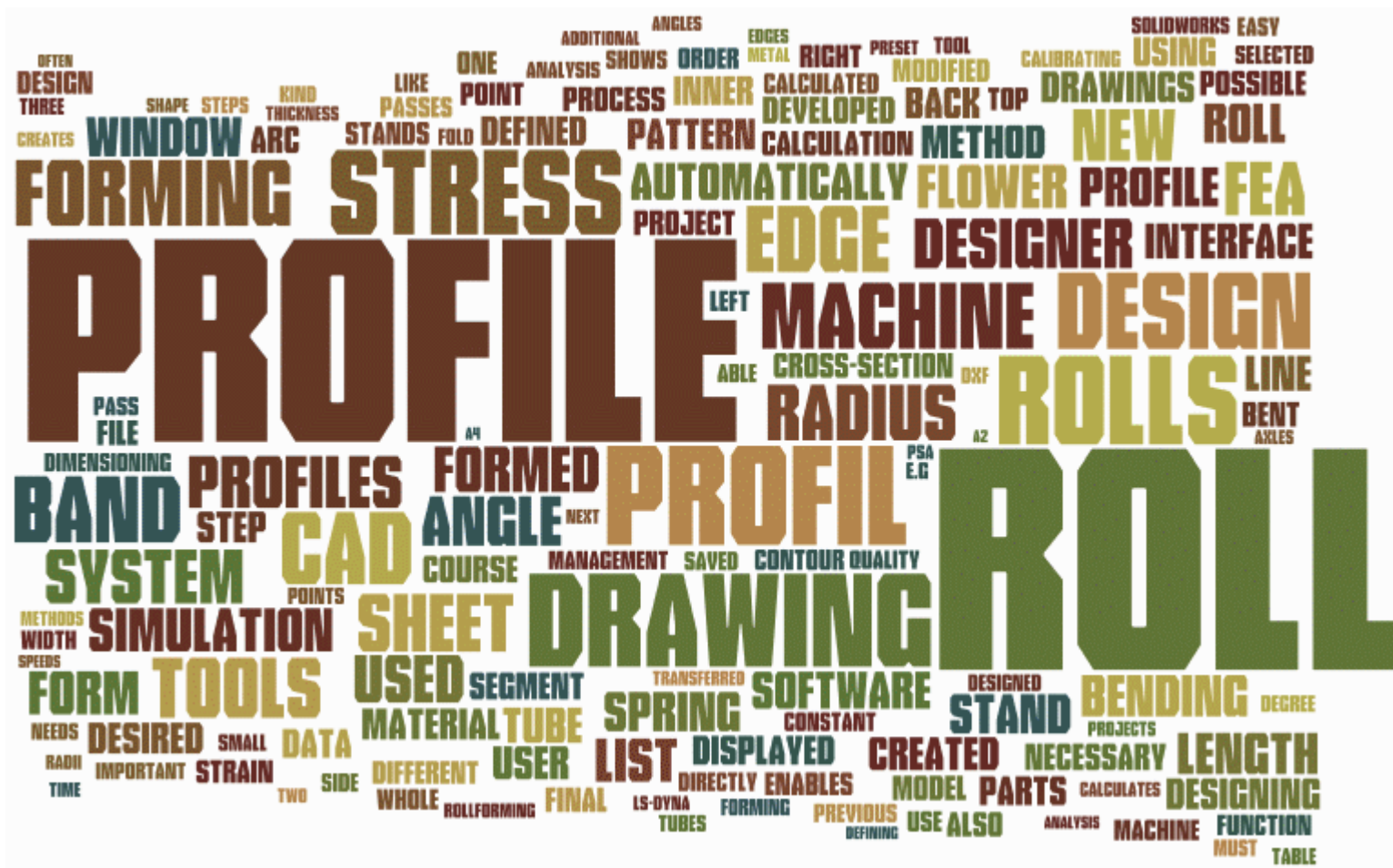


**PROFIL** Rollform Design Software**PROFIL** Rollform Design Software

# User Manual

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This document can also be called as a context dependent help directly from the software.



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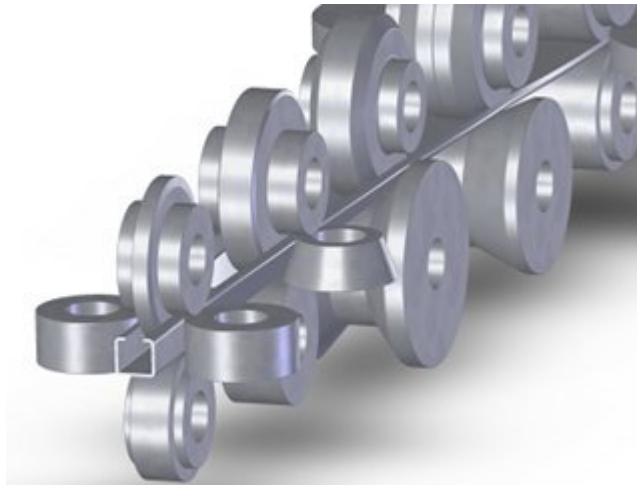
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# 1 General

## 1.1 Roll Forming



### Roll forming

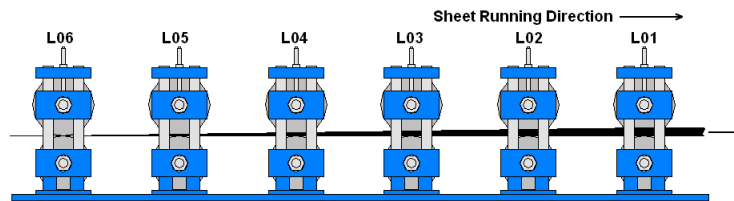
Roll forming is a **continuous bending operation** with rotating tools in which sheet or strip metal is gradually formed in tandem sets of rollers until the desired cross-sectional configuration is obtained. During this operation, only the cross-section is modified, not the sheet thickness. Roll forming is ideal for producing parts with long lengths or in large quantities.

### Profiles



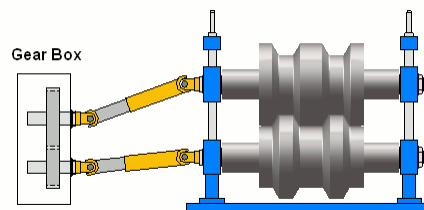
Any kind of shapes can be roll formed, simple and complicated ones. Some examples for roll formed profiles are: U and C channels, door frames, shutter profiles, trapezoidal profiles, corrugated sheet, screen doors, wall and roof cladding, roof bows and trusses, panels, gutters, purlins, fence posts, greenhouse profiles, grape stakes, logistic tracks, drawer slides, studs, beams, beads, shelf racks, sheet piling, guard rails, seat tracks, bumpers, truck and trailer components, window guide channel, seal retainer, cross-members, heat transfer pipes, garage doors, rack beams, duct flanges, drywall profiles, cable trays.

## Machine



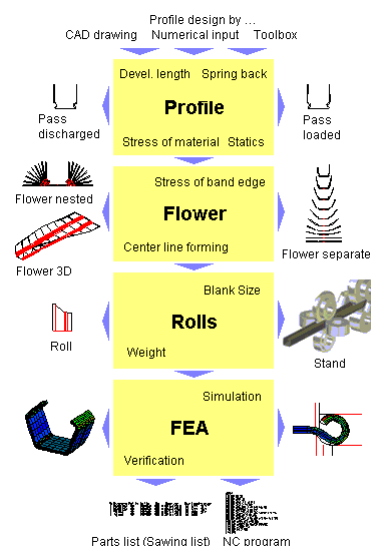
The **roll forming machine** consists of a machine base with a set of **roll forming stands**, arranged in series. Mostly between 6 and 32 stands are needed, dependent on the complexity of the profile.

## Stand



Each stand has a driven top and bottom shaft with force-fit tied roll tools. Often non driven side rolls with vertical axis are used, these side rolls are fastened on special auxiliary holders within or behind the stand. A **common motor** drives the top and bottom shafts of all stands via a **gear box** behind the machine. In order to allow adjusting the vertical position of the rolls, the shafts of the stands are connected to the gear box by using **cardan shafts**. If the gear transmission ratio between top and bottom shaft is 1:1, then the top and bottom rolls have the same working diameter (pitch diameter) in order to have the same circumferential speed at the profile web at least. Mostly however the transmission ratio 1:1.4 is used in order to have a larger top roll diameter (= bottom roll diameter x 1.4). This enables to produce profiles with larger vertical legs.

