

MÜPRO

profile calculation program

user manual

Table of Contents

Table of Contents	2
1. Preface.....	5
1.1. Motivation	5
1.2. Verification.....	5
1.3. Coordinates system	5
1.4. Used terms, abbreviations and definitions.....	6
2. User Interface.....	8
2.1. Systems / register / Tabs	8
2.2. Image rail profile	9
2.3. Rail graphic.....	9
2.4. (Result) Memo	10
3. Dialogues	11
3.1. Projekt information	11
3.2. Set type of profile	12
3.3. Support conditions	13
3.4. Node loads	15
3.5. Pipe selection	17
3.6. Distributed loads	18
3.7. Calculation of distributed loads	19
3.7.1. Component	19
3.7.2. Air duct.....	20
3.8. Selection of static systems.....	21
3.9. Options	22
3.9.1. Units.....	22
3.9.2. Various.....	22
3.9.3. Font sizes.....	22
3.9.4. Pre-settings.....	22
3.9.5. Language	23
3.10. Forces and Moments.....	23
3.11. Stress.....	24
3.12. Info.....	25
3.13. Print	25
3.14. Printer setup.....	26
3.15. Export to Microsoft Word	26
4. Menu bar	28
4.1. File.....	28
4.1.1. New.....	28

4.1.2.	Open	28
4.1.3.	Save.....	28
4.1.4.	Save as	28
4.1.5.	Export to.....	28
4.1.6.	Print	28
4.1.7.	Printing range.....	29
4.1.8.	Printer Setup	29
4.1.9.	Exit.....	29
4.1.10.	filelist	29
4.2.	Project	29
4.3.	View.....	29
4.3.1.	Symbols / Toolbar	29
4.3.2.	Show values in graphics.....	29
4.3.3.	Overlay loads	29
4.3.4.	History / Undo / Redo	29
4.4.	System	30
4.4.1.	Selection of static systems	30
4.4.2.	Add Tab	30
4.4.3.	System new	30
4.4.4.	Copy system	30
4.4.5.	Insert system.....	30
4.4.6.	Delete tab.....	30
4.4.7.	Delete all Tabs	30
4.4.8.	Systems	30
4.4.9.	Project information	30
4.4.10.	Selection of profile type.....	30
4.4.11.	Support conditions.....	30
4.4.12.	Node load	30
4.4.13.	distributed load	30
4.5.	Options	31
4.5.1.	Change calculation level	31
4.5.2.	Automatic FEM-Calculation	31
4.5.3.	Automatic admissible load calculation	31
4.5.4.	Automatic suitable profile choice	31
4.5.5.	Considering dead load	31
4.5.6.	Settings.....	31
4.6.	Calculation.....	31
4.6.1.	FEM-Solver	31
4.6.2.	Forces, Moments	31

4.6.3.	Stresses	31
4.6.4.	Project / profile type / support conditions / a.s.o.	31
4.7.	Help	31
4.7.1.	use Help	31
4.7.2.	support conditions	32
4.7.3.	Fast help on / off	32
4.7.4.	Info	32
4.7.5.	Homepage	32
4.8.	Language	32
5.	Toolbar	33
5.1.	Open file	33
5.2.	Save file	33
5.3.	New file	33
5.4.	Add system	33
5.5.	Export to Microsoft Word	33
5.6.	Print	33
5.7.	Printer Setup	33
5.8.	Setup	33
5.9.	Data from project manager	33
5.10.	Change of calculation level	33
5.11.	Selection of static systems (+ selection list)	33
5.12.	profile type (+ selection list)	33
5.13.	Dead load	33
5.14.	Support conditions	34
5.15.	Node loads	34
5.16.	Distributed loads (+ selection list)	34
5.17.	FEM-calculation (+ selection list)	34
5.18.	Element forces	34
5.19.	Tensions	34

1. Preface

The MÜPRO-profile calculation program is used as dimensioning aid for choosing MÜPRO-support channels.

1.1. Motivation

It was committed to make also unexercised users able to create in short time completely described calculation models. The possibility to choose from standard systems, support channels - in the following also referred to as rails - or loads from defaults, enables you to get results pretty fast. These can be adjusted easily to your needs. Additionally there are options for a suggested profile or automatic calculation of maximum possible loads.

Object of this manual is only handling of the program, not evaluation of input or results.

1.2. Verification

Already while entering values, you can determine resulting tensions, deflection, reaction forces and anchor extraction forces. These are calculated from entered profile type, supports, node loads and distributed loads by FEM. For verification of load bearing capacity stress is proofed - direct stress, shear stress, bending stress and combination of direct and bending stress. Checks of admissible deflections are made in selected positions. Loads are set each in relation to resistance of the system, so that a fast evaluation is possible. For complex analysis, like flexural buckling, local buckling or dynamic loads please use other programs.

Although of easy handling it is indispensable, to have knowledge about the meaning of input and output values. The idealisation of the real existing construction into a system for calculation, its loads and other boundary conditions can not be made by the program and is duty of the user.

1.3. Coordinates system

Input was reduced to a one way, horizontal system.

The x-axis runs here horizontally in direction of rails length, z-axis runs downwards and y-axis runs horizontally at right angle to the rail. Because of this x-z-coordinate system is used with dead loads as default, and x-y-coordinate system not.

If you change calculation layer from x-z-coordinate system to x-y-coordinate system then the rail is strained in the other orthogonal axis. Namely Iz, Wz, a.s.o. are used now instead of Iy, Wy, a.s.o. resp. the other way round.

Remark: If a rail is used rotated by 90°, you have to adapt dead load accordingly.

Remark: If a rail is used without regard of dead load, it is drawn unfilled.

1.4. Used terms, abbreviations and definitions

Following mostly established terms and abbreviations are used, because long explanations every time, what has to be used how and when or how it is meant, will deflect attention more as it would be helpful - not to mention overlong, nested sentences.

If these are not familiar to you, you can look them up here. Experienced (Windows-) users will probably know a lot of things mentioned in this section.

General:

Action	With this is meant the execution of an activated program cycle.
ToolTips	Small yellow text hints, which are shown if the mouse stays longer time over a certain element. Remark: In the rail calculation program it is named ‚fast help‘.
Button	This term is from english ‚Button‘. It means a switch, which is often bordered by a rectangle. If you click it, a predefined action is activated.
[X]	Square brackets surround often units of values or keys or key combinations to be pressed.

