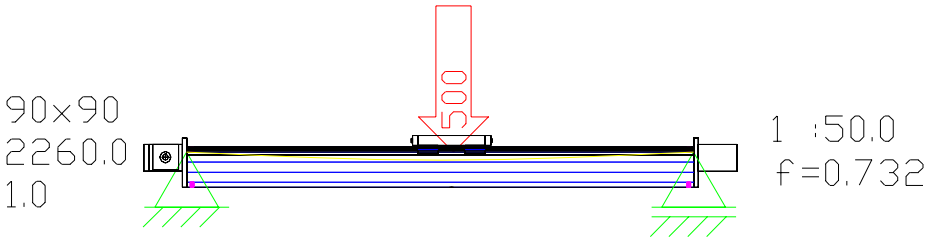
The required number of small parts such as screws or connecting elements can be calculated before creating the parts list. Nothing is omitted!

Calculation functions cancelled!

The calculation and depiction of the elastic lines of profiles which are loaded by any number of forces, gives you a reliable evaluation of your construction.

The calculation of the buckling of axially pressure-loaded profiles is also possible.

Linear guides can be designed optimally using MGEsoft. Your constructions are cheaper and safer.

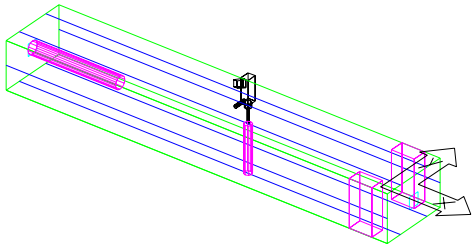


Depiction of sections

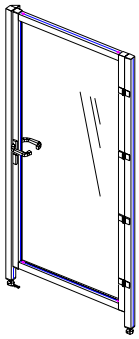
Sections through frames allow detailed investigations and the generation of 2D drawings for production. By means of additionally applied profile cross-sections there are no unanswered questions left.

Special finishing

A special function assists you in drawing in special finishes such as a bore in the profile. The parts list information is updated such that in fact no additional manufacturing drawings have to be made. However, you can generate detailed drawings which are fully dimensioned and you can obtain all information necessary for the finishing in the form of a table.



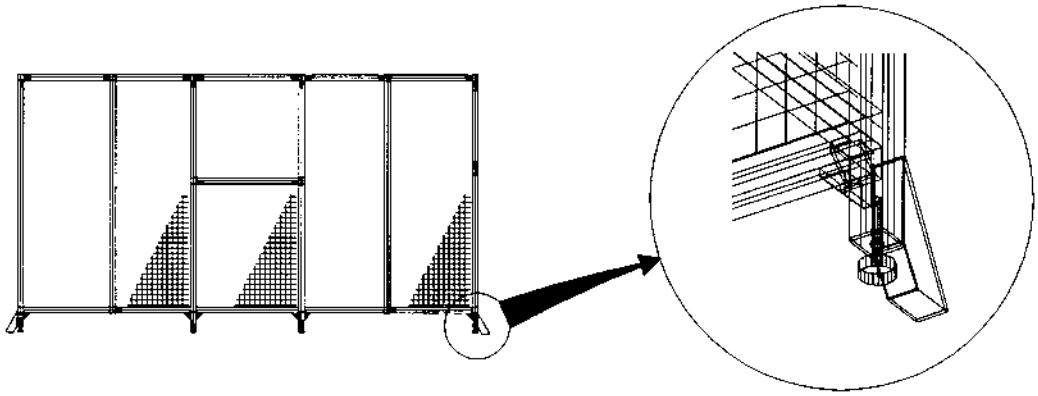
Macros

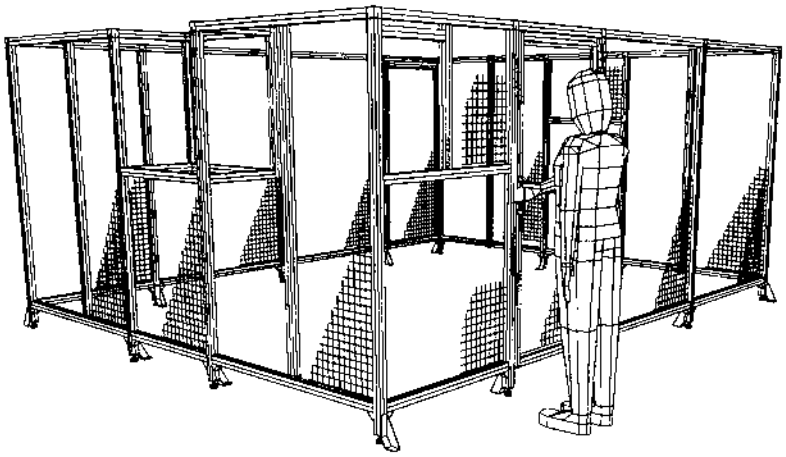


If you frequently have similar frames such as protective barriers, table frames or protective hoods to design, you can save 90% of construction time by using macros. The programs are written by you in AutoLISP and save you the tedious job of constructing similar frames. At the same time the quality is enhanced by using standards. Ready-to-use macros for tables, doors and material levels increase your productivity and serve as patterns for your own applications.

3D-drawings in manufacture

For your manufacture you no longer need to make detailed drawings! Experience has shown that 3D drawings and parts lists are fully sufficient for the fitter.





3.2 This section provides information about how to find the components for frame design in your menus and how the pull-down-menus are organised. MGEsoft menus

3.2.1 *Pull-down-Menu*

Two submenus have been added to the standard AutoCAD menu: *MODULES* and *TOOLS*

3.2.2 **MODULES**

The ***MODULES*** menu is for selecting and inserting construction elements and, like the current MGE catalogues, is divided into:

Looking for parts
BOSCH Catalogue

MGE-components
Profiles...
Connections
Joint...
Doors...
Prot. Barriers/sep. Walls...
Feet and wheels...
Air supply
Linear guide...
Conveyor Sections
Installation...
Miscellaneous....

3.2.3 **TOOLS**

The ***TOOLS*** menu contains all other functions.

Moving from one FMSsoft package to another:

Package
MGEsoft
MASsoft
TSsoft

Draw
Cnode
Form
Preplot

DIM Layer
Planview
Sideview
Frontview
TEXT Layer
Planview
Sideview
Frontview
GRAPHIC Layer

Modblock

Image
Fade
Clip

	Switch
	Modlay
Change	Change Module
	Stretch Module
	Module Name
	Disass
	Join

Parts list functions:

Partlist	Plist
	Magnifier
	Item no.
	PriceEdit
	PriceUpdate
	ChangeLanguage
	Assembly

Multiple windows:

Views	Initialise
	Reset
	Single viewport
	Swap

Edit functions

Construct	Position
	Connect
	Conangle
	2D-move
	2D-copy
Layer	Change layer
	Set layer

MGEsoft special functions

Planning functions	Special finish

Online command reference

MGEsoft help

About MGEsoft

MGEsoft info

and for a little light entertainment....

Game

Information about the planning functions and how to use the functions for the multiple window (see TIPS on page 18) is provided in this manual. For information about the other functions please see the FMSsoft manual 5.9b or the Online Help.

3.3 Where can I find the components in the menu?

You have found a component in the FMS catalogue you would like to use and now you want to know how to find it as quickly as possible in FMSsoft.

There are several ways of looking for a component:

- Using the Search function you can look for the part by order number or name. The command to call the search function is: **BOSCH**.
- You can use the CHOOSE command to display all the modules available (or just certain modules):
  Command: **CHOOSE**
  Module name ?/<name>: **?**
  Search command <*>: ***** *for all modules or a string (wildcard character...)*
  see also: CHOOSE
- You can also use the pull-down menu *Modules*. The submenus are arranged in the same way as the articles in the catalogue.

The following table gives an overview of the module names and their assignment to a submenu of the *Module* menu.