## Roll Design

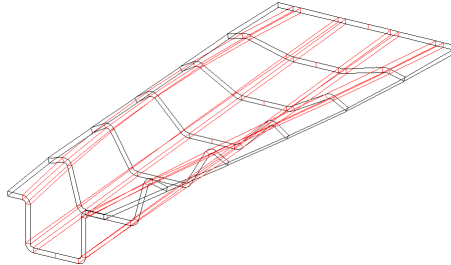


Each profile cross section needs a customized set of roll tools. First, after defining the final profile cross-section, the flower pattern has to be designed. This means, starting with the final section, the cross section of the profile in each stand is defined by unbending the arcs. Afterwards, the rolls have to be designed by deriving the roll contour from the profile contour in each stand. A special roll forming problem is the longitudinal strain within the sheet (see below). To avoid remaining strain and unwanted deformations, it is necessary to check just in time, if the longitudinal stress does not meet or exceed the yield stress. During flower pattern creation, the stress can be checked approximately by the **Stress of Band Edge Calculation** or by the **Profile**

### Stress Calculation (PSA).

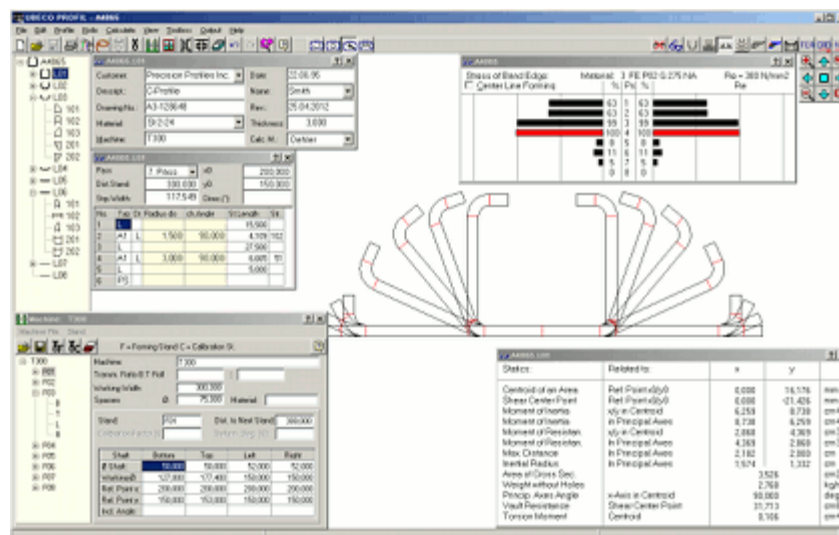
After completion of the roll design, the **FEA** method can be used for a final check. It also helps to evaluate if the designed roll tools are able to form the desired profile with the given allowances. The last step is to export the manufacturing data like parts list (sawing list) and the program for the CNC lathe.

### Strain and stress in the longitudinal direction



Exactly this is the problem: if single points of the sheet cross-section are tracked, movements on curves with different lengths are observed. The result is different strain and stress of the material. As long as this occurs within the elastic bounds, strain disappears again after the profile leaves the final stand of the roll forming machine and the desired profile form can be obtained. If, however, the yield stress is exceeded, remaining strain arises. The local "too much" of material causes unwanted deformations like rippled edges, mostly in case of symmetric profiles. If the profile is non-symmetric, twists around the longitudinal axis or curved profiles can result. When this occurs, time-consuming and expensive modifying of the roll tools is necessary.

## 1.2 What is PROFIL?



**PROFIL** is the roll design software for every manufacturer of cold roll-formed profiles or seamed tubes from sheet metal and for designers of rollformers and tube forming machines.

**PROFIL** enables quicker working and cost reductions in planning, design, calculation and drawing of the profile, the flower pattern (bending steps) and the roll tooling.

**PROFIL** is running under all WINDOWS platforms and has an easy to use WINDOWS-based user interface, which enables experienced WINDOWS users to learn through self-tuition.

**PROFIL** has built in CAD interfaces (DXF, IGES and MI), which can be used for generating drawings in any CAD system. For AutoCAD, SolidWorks, SolidEdge, and BricsCAD the very

comfortable ActiveX-interface is used. 3D models can be transferred to any 3D CAD system via the STEP format in accordance with DIN ISO 10303.

## 1.3 What are the aims of PROFIL?

**PROFIL** will not replace the engineer. This is not possible. But it will give him practical help for a quicker and safer design.

To achieve these aims, the software must relieve the designer of tedious work, e.g. calculating the developed length, drawing the profiles and roll tools, compiling the parts list, etc. So the designer is able to fully concentrate on the design.

A designer, in most cases, is not a computer specialist. Therefore the software must speak the designer's language. And the software must be easy to operate and easy to learn.

The software must fulfil practical requirements and must have great flexibility to adapt to specific requirements. It also must contain the latest results of scientific research.

## 1.4 What does PROFIL do?

**PROFIL** performs these tasks:

- **PROFIL** supports the section definition. Create new profiles by entering the data into the system, by importing a CAD contour or by using the design toolbox for standard profiles, or by combining these powerful tools to get your design.
- **PROFIL** works as your assistant while designing. Besides determining the neutral line and calculating the developed length of the sheet metal, PROFIL calculates all the important data for the bending process: spring back, statics and the stress of the band edge.
- **PROFIL** speeds up your work designing the flower. Simply change angle or radius values with the editor or by using the modify toolbox to create the bending steps. You may select constant developed length or a constant radius method.
- **PROFIL** supports modification and optimization of the flower. After every modification of a bending angle, the stress of the band edge is recalculated and displayed.
- **PROFIL** supports drawing with automatic creation of flower pattern, in a nested or separated view or as a perspective 3D drawing. Drawings can be dimensioned associatively.
- **PROFIL** speeds up the roll design by directly using the profile contour or any contour drawn in CAD. Use the powerful commands to modify the rolls to your individual needs. Create the drawing of the rolls just by key stroke. Roll tool drawings are dimensioned automatically.
- **PROFIL** supports manufacturing of the roll tools by creation of parts lists and NC-programs (DIN 66025).
- **PROFIL** helps searching for suited rolls, if existing old rolls from the roll stock should be re-used in a new project.

## 1.5 Profit by PROFIL

In the past, cold rolled profiles or tubes were designed by manual drawings or by CAD only. All calculations were made by hand. The designer needed a lot of experience and much intuition. To get drawings for very similar profiles or rolls he had to repeat the whole process.

**PROFIL** supports the design as far as possible. Saving you considerable time in creating drawings and using the technological power to check the design during design.

Thus the designer's profit is:



- better design, because calculations are done with high accuracy, input errors can be changed at the stroke of a key,
- quicker design, because calculation- and drawing work is done in the background,
- safer design, because simulations will find out critical points,
- reduce cost by more economical roll design,
- more systematic than pure experience,
- never repeat the same work - use existing designs for easy modifications,
- improve the profile quality; by having more time to consider the forming method while less time is spent on routine calculations.

## 1.6 The PROFIL Story

**PROFIL** has been designed by **UBECO** in co-operation with the German roll forming industry. When it was presented at the Euro-BLECH exhibition in 1986, the attendees were impressed by the capabilities of the software. Soon **PROFIL** became a leading software for roll forming design in Germany.

The first version was running under MS-DOS and HP-UNIX. Since 1997 the WINDOWS version has been available. Because of its very easy to handle graphical user interface **PROFIL** became famous world-wide. Until now more than 500 systems have been sold. Customers in more than 25 countries are using **PROFIL** for their design.

In 2001 the simulation of the roll forming process by using FEA (Finite Element Analysis) was possible for the first time in the designing office of roll forming companies. This is a milestone in the development of design methods, since user without FEA experience are enabled to get the benefit of the FEA simulation to enhance the product quality. When later the PSA (Profile Stress Analysis) was added, a three-step quality management concept was available.

## 1.7 Remarks for Licence

Copyright and connected rights for delivered programs and documentation are reserved by **UBECO GmbH**, Unternehmensberatungsdienst für Computeranwendungen, Iserlohn, Germany.

By signing the bill of sale the customer obtains the right to install and use the program-system **PROFIL** on a single workstation. He is not allowed to install **PROFIL** on further workstations, neither handing **PROFIL** to third parties nor copying original **UBECO** disk with the exception of copy ensuring (one backup).

## 1.8 Support

If you are owner of the **PROFIL** Service Agreement, you not only will regularly get **PROFIL** updates, but you also can contact **UBECO** if you have questions or problems while using **PROFIL** or the CAD-interfaces. Best way is contact by E-Mail, you can send us your project file and we will examine it. Please do not forget to give us the **PROFIL** release number and the CAD-system you are using.

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## 1.9 What's new - Releases 5.x

### PROFIL - Rel. 5.3 - 01-Dec-2015

#### New features:

- [Output, FEA, LS-Dyna, Profile](#): Element type SOLID alternatively to element type SHELL. With it also simulation of roll formed processes are possible that contain massive forming, either wanted or unwanted.
- [Output, FEA, LS-Dyna](#): Output of a logfile for documentation of the FEA settings.
- [Output, FEA, LS-Dyna, Start](#): Configurable FEA project name.
- [Options ActiveX](#): New ActiveX interface to SolidEdge and BricsCAD.
- [Options, General](#): Preview pictures can be switched off, this causes faster screen refresh for larger projects.
- [Partial Project, Add-On](#): Roll number and part number are locked against renumbering.

#### Bugfixes:

- [File New](#): In the path name also a . (dot) is allowed now.
- [Print](#) (directly, without preview): No form feed anymore behind each line.
- [Profile List Window](#), [Strip Width](#) is now also updated if the window is not active.
- [Output, FEA, LS-Dyna](#): Round-off error in \*CONSTRAINED\_GLOBAL, z-Position is now fixed. This causes correct sheet movement.
- [Profile, Read CAD Contour](#): Invisible colors now are replaced by visible colors.
- [Profile, Read CAD Contour](#): Also arcs with reverse rotation now are imported correctly.

### PROFIL - Rel. 5.2.1 - 01-Mar-2015

#### New features:

- [Output, FEA, LS-Dyna, Others](#): Guiding the first node row at the profile lead and tail end, forces safe threading of the profile into the next roller stands and prevents up and down oscillation.
- [Output, FEA, LS-Dyna, Contact](#): Check self-contact starting with a preset stand number. This is useful to avoid penetration in case the profile touches itself.
- [Partial Project Add On](#): Added rolls are locked against automatic roll renumbering.