Mouse actions + buttons:

Click	Means pressing the mouse button. If nothing additionally is mentioned, please use the left mouse button.
Draw	While moving the mouse the mentioned keys stay pressed from start point until end point! If no key is mentioned additionally, then (again) use the left mouse button.
Left Click	Means pressing the left mouse button. (LMT)
Right Click	Means pressing the right mouse button. (RMT)
Doubled Click	Means two fast after another executed clicks. If nothing additionally is mentioned, use left mouse button twice.
LMT	see Left Click
RMT	see Right Click
DLMT	Doubled Click with left mouse button
MR	Scrolling with mouse wheel Remark: not to be confused with pressing mouse wheel (= Middle mouse button)

keyboard:

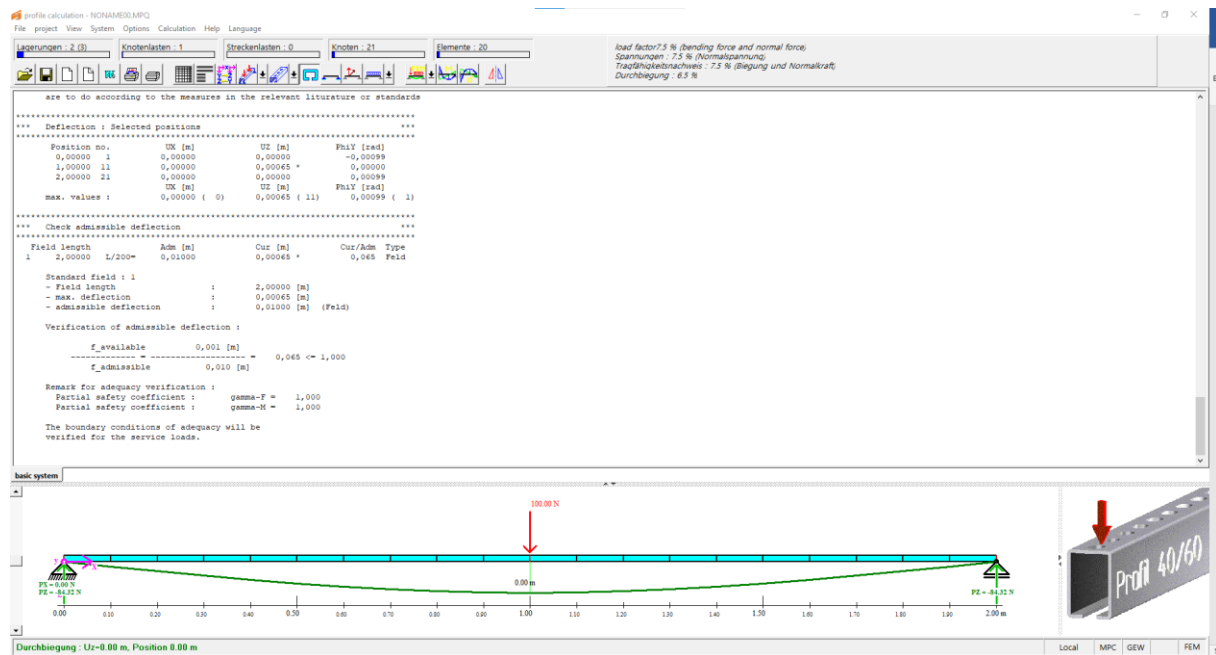
arrow	see arrow key
arrow key	Most times is meant a cursor key. These are located between the normal and the number pad. There are for different directions: <ul style="list-style-type: none"> • up(wards) • down(wards) • (to the) left • (to the) right
ArrowUp	arrow key up
ArrowDown	arrow key down
Strg	Press Control key; on keyboards often shortened to Ctrl
Alt	Press Alternate-key; on keyboards often shortened to Alt Not to be confused with AltGr (Alternate Graphic; alternate font type) !
Shift	Press (case) shift key; on keyboards often labelled with an unfilled arrow key; not to be confused with caps lock key (labelled with A and arrow) Remark: There is one shift key in the left and one in the right, but because of hand size, normally the left one is used.

- Return** Press return key; on keyboards often labelled with a special arrow (arrow to the left, which runs upwards at the end; symbolizes a new line with a jump to the beginning, because the return key was called carriage return in the past); this key is most (but not every) time equivalent to the key 'Enter' on the number pad.
- Remark:** If text says 'With key X', it is meant pressing this key or keep it pressed while doing another action. Sometimes it is shortened to 'With X', 'With [X]' or simply [X].
- Remark:** Strg, Alt and/or Shift are often used only in combinations with other keys and / or mouse actions.
- combinations:**
XXX + YYY This means a simultaneously use of several keys and / or mouse actions. These can be more than two and have to stay pressed during the whole period.
 Is no period mentioned, then it depends on the action if it is a short tip or a begin to end action (e.g. drawing).
- Example:** Strg + Alt + LMT means pressing simultaneously control key, Alternate-key and left mouse button.

2. User Interface

The programs main window is organised in several areas. These contain the common elements like menubar, toolbar and status bar. Additionally an image of the used profile incl. load direction, a rail graphic of the used system and the memo with results are shown by default. You can switch through the entered rail systems by a register bar at the bottom of the memo.

Input can be done by opened dialogues, toolbar and partly by the rail graphic. The dialogues can be opened by menubar, toolbar, hot keys and doubled click on some elements.



2.1. Systems / register / Tabs

The entered systems are organised in register - also called tabs. For normal speech the register and its content are often equivalent. But to execute an action it is important, if you delete a register or its content. Because of this it was tried to distinguish register and their content conceptual. So tab will name a register and system will name the content of a register. Systems in different tabs don't influence another in calculation and could also be placed in separate files.

Remark: With a doubled click on a tab you open the dialogue 'projekt information'.

2.2. Image rail profile

In this image the chosen profile type is shown schematic with load direction.

Remark: If you click on the image, the dialogue ,Info' is opened.

2.3. Rail graphic

Main task of the rail graphic is to show input to the system visually.

This allows fast feedback about the significant inputs of the system - equal if supports, loads or deflection. By clicking left (LMT) the values (load, suspension) are shown resp. hidden.

Additionally the symbol of the mouse cursor changes depending on position over the graphic. With a doubled click (DLMT) you can open the corresponding dialogue. With the control key (Strg) respectively control key + shift key (Strg+Shift) you can move supports, single loads, distributed loads or length of the rail depending on the shown symbol. At several combinations you can optional change the position with the mouse wheel (MR) in 0,5 m steps. Last one is especially helpful, if the new position lays outside the rail graphic.

The combinations result in actions as follows:

support symbol:

DLMT: Opens dialogue ,Support conditions'

Strg + LMT (+ MR): Moves support to position
Moving beyond rail length deletes the support

Strg + Shift + LMT (+ MR): Copies support to position
Copying beyond rail length elongates the rail

Remark: DLMT + Strg is not intended. Possibly the support will be moved additionally!

single load symbol:

DLMT: Opens dialogue ,Node loads'

Strg + LMT (+ MR): Moves single load to position
Moving beyond rail length deletes the single load

Strg + Shift + LMT (+ MR): Copies the single load to position
Copying beyond rail length elongates the rail

Remark: DLMT + Strg is not intended. Possibly the Node load will be moved additionally!

distributed loads symbol:

DLMT: Opens dialogue ,Distributed loads'

Remark: Distributed loads symbol is only shown, until LMT is pressed.
DLMT + Strg is not intended; Because of this the symbols distributed load left, middle and right will move the distributed load instead; border symbols will possibly change length of the distributed load!

distributed loads symbols left border + right border:

Strg + LMT (+ MR): Edits the position of the chosen border of the distributed load to the new position
Editing beyond rail length deletes the distributed load

Strg + Shift + LMT (+ MR): Copies the distributed load and edits the position of the chosen border of the distributed load to the new position
Editing beyond rail length elongates the rail

Remark: If the left border is moved farther right then the right border, then the right border becomes the left one and the new position the right border. At the right border it will be changed accordingly the other way round.

distributed loads symbols left + middle + right:

Strg + LMT (+ MR): Moves the distributed load to position
Moving beyond rail length deletes distributed load

Strg + Shift + LMT (+ MR): Copies distributed load to position
Copying beyond rail length elongates the rail

Remark: The symbol distributed load left appears, if the mouse is moved over the left third of a distributed load. (Exception: left border)
Accordingly symbol distributed load middle is used in the middle third.
The symbol distributed load right is used in the right third. (Exception: right border)

rail profile symbol:

DLMT: Opens the dialogue 'Set type of profile'

Strg + LMT (+ MR): Edits rail length to the position
supports block the shortening; if a bearing rested on the prior rail end, it isn't moved

Strg + Shift + LMT (+ MR): Edits rail length to the position
At shortening the bearings will be set to the end resp. surplus the additional ones deleted; if a bearing rested on the prior rail end, it is moved to the new rail end

Remark: Loads block shortening of the rail. Namely you can't edit a rail shorter than the farthest right existing load (resp. the farthest left existing load at shortening on the left). This has to be deleted / moved first.