Modul name	**Submenu....**
A	
Adapter	Linear guide
Air_bar	Air supply
B	
Bearing_pedestal	Joints
BF_Assembly_Set	Conveyors
BF_Belt	Conveyors
BF_Cover_Strip	Conveyors
BF_Drive	Conveyors
BF_Gear	Conveyors
BF_Node_Plate	Conveyors
BF_Roller_Set	Conveyors
BF_Slide_Plate	Conveyors
Bracket	Connecting technology
Bracket_accessy	Connecting technology
Bumper	Doors
C	
Cable	Doors
Cable_accessories	Installation
Cable_duct	Installation
Cap	Profiles
Cap	Doors
Cap	Feet and wheels
Cap	Linear guide

Modul name	Submenu....
Clamp_busching	Prot. Devcs/sep. walls
Clamping_claw	Connecting technology
Clamping_lever	Joints
Connection_piece	Air supply
Connector45	Connecting technology
Corner_piece_30	Connecting technology
Corner_piece_45	Connecting technology
Counterbalance	Doors
D	
Damping_ring	Feet and wheels
DIN_Parts	Connecting technology
DIN_Parts	Doors
DIN_Parts	Feet and wheels
Divot_Bearing	Joints
Door	Other
Door_lock	Doors
E	
End_connector	Connecting technology
F	
Fastening_element	Connecting technology
Fastening_element	Feet and wheels
Fastening_set	Feet and wheels
Flanged_plate	Air supply
Floor_dowel	Connecting technology
Floor_dowel	Feet and wheels
Folding_door_elem	Doors
Foundation_bracket	Connecting technology
FRAMETRAY	Other
H	
Handle	Doors
Handle_bar	Doors
Hexagonal_shaft	Linear guide
Hinge	Doors
Hinged_foot	Feet and wheels
I	
Inner_bracket	Connecting technology
J	
Joint	Joints

Modul name	Submenu....
L	
LF_Belt_connector	Linear guide
LF_Clamp_profile	Linear guide
LF_Cover_profile	Linear guide
LF_Drive_head	Linear guide
LF_Guide_profile	Linear guide
LF_guide_rod	Linear guide
LF_Head_plate	Linear guide
LF_Support_bearing	Linear guide
LF_Synchronous_shaft	Linear guide
LF_Tensioning_Head	Linear guide
LF_Toothed_belt	Linear guide
LF_Trolley_profile	Linear guide
LG	Other
Lock	Doors
Longit_end_connector	Air supply
M	
Magnet_catch	Doors
P	
Pane	Prot. devcs/sep. walls
Panel	Prot. Devcs/sep. walls
Pinion	Linear guide
Plastic_profile	Profiles
Plastic_profile	Doors
Plastic_profile	Prot. Devcs/sep. walls
Plate	Prot. Devcs/sep. walls
Plate_holder	Connecting technology
Profile	Profiles
Profile	Doors
Profile	Prot. Devcs/sep. walls
Profile	Air supply
Profile_connector	Connecting technology
Profile_connector	Conveyors
Profile_node	Connecting technology
Prot_grille_corner	Prot. devcs/sep. walls
Prot_grille_prof	Prot. devcs/sep. walls
Protective_Grille	Prot. devcs/sep. walls

Modul name	Submenu....
R	
Rack	Linear guide
Rect_tube	Profiles
Roller_element	Conveyors
RS_Roller	Conveyors
RS_Roller_Fix	Conveyors
S	
Screw	Connecting technology
Screw_connector	Doors
Sealing_profile	Doors
Section_profile	Conveyors
Slide_hanger	Linear guide
Slider	Doors
Slider	Linear guide
Slot_plate	Profiles
SP_Connector	Profiles
Special_profile	Profiles
Special_profile	Doors
Special_profile	Prot. devcs/sep. walls
Spring_hook	Linear guide
Star_knob	Joints
Strap_shpd_handle	Doors
Support_Bracket	Connecting technology
Section_profile	Profile
Support_wheel	Linear guide
Suspension_piece	Prot. Devcs/sep. walls
Swivel_bearing	Joints
Swivel_fastening	Joints
T	
TABLE	Other
V	
Variofix_block	Prot. devcs/sep. walls
W	
Wheel	Feet and wheels

4 TIPS

When you use a computer for design work things tend to get very complicated, and you often feel there are so many lines that you can't see anything. Here are a few tips to help you keep track of things and work more quickly with FMSsoft:

4.1 Several viewports for greater clarity!

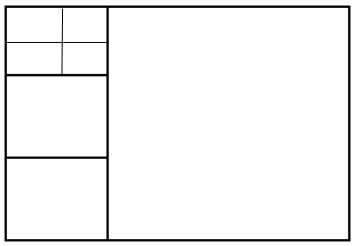
You should try and begin to work in several viewports as early on in the design process as possible e.g. as soon as you have inserted a work table or a ManModel. In FMSsoft you can use the **Views** option in the ***Tools** pull-down menu for this purpose.

This includes three functions:

Initialise Initialises arrangement of the viewports with the front, top and side views in three smaller viewports and the ISO view in the main viewport

Reset Returns to the last viewport arrangement saved after the last initialisation.

Swap Swaps the view in a smaller viewport to the main viewport.
Example: the top view is in a small viewport. Your cursor is also in this viewport. On the SWAP command, the top view is also shown in the main viewport.



Single Viewport Use this to return to single viewport viewing. The view for the resulting single viewport is inherited from the current viewport.

It is also very sensible and useful to use several viewports during ergonomic tests with the Manmodel.

4.2 Editing in the top view?

If so, you should certainly use the 2DMOVE and 2DCOPY commands. These commands ignore the Z co-ordinates of the digitised points and prevent the elements which were previously on the table ending up on the floor.

4.3 Specified user coordinate systems

When positioning modules, e.g. at table-top height, you should first set the UCS to the right height. To do so, use the AutoCAD command UCS |Origin and select a new point of origin, e.g. the top left-hand corner at the front of the table top. Then save this UCS with the UCS command Save option. You can use a name like T1000, where T stands for table and 1000 for the table height.

4.4 Macros for selecting and inserting modules!

The following macros are included in TSsoft:

FRAMETRAY
DOOR
TABLE
CANGLE
LG

You can use these to insert not only individual elements, but also accessories and related modules which speeds up the entire selection and positioning process considerably.

In addition to the macros, which do not have any superior parts list information, the following module(s) are realised in MGEsoft as complex module(s):

Profiles with SPECIAL FINISH (☞ p. 31)
AIR_BAR
DIVOT_BEARING

4.5 Different layers structure the drawing

It is standard for all modules to be inserted in the MAS layer. They are then shown in the colour of this layer.

In order to work with certain elements only, for example all those on the table top, proceed as follows:

1. Call CHGLAY and select all the parts you want to arrange on the table. Don't forget the table top!
2. Give these parts the new layer name ONTABLE and select the colour CYAN.
3. All the selected elements will now be shown in CYAN.
4. Set the current layer to ONTABLE. This is easiest with the SETLAY command. Now select one of the cyan-coloured elements.
5. Now freeze the MAS layer (AutoCAD-command _DDLMODES)

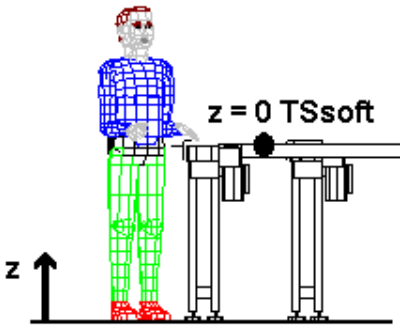
You will now see only the table top and the elements you want to position.

You can, of course, also use this method in a larger scale.

4.6 Changing between the TSsoft, MAS and MGEsoft packages

When changing between the TSsoft, MAS and MGEsoft packages (*TOOLS* | packages...), you must take the following into account.

1. The origin of the co-ordinate system in packages MAS and MGEsoft is at a level of z=0 in the world co-ordinate system. In other words, you could say it is lying on the floor.
2. In TSsoft the origin of the co-ordinate system is at transport level. The (user) co-ordinate system is set by the **START** program using the transport level given there.



Procedure:

- **TSsoft ➔ MAS or MGEsoft**

If you want to move from TSsoft to MAS or MGEsoft, first save your current UCS

Command : **UCS** ↵
Origin/.../Save/..../<World> **S**↵
?/Desired UCS name: *enter a name, e.g. TS980*

Then reset the co-ordinate system to world, so that, for example, a work-table is standing on the floor:

Command: **USC** ↵
Origin/..../<World> ↵

- **MAS or MGEsoft ➔ TSsoft**

If you want to move from MAS or MGEsoft to TSsoft, retrieve the UCS you have stored:

Command: **UCS** ↵
Origin/.../Restore/..../<World> **R**↵
?/Name of UCS to restore: *enter the name of the UCS you have saved: :e.g. TS980*

or use the **DDUCS** command.

5 MGEsoft PLANING functions

MGEsoft's special commands offer comprehensive planning and calculation functionality within MGEsoft:

With CONNNECT components are placed on profiles practically automatically. Assembling frames is made very easy and in particular is very fast.