#### Bugfixes:

- [Output, FEA, LS-Dyna](#): No Runtime-error anymore in case a roll has no bore hole.
- [NC-Program](#): The G-code is now correct if a roll has an edge fillet radius and a neighboring roll exists.
- Creating [Roll Extensions](#) and [Roll Move](#) do not move the next roll anymore if enough space exists between the rolls.

### PROFIL - Rel. 5.2 - 01-Dec-2014

#### New features:

- [Postprocessor](#) to the FEA systems **LS-Dyna** from Livermore Software Technology Corp. The FEA result can be analyzed in **PROFIL** directly.
- [Graph for Stress, Strain, Sheet Thickness](#), shows the LS-Dyna simulation result dependant on the sheet position in the machine.
- Importing a stress-strain-curve from a text file, e.g. from a tensile test, in [Output, FEA](#).
- [Options, Files](#): Presetting a drawing scale while importing a profile or roll contour.
- [Roll, Double Fillet](#): Creating rounded roll corners with two radii and tangential connections.
- The context menu of the [Machine Window](#) and the [Profile List Window](#) enables copying of certain parameters to the corresponding items of all other stands or profile lists.

- The input field [Working Diameter](#) of the [Machine Window](#) enables parameterizing of increasing diameters for keeping the strip under tension.
- During processing a quotation the approximate [Required Count of Stands](#) for a given profile is calculated for the cost assessment.
- Double-clicking on the mouse wheel fits the drawing into the [Drawing Area](#).

#### Bugfixes:

- Changing roll corner radius with [Toolbox Modify](#) does not causes overlaps with neighboring radii anymore.
- Split rolls on multi side axles now can be dimensioned, too.

### PROFIL - Rel. 5.1 - 01-Dec-2013

#### New features:

- [FEA Interface](#) to the leading FEA systems **LS-Dyna** from Livermore Software Technology Corp. The simulation of the roll forming process enables the designer to validate and optimize his roll form design at an early stage before the rolls are manufactured to ensure that the final product meets the particular needs.
- [Curve Generator](#) for quick creating a stress-strain-curve for FEA simulation by defining three characteristic curve points in case the exact curve is not available.
- Bottom/top rolls with arbitrary [Inclination Angle](#) optionally.
- [Profile, Read CAD-Contour](#): Besides scanning the bottom profile side also scanning the top side and the sheet center line is supported.
- [Grid Lines](#) in the drawing area, preset in [Options, Drawing](#) and [Options, Colors](#).
- Showing the [Previous/Next Pass](#) also in the [Pass View](#).
- [Printing](#) all rolls of a certain stand or of the whole project.
- [Printing](#) rolls also with [NC program](#).
- [Printing](#) profile list, radius and angle bold for determination [Loaded/Discharged](#) state.

#### Bugfixes:

- ActiveX-Output to AutoCAD now creates correct width dimensioning in case of an [Inclination Angle](#).
- Export of [Additional Side Axles](#) to the machine file now is possible.
- [Dimensioning](#) between different rolls on the same axle is possible again.
- [Roll Read CAD Contour](#): Contour tracking does not change the profile side anymore, if a the end of a line touches another line.

### PROFIL - Rel. 5.0.1 - 01-Feb-2013

#### Bugfixes:

- [Roll, Arched extension](#) now works fine again.
- If increment is selected in the roll number key in [Options, Rolls](#) or [Options, Spacer Rolls](#), the variable \$RW now is replaced by the correct roll width.
- In the PROFIL LT version, the [Sheet Thickness](#) can be modified again.
- [Emptying Profile](#) will not delete or modify roll dimensioning of the same pass anymore.
- If **Use layer numbers instead of names** is set in [Options, Files](#), the spacer rolls now get the correct layer numbers.
- No runtime error anymore after deleting the first corner point of a roll ([Roll, Comer, Remove](#)) and after selecting other rolls by using the [Explorer](#).
- [PSA](#) and [View, Flower 3D](#) is now displayed correctly in case the function [Modify Develop Point](#) was used previously.
- In the [Roll Assembly Plan](#) all [Spacer Rolls](#) now are shown completely, in case **Use layer numbers instead of names** is preset in [Options, Files](#).

**PROFIL - Rel. 5.0 - 01-Dec-2012****New features:**

- [Modify Sheet Thickness](#) with constant inner or outer radius, constant neutral line, or constant geometric center line.
- [Modify Strip Width](#) for the current pass or all passes of the whole flower pattern.
- [Modify Reference Point](#) for the current pass or all passes of the whole flower pattern.
- Expanded [Undo /Redo](#) functions: The function name of the next step is shown and the count of steps can be preset by the user.
- [Spacer Rolls](#), that are objects like forming rolls. They can be dimensioned, designated by a special number key, and modified geometrically like forming rolls.
- [Roll Angle](#) in the [Roll Tool Window](#) can be modified.
- [Clearance Angle](#) can be defined relative to the actual angle or to the roll axle.
- [Parallel Gap](#) between roll and profile.
- [Modify Roll Reference Point](#) (context menu): Axial only, radial only, or both.
- [Modify Develop Point](#), in order to redefine the develop point during flower pattern creation.
- For [Output 3D Model -> CAD](#) via ActiveX the rolling direction can be selected.

**Bugfixes:**

- [Undo /Redo](#) is available now for all commands that modify the project, also after changing the view.
- [Import Profile Lists/Roll Tool Files](#) now imports the machine data correctly.

## 1.10 What's new? - Releases 4.x

**PROFIL - Rel. 4.8 - 01-Dec-2011****New features:**

- New snap points for setting the profile reference point in the [Window Read CAD Contour](#): Line Center Point, Arc Quad Point 270° and 90°.
- Arc type A4 with modifying angle and radius, see [Arc Types](#).
- [Draft Modus](#) for modifying angles and radii of arc segments without bending or unbending the segments.
- [Splitting](#) and [Joining](#) profile elements (lines and arcs).
- [Modify Start Element](#), for optimizing the orientation of the profile in the machine, also for changing the opening direction.
- [DXF Output](#): Objects not only organized on layers, but optionally also in blocks.
- [Partial Project Add On](#) and [Partial Project Save as..](#) for combining new profile projects from parts of existing projects.
- For the spacers, [Output to CAD](#) now creates unique layer names that contain the pass number.

**Bugfixes:**

- [NC DXF Output](#) and [NC Program Output](#): Separate files are now created with unique file names.
- [Profile, Read CAD-Contour](#) now also is possible in case the first drawing element is an arc with a very small radius.

### PROFIL - Rel. 4.7 - 01-Dec-2010

#### New features:

- [Machine Data](#) are handled in the [Project](#) and saved in the project file, new machine explorer, interactive handling of the [Machine Window](#).
- [Multi Axles](#): Additional side axles for the precise forming of inner contours that are difficult to access.
- [Print Preview](#): "Scale Fit" adjusts the drawing scale in order to have the whole print-out on one page only (in case the table lengths are not too large).
- [Shaped Tube Calibration](#) also starting from an elliptic cross-section in the welding station. Useful if the shaped tube is either quite large and flat or quite high and narrow.
- [Shaped Tube Calibration](#) also by keeping the cross-section of the shaped tube unchanged. This means, the shaped tube is formed as an open profile, welded, and then calibrated.
- Beside the previous pass also the [Next Pass](#) can be inserted in the roll tool drawing.
- New [Variables](#) for consecutive numbering of the rolls of a stand and of a shaft type.

#### Bugfixes:

- The fixed scales in [Print Preview](#) and [Roll Assembly Plan](#) are adapted to DIN ISO 5455.

### PROFIL - Rel. 4.6 - 01-Dec-2009

#### New features:

- [Options, Drawing](#): Profile element separator lines can be switched off, thus better manual extruding in 3D CAD.
- [Radius Dimension](#): Function [Move Dimension](#) now also rotates dimension line and text.
- [Roll, Read CAD Roll](#) imports a complete roll from CAD and positions it on the selected shaft without modifications.
- New [Space Mouse](#) integration with improved rotation function in 3D.
- [Output to CAD](#): No layer name limitation to 8 characters anymore, thus longer roll numbers possible.
- [Print Preview](#): The pathname is displayed in short form in case it is too long.

#### Bugfixes:

- No "Stream-Error" anymore during loading INI files from older releases.
- [Automatic Flower Creation](#): Alternating bending methods are handled correctly now.

### PROFIL - Rel. 4.5 - 01-Dec-2008

#### New features:

- Creating the [profile pass](#), the [PSA model](#), the [roll tool stand](#), or all stands as 3D models in [STEP](#) format in accordance with DIN ISO 10303.
- Extended [Import](#) functions: DXF, KTR, Profile Lists, FEA Result.
- Extended [Export](#) functions: DXF, IGES, MI, A11, Profile Lists, Parts List, NC-Program, FEA Model.
- New drivers for the [USB-Hardlock](#), by this fit for Windows XP/64bit.
- Integer places (digits left of the decimal separator) can be preset in [Options, Calculate](#).
- [Mirror Roll](#): Roll No. and Part No. are kept if this is preset in [Options, Rolls](#).
- [Roll Stock Management](#): New import function.
- [FEA](#): General Contact.
- [FEA](#): FIL files created on Dual-Core-Processors can be opened now.
- [FEA](#): Toggling to sequential reading of a FIL file, in case the file is too big for direct opening.
- [FEA](#): Mass scaling time interval and recommended value with 2 decimal places.