Remark: DLMT + Strg is not intended. Possibly a bearing is moved additionally!

Remark: If the start position lies on the left half of the rail, it will be shortened left, analogue at a start position on the right half of the rail it will be shortened right. For elongations this distinction would be unnecessary, but these underlie the same rule. At elongations on the not expected side, the process cancels.

2.4. (Result) Memo

The biggest element is by default the memo with results. In this are shown the input data and results belonging to the current system - that means the system in the currently chosen tab/register - as text.

Remark: The result memo with results is a text output, from which can be copied - but not written to. Because of this graphics are never shown there. They can be shown in separate elements.

Remark: The content of the result memo complies with the selections in the dialogue 'Print'.

3. Dialogues

3.1. Projekt information

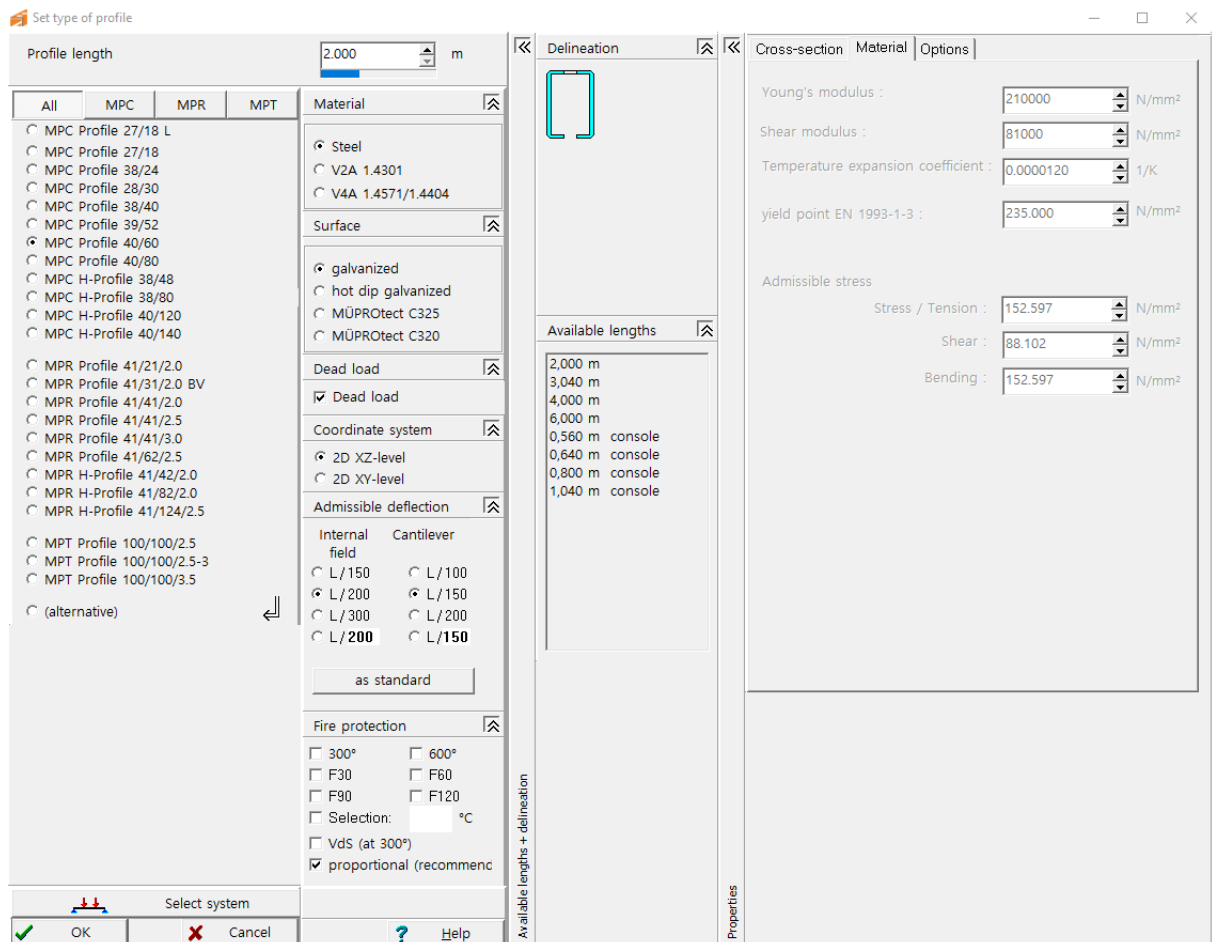
In this dialogue you can insert data for the project to which the system belongs.


The button ,Project' opens a subdialogue with a list of projects. These are from a mirror of SAP. By choosing a project some fields are pre-populated with there available data. With the triplearrow you can show or skip additional input fields. With the button ,remarks' you can choose default texts, which shall be notified.

With the button ,Copy from' you can copy data from the chosen tab into the current tab.

3.2. Set type of profile

In this dialogue you can choose the rail profile incl. material, surface and length. Here you can also adjust dead load, load layer, admissible deflection and fire protection.



With the Buttons with to arrows , you can enlarge or reduce the part of the dialogue in the shown direction. The triggered arrows stand for shown, and not activated for a hidden section. With the windows symbol ,minimize' the dialogue is shrunk to the smallest size. This was needed, so in some monitor resolutions – because of the multiplicity of choosable profiles – the buttons on the bottom couldn't be seen. With the windows symbol ,maximize' it are shown all sections, but the dialogue will be very big.

In the section ,Available lengths an delineation' you see the shape of the profile section and the standard lengths according to the catalogue (state 04/2022).

In the section properties are the register cross-section, material and options.

At choosing a rail profile the according values change and are shown on the right part under the register cross-section. Not every rail profile is selectable in every different material and surface. So for example MPT are found mainly only under hot dip galvanized.

In the filter line you can limit the view to profiles MPC, MPR or MPT. This has no influence on the automatic search of profiles.

By choosing a material, a surface or fire protection (temperatures resp. calculation type) values of material are changed according to the chosen rail and are shown under the register material. The temperatures for F30 (842°), F60 (945°), F90 (1006°) and F120 (1049°) comply with EC3, section fire (DIN EN 1993-1-2:2005). For fumes and sprinkler systems are 300°

and 600° typical. If you need an other temperature, you can enter this under ‚selection‘. For reliability the result is double-checked in cold conditions (normal temperature 20°), because it shouldn't collapse there either. A temperature check under 20°C is not possible.

‚VdS (at 300°)‘ uses - if chosen 300° - a factor on loads of 1,4 instead 1,0. (Remark: To reduce failures it is now coupled to with 300°C, and can no longer be chosen at other temperatures – especially cold case.)