Editing (rotating, moving, copying) elements in space is supported by the POSITION command. It is then no longer necessary to pre-set the coordinate system.

Special finishing such as additional bores can be generated with SPECFIN.

The elastic line of profiles is calculated by the ELINE program.

The calculation and depiction of linear guides is supported by the LG command.

Frame corner reinforcements with joints and 45° connectors are most easily drawn using CANGLE.

The FRAMETRAY macro draws trays such as those used in material shuttles.

DOOR and TABLE are two other macros with which you can save a lot of construction time.

5.1 CONNECT

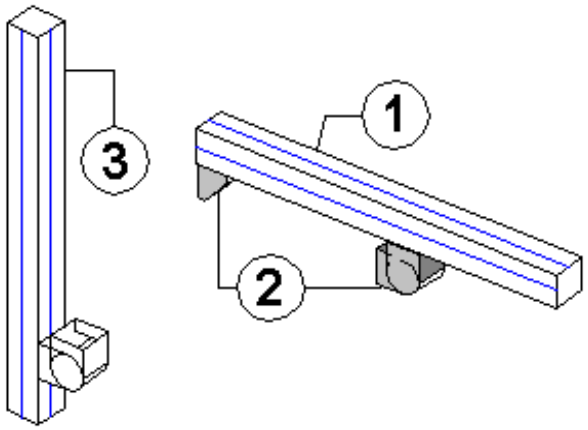
MGE Connects elements at profiles

If one takes a look at the components of the MGE catalogue one notices that in terms of connections a large proportion of the elements are geared towards the strut profile. Thus, angles, hinged feet, joints etc. and of course the profiles themselves, are fixed at defined places on a strut profile. These places are usually the grooves of the profile and the ends. At the ends of the profile there may be butt and corner joints.

From the menu *Tools* Construction select **CONNECT** .
CONNECT is also an option of the Choose command, which you start automatically when selecting a module via the *Module* menu.

At the Command prompt enter **CONNECT**.

Format:
CONNECT
Base element: *Select the element to be connected to another element ❶*
Select objects: *Select additional elements ❷ or* ↵
to object: *Select the element you want to connect the objects previously selected to ❸*



☞ Notes

If you want to connect your component to a profile, once you have selected it the following menu appears:

<Exit>/Rotate/Move/Copy/Previous/Next position/Flip/Groove/Butt/Lcorner
or select connection point:

If you do not connect a component to the profile, e.g. a spring hook with a suspension caster, this menu appears:

<Exit>/Rotate/Next position or select connection point:

5.1.1 *Notes*

- With **CONNECT** as an option of the FMS_MODULE or the CHOOSE program the current element is automatically defined as the base element.
- If **CONNECT** is called as an independent program, you have the option of connecting together entire assemblies. To do this, first select a base element ❶ (e.g. a profile) and then the objects

belonging to the assembly. The side on which you digitise profile ❶ is then the relevant side for connecting, i.e. the base element hangs at a connection point on this side.

5.1.2 *Connection point*

Calculate the mutual positioning of the components. If you enter a connection point near a profile groove, the connection program automatically aligns your component with this groove. If you do not have a profile you want to connect to, your component is aligned with defined connection points. That means that you do not give much thought to the positioning of your UCS, but you connect together the components intuitively as when assembling. Connection points can be digitised as often as you like and using different components!

5.1.3 *Rotate*

With Rotate the components are rotated 90° around their z-axis. For profiles, this is the longitudinal axis, for example.

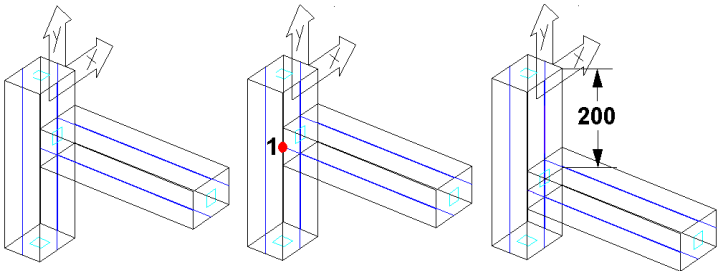
5.1.4 *Move and Copy*

The components can be displaced or copied to their exact end position after being connected to the profile.

On displacing and copying only the groove direction (x-direction) is automatically accepted. The y- and z-coordinates of the points entered are ignored!

That means that you can only enter a point 200 mm from the end of the profile by entering -200 - with reference to the coordinate system.

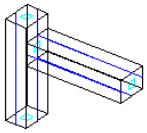
If a component has been copied, it is then available again for other connection processes.



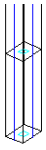
from point: *first point*
to point: *enter value (-200 in the example) or select point*

If a component has been copied, the copy becomes the active component. The component which was active prior to copying is then no longer available for manipulation.

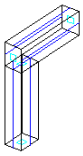
5.1.5 *Groove/Butt/Lcorner/Flip*



"Groove", changes the connection place to the profile grooves.



"Butt" creates a butt connection and



"Lcorner" creates a corner connection at the current profile end.

With "Flip" you can change to the opposite end of the profile.

5.1.6 *Previous or Next position*

If you are in a groove connection you can use the **Previous** and **Next** options to jump from groove to groove until you have found the right insertion position. With corner connections the next possible corner connection is created accordingly. With components which are not profiles, this option is used to jump to the next connection point.

5.1.7 *Exit*

With Exit or ↵ you leave the connection program.

5.2 POSITION

MGE Rotates, moves and copies objects in space

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

The **POSITION** command simplifies the positioning of objects in space and considerably reduces the number of entries to be made.

 From the menu *Tools* Construct, select **Position**

 At the Command prompt enter **Position**.

Format:
Position
Base element: *Select a point on an object to which all subsequent operations refer.*
Select objects: *Additional objects to be moved along with the base element.*
<Exit>/Align/Xrotate/Yrotate/Zrotate/Move/Copy/Undo:

This menu allows you to position a module in space without having to manipulate the UCS using standard commands. And you do not need to set any UCS beforehand either!

5.2.1 *Options:*

Exit:	↵ leaves this menu
Align	Realigns the element. The new X-direction arises from the current base point (zero point) and the alignment point (3D!)
X/Y/Zrotate:	Rotates the module around the appropriate axis with reference to the overlaid UCS.
Move:	Moves the module
Copy:	Copies the module
Undo:	Undoes the "Operations" up to the start of the command.

5.3 SPECFIN

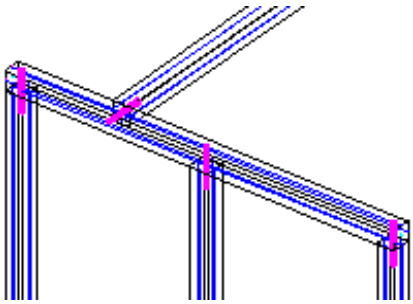
MGE Applies special finishes (e.g. bores) to profiles

Profiles often require special finishing in addition to standard end finishing in the form of cross bores of various kinds. With the **SPECFIN** command these bores can be applied to MGE profiles.

From the menu *Tools* Planning functions select **Special finish**.

At the Command prompt enter **SPECFIN**.

Format:
SPECFIN
select profile: *Click on the profile you want to edit*
Special finish Erase/Add/Detail/<eXit>: *Select an option*



Notes on the parts list

5.3.1 *Add*

This option adds bores to the profile which cannot be ordered with bores at this position or of these sizes. The entire designation must always be entered, e.g. D9.5 or D17-26.

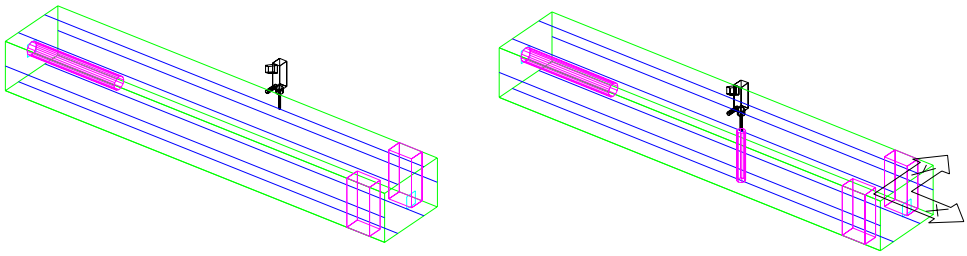
Special finish Other/D5/D6.5/D9.5/D13-20/D17-26/D20-31: *Select an option*

Other
Diameter1:
Height1:
Diameter2:
Height2:

select location: *Click a groove, for example*
<Exit>/Rotate/Move/Copy/Previous/Next position/Flip/Groove or select connection point:

All options are identical to those of the CONNECT command.