**Bugfixes:**

- [Roll Assembly Plan](#) templates now are saved again with variables.
- [Open Fold](#) now calculates the developed length correctly, if the profile starts with a 90 degree bend.
- For the roll corner point functions shortcuts can be defined as well.
- [Center Line Forming](#): Warning message in case of empty profile elements is now added.
- [Dimensioning](#): No wrong geometry reference anymore if mirrored profile elements are dimensioned and profile elements are removed.
- [Radius Dimensioning](#) is now removed if the profile is unbent to flat.
- [Part List](#): The gross and final weight is not doubled anymore in case of saving to text file.
- [SolidWorks Interface](#): 3D Output of a fold with inner radius 0 is now correct.
- [FEA](#): No penetrations anymore in the display of sharp inner corners.
- [FEA](#): The stand drawing is now displayed in case of missing bottom rolls as well.

**PROFIL - Rel. 4.4 - 01-Dec-2007****New features:**

- [3D Model -> CAD](#) transfers the rolls of the current stand or all rolls of all stands to AutoCAD or SolidWorks.
- Automatic [Trapezoidal Profile Forming](#), either with cosine band edge course or linear course with user defined fillet radii.
- [Trapezoidal Profile](#) as a new component of the [Profile Design Toolbox](#).
- [Copying](#) of marked profile elements as a block via the clipboard, also into other profile lists.
- [Angle Dimensioning](#) also related to the horizontal or vertical axis, with selection from the context menu.
- [Angle Dimensioning](#) also for rolls.
- [Mirroring](#) side rolls to the opposite side.
- [FEA](#): improved message handling, when critical ratio length/width is exceeded.
- [FEA](#): reduction factor "pressure overclosure" can now be preset, in order to simulate extreme small sheet thickness.
- New HTML based help system, by this fit for Windows Vista.

**Bugfixes:**

- Roll assembly plan templates now always are saved with file extension .DXF.
- DXF files again contain dimensioning.
- [Assembly Plan, Drawing -> CAD](#) transferred a template in error.
- Invisible buttons in [Window Read CAD-Contour/ Scan Profile Drawing](#) now are visible again, when the screen resolution is set to 120 dpi.
- [Drawing -> CAD](#) to ME10 via MI file is now also possible if spacers exist on top or bottom shaft only.

**PROFIL - Rel. 4.3 - 01-Dec-2006****New features:**

- [Plotting](#) a standardized roll assembly plan without CAD by using a [Drawing Template](#) (drawing frame with title block).
- Extended [Variables](#) for Number Keys in [Options Rolls](#) and Title Block in the [Drawing Template](#).
- Context menu to speed up the profile and roll design and for adapting the machine parameters by using the right mouse button in the [Drawing Area](#).

- Improved network support: An opened project file can be opened by another user “on approval”, see [Open Project](#).
- [Partial Ellipse](#) as a new component of the [Profile Design Toolbox](#).
- [Read CAD-Contour](#) imports partial ellipses and converts them to arcs approximately.
- Shaped tube calibration now also is possible behind round tubes calibration. For it, the [Deformation Degree](#) is set to 0.
- The [Machine](#) can be removed from the [Project Data Window](#) by using the Del key.
- [Drawing -> CAD](#) creates a layer named “Pass Number” instead of “Profile List Number”, if the [Explorer](#) is switched to pass number.
- Improved editing functions of the numerical values in the input fields.
- Scaling factor of the z axis in [View Flower 3D](#) can be preset.
- Reversible zoom direction when spinning the mouse wheel (see [Options Mouse](#)) and mouse position dependent zoom.
- Support of the [Space Mouse](#) from 3DConnexion as navigation input device.
- Either [USB Hardlock](#) or hardlock for the parallel interface.

#### Bugfixes:

- Saving the [Machine File](#) is possible now in case of an empty deformation degree field of calibration stands.
- No data loss anymore when in the [Machine Window](#) forming stands are appended behind calibrating stands.
- [Roll Table](#), Insert Roll from CAD is now possible again.
- For the [Weight](#) in the [Statics Table](#) holes/cut-outs now are not considered since input of count and length is not supported.

### PROFIL - Rel. 4.2 - 01-Dec-2005

#### New features:

- User defined shortcuts for any menu item, see [Options Keyboard](#).
- Parts list can be created not only in a fixed output path, but also in the path of the belonging project file, see [Options Parts List](#).
- Changing the radius of roll corner points (fillets) now is possible in case of pure arc transitions to the neighbouring corner points, see [Roll Corner Point, Radius](#).
- [Plausibility Check](#) recognizes and flags implausible profile lists and roll contour errors.
- The [Calculator](#) can be opened from the context menu of any numerical input field.
- The [Explorer](#) is able to show either the profile list number (counting against the sheet running direction) or the pass number (counting in sheet running direction).
- The [Development Table](#) represents the angle sequence of the flower pattern. Bending angles are shown either in degree or in percent related to the final angle.
- [Automatic Flower Creation](#) by using the development table to speed up the design of similar profiles.
- [Converting Line to Arc](#) (L to A1).
- [Center Line Forming](#) lowers all profile lists to constant height of the centroid or by any other amount.
- Improved roll design: By using the functions [Roll Paste](#), [Roll Read CAD-Contour](#), [Roll, Scan Profile Drawing](#), and [Roll Move](#) also certain rolls can be modified and replaced subsequently.

#### FEA interface extensions:

- Self-contact sheet with sheet can be considered, see [Output FEA](#), Others.



**Bugfixes:**

- After [Profile, Remove](#) it is possible again to save the project.
- [Profile, Read CAD-Contour](#) now also is possible for extreme small profiles.
- No sporadic moving of the roll anymore after negative [Cylindrical Extension](#) and [Conical Extension](#).
- [Roll Scan Profile Drawing](#) now ignores the cross-hair circle of a roll corner point.

**PROFIL - Rel. 4.1 - 01-Dec-2004****New features:**

- [Roll, Mirror](#) for easier designing of rolls for symmetrical profiles.
- Improved [Window Read CAD-Contour](#) and [Window Scan Profile Drawing](#) respectively: zoom and move functions, wheel mouse support, manual controlling of the automatic contour tracking simply by clicking on the next drawing element.
- Search paths to the system files [Material File](#) and [Factor file](#) can be preset in [Options Calculate](#).
- Spacer's material for the [Parts List](#) can be preset in the [Machine Window](#).
- Stand number is shown in the table in the [Machine Window](#).
- [Profile, Insert](#) or [Profile, Append](#) copy rolls too, if existing. The user is asked for confirmation.
- [Roll Renumber](#) creates new roll and part numbers dependent on the number keys.
- Improved DXF interface from CAD: [Profile, Read CAD-Contour](#) and [Roll Read CAD-Contour](#) now open any [DXF format](#) created by CAD. Contour definition and tracking is done in PROFIL now.
- New help assistant for easier learning the software.
- [Profile Explorer](#) gives clear overview of the project, the passes, stands, and rolls. Useful for quick selection of any component. Options Explorer for clear selection of option settings.
- The status bar shows the maximum stress of the [PSA - Profile Stress Analysis](#).
- [Check for Update](#) online.
- System and user settings can be stored in an INI file as well (Setting in [Options General](#)).
- [Profile, Mirror](#) mirrors unsymmetrical profile lists at the reference point.

**Bugfixes:**

- No stand number change anymore after saving the machine data.
- Scrolling of the content of the Stress of Band Edge Window now possible.

**PROFIL - Rel. 4.0 - 01-Dec-2003****New features:**

- [PSA - Profile Stress Analysis](#), calculation of the approximate stress within the whole profile cross section with colored 3D display.
- [Profile Catalogue](#) (profile database), gives an overview of all earlier produced profiles, with quick search functions, graphic display of the final profile, and quick access to the project file.
- ActiveX-Interface to SolidWorks 2003 (2D Drawing) ), see [Options ActiveX](#).
- [Track Holding Method A4](#) with constant intersection point on the inside or outside as selected (preset in [Options Profile List](#)).
- [Calculation Method](#) DIN 6935 of the developed length by table or formula as selected (preset in [Options Calculate](#)).

**Bugfixes:**

- No empty entries anymore in the [Parts List Columns](#).
- Statics parameter table can be [Copied](#) to the clipboard now.



- New representation of arcs by polylines in the [Drawing Area](#). Thus, the arcs do not disappear anymore while zooming by the [Navigator](#) and more enlargement is possible.

## 1.11 What's new? - Releases 3.x

### PROFIL - Rel. 3.4 - 01-Dec-2002

#### New features:

- [Shaped Tubes](#): automatic creation of the calibrating stands for any shaped tube cross-section, dependent on given calibrating factor and forming degree.
- [Bending Method A4](#) as a combination of A2 and A3 with user defined distribution of the residual lengths to the previous and next segment and with automatic distribution for guiding the strip straightaway (track holding method).
- Perspective angle 0 (side view) for [View Flower 3D](#) (Setting in [Options Drawing](#)).
- Automatic adaptation of the rolls when changing the working diameter or the reference point (Setting in [Options Rolls](#), "Change Machine Data").
- Search path for the bore hole, bushing, identification groove, and material files are remembered (see [Expanded Roll Tool Window](#)).
- Reference point of automatic roll dimensioning can be set to the left or right side (see [Automatic Roll Dimensioning](#)).
- Main line color of the profile and the rolls can be set separately (Setting in [Options Drawing](#)).
- Decimal places of the co-ordinates of the NC program can be preset (Setting in [Options NC](#)).

#### Bugfixes:

- In the parts list column "Actual Diam." the options setting "Diameter from Intersection Point/Actual Maximum" now is considered (see [Parts List Columns](#)).
- [Open Fold](#) now calculates the correct strip width in case of symmetrical profile.