‚proportional‘ controls the used reduction ratio for yield point in case of fire. In a meeting of the Anwendungstechnik they agreed to use by default the smaller value (conservative results).

If you want to calculate a profile, which isn't common available, you can select ‚(alternative)‘ to use a customised profile. The values can be entered in register cross-section and material. Simply choosing material and surface is then (at the moment) no longer possible, because the according sort and special confirmations of quality depend on the used rail. At custom profiles classifications had to be guessed, because there is for example no possibility to enter a specific steel type.

Remark: With the Button on right bottom (angled double arrow) you can copy the values of the prior chosen rail into the alternative rail.

The check mark dead load switches, if dead load is considered in the system or not.

By choosing a coordinate system the section values change accordingly. The direction is illustrated red in the coordinate plane in the register options resp. by an arrow in the image rail profile of the main window.

The admissible deflection can be changed by selection resp. entering a value under the fraction line in the last selectable point. You can overwrite the by default used value with the current selection by clicking ‚as standard‘.

In options you can change the amount of the node points used in the calculation. It is recommended to keep the automatic calculation of node points.

With ‚automatic suitable profile choice‘ profiles are used automatically depending on the entered load.

With the button ‚Select system‘ you open the dialogue ‚Selection of static systems‘.

Remark: With [Return] you set focus on OK-Button. So you can accept and close the the dialogue by doubled [Return]. [Alt+F4] closes the dialogue (without prior focus on close-button).

3.3. Support conditions

In this dialogue you can adjust all supports of the system.

While in the rail graphic of the main window only existing bearings can be moved or copied, you can edit in this dialogue all supports. These contain - along several choosable fixed / loose combinations - also ‚special points‘ and ‚joints‘ in the rail.

With ‚support No‘ you choose the support number, which you want to edit. In the rail graphic of the main window the chosen support is now marked with the number. If the support already exists, the corresponding position will be shown in the edit field ‚Position‘. New supports start with value 0. You can overwrite here a position directly with a new one, to

move the support. For a new support you edit the support number to a not already used value. (In this case, all values are shown with 0.)

As already mentioned there is a support called 'special points'. With this you can evaluate a specific position- additionally to the automatic determined positions. These are shown then in the result memo.

If you choose a not already used position - and support number - you can mark this with the first button from top ('special points marking'). Thereby changes the description of the button to 'Delete special points', with which you can countermand such additional marks again.

Remark: This special point has no impact on the system, but only on the amount of evaluation points .

Remark: You can mark already existing points, but these will be evaluated already anyway.

Support conditions

Nodes suspension

Lagerung Nr. : 1

Position : 0,000 m

☒ : PX ☒ : MX

☒ : PY ☒ : MY

☒ : PZ ☒ : MZ

☒ : Pxyz

e1y (Console)

e1z (Console)

☒ : Restraint is a MÜPRO wall hanger bracket

☒ Müpro-Konsole

☐ : Hinge joint

Regenerate graphic

OK Cancel Help

The last button at the bottom ('insert joint' resp. 'delete joint') creates a joint in the rail resp. deletes it again. It breaks the rail and can transfer no moments in this point.

Remark: The checkbox at 'hinge joint' is coupled with that button.

Remark: Because the joint is in the rail - and not at the rail - you cannot convert a clamped support into a fixed support or the other way round.

The remaining supports are bearings. That means there are transferred loads from the rail into the bearing (connections, hangings, fastening, subsoil). Depending on the chosen button the transferrable forces (PX, PY, PZ) and moments (MX, MY, MZ) are checked in the selection on the left. Reciprocal checking or unchecking switches between the different symbols.

The free support is a special support, which can't transfer forces or moments in any direction. It marks for example the right end of a cantilever, or removes existing supports, without deleting joints or markings.

If the bearing can transfer moments about the y-axis and forces in x-direction, then will be prompted, if the element is a console and if yes, if it is a MÜPRO console. If you confirm the second one, then the values for eccentricity of the wall plugs in e1z will be predefined with values of the console suitable to the rail profile. If only console is selected, they can be chosen freely (Remark: This means a special construction usually implemented on site). If there are values for eccentricity of the wall plugs given, then additionally to the existing reaction forces the loads for each (pair of) plug is printed. (Remark: At standard value 0 a hint is shown, that it isn't eccentric.)

The supports which are printed as triangle are in each case pin-jointed, and can't transfer a moment because of this. If the support can transfer moments, then a connection on a orthogonal tile is used in the symbol. The hatching in the symbols signs a fixed connection for forces in both directions of the area, besides they are separated by an additional line. If the support is marked with an additional line in the symbol (see movable supports), then the support can transfer forces at right angles to the line, but not parallel to it (independent, if it contains a hatch or not).

Remark: The symbol for the fixed support is the well-established use. In few books however it is used as clamped support instead.

The fixed support cannot transfer moments!

Remark: With [Return] the focus is set on the OK-button. So with a second [Return] you close the dialogue. [Alt+F4] cancels the dialogue (without prior focus on the cancel-button).

Remark: If a edit field is selected, then you can switch between the support numbers with [Shift+ArrowUp] resp. [Shift+ArrowDown].

3.4. Node loads

In this dialogue you can enter the single loads of the system.

As already known from the dialogue 'support conditions', you can choose and edit by node number the corresponding load, and with 'position' the corresponding position.

The stated forces and moments are loads, which are applicated in this position on the system. The disabled fields depend on the chosen coordinate system. Instead of entering a value in an edit field for forces, you can select this and then click the now enabled button 'select'. By this opens the dialogue 'pipe selection'. After closing this the determined value from there is entered into the currently chosen field automatically.

If there are used different support distances in pipe selection for multiple single loads, then an 'i' is shown near the selection. Hereby support distances can be adjusted.

Remark: If data are not taken from the data base resp. pipe selection, they possibly cannot be adjusted by pressing the button.

„Lower point of load application“ refers to the forces, too. With this they can be drawn above or under the rail in the rail graphic of the main window.

Remark: If „Overlay loads“ is selected, then forces are always drawn above the rail, and never under it.

„Delete values“ deletes the values of the current chosen single load.

„Delete all point loads“ deletes all existing node loads in the system.

„Regenerate graphic“ redraws the rail graphic of the main window.

Remark: The designation noad loads is misleading. Though every single load has a point for calculation (in FEM named node), however they have nothing to do with connections between real existing elements (for example frameworks).

But because this program performs only single-axis calculations, there is no danger to confuse them.

Remark: With [Return] the focus is set on the OK-button. So with a second [Return] you close the dialogue. [Alt+F4] cancels the dialogue (without prior focus on the cancel-button).

Remark: If a edit field is selected, then you can switch between the node loads numbers with [Shift+ArrowUp] resp. [Shift+ArrowDown].

3.5. Pipe selection

With this dialogue you can choose preset pipes from several DINs and manufacturers.

Hereby the recorded values for this pipe are entered.

If a value is changed, then automatically the pipe selection is nullified and the material is set to ,custom'.

You can define here pipes like wished, equal if diameter, wall thickness, material (density), pipe weight, isolation, isolation thickness or filling. With a click on wall thickness it changes to the value for inner diameter, and the other way round.

The pipe weight is calculated from the input and multiplied with the entered support distance to the clip load. After closing this will be written in the chosen field in ,node loads'.

You can also change directly pipe weight, support distance and clip load.