☞ In order to define the exact **position of the bore**, use the Move option.
When prompted "from point" select the object snap "Insertion" and click the drill.
The coordinates of the target point "to point" refer to the overlaid coordinate system. When entering the point you only need to enter the X-coordinate (e.g. -200), as the Y- and Z-coordinates of the point are automatically set to zero.



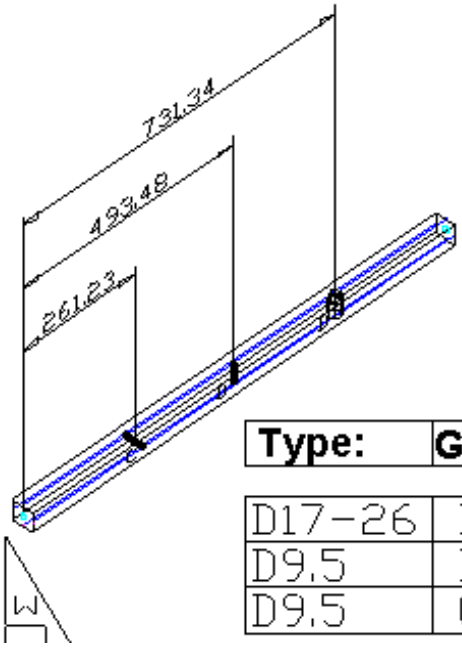
5.3.2 *Erase*

select special finish: *Click on a special finish*

5.3.3 *Detail*

The detail drawing is generated in the paper area. It consists of a depiction of the profile which for this purpose is copied, dismantled into its component parts and dimensioned and a table containing the short designation, a page number and the dimensions.

Basepoint of insertion: *Select a point at which the detail drawing is to be inserted*



Type:	Groove	Distance:
D17-26	D	731.34
D9.5	D	493.48
D9.5	C	261.23

The table is shown here enlarged. You can adapt its size using the AutoCAD command SCALE.

See also: **PSPACE (AutoCAD)**
SCALE (AutoCAD)

5.3.4 *Exit*

With EXit or ↵ you leave the special finish program.

5.3.5 *Notes on the parts list*

The special finishes are each stored as individual blocks with parts list information, just as modules can be created with the MODBLOCK command. The module (profile) which is given a special finish is unchanged in the

process! For the special profile to become a special profile the basic profile and the special finish are summarised once more in a block.

The parts list depicted shows this structure. In order to obtain all information about a special profile of this kind it is thus necessary to create the parts list (PLIST command) with the type **STRUCTURE**.

```
+-------------------------------------------------------------------------+
|  1  |  2  |        3          |          4           |      5       |
|-----+-----+-------------------+----------------------+--------------|
| Item| QTy.|     Description    | Orderno.              |   Remarks     |
|-----+-----+-------------------+----------------------+--------------|
|     |    1| SPECIAL PROFILE    | XXXXXXXXXXXXX         |              |
|-----+-----+-------------------+----------------------+--------------|
|>    |    1| PROFILE 60X90      | 3 842 990 472         |              |
|     |     |L= 500.0 MM;F2/M16; |                       |              |
|-----+-----+-------------------+----------------------+--------------|
|>    |    1| SPECIAL FINISH     | XXXXXXXXXXX           |              |
|     |     |D17-26;SIDE=B;X=191.3MM;                    |              |
|-----+-----+-------------------+----------------------+--------------|
|>    |    1| SPECIAL FINISH     | XXXXXXXXXXX           |              |
|     |     |D9,5;SIDE=A;X=300.0MM;                      |              |
|     |     |                   |                      |              |
|-----+-----+-------------------+----------------------+--------------|
```

5.4 ELINE

MGE Calculates elastic lines and max. profile deflections

The elastic line program ELINE calculates the maximum deflection of the profile and draws the elastic lines.

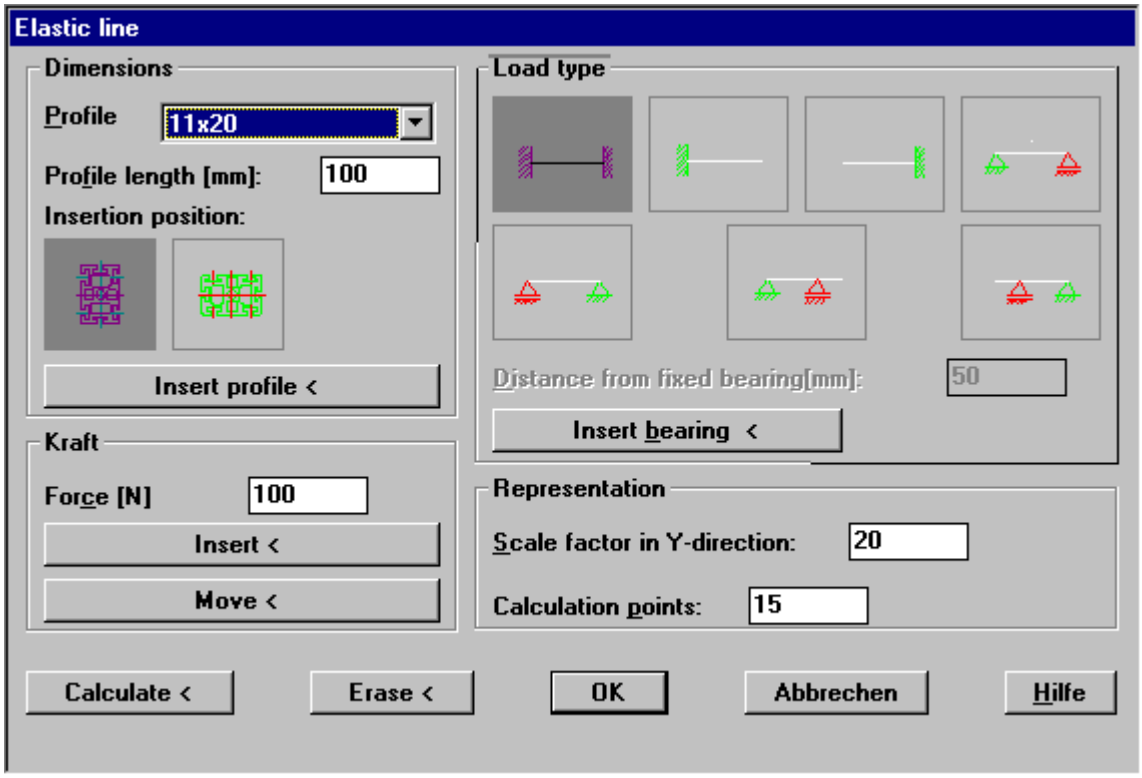
From the *Tools* menu Planning functions select Elastic line .

At the Command prompt, enter **ELINE**.

Procedure:
On entry of the command a dialog box is opened, in which a load type can be defined. A profile must be completely defined in order to be able to calculate it, i.e. selection of the profile, length and insertion position for rectangular profiles, definition of the load type and insertion of the forces acting on the beam (this can be omitted in which case only the deflection caused by the unloaden weight will be calculated).

The program calculates the deflection but not the diffusion of stress and moments or the pressure due to the load. The unloaden weight of the profiles is taken into account in the calculation.

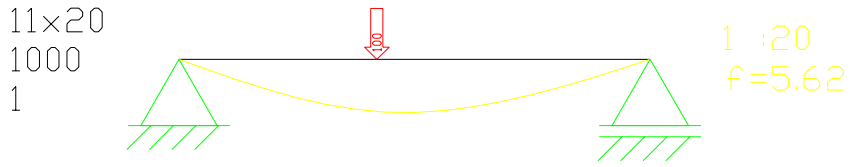
An elastic line can be modified by defining new forces, moving the forces on the profile or deleting existing forces.



Description of the dialog box

- Dimensions
- Profile length
- Insert profile
- Insert force
- Calculate
- Erase
- Representation
- Profile
- Insertion position
- Load type
- Force
- Insert end condition
- Move

Result:



5.4.1 *Dimensions*

The profiles are defined with the functions Profile, Profile length and Insertion position.

5.4.2 *Profile*

BOSCH REXROTH strut profiles are predefined in a list. If a different profile is to be calculated, "Other profile" is selected from the list. This command causes a second dialog box to be opened, in which another profile can be defined. The height, width, elastic modulus, weight (kg/m) and geometrical moment of inertia Ix and Iy must be entered. The values entered are stored by pressing <OK> and the dialog box "Other profile" is closed.