### PROFIL - Rel. 3.3 - 01-Dec-2001

#### New features:

- [FEA Interface](#) to the leading FEA system ABAQUS/Explicit from SIMULIA Dassault Systèmes. The simulation of the roll forming process enables the designer to validate and optimize his roll form design at an early stage before the rolls are manufactured to ensure that the final product meets the particular needs.
- [Undo /Redo](#) command, for reversing the 5 most recent operations and for reversing the effect of the previous undo command.
- [Copying](#) of the drawing as a pixel image to the Windows clipboard for transfer it to any other Windows application.
- [Open Fold](#) command, useful if in the last pass a 180 degree fold with inner radius 0 should be created by pressing together the legs of a bend.
- [Clearance Angle](#) by entering a desired angle value and by extending or shortening the arc.
- [Arched extension](#) for rolls that end with an arc or a line. In case of arc the arc is lengthened; in case of line an arched extension with a selectable radius is added.
- Extended [extension](#) settings: by width (relative), to width (absolute) and to diameter (absolute).
- New [Roll Properties](#), separate for each roll: [Bore Hole](#), [Bushing](#), [Identification Groove](#), [Material](#), [Treating](#), [Surface](#), [Addition](#), [Remark](#). Default values can be preset.
- User defined composition of the [Parts List Columns](#) with selectable sorting and sum fields. As well as rolls the parts list can contain spacers and bushing. Rolls with same parts list data are summarized to one parts list entry, if the column "Count" is selected.
- The addition for calculating the blank size of the rolls is taken either from the intersection point of

the tangents or the actual maximum diameter by user preset, see [Options Parts List](#).

- The parts list can be transferred to MS-Excel by ActiveX, starting in the marked cell, see [Options Parts List](#).
- Bore hole designation is shown in the drawing, all texts aligned left or center, see [Options Drawing](#).
- Selecting layer numbers instead of layer names, when the output file is created ([Options Files](#)).
- Improved online help.
- User defined text editor instead of NotePad, see [Options General](#).

#### **Bugfixes:**

- No sporadic error message anymore when [removing](#) a profile list.
- Bug fixed: When splitting a roll [between corner points](#) the input of the limits sometimes is not possible.
- No sporadic error message anymore during the function [3D-Stand -> AutoCAD](#).

### **PROFIL - Rel. 3.2 - 01-Nov-2000**

#### **New features:**

- Creating a [Photo-Realistic Image](#) of a stand or of the profile (if PROFIL is connected to AutoCAD R14 or higher), suited for demonstrating the company's products e.g. in presentations, offers and advertising brochures.
- Wheel mouse support, using the wheel to [Zoom and Move](#) the drawing in the drawing area.
- Output of the roll drawings in [Separate DXF Files](#) with polylines, suited for interfacing to an NC programming system.
- Improved path selection in the [Options Window](#).

### **PROFIL - Rel. 3.1 - 15-May-2000**

#### **New features:**

- Automatic roll design for any kind of profile, folded and complicated, too, and for every kind of shaft, without using CAD. The new function is called [Roll, Scan Profile Drawing](#) (replaces the function "Roll, Read Profile List") and proceeds a hidden line algorithm for the drawing in the drawing area. Existing rolls on other shafts are considered, too.
- In the roll tool drawing, the [Previous Pass](#) can be inserted. This is useful for checking out if the incoming profile is treated correctly by the rolls and if no conflict occurs.
- In the [Print Preview](#) now any user defined scale can be selected.

#### **Bugfixes:**

- Side rolls with an inclination angle now are dimensioned correctly.
- The diameter of the cross-hair circle of the marked roll corner point now is independent of the zoom factor.
- In the [Print Protocol](#) of the rolls the widths now are related to the corner of the roll.

### **PROFIL - Rel. 3.0 - 14-Nov-99**

#### **New features:**

- [ActiveX Interface](#) to AutoCAD 2000
- Fully associative [Profile and Roll Dimensioning](#).
- [Automatic Roll Dimensioning](#).
- Transfer of the dimensioning to AutoCAD R14 and 2000 via [ActiveX](#).
- Extended functions for [Tube Forming](#): Welding pass, fin pass, break down pass, automatic generation of rolls for all passes.

- [Roll tool Stock Management](#) with quick searching for suited rolls for re-use (option).
- [Project Select Window](#) with graphic preview.
- Interface to the [FEA Technologic Processor](#) of the Institute of Production Engineering and Forming Machines (PtU Darmstadt, Germany) for FEA calculation.
- Maximum count of [Profile Elements](#) increased from 99 to 199.
- Presetting of the edge rounding radius in [Options, Roll](#).
- Extended online-help, with new table of contents.
- Button [Save](#) only enabled, if the project was modified.
- Single passes or rolls can be transferred to CAD.
- Text height can be preset in [Options, Drawing](#).
- If the input fields "Top Roll, Ref. Point x/y" of the machine data window are empty, the [Roll Reference Point](#) for the top rolls now is set to the upper side of the sheet automatically.

**Bugfixes:**

- No runtime-error anymore while removing a roll and the [Roll Tool Window](#) is open.
- No sporadic error message anymore after opening a [Profile Design Toolbox](#) window.
- The [Parts List](#) in the Imperial System now contains blank sizes and bore diameter with two decimal places.

## 1.12 Frequently Asked Questions

**Q:** The windows are too small, some contents cannot be read.

**A:** Please set the WINDOWS screen settings to "Small Fonts".

**Q:** In the print-out vertical and diagonal dimension texts are absent.

**A:** Some older printers support text angles of 0° only. The same problem can occur in the landscape format.

**Q:** If I select another working diameter in the machine file, the rolls move away from the profile. How can I modify the diameter afterwards without designing the rolls newly?

**A:** Uncheck the box "Keep Roll Data" in "Options, Rolls" previously.

**Q:** During driver installation for the parallel hardlock I got the error message "System Error 1275". What is the reason?

**A:** You tried to install a 32 bit driver in a 64 bit system. This is refused by WINDOWS 7. We recommend to update to the topic PROFIL release by changing the parallel hardlock against an USB hardlock concurrently.

**Q:** During bending with arc type A2 the profile form keeps unchanged, only the material is moved to the neighbor arc.

**A:** The profile is bent up only if the previous segment is a line. So insert a line segment with length zero in front of the A2 arc, before you bend up the arc.

**Q:** Bottom and top rolls are combined in one parts list row, left and right side rolls, however, not. Why?

**A:** Also left and right side rolls are combined in case of equal values in all visible columns. Check if they have the same designation! (Set-Up in "Options, Parts List, Set-Up Columns, Designation").

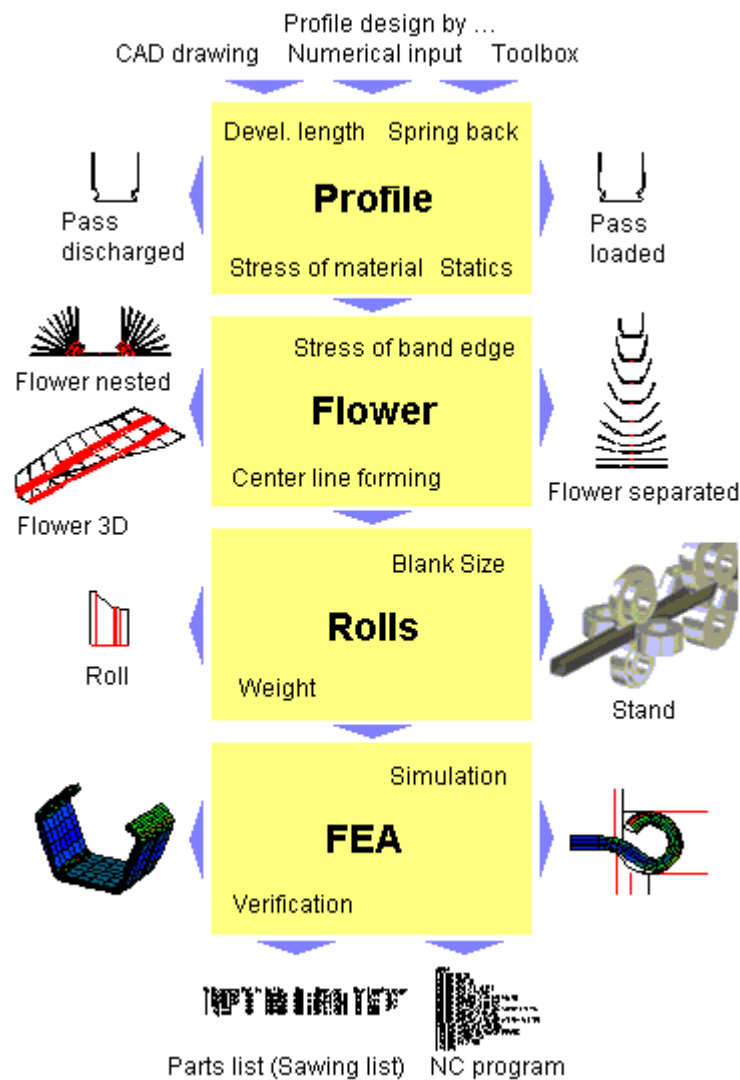
- Q:** I want to use Docol DP1200 for roll forming. Do you have the specifications for the material file?
- A:** I'm sorry not. If you want to calculate the stress of band edge or the PSA, you only need to select a similar material from the material file. Check and modify the yield stress dependent on the supplier's specifications. If you want to calculate the spring back, more effort is necessary. You need the spring back factors from two bending experiments with two different inner radii. More see PROFIL user manual.
- Q:** Although I checked "Spacers" in Options, Drawing the spaces are not shown in the drawing. Why not?
- A:** You probably forgot to fill out Working Width or Spacer Diameter in the machine window. Without these data the spacers cannot be shown.
- Q:** The window "Options, Files" is disabled. Why?
- A:** You have enabled ActiveX. Uncheck the boxes in "Options, ActiveX". Then you can set-up the file interface.
- Q:** In "Options, Rolls", I selected "Keep Roll/Part No." for mirroring. In the parts list, the top and bottom rolls appear with count 2 correctly, the side rolls, however, not. What is wrong?
- A:** Count 2 appears if and only if the parts list rows are fully identic, also in the column "Designation". Please check "Options, Parts List, Set-Up Columns, Designation" if the designation for left and right side rolls are identic. Alternative: Remove the column "Designation". Then the designation is not checked for identity.
- Q:** How can I install a PROFIL update without losing my settings?
- A:** Do not uninstall the old release. Simply install the new release by overwriting the old one, then the settings from the Windows system registry keep alive. Alternatively, export the setting from the old release into an INI file (Options, General, Save INI file settings) and reload it in the new release.
- Q:** Why do the short keys Ctrl-C and Ctrl-V for Copy and Paste not work in the PROFIL input fields as in other Windows programs?
- A:** By default, these short keys are assigned to "Roll, Copy" and "Roll, Paste". Simply remove the assignment in "Options, Keyboard", then you can use the short keys Ctrl-C and Ctrl-V in input fields as usual. Alternatively open the context menu by right mouse click in the input field: The functions Copy and Paste work independent on the assignment.
- Q:** Why is it not possible to dimension the distance between top and bottom roll?
- A:** The dimensioning works object orientated and associatively in contrast to many CAD systems. This is why both dimension points must belong to the same dimension object (i.e. roll or axle). Use the function "Measure" instead, which has not this object restriction.
- Q:** I want to copy the whole left half of a profile from an old to a new project. Only the right half should be created newly. How should I proceed appropriately without using a CAD system?
- A:** Copy the whole left half into the clipboard. For it select the first profile element in the profile list window, press and hold the "Shift Key" and press the "Arrow Down Key" until all profile elements of the left half are marked. Then copy the whole marked set by using "Profile, Element, Copy" into the clipboard. In the new project, select the desired row and call "Profile, Element, Insert".
- Q:** I cannot enter machine data into the empty machine window. What is going wrong?
- A:** You forgot to create a stand. Press the button "Append Forming Stand" and the final stand F01 appears in the machine explorer. Now enter the stand data. Afterward append further stands; the stand data are also copied and you can modify them if needed.