Pipe selection

Auswahl Rohrleitungen

Norm/manufacturere : (ALL) (all)

Pipe selection :

Outer diameter : 0,000 mm

Wall thickness : 0,000 mm

benutzerdefiniert ρ 0,000 kg/dm3

Pipe weight (empty) : 0,000 kg/m

☒ Isolation : 0,000 kg/m

Isolation thickness : 0,000 mm

☒ Pipe Filling : ρ 1,000 kg/dm3

Piping weight : 0,000 kg/m

Support distance : 0,000 m

Clip load : 0,000 N

Apply Cancel

3.6. Distributed loads

In this dialogue you can enter distributed loads of the system.

As already known from dialogue 'support conditions', you can choose and change by distributed load number the corresponding load and with 'Position' the corresponding position. The only difference is, that the load is defined by two points this time; one start point left (from) and an end point right (to).

The value of the load in these points, can be entered in y-direction (Qyl and Qyr) or z-direction (QZl and QZr), depending on how the coordinate plane is rotated. The section between is linear, and interpolated with left and right value. Triangled loads are a special case, which has value 0 at one side. Other cases of loads have to be assembled or approximated with multiple distributed loads.

Near the input field for loads of QZl is a button, with which you can enter loads in z-direction by an additional dialogue 'Calculation of distributed loads'.

Remark: In y-direction (in a right angle to the rail) this possibility doesn't exist.

With 'Lower point of load application' you can change the distributed load similar to the single loads visual under the rail instead setting it above.

Remark: If 'overlay loads' is selected, then loads are always drawn above the rail, and never under it.

'Delete values' deletes values of the current chosen distributed load.

'Delete all distributed loads' deletes all existing distributed loads in the system.

'Regenerate graphic' redraws the rail graphic of the main window.

Remark: With [Return] the focus is set on the OK-button. So with a second [Return] you close the dialogue. [Alt+F4] cancels the dialogue (without prior focus on the cancel-button).

Remark: If a edit field is selected, then you can switch between the distributed loads numbers with [Shift+ArrowUp] resp. [Shift+ArrowDown].

3.7. Calculation of distributed loads

Unfortunately for labelling the registers there have been chosen 'Air duct' and 'Component'. I say unfortunately, because this is based on the assumption, that an air duct is always extended over many supports (endless / multi span beam), and a component always only over few supports (single span, two span and triple span beam). But the used element is not mandatory an air duct - although it is extended over many supports - and the other way round a short air duct can be supported only by few girders. On the other side these terms are short and pithy; and they distinct from terms of the static system - which means the rail or console (support girder) to be calculated and running orthogonal to it.

3.7.1.Component

Knowing this we have a look on the single span, two-span and triple span beams (component).

Calculation of distributed loads

Air duct

Component

Distributed over n support channels with equal spacing:
n
4

Width of the base on the support channel:
b
0,3
m

Weight of component:
F
24
kg

Weight referring to one support channel in internal Load:
G
8,800
kg

p
86,299
N

Distributed load: QZl = QZr =
q
0,288
kN/m

Distribution of the load on the support channels in relation to 1 / 2.75 / 2.75 / 1

Apply

Close

The loads are 'Distributed over n support channels with equal spacing'.

At single span beams ($n = 2$, because $n - 1 = 2 - 1 = 1$) the 'weight of component' is simply divided in halves for 'weight referring to one support channel'.

The 'load' [unit N] results from the product of 'weight referring to one support channel (in the internal field)' [unit kg] and gravitation acceleration.

The 'distributed load' results from 'load' divided by the 'Width of the base on the support channel'. So it is assumed, that the load is equally distributed on the rail in this section.

Remark: At single span beams there is no internal rail or field.

Remark: The gravitation acceleration is used rounded to some digits. This is about 2 percent lower than the factor 10 familiar from calculation by hand.

At two-span ($n = 3$) and triple span beams ($n = 4$) the load is - because of the moments at the inner supports - unequally divided on the support girders. This apportionment is shown on the right bottom. The evaluation is carried out for the most stressed inner girder.

The load factor of the rail load results directly from the ratio (apportionment of this rail / sum of all apportionments).

Example: At two-span beams its because of ratio $1 / 3.33 / 1$ for the inner girder $3.33 / 5.33 = 0,625$. In comparison to the perhaps expected third of the total load it is nearly doubled.

Remark: So much higher the amount of spans - and therefore girders with moment at supports - so much more drops the factor for this unequal distribution.

3.7.2.Air duct

Calculation of distributed loads

Air duct

Component

Distance between fastenings: a

2

m

Width of air duct: b

0,3

m

Weight of air duct: p

4,4

kg/m

Weight referring to one support channel: G

8,800

kg

Load: p

86,299

N

Distributed load: QZl = QZr =

q

0,288

kN/m

View at the continuous beam as

☒ Internal field of an endless beam
 ☐ Edge field of an endless beam

Apply

Close

The 'distance between fastenings' multiplied with the 'weight of air duct' and a factor for the position on the endless beam results to 'weight referring to one support channel'.

In the outer field - see options of selection on the right - this is a little bit higher.

The load [unit N] results from product of 'weight referring to one support channel' [unit kg] and gravitation acceleration.

The 'distributed load' results from 'load' divided by 'width of air duct'. So it is assumed, that the load is equally distributed in this section on the rail, although the air duct is hollow.

Remark: The gravitation acceleration is used rounded to some digits. This is about 2 percent lower than the factor 10 familiar from calculation by hand.

Remark: An edge field of an endless beam – supported in Point B – has an enhancement factor of 1.133975

An internal field of an endless beam has an enhancement factor of 1.000

3.8. Selection of static systems

Selection of static systems

Profile length : m

MPC Profile 40/60

MPC Profile 40/60

Load P in N :

☐ Automatic admissible load calculation

☐ Automatic suitable profile choice

☒ Considering dead load

OK Cancel Help

In this dialogue you can choose preset static systems. If you choose a system, the two arrows appear at single span beams. With this you can increase the amount of spans up to 7 resp. Lower them down to 1 again. The total length will be divided on all fields.

Additionally there are possibilities to change the rail profile and its length. In the graphic beneath, it is shown what the current chosen system looks like. This later can be adopted - by other dialogues - as wished.

After opening the dialogue ,Given loads' is written beneath the graphic.
If you chose a system, the designation changes and you can enter the load into an edit field.
At distributed loads a button is visible for the dialogue ,Calculation of distributed loads'.

With ,Automatic admissible load calculation' the arranged loads are maximised. With the button on the left you can start a calculation.

The option ,Automatic suitable profile choice' suggests a rail profile and cannot be chosen simultaneously together with the automatic calculation of the admissible load.

,Considering dead load' switches this - for this coordinate plane - on or off.

Remark: With [Return] the focus is set on the OK-button. So with a second [Return] you close the dialogue. [Alt+F4] cancels the dialogue (without prior focus on the cancel-button).

3.9. Options

In this dialogue you can change the settings of the rail calculation program in several register.

Remark: With [Return] the focus is set on the OK-button. So with a second [Return] you close the dialogue. [Alt+F4] cancels the dialogue (without prior focus on the cancel-button).

3.9.1.Units

Here you can change the dimensions of the units and the amount of digits.

3.9.2.Various

These values control the view of the rail graphic on the screen and in print.
Resetting sets changed values back to standard ones.