5.4.3 *Profile length*

At Profile length the length of the profile to be calculated is entered.

5.4.4 *Insertion position*

If a non-quadratic profile is selected, the profile can also be rotated by 90° and calculated. If a quadratic profile is selected this function is disabled.

5.4.5 *Insert profile*

When all measurements have been entered, the profile is inserted into the drawing.

Base point of insertion: *click on screen or enter value*

After insertion the origin of the UCS is at the point of application of the force!
On the left are the description of the profile, its length and a 1 for a vertically inserted profile and a 2 for a horizontally inserted profile.

5.4.6 *Load type*

Seven different load types are available. A load type must be selected by clicking one of the pictures. Movable bearings can be positioned later at a certain the distance from the fixed bearing.

The following fixing options are available:

- Profile fixed at both ends
- Fixed bearing on the right
- Fixed bearing on the left
- Fixed bearing on the right and movable bearing on the left
- Fixed bearing on the left and movable bearing on the right
- Fixed bearing on the right and movable bearing at a distance
- Fixed bearing on the left and movable bearing at a distance.

If a movable bearing is to be displaced, "Distance from fixed bearing“ is enabled.

5.4.7 *Insert bearing*

Inserts the selected bearing into the drawing.

Select profile: *click on profile*

5.4.8 *Force*

After the profile and the fixing method have been selected, the origin of the coordinate system is at the starting point of the profile.
The force to be applied can only act vertically on the profile. The point of application of the force can be entered by selection with the mouse (OSNAP next) or by entering the point as an absolute value.
One or more forces can be defined and drawn on the profile.
When entering the point the x- and the y-coordinates must be entered, and the y-value has to be zero!!

Example :
Point of application of force : **2000.0**

Hint: If the point of application of the force is too near to the bearing, the force should be applied to the profile and then moved.

Insert force

5.4.9 *Insert force*

Inserts the defined force into the drawing.

Select profile: *click on profile*

Point of application of force: *enter value or click on profile*

5.4.10 *Erase*

For erasing any object. Functions like the AutoCAD ERASE command.

5.4.11 *Move*

For moving forces or the movable bearings. Functions like the AutoCAD MOVE command.

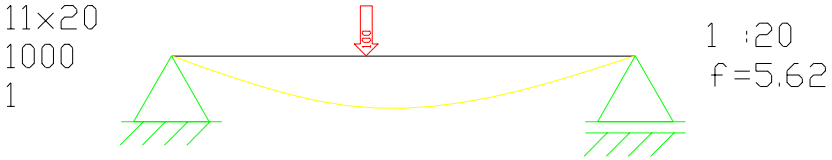
5.4.12 *Calculate*

Once a profile has been defined (Dimensions/Load type/Force) the calculation can be carried out.
If an elastic line runs very close to the beam, it is sometimes not possible to click on the profile.
This problem can be bypassed by selecting one of the numerical data at the profile. When the line is marked by clicking a prompt may appear to erase the old elastic line. Pressing <Yes> or ↵ will erase the old elastic line and draw a new one.

5.4.13 *Representation*

Scale factor in Y-direction:
As the deflection of the profiles is generally very small, to make the elastic line visible it is necessary to scale the functional values (Y values) of the elastic line with a factor. The edit field "Scale factor in Y-direction:" is used for this purpose.
A value of 20 has proved practical in many cases. Values between 1 and 100 can be entered.

Calculation points:
To represent the elastic line and to determine the maximum deflection f a certain number of calculation points is required. The more points there are, the more accurately the elastic line will be drawn, but the longer the calculation takes as well. The option "Calculation points:" lets you enter the number of points to calculate.
The default value is 15.



5.5 BCALC

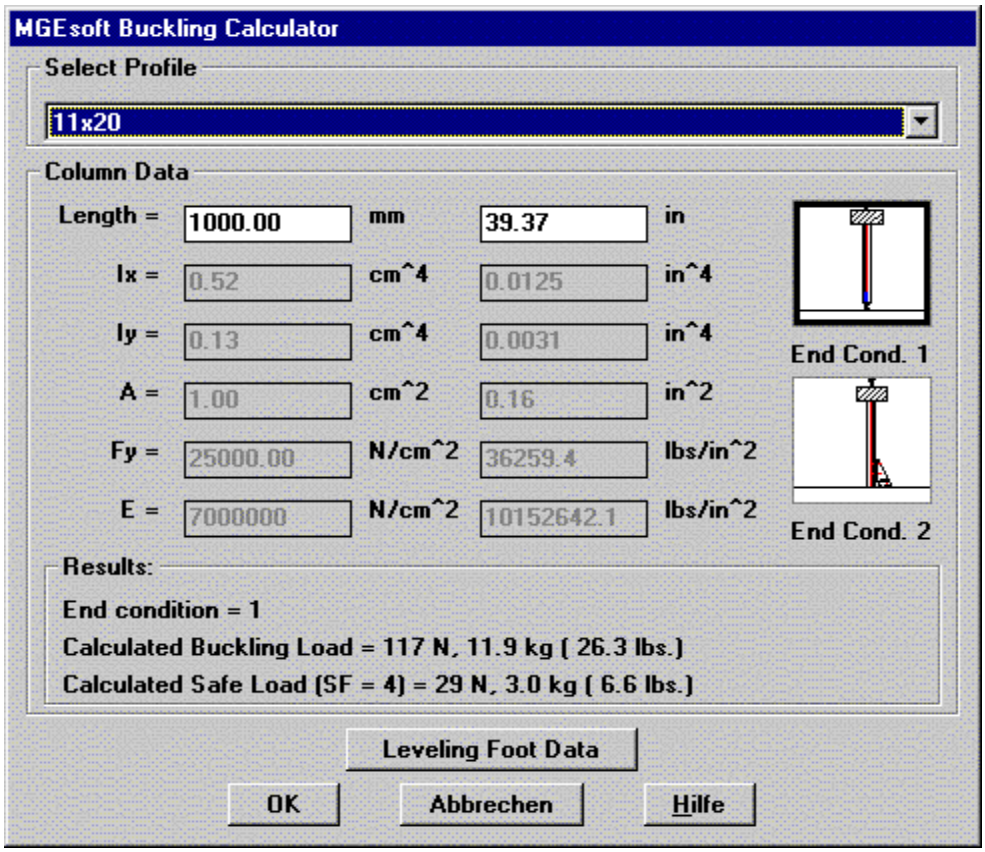
MGE Calculation of the buckling load of profiles

The BCALC program calculates the permitted load with pressure loading before failure due to buckling occurs. Depending on the profile and the type of fixing the program calculates the buckling load and the safe load (with safety factor 4). However, here, the maximum permitted compressive forces of the levelling feet are not taken into account. You can get the Levelling foot data by pressing the button labelled thus.

From the menu *Tools* Planning functions select **BCALC** .

At the Command prompt enter **BCALC**.

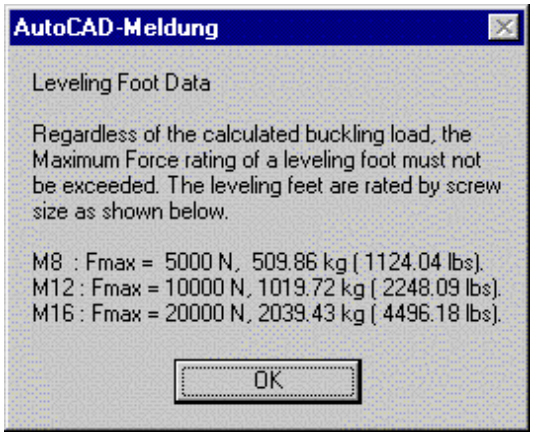
Procedure
Levelling foot data
Basis for the calculation:



5.5.1 *Procedure:*

- Select a profile.
- Enter a length
- By clicking one of the pictures select an end condition similar to the problem (1 or 2).
- The calculation results are displayed in the dialog box area labelled thus.