- Q:** The automatic trapezoidal profile forming creates an improper result. Why?
- A:** Check if the final profile has arcs with inner radius 0. If yes, modify them to a realistic value of 0.2 mm at least. Moreover check if the profile has arcs at the top and the bottom. These arcs have to be split at the top and bottom point (90° and 270°).
- Q:** How can I avoid the springback of the profile after leaving the machine?
- A:** By [overbending](#) each arc in that pass in which it is bent to the final angle. Another smart measure is to use the [Arc Type A4 with the Modify angle and radius option](#) in an additional calibrating stand at the end of the forming process. This method overbends parts of the arc and bends them to flat again in order to compensate the spring back. If the corner of the profile should be formed to a hem (180° fold with inner radius 0), use the [Open Fold](#) function in order to avoid spring back.
- Q:** How can I avoid the end flare effect after cutting the profile?
- A:** End flare means one end opens and the other closes after the split cut. In order to decrease this effect, use smaller bend angle change from pass to pass, i.e use more stands. Another measure is to insert side rolls between the stands. They prevent spring back of the profile legs between the stands. Also using of the [Arc Type A4 with the Modify angle and radius option](#) decreases the end flare effect.
- F:** The ME10 macros do not work under PTC Creo Elements/Direct Drafting rel. 19. What can I do?
- A:** In rel. 19 new internal variables are used, whose names conflict with macro names. The new adapted macro rel. 2.4 is available in the download section (<http://www.ubeco.com>).
- F:** The LS-Dyna simulation modifies the sheet thickness, although the roll gap equals the sheet thickness. Why?
- A:** You are using an older LS-Dyna release. Update to the solver R7 or later.
- F:** I am just designing a quite complicated profile with 32 passes. The more rolls I create, the slower is the screen refresh after each roll modification. How can I accelerate it?
- A:** Switch off the preview pictures in [Options, General](#).



## 2 Tutorial

### 2.1 How to Work



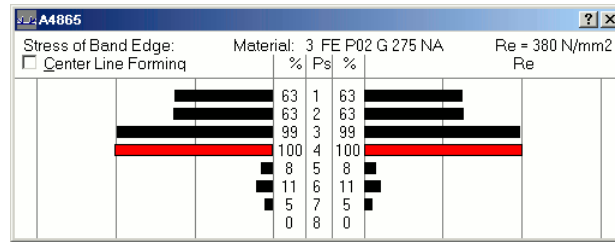
To design the roll tools for an open, cold rolled profile, proceed the following steps:

- defining the desired [Profile or Tube Cross-Section](#) and calculating the initial strip width
- defining the [Bending Steps \(Flower Pattern\)](#) dependent on the permissible longitudinal strain
- designing of the [Roll Tools](#)
- if needed: verifying the design by [Finite Element Analysis](#)
- output of the [Manufacturing Data](#)

To design the roll tools for a cold rolled seamed tube or a shaped tube, use the [Toolbox Tube Design](#)

## 2.2 Quality Management

PROFIL has a three step concept for quality management:

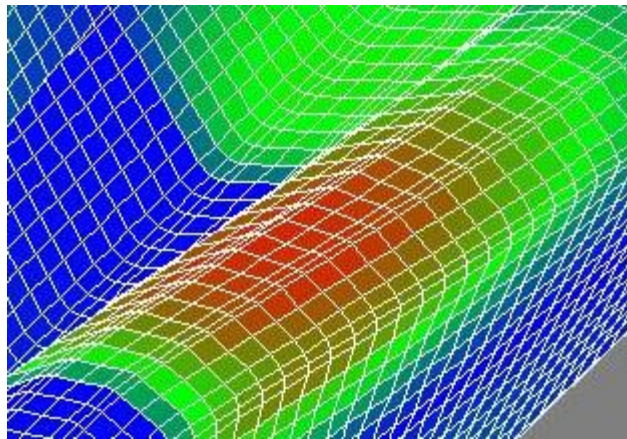


Stress of band edge as a bar diagram

### Step 1: Calculate Stress of Band Edge

Strain and stress at the band edge are calculated and displayed as a bar diagram. Properties:

- quick and approximate check of keeping the dangerous yield point limit
- can be displayed simultaneously while designing and optimizing

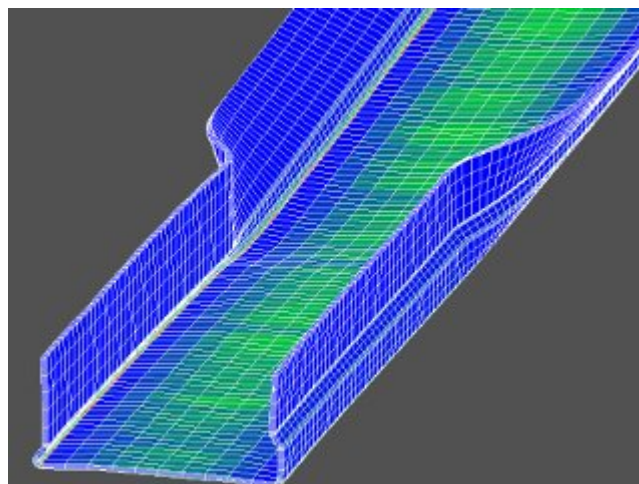


PSA - Profile-Stress-Analysis, shown in the PROFIL drawing area

### Step 2: PSA - Profile-Stress-Analysis

Stresses in the whole profile are calculated while passing through the roll forming machine and are displayed as colored faces in a 3D-graphic. Properties:

- quick and approximate check of keeping the dangerous yield point limit, especially when extreme stresses are not at the band edge, e.g. if edges are folded and the folds are bent
- can be checked by key stroke





Result of an FEA simulation with ABAQUS/Explicit, shown in the PROFIL drawing area

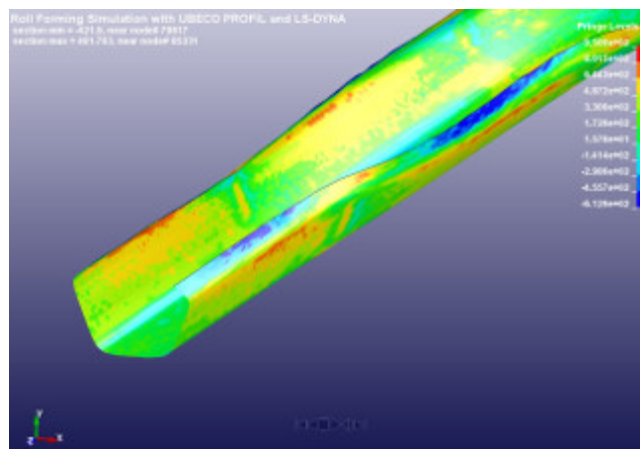
### Step 3: FEA - Finite-Element-Analysis