Here you can change how history of the restore points is reset, the detailed view in open and save-dialogues, as well as the preset value for the view of project data.

Beneath there is the amount of filelist entries for the used files and the button for linking extensions of the rail calculation program with exactly this program (folder and name).

3.9.3.Font sizes

The output of text is structured in different sizes. Here you can enter, in which font size each of them are to be printed.

3.9.4.Pre-settings

Under this simple designation are hiding essential assumptions for the calculation. These contain the admissible deflections by default and the safety factors on loads. So these are to be changed with additional care and to be adjusted again for new calculations!

3.9.5. Language

Under language you can self-explanatory adjust the language used in the program.

But additionally you find here the settings for the used database. In fact not only how these are named (incl. extension for language), but also, if these are loaded from server, local or automatically chosen.

The screenshot shows the 'Options' dialog box with the 'Language' tab selected. The dialog has a title bar with a close button (X). Below the title bar are tabs: 'Units', 'Various', 'font sizes', 'Pre-settings', and 'Language'. The 'Language' tab contains the following settings:

	horizontal	vertical
Start :	0,0400	0,5000
Delta :	0,9000	0,5000
Scale :	1,0000	1,0000
Graphic :	275	
Deflection line :	10	50

Below the table is a button labeled 'Reset to screen standard-values'. Underneath are three checkboxes:

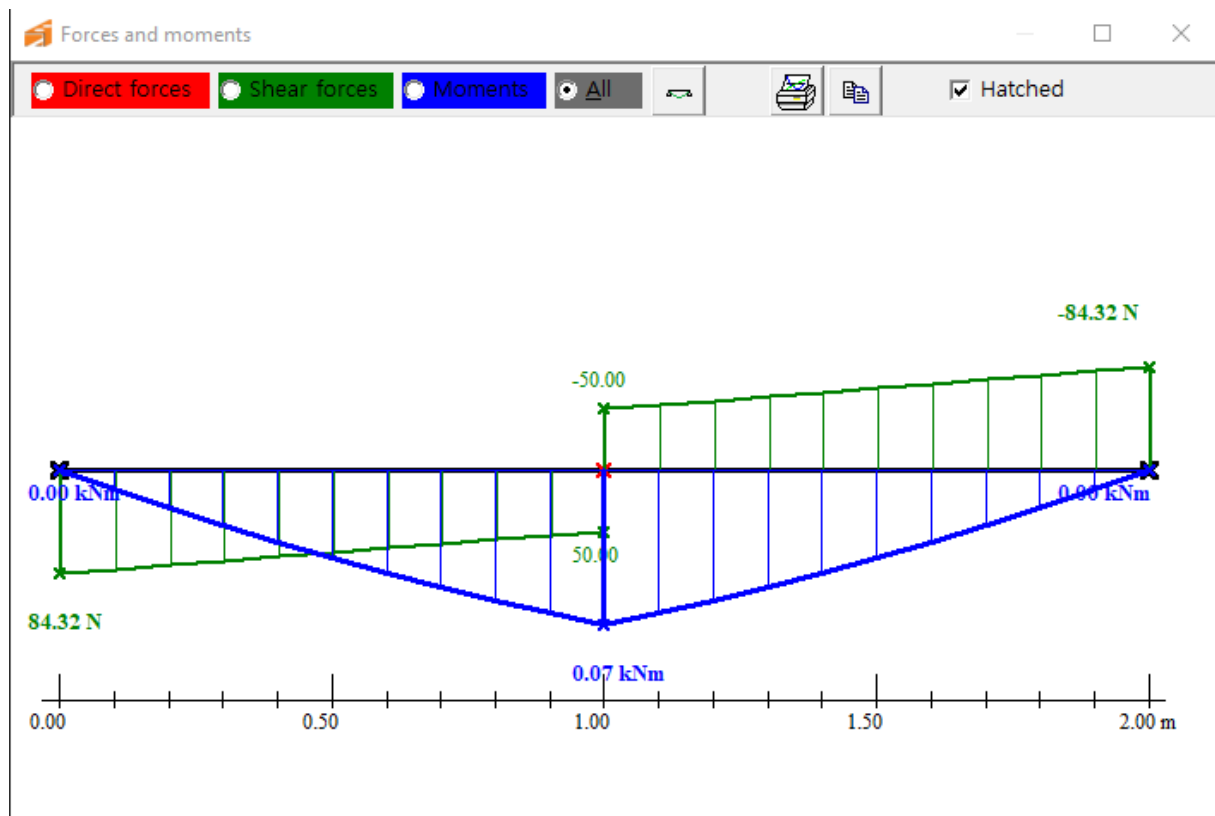
- ☐ Reset history when system is changed
- ☐ Activate more detailed view in dialogs
- ☒ Erweiterte Ansicht in Projektdialog aktivieren

Below the checkboxes is a label 'Amount of files in history list' and a spinner box set to '5'. At the bottom of the dialog is a button labeled 'Link file extensions *.mpo and *.mpq'. The bottom of the dialog has three buttons: 'OK' (with a green checkmark icon), 'Cancel' (with a red X icon), and 'Help' (with a blue question mark icon).

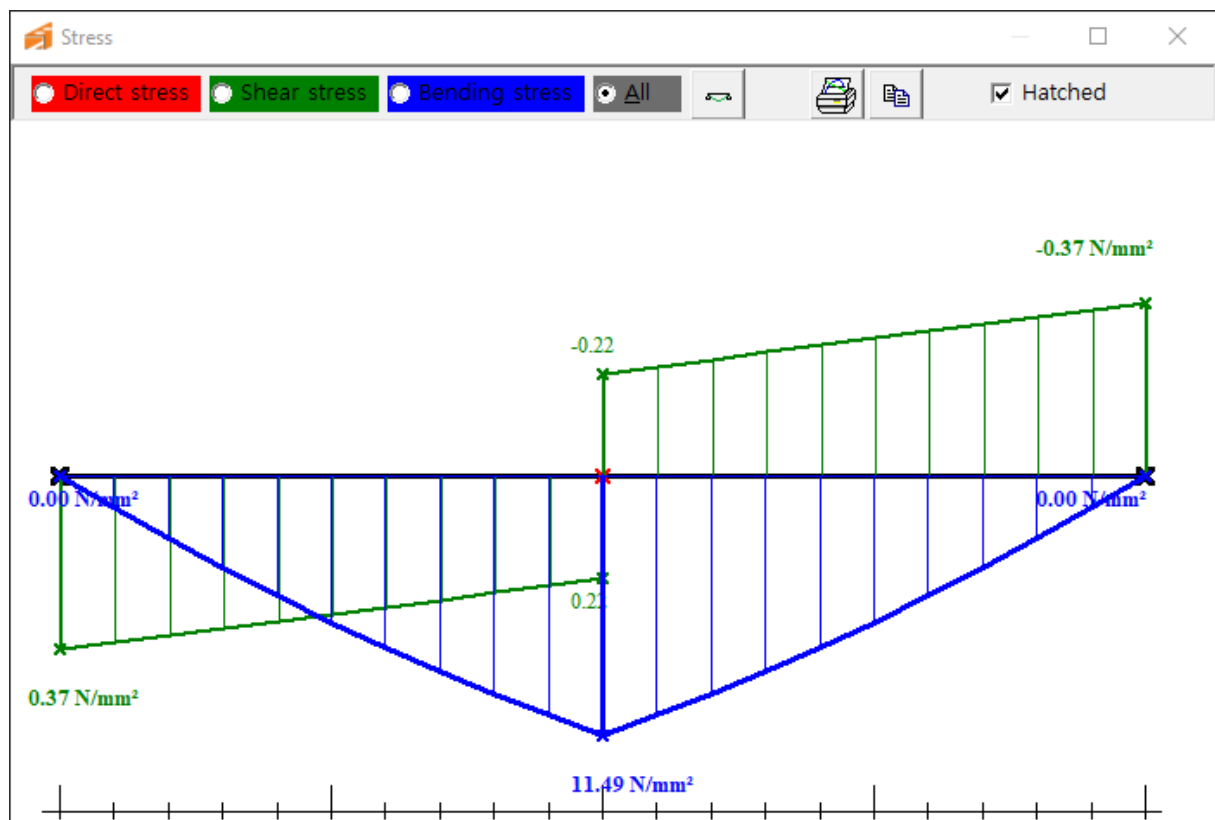
3.10. Forces and Moments

In the graphic of this dialogue are shown the forces and moments in the current system.