5.5.2 *Levelling foot data*



5.5.3 *Basis for the calculation:*

- **MGE profile data and material data**
- **Formulae:**

If *Kl/r* < *Cc*

$$Fa = \frac{\left[1 - \frac{(Kl/r)^2}{2Cc^2}\right]Fy}{\frac{5}{3} + \frac{3(Kl/r)}{8Cc} - \frac{(Kl/r)^3}{8Cc^3}}$$

If *Kl/r* < *Cc*

$$Fa = \frac{12\pi E}{23(Kl/r)^2}$$

where

Fa = Permitted buckling strain

K = Factor for effective buckling length:
K = 1 for buckling case 1 (hinged below, above only movable along axis)
K = 2 for buckling case 2 (fixed below, free above)

l = Length

r = Radius of gyration

Fy = Yielding point

E = Elastic modulus

Cc= Coeff. of fineness $Cc = \sqrt{\frac{2\pi^2 E}{Fy}}$

Ref.: Manual of Steel construction, 7th Edition, Courtesy of America Institute for steel construction

5.6 LG

MGE Lays out and draws BOSCH REXROTH linear guides LG

Here it is possible to check the load on the supporting bearing for linear guides LF6,12 and 20 by calculation on the basis of a given stress. The calculation is carried out on the basis of a calculation method dictated by the bearing manufacturer.

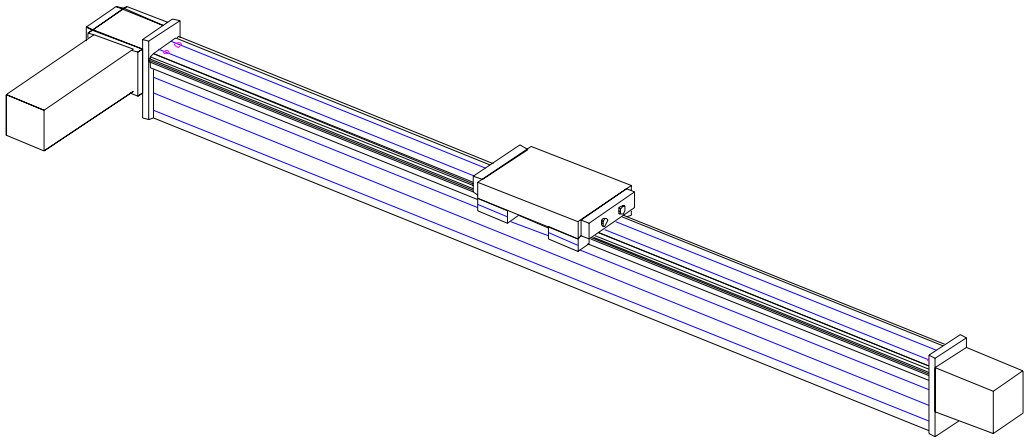
From the menu *Tools* Planning functions select **Linear guide** .

At the Command prompt enter **LG**.

Format:
LG
Linear guide Type/Dimensions/Load/Status/Generate/<eXit> :

Additional information:

Procedure p. 41
Determine stresses p. 44
Depiction and parts list p. 44
What conditions must be fulfilled by the bearing arrangement? p. 44
Interpretation of the calculated safety factor p. 47
Tips:
Insert bearing correctly - trolley vertical! p. 45
For bearing distance use "List“ p. 45



5.6.1 *Procedure*

1.Define the Type and Dimensions - these are obligatory entries.

2.Select the Load option and enter the load data.
First use the bearing distance option <List>. This will give you a good overview as you can see the safety factor and the running performance of 20 different trolley lengths!

3.a)If errors have now occurred (bearing overloaded (SD = 0) etc.) continue at item 1. - but select a larger LG type.

3.b)If there is a trolley length L1 at which no bearing is overloaded and with which the running performance also has the required value, select a suitable trolley length.

4. Now you can either conduct a detailed calculation with the Load option, bearing distance <Fix> or with the Generate option have the linear guide generated directly.

5.6.2 *Type*

Here you select the supporting bearing type 6, 12 or 20 in the version supporting

bearing alone	6, 12, 20
with rail	6S, 12S, 20S
with clip profile	6C, 12C, 20C.

If supporting bearings alone were selected you also have to enter the breadth of the guide rails.

5.6.3 *Dimensions*

Two important quantities for linear guides are the length of the trolley L1 - this is also dependent on the load - and the stroke. These two quantities are defined here.

Note: When conducting a recalculation adopt the default for the minimum trolley length (= 2* supporting bearing breadth)

5.6.4 *Load*

With this option you can check a given LG configuration (type, trolley length, load) by means of calculation.

First the following is entered

Type of load Static/Dynamic:

The next question is to do with the bearing distance:

Bearing dist. Search/List/<Fix>:

- With the **Search** option there follows a query of the safety factor desired. After the loads have been entered the program will try to find a bearing distance at which the bearing with the lowest safety factor just exceeds the desired safety factor.
- With the **List** option after the loads have been entered a list is displayed showing the safety factor SD for 20 different trolley lengths L1 and for dynamic loads also the running performance L in km. If SD is zero, this means that the calculation was not possible as a bearing limit was exceeded. The list is a very good way of obtaining an overview of the load conditions of the selected LG configuration.
- With the **Fix** option, after the load has been entered the selected LG configuration is calculated. The results of this calculation are the bearing forces Fa (axial), Fr (radial) and the equivalent

bearing load P for all four bearings, and the result for the overall system is the safety factor SD of the bearing under the greatest load. For dynamic loads the running performance in km is also displayed.

Before the calculation starts the **loads** have to be entered. These are **forces Fx, Fy and Fz** with the respective **force application points** with reference to the coordinate system of the linear guide - see MGE catalogue - and the **radial stress-free moments** around the X, Y and Z axes.

In addition the results of the LG calculation can be outputted in a file.

5.6.5 *Bearing distance Fix*

The following are displayed

- The load type
- The forces and moments entered
- for the 4 bearings:
  the respective axial load Fa
  the radial load Fr and
  the equivalent bearing load P.

  For Fa and Fr the permitted maximum values are also displayed.
- Result:
  S_min , the safety factor for the bearing under the greatest load
  and for dyn. loads L , the nominal life as a running performance in km.

Example:

```
BOSCH REXROTH LF-roller guide - 1.95 -  LF12  ;90 mm ;L3=    215 mm
Dynamic load
Forces  :      Value[N]    x [mm]     y [mm]     z [mm]
Fx      :       250.0        0.0        0.0       70.0
Fy      :         0.0        0.0        0.0        0.0
Fz      :     -1000.0        0.0        0.0       70.0

Moments :          X          Y          Z
M  [Nm] :        0.0        0.0        0.0

Bearing :   1          2          3          4          max.
Fa  [N] :  209.3      290.7      209.3      290.7      600.0
Fr  [N]:     0.0        0.0        0.0        0.0     5100.0
P   [N] :  795.3     1104.7      795.3     1104.7

Results :
S_min   :       7.51
L [km]  :      43971
```

5.6.6 *Bearing distance List*

Displays the safety factor as a function of the trolley length. This option is above all meant for obtaining a quick overview: ***Is it possible or not?***

L1 [mm]	***S [-}***	
180	***0.0***	
205	***0.0***	
230	***0.0***	***S= 0 : Failure!!***
255	***0.0***	
280	***3.93***	
305	***4.00***	
330	***4.06***	
355	***4.11***	
380	***4.15***	
...		
...		
530	***4.31***	
555	***4.33***	
580	***4.34***	
605	***4.36***	
630	***4.37***	
655	***4.38***	

5.6.7 *Status*

Status displays the current LG type, the breadth of the guide rail(s), the stroke and the length of the trolley L1.

5.6.8 *Generate*

With **Generate** the selected linear guide is generated. After that only the insertion point and the angle of rotation are queried.

The insertion point is in the middle at the underside of the basic profile.

The linear guide is drawn in the set UCS plane.

Rotation around the longitudinal axis is only possible after exiting the program - e.g. with the POSITION command. The linear guide is shown in accordance with the MGE catalogue, the toothed belt is applied here as an accessory.

5.6.9 *eXit*

With ↵ or **X** you leave the LG program.

The data entered are only available temporarily, i.e. as long as you are inside the program.

5.6.10 *Determine loads*

The more accurately the loads which occur are known, the more

reliable the calculation

Loads can be:

- Forces of gravity
  - Accelerating force: Entire moved mass * Acceleration
    Note: *only to be taken into account with dynamic loads*
- external forces
- radial stress-free bending moments

Notes:

- Points of application of the forces: The coordinates refer to the local coordinate system of the trolley
- For masses the common centre of gravity is to be calculated (parallel-axes theorem)

The resulting moments, which arise because forces do not act at the origin of the local coordinate system of the trolley do not need to be calculated!