The FEA simulation calculates how a flat sheet is formed by the designed roll tools. Properties:

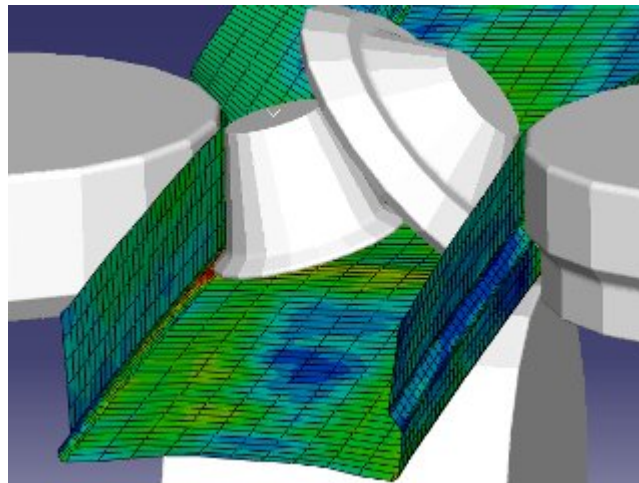
- simulation of the roll forming process, very precise calculation of strain and stress
- very precise calculation of the cross section of the final profile that will be produced by the designed roll tools. Check if the deformation is within the given allowances.
- because of the computation time the simulation makes sense at the end of the design process

All three steps can be combined excellently: step 1 can be used simultaneously while designing, step 2 can be called by key-stroke in critical situations and step 3 should be used at the end of the design process for a final check if the designed roll tools are able to produce the desired profile.

## 2.3 FEA (Finite Element Analysis)



Result of an FEA simulation, shown in **LS-PrePost**, the user interface of **LS-Dyna**



Result of an FEA simulation, shown in **ABAQUS/CAE**, the user interface of **ABAQUS**

The simulation of the roll forming process by using FEA is the third step of the three step [Quality Management](#) concept. The FEA (Finite-Element-Analysis) simulation gives the designer very precise information about stress and strain within the whole profile and about the cross sectional pattern of the final profile. This enables him to validate and optimize his roll form design at an early stage before the rolls are manufactured to ensure that the final product meets the particular needs.

To proceed this very precise calculation of the stress and the longitudinal deformation within the

whole profile, an interface has been provided to the leading FEA systems **LS-DYNA** from Livermore Software Technology Corp. and **ABAQUS/Explicit** from SIMULIA Dassault Systèmes.

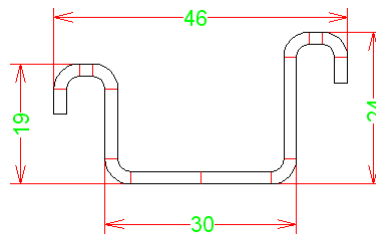
The necessary steps are:

- Design the profile, the flower pattern and the roll tools, afterwards create the interface files by using the function [Output FEA LS-Dyna](#) or [Output FEA ABAQUS](#).
- Proceed the FEA-simulation by using **LS-DYNA Solver** or **ABAQUS/Explicit Solver**.
- Check the result of the simulation by using the function [View FEA Result](#) or with **LS-PrePost** (see picture) or with **ABAQUS/CAE** (see picture).

The result of the FEA-calculation is the a CAD-drawing of the profile, which can be dimensioned and checked, if the deformation is within the allowances. If not, the design has to be modified. All this happens before the rolls are manufactured.

## 2.4 Profile

### 2.4.1 Designing the Profile



Create a new, empty profile list by using the function [File New](#), enter a suitable file name that allows easy identification of the project. Enter the project data into the [Project Data Window](#).

Select among 3 different methods, the toolbox, the graphical or the numerical method for data input:

- Use the [Toolbox Profile Design](#) for quick and easy design of standard profiles like U, C, Hat, etc. without CAD or for applying extensions to them.
- Use the [Graphical Method](#) to design a more complicated profile, which is better to be drawn in CAD.
- Use the [Numerical Method](#) to design a simple, rectangular profile or a tube. Because in this case you know all input data (or you easily can calculate them), which are directly entered into the profile list.

If after defining the profile cross-section modifications in the profile list should be necessary, toggle to the [Draft Modus](#). You can modify angles and radii of arc segments without bending or unbending the segments..

Function [File Save](#) saves your profile project.

Use the function [View Pass](#) to view the drawing of the profile in the [Drawing Area](#). The [Navigator](#) helps you to zoom, to move and to fit the drawing.

Transfer the drawing, which is displayed in the [Drawing Area](#), to your [CAD System](#) by using the function [Drawing -> CAD](#).

Use the function [Calculate Statics](#) to get a table of all static parameters of the cross sectional area. So you can examine, if your profile satisfies given static parameters. The function [View Statics](#) generates a drawing of the profile and the statics.

With the function [Abs. Angle](#) you can examine, if specific parts of your profile have the desired angle related to the horizontal line.

#### Hints:

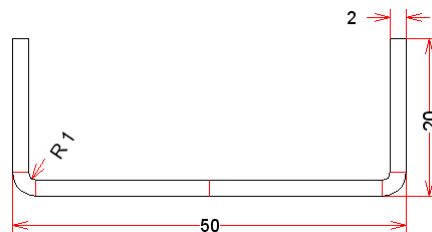
- After designing the cross-section of the final profile, your profile project contains a profile list with the name **L01**. This name is shown in the [Explorer](#).
- The next step is [Designing the Flower Pattern](#). By doing this, the profile lists **L02**, **L03**, etc. are created. The last profile list, e.g. **L16**, should contain the flat strip as it is decoiled from the coil.

## 2.4.2 Numerical Method

Use the numerical method to design a simple, rectangular profile or a tube. Because in this case you know all input data (or you easily can calculate them):

Enter the [Profile Elements](#) directly and manually into the [Profile List Window](#).

### Example 1

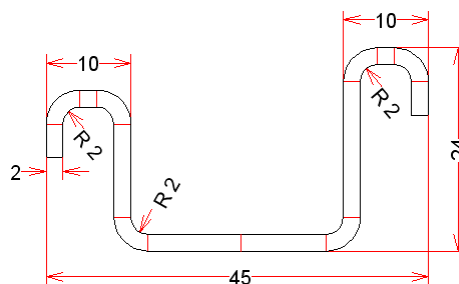


A profile U 50x20x2 (50 wide, 20 high, 2mm sheet thickness, 1mm inner radius) is described by the following input into the profile list:

Thickness = 2				
L				Length 22
A1	L	Radius 1	Angle 90	
L				Length 17
PS				

(For explanation:  $L = \text{LineLength} = 50/2 - 2 - 1 = 22$   
 A = Arc, Left, 1mm inner radius, 90 degree  
 $L = \text{LineLength} = 20 - 2 - 1 = 17$   
 PS = Point symmetrical)

### Example 2



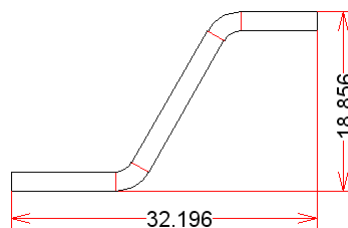
An unsymmetrical hat profile 45x24x2 (45 wide, 24 high, 2mm sheet thickness, 2mm inner radius) is described by the following input into the profile list:

Thickness = 2				
L				Length 10
A1	L	Radius 2	Angle 90	
L				Length 16

A1	R	Radius 2	Angle 90
L			Length 2
A1	R	Radius 2	Angle 90
L			Length 4
P			
L			Length 11
A1	R	Radius 2	Angle 90
L			Length 11
A1	L	Radius 2	Angle 90
L			Length 2
A1	L	Radius 2	Angle 90
L			Length 4

(For explanation: P = Point unsymmetrical, below it is the description of the left half)

### Example 3



In case of a profile with oblique angles (not 90°) you normally do not know the lengths of the line segments. Enter estimated values for x, y, and z:

Thickness = 2			
L			Length x
A1	L	Radius 2	Angle 60
L			Length y
A1	R	Radius 2	Angle 60
L			Length z
PS			

Afterwards dimension the profile as it is dimensioned in the specification drawing. Now modify the lengths of the line segments by using the [Toolbox Modify](#) or by pressing the **Pg Up/Dn** keys of the keyboard until the shown dimensions fit to the specification drawing. If necessary, set a small **Step Interval Length** in [Options Mouse](#) in order to reach the desired value exactly.

After the input of each profile element the drawing of the profile is displayed in the [Drawing Area](#) for examination.

You can correct your input by using the functions [Element Insert](#), [Element Append](#), [Element Remove](#) and [Profile Empty](#).

### Hints:

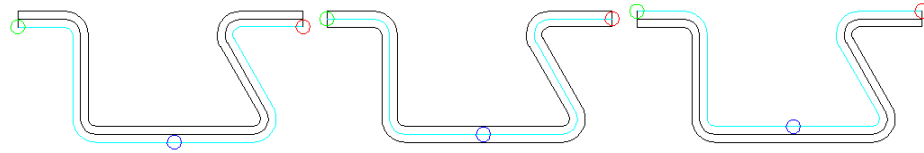
- If you have made a mistake in entering an arc, you must not modify the angle or the radius of the arc. By doing so you would bend up the arc with constant straight length. For correction toggle to the [Draft Modus](#), which enables to modify angle and radius without bending the arc.
- By using the numerical method, it is not so easy to enter profiles that have not 90° angles (see example 3). Better use the [Toolbox Profile Design](#) or the [Graphical Method](#).