There is a possibility to show separately only direct forces, shear forces or moments, or all of them. Furthermore you can print the graphic, copy it, (not) hatch it, show/hide bending line or close the window. The values can be shown resp. hidden by a left click (LMT).



3.11. Stress



In the graphic of this dialogue are shown the stress in the current system.

There is a possibility to show separately only direct stress, shear stress or bending stress, or all of them. Furthermore you can print the graphic, copy it, (not) hatch it, show/hide bending line or close the window. The values can be shown resp. hidden by a left click (LMT).

3.12. Info

In this dialogue stand informations to the rail calculation program, like for example the version.

3.13. Print

By selection of themes in the dialogue you control the extent of the result to be printed.

On the top left you can set the printer to use and the option 'print as ASCII-text' selectable as needed and setup.

On the top right you can limit the range of print. This influences which themes are selectable beneath in output controls.

With reduce resp. enlarge you can control the view of section output controls in the window. At system you can choose, which tabs / systems shall be printed.

Output control divides themes in three blocks: system data, results and graphic. These you can switch on or off in blocks with the ,+‘ - and ,-‘-symbols near them. In block graphic there is the option to overlay loads or not. If loads are not overlayed the options for sums of node loads and distributed loads are disabled.

Remark: The settings for printing also partly influence the view of the result memos and the rail graphic.

3.14. Printer setup

Hereby you can change settings of the printer.

3.15. Export to Microsoft Word

Hereby you can create an output to Word.

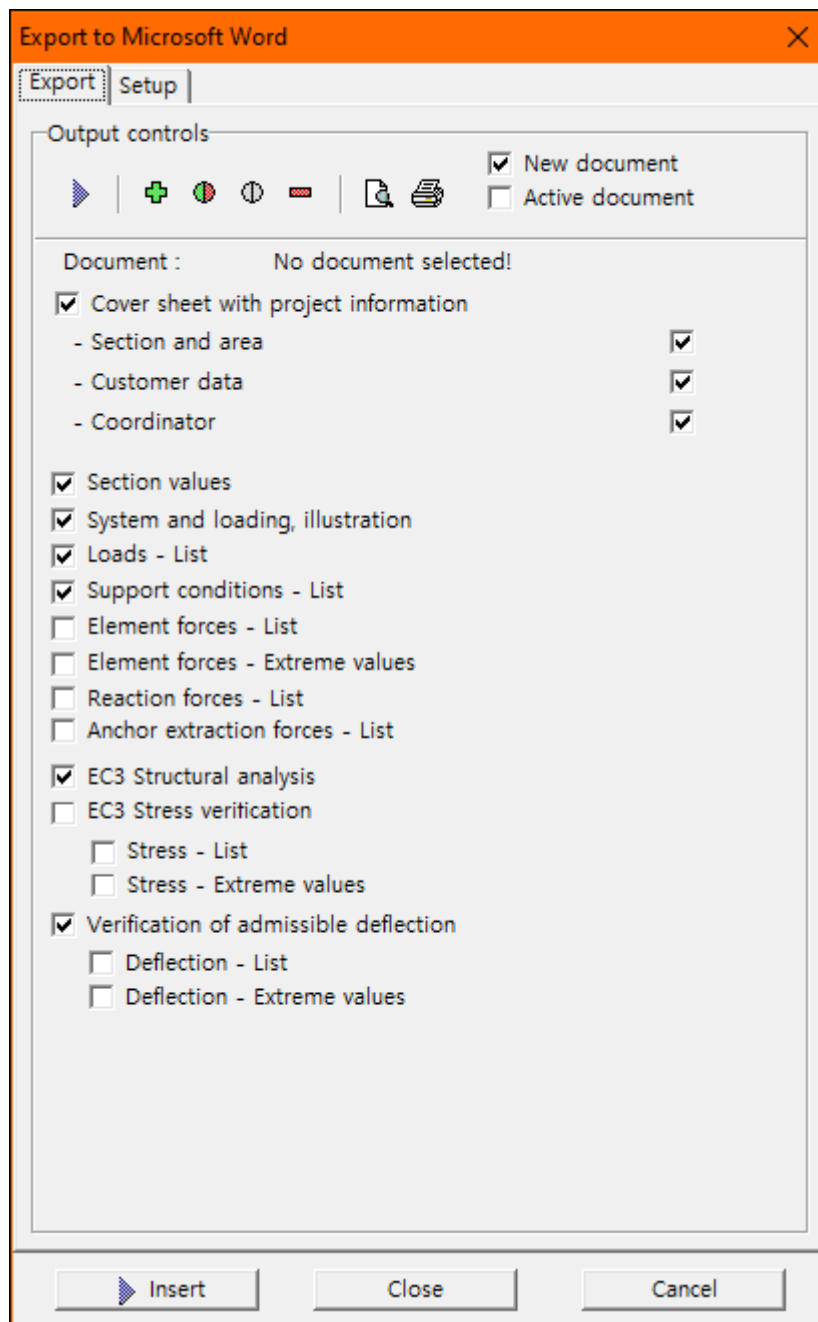
This happens by the button with the arrow or by button ,insert‘ at the bottom. Before that you can change the extent of the export like you wish with several hooklets. To make this easier there are the buttons ,activate all‘ (plus-symbol), ,standard‘ (coloured hemisphere), ,user‘ (uncoloured sphere) and deactivate all (minus-symbol). The last one only makes sense if you want to set new single hooklets afterwards, because otherwise maximum the cover sheet is printed.

The buttons preview and print refer to the already active word document.

The hooklet at new document and active document control, if the export shall be done into an existing or a new document.

Remark: At the moment there is an additional request before the export, so that this selection is without effect.

In register ,options‘ you can change the format of the document as needed.



4. Menu bar

The menu bar provides selectable actions sorted in following different sub menus:

4.1. File

Like familiar from Windows under the menu item file are listed the commands for creating a new file, open, save, save as, print and exit program. Furthermore you can export the result to word or into a text file, set up the printer or range of print (extent of print).

4.1.1.New

Closes the file and creates a new unsaved one.

4.1.2.Open

Opens a windows typical selection dialogue for opening a file.

4.1.3.Save

Saves the data into the corresponding file. If the file was never saved, instead of this ,Save as' is opened.