Finally you should obtain a table containing the resulting forces in the X, Y and Z direction and the coordinates of the points of application of forces and the radial-stress-free bending moments around the X, Y, and Z axes

5.6.11 *Depiction and parts list*

Next the selected linear guide is drawn with the Generate option and the parts list is generated

(PLIST Generate)

5.6.12 *What conditions must be fulfilled?*

The bearing arrangement must fulfil the following conditions:

- **Static safety factor S >= 4**

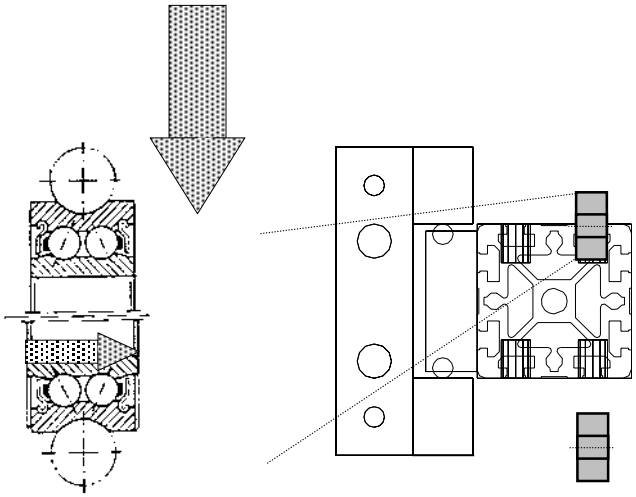
and in addition for dynamic loads

- **Dynamic safety factor S >= 4**
- **and running performance >= 10,000 km**

5.6.13 *Tips*

5.6.13.1 Insert bearing correctly: trolley vertical!

The carrying capacity is much higher in the radial direction than axially!



5.6.13.2 Use bearing distance LIST

In the calculation it has proved particularly useful to use the option **Bearing distance** List.

With this the arrangement of the supporting bearings is not only calculated once, but about 20 times, for the various bearing distances. This list is displayed as a result of the option. This list is the numerical representation of the safety factor as a function of the trolley length.

5.6.14 *Tasks in the LG calculation and the relevant program options*

Task	Program	Option	Queries
1. Preselection	LG	Type	6,12,20, 6S,6C, 12S,12C, 20S,20C only for 6, 12 , 20 : Breadth of the guide rail(s):
		Dimensions	Trolley length L1 Stroke
2. Loads			manually
3. Selection	LG	Load	Type or load: Static
			Bearing distance List
			Forces Fx(x,y,z) Fy(x,y,z) Fz(x,y,z)
			Moments Mx My Mz
			Output of results also in file Yes/<No>: Filename
			Trolley length L1 <
4. Change bearing type			-> 1)
5. Draw LG	LG	Generate	Basepoint of insertion
			Rotation angle
6. Detailed calculation	LG	Load	first Type or load: Static
			Bearing distance Fix
			then Type or load: Dynamic
			Bearing distance Fix

5.6.15 *Interpretation of the calculated safety factor*

Safety S	**Meaning** / measures
S = 0	**Failure of at least one bearing** Select larger bearing type or larger guide rail breadth or larger bearing distance or trolley length *-> Change bearing type*
S >> 4	**Supporting bearing arrangement too large.** Select smaller bearing type or smaller guide rail breadth or smaller bearing distance or trolley length *-> Change bearing type*
S >= 4	**Supporting bearing arrangement has adequate size.** If S is greater than or equal to 4, at the query of the trolley length enter the value next to it. *-> Detailed calculation*
S < 4	**Supporting bearing arrangement too small.** Select larger bearing type or larger guide rail breadth or larger bearing distance or trolley length *-> Change bearing type* or Change insertion position (trolley vertical) *-> Conversion of forces, moments and point coordinates when the insertion position of the trolley changes* or Change construction: use support bearings *->Recalculation of the forces acting on the trolley necessary!*

5.7 CANGLE

MGE Frame corner reinforcements with 45° connectors and joints

The CANGLE program draws frame corner reinforcements consisting of 45° connectors or joints. At least one connector or joint must already be present. All angle and profile lengths are calculated automatically.

From the menu *Tools* Construct... select **CANGLE** .

At the Command prompt enter **CANGLE**.

Format:

CANGLE

Pick 45deg connector or multi-angle joint: *Click one of the two (1)*

- **Joint selected:**

  pickSecond/Newangle/<pickProfile>:

  Option **pickSecond**

  Pick second multi-angle joint: *(2)*

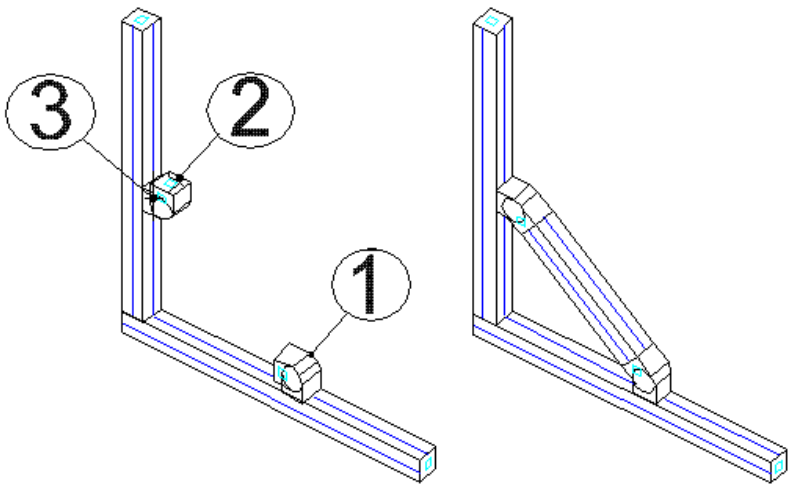
  Option **Newangle**

  Pick second multi-angle joint: *(2)*
  New angle: *type angle (z.B. 45)*

  Option **pickProfil**

  Pick a profile centerline: Select a point on a profile groove (3). At this point the second joint will then be drawn.

  In each of the three cases the joints are re-adjusted and a profile with the correct length and of the correct type inserted.



- **45° connector selected**

45deg connector - Xdist/Ydist/profileLength/<pickProfile>:

Option **Xdist** and similarly **Ydist**

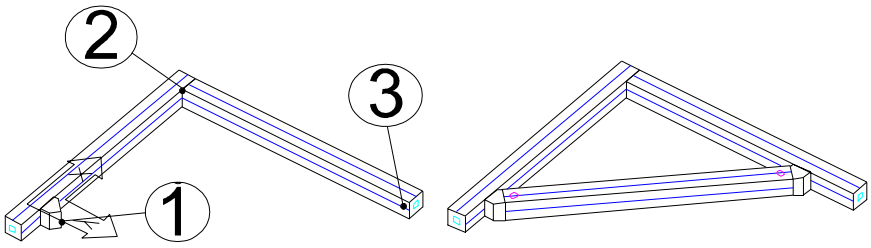
Distance in X direction: Enter a value or click a point. The coordinate system is visible.

Option **profile length**

Profile length: Enter a value or click a point.

Option **Profile**

Pick face point on profile centerline: *Select a point on a profile groove (3). At this groove the second connector will then be positioned.*



5.8 FRAMETRAY

MGE Draws a tray made of frame profiles

With the FRAMETRAY macro a frame tray can be constructed. For this, the length, breadth and thickness of the plate to be integrated are required.

From the menu *Module* Other... select FRAMETRAY

At the Command prompt enter FRAMETRAY.

Format:

Frametray

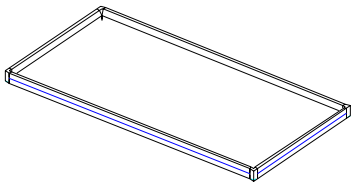
Tray length: *Enter the length of the tray in mm*

Tray breadth: *Enter the breadth of the tray in mm*

Tray depth 1.5/3.0/5.0:*Enter one of the options for the depth*

Basepoint of insertion : *Define a point*

The insertion point is the front left corner of the tray bottom.



5.9 DOOR

MGE Draws a door

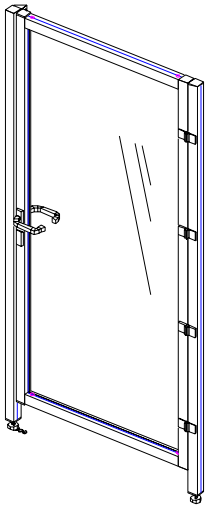
The **DOOR** macro draws a door according to the previously defined dimensions. The door comprises door beams, door frame, cross strut, door lock, panels, cover caps, hinged feet and mounting hardware as accessories.