Remark: The program creates a temporary file (with the extension *.tmp) while working, thereby you don't need to answer a save request whenever you switch to another system. Additionally overwritten files are sometimes saved with the extension *.SAV.

At the moment it does these saves to often. Especially at not already once saved files, they afterwards often remain.

Caution: If you want delete temporary files, you should close the rail calculation program first, because you could possibly delete unsaved data!

4.1.4.Save as

Opens a windows typical selection dialogue for saving a file. Hereby you can save a not already saved file or save it under another name or version than the source file.

4.1.5.Export to

- Word: Opens the dialogue ,Export to Microsoft Word'.
- text file: Saves the result memo into a text file with the same filename, but file extension ,txt'.
- Copy text field to the clipboard:
The content of the result memo is copied into the clipboard.
- Copy graphic into clipboard:
The rail graphic is copied as image into the clipboard.

4.1.6.Print

Opens the dialogue ,Print'.

4.1.7. Printing range

- All
- limited (= selected themes)
- only input data

4.1.8. Printer Setup

Opens the dialogue ,Printer Setup'.

4.1.9. Exit

Closes and ends the program.

4.1.10. filelist

Attached to the end of the menu are the last used files.

4.2. Project

This element is no sub menu, but shows the dialogue ,project information'.

4.3. View

In this sub menu there are control over the toolbar, some view options and the commands for the restore points:

4.3.1. Symbols / Toolbar

This menu item rotates in sequence through

- | | |
|---------------------------|---|
| • Symbols / show toolbar: | Shows the toolbar |
| • Expand toolbar: | Shows additionally the amount of node loads, distributed loads, supports, nodes and elements of the system in the bar |
| • Hide toolbar: | Switches the toolbar off |

4.3.2. Show values in graphics

Here you can activate and deactivate showing values in the rail graphic.

4.3.3. Overlay loads

If switched off, loads can be drawn above and under the rail.

If switched on, these are always drawn summed up above the rail.

4.3.4. History / Undo / Redo

History opens the dialogue with the selection list of the restore points.

Undo switches back to the last step.

Redo reverses Undo again.

4.4. System

4.4.1. Selection of static systems

Opens the dialogue ‚Selection of static systems‘.

4.4.2. Add Tab

Here you find a sub menu, with which you can insert new tabs:

- New: Adds a tab with a default system at the end of all tabs
- Copy Adds a tab with a copy of the current tab at the end of all tabs
- insert New Adds a tab with a default system ahead of the current tab
- insert Copy Adds a tab with a copy of the current tab ahead of the current tab

4.4.3. System new

This command deletes the content of the current tab and inserts a default system instead.

4.4.4. Copy system

Copies the current system into a special storage.

4.4.5. Insert system

Paste the system from a special storage into the current tab.

Remark: You can copy and paste also into a second started rail calculation program.

4.4.6. Delete tab

Deletes the current tab.

4.4.7. Delete all Tabs

Deletes all Tabs. (In the Moment) this equivalent the command new file.

4.4.8. Systems

Shows a sub menu with the existing tabs for selection. By selecting a tab the view jumps to this tab.

4.4.9. Project information

Opens the dialogue ‚project information‘.

4.4.10. Selection of profile type

Opens the dialogue ‚Selection of profile type‘.

4.4.11. Support conditions

Opens the dialogue ‚support conditions‘.

4.4.12. Node load

Opens the dialogue ‚node loads‘.

4.4.13. distributed load

Opens the dialogue ‚distributed loads‘.

4.5. Options

4.5.1.Change calculation level

changes the calculation level from x-z-coordinate system to x-y-coordinate system, and the other way round. (see also section 1.3)

4.5.2Automatic FEM-Calculation

Activates/deactivates the automatic calculations after editing the system.

4.5.3Automatic admissible load calculation

Calculates the maximum admissible load to the chosen system.

Therefor the entered loads are scaled, until the system collapses. Hereby you can choose a ratio between the loads among another.

Remark: Because the loads depend on the chosen profile, a combination of this function with the automatic chosen profile is not possible.

4.5.4Automatic suitable profile choice

Trys to use a profile, which can carry the entered loads.

Remark: Because the loads influence the needed profile, a combination of this function with the automatic calculated maximum loads is not possible.

4.5.5.Considering dead load

Switches on and off, if dead load is considered.

4.5.6.Settings

Opens the dialogue ,Options‘.

4.6. Calculation

4.6.1.FEM-Solver

Starts the FEM calculation.

Remark: This command isn't necessary, if automatic calculation is activated.

4.6.2.Forces, Moments

Opens the dialogue ,Forces and moments‘.

4.6.3.Stresses

Opens the dialogue ,Stress‘.

4.6.4.Project / profile type / support conditions / a.s.o.

Jumps to the chosen theme in the result memo, if existing.

4.7. Help

4.7.1.use Help

Opens this PDF (resp. the PDF in corresponding language).

4.7.2.support conditions

Opens a PDF for the theme supports.

4.7.3.Fast help on / off

Switches the hints on resp. off.

4.7.4.Info

Opens the dialogue ,Info'.

4.7.5.Homepage

Starts a browser to show the Müpro-Homepage.

4.8. Language

This element is no sub menu, but opens the selection box for the language to use in the program.

5. Toolbar

5.1. Open file

Opens a windows typical selection dialogue for opening files.

5.2. Save file

Saves the data into the according file. If the file was never saved before, instead ,Save as' is opened. (see also menubar section 4.1.4 **Fehler! Verweisquelle konnte nicht gefunden werden.**)

5.3. New file

Closes the file and creates a new unsaved one.

5.4. Add system

Adds a new tab with a default system.

5.5. Export to Microsoft Word

Opens the dialogue ,Export to Microsoft Word'.

5.6. Print

Opens the dialogue ,Print'.

5.7. Printer Setup

Opens the dialogue ,Printer Setup'.

5.8. Setup

Opens the dialogue ,Options'.

5.9. Data from project manager

Opens the dialogue ,Project information'.

5.10. Change of calculation level

Changes the calculation level from x-y-coordinate system into x-z-coordinate system, and the other way round. (see also section 1.3)

5.11. Selection of static systems (+ selection list)

Opens the dialogue ,Selection of static systems'.

In the attached selection list you can switch on or off automatic load maximising, or start a single calculation.

5.12. profile type (+ selection list)

Opens the dialogue ,Set type of profile'.

In the attached selection list you can switch on or off automatic selection of the profile, or start a single suggestion. Furthermore you can choose here from a list of profiles which is to use.

5.13. Dead load

Switches on resp. off, if dead load is considered.

5.14. Support conditions

Opens the dialogue ‚support conditions‘.

5.15. Node loads

Opens the dialogue ‚node loads‘.

5.16. Distributed loads (+ selection list)

Opens the dialogue ‚distributed loads‘.

In the attached selection list you can switch on or off overlay of loads.

5.17. FEM-calculation (+ selection list)

Starts the FEM calculation.

In the attached selection list you can switch on or off automatic calculation, and search for listed themes in the result memo.

5.18. Element forces

Opens the dialogue ‚Forces and Moments‘.

5.19. Tensions

Opens the dialogue ‚Stress‘.

5.20. Mirror

With this the hole system is mirrored.

Remark: This is only possible by this Toolbutton.

Remark: The automatic query, if the channel is a MÜPRO wall hanger bracket, only takes place when it is a clamped support on the left. But you can choose it by dialogues as normal.