From the menu *Module* Other... select **Door**.

At the Command prompt enter **Door**.

Format:

Door
Door breadth <900.0>:
Door height <2000.0>:
Cross strut over door Yes/<No>:
Height offset<150.0>:
Handle height <1040.0>:
Cross strut height <without> or value: *this means a cross strut in the door leaf!*
Hinged foot height <45.0>:
Door lock with knob/<door handle>:
Top surface element PLate/PAn/None/Grille/<Makrolon>:
Lower surface element PLate/Makrolon/Grille/None/<PAn>:
Basepoint of insertion :



5.10 TABLE

MGE Draws a table

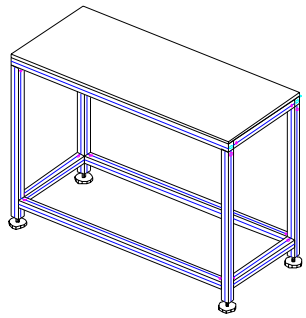
The Table macro draws a table according to previously defined dimensions. The dimensions of the table and the profile cross-section can be selected.

From the menu *Module* Other... select Table.

At the Command prompt enter **Table**.

Format:

Table
Breadth:
Length:
Heigth:
Leg profile breadth 45/60/90/180:
Leg profile depth 45/60/90/180:
Type of profile 1N/2N/3N/2NVS/Light/<4N>:
Please wait....
Basepoint of insertion :
Rotation angle<0>:



5.11 PROTECTIVE BARRRIER

MGE Draws protective barrier segments

This macro draws protective barrier segments with the currently selected values of the individual components. The components are:

- Protective frames
- System supports
- Suspension brackets

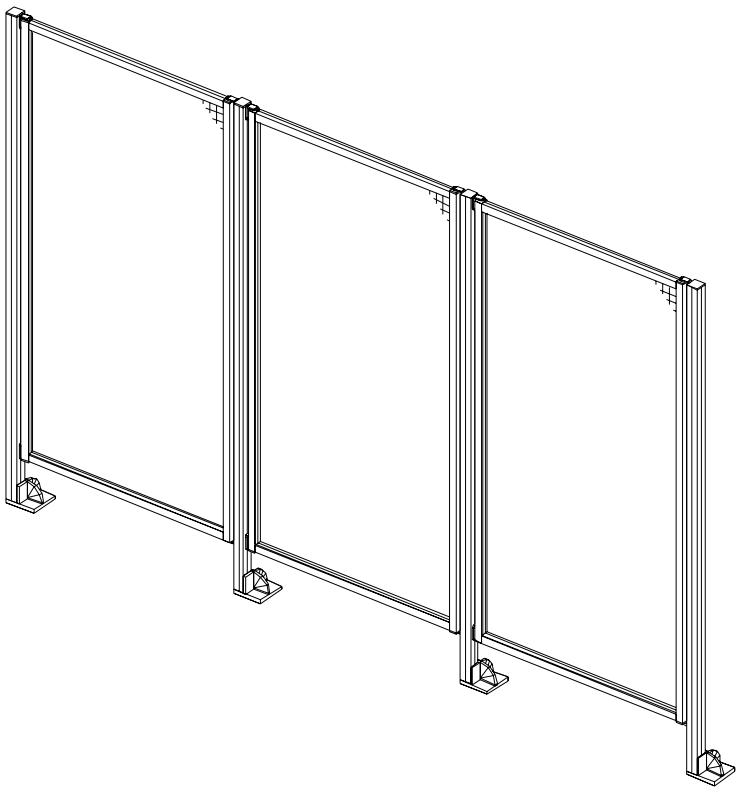
The macro needs the left system support as a start module, the angle of insertion and the quantity of the segments to be drawn. If you intend to change the length and height of the segments, call the Choose program of the individual components first and select the desired dimensions there.

From the menu *Module* Other... select Protective barrier.

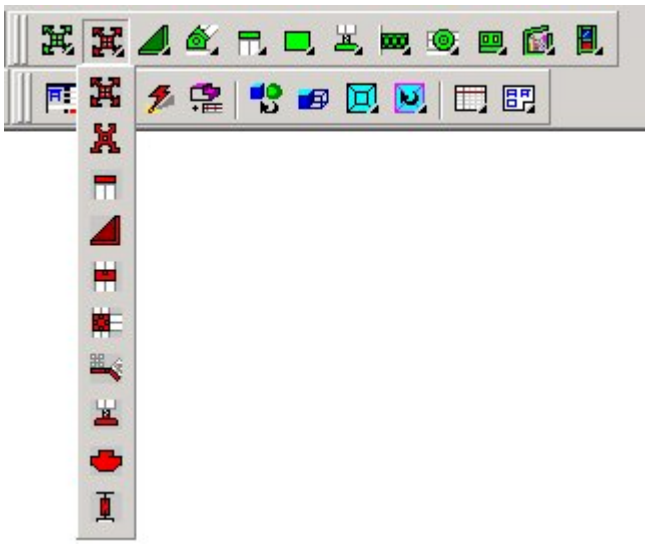
At the Command prompt enter **Pbarrier**.

Format:

Pbarrier
Choose left system support:
Quantity of segments <1>:
Angle to system support <0.0>:



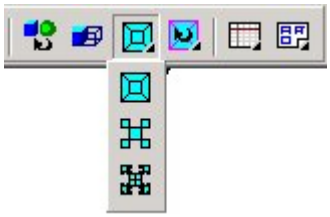
5.12 New Technology starts with 40/50 Profiles



With FMSsoft 7.0 you have the possibility to insert objects as 3D-solids. You can export the solid models to other 3D-CAD systems. This is not possible with the 3D wire frame models which stem from previous FMSsoft versions. As long as not all MGE parts have been implemented in the new you will have two different pulldown menus. To find the new parts easier we changed the icon color for the new objects into red.

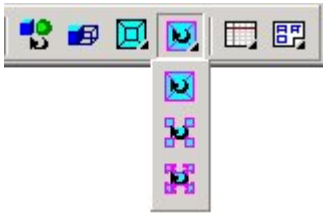
Another improvement of FMSsoft 7.0 is the possebility to change the level of detail. We implemented in MGE three different levels of detail. There are two different menus for this purpose:

Change the level of detail for the drawing.:

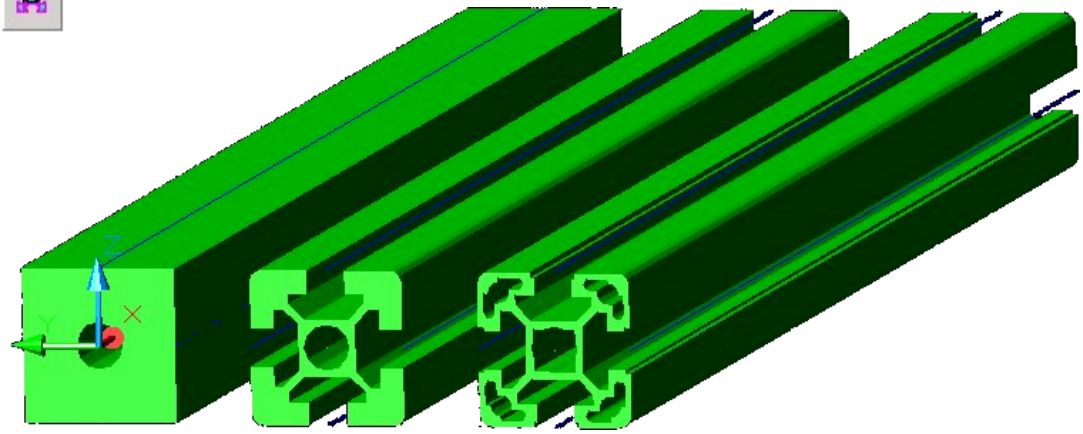


After you change the level of detail for the drawing, all following objects will be drawn in the new level. Parts you included before will not change their level of detail.

Change the level of detail for parts:



You want to change one or more parts after you inserted them? After you chose the command **‚fms_ChangeDetailLevel‘** you can select several parts. Afterwards the system will change the level of detail for those.



5.13 Technical Support

Wie give you technical support if you have problems with the installation via our FMSsoft Hotline.

Tel. +49 (0)711 / 811 - 32990
Fax: +49 (0)711 / 811 - 7793
E-Mail: FMSsoft@de.bosch.com
Internet: www.boschrexroth.com/FMSsoft