### 2.4.3 Graphical Method

Use the graphical method for data input of more complicated profiles, if you do not know all data and you cannot simply calculate them:

#### Preparing the profile contour inside the CAD system

First step is to draw the contour of the desired profile inside your CAD system.



Optionally, drawing the profile bottom side, the sheet center line, or the profile top side (bottom and top related to the reference point) is possible.. If you want to design a symmetrical profile, you only need to draw one half. If you design a symmetrical profile, take care that the web line is split in two halves, so that the split point defines the profile reference point.

#### Reading the contour

Use the function [Profile, Read CAD-Contour](#) to read the contour file and to generate a [Profile List](#). In case of a symmetrical profile append a profile element PS; in case of an asymmetrical profile enter P instead and repeat contour import for the second half of the profile as described above. Before reading the contour file, activate the next row behind P. If you use the [ActiveX-interface](#), both halves and the point P are created at the same time.

#### Example

You create the following drawing inside your CAD system:



After reading the contour into **PROFIL** and after appending a profile element PS at the end of the profile list, you get the following profile:



### 2.4.4 Searching for similar profiles

If a similar profile is manufactured already in the past, the designer wants to base upon the experience. To find quickly the older profile project, the profile catalogue is helpful.

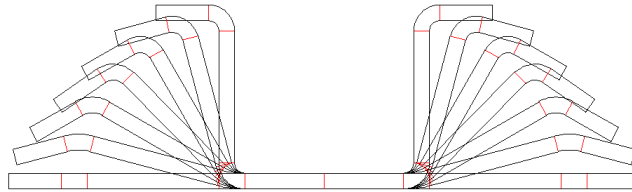
To search for a similar project, proceed as follows: Open the [Profile Catalogue](#) and create a [Filter](#) with the desired filter conditions, e.g. outer dimensions or classification key. If a suitable filter already exists, you only need to select it from the drop-down-list. See [Filter](#) for more info.

After defining the filter click on the button **Filter On**. This causes the profile catalogue to show only the profiles that match the filter conditions.

Browse through the filtered profiles and select a suitable one. Click on **Open Profile Project**.

## 2.5 Flower Pattern

### 2.5.1 Designing the Profile Flower Pattern



For designing the flower pattern, the [Draft Modus](#) must be switched off.

First define the [Bending Method](#). You can change the bending method **A1** to **A2** or **A3** or **A4**, if you want to use a constant radius method. The bending method affects the following bending.

Afterwards you can define the bending steps, each for every stand. Use the function [Profile Append](#) to create the next profile list for the next stand (opposite the sheet running direction). Activate an arc element, either by clicking into the drawing or into the field **Angle** or **Radius** of the profile list. Now select among 3 possibilities to bend up the arc:

- enter a new angle or radius into the field **Angle** or **Radius** of the profile list;
- press the buttons **Pg Up/Dn** on your keyboard until the desired angle/radius is obtained.
- select **Angle** or **Radius** in the [Toolbox Modify](#), click on the button **smaller** or **10x smaller** until the desired angle/radius is obtained.

If you have selected the bending method **A2** or **A3** or **A4**, the angle can be modified only.

After proceeding this for all bending steps, use the function [View Flower Nested](#) to view the drawing of the flower pattern.

The function [View Flower Separated](#) gives to you a clearly laid out view of all bending steps.

The function [View Flower 3D](#) gives you a perspective view of the bending steps. So you are able to examine, if the course of the band edge is smoothly without detours.

Use the function [Calculate Stress of Edge](#) to get the bar diagram of the stresses between all stands, on the left and on the right side of your profile. Examine, if all stresses do not exceed the yield point (100% in the bar diagram). Otherwise correct the bending steps. The stress of edge can be examined while bending yet.

If the calculation of the stress of edge is not sufficient use the function [PSA - Profile Stress Analysis](#) to calculate the stress within the whole profile. This is important when the maximum stress is not at the band edge, e.g. when edges are folded and the folds are bent.

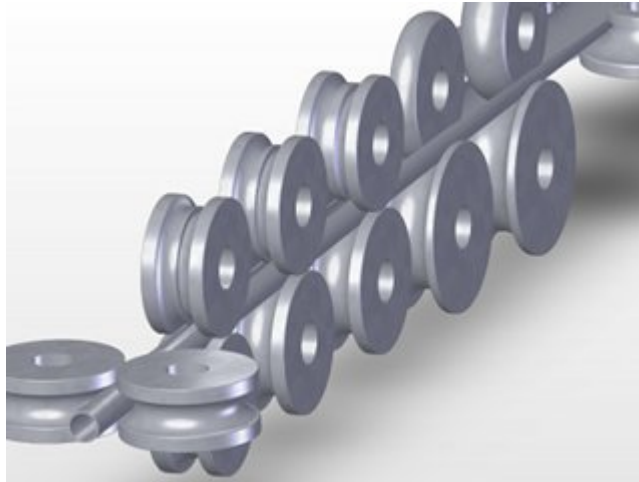
The function [File Print](#) prints out the drawing, the profile list, the table of statics and the bar diagram of the stress of band edge.

Use the function [Drawing -> CAD](#) to transfer the drawing, which is displayed in the [Drawing Area](#), to your [CAD System](#).

#### Hints:

- During designing the flower pattern, you get a set of profile lists **L01**, **L02**, **L03**, etc. The last profile list, e.g. **L16**, should contain the flat strip as it is decoiled from the coil. The names are shown in the [Explorer](#). The number sequence is in design direction, this means opposite the sheet running direction.
- The next step is [Designing the Roll Tools](#).


## 2.5.2 Designing the Tube Flower Pattern



By using the Toolbox Tube Design you quickly can design the flower pattern for seamed tubes and the roll tools without a CAD system.






### Calling the function

Call this function optionally by:




- Main menu: **Toolbox, Tube Design**.
-  Button **Toolbox Tube Design** in the [Button Key Bar](#).

### Content

**Section 1** contains functions for creating the different passes for tube forming and for tube calibrating:

-  [Shaped Tube Calibration](#)
-  [Welding Pass](#)
-  [Fin Pass](#)
-  [Break Down Pass](#)
-  [Break Down Pass, W-Forming](#)

**Section 2** contains functions for generating the rolls for tube forming:

-  [Fin Pass, Top Roll](#)
-  [Fin Pass, Bottom Roll](#)
-  [Break Down Pass, Top Roll](#)
-  [Break Down Pass, Bottom Roll](#)
-  [Fin Pass, Side Rolls](#)



### [Break Down Pass, Side Rolls](#)

#### Principle of operation

- **Preparing:** Create a new profile project by using [File New](#). Open the [Machine Window](#) and enter the machine data or import a [Machine File](#) that you exported from a previous project. If the welded tube should be formed to a shaped tube, the machine must contain calibrating stands.
- **Defining the tube dimensions (round tube):** Call the function [Welding Pass](#) of the Toolbox Tube Design and enter the diameter, the sheet thickness and the necessary addition for welding.
- **Defining the tube dimensions (shaped tube):** Use the [Toolbox Profile Design](#), the [Graphical Method](#), or the [Numerical Method](#) to define the cross-section of the shaped tube. Afterwards call the function [Shaped Tubes Calibration](#) of the Toolbox Tube Design. The cross section patterns for the calibrating stands and the welding stand are created automatically. By using the function [Welding Pass](#) you can attach the addition for welding.
- **Creating the passes:** For each stand create a pass by using the function [Profile, Append](#) one after another and call in each pass one of the functions [Fin Pass](#), [Break Down Pass](#) or [Break Down Pass, W-Forming](#), dependent on whether it is a fin pass or a break down pass. All these functions bend open the existing profile dependent on the entered parameters as desired.
- **Generating the roll tools:** For each pass call the functions [Fin Pass, Top Roll](#), [Fin Pass, Bottom Roll](#), [Break Down Pass, Top Roll](#), [Break Down Pass, Bottom Roll](#), [Fin Pass, Side Rolls](#) or [Break Down Pass, Side Rolls](#), dependent on whether it is a fin pass or a break down pass or whether you want to create top, bottom or side rolls. The rolls for the calibrating stands can be created by using the function [Roll, Scan Profile Drawing](#).

#### Properties

Except of the function **Welding Pass** (which is called in a new project with an empty profile list) all other functions use the pass in which they are called, i.e. the functions **Fin Pass** and **Break Down Pass** bend the tube profile dependent on the entered parameters (This is why you should have called the function [Profile, Append](#) previously). The functions **Top Roll**, **Bottom Roll** and **Side Rolls** generate rolls for the current pass.

All functions of the Toolbox Tube Design (except **Shaped Tube Calibration**) are designed for symmetrical tubes with two arc segments on each side, i.e. the profile list belonging to it **must** look like this:

A1  
A1  
PS

With other kinds of profile lists the Toolbox Tube Design will not work. Modifications by hand or by the [Toolbox Modify](#) are allowed, however.

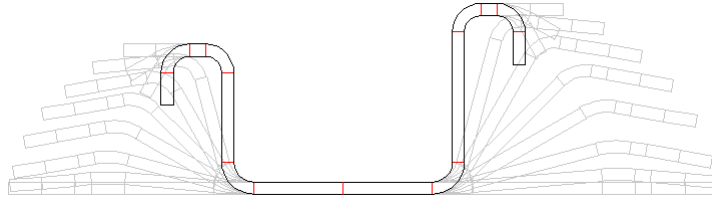
- The function **Shaped Tube Calibration** needs a closed cross section pattern, which can be symmetrical or unsymmetrical with any count and type of profile elements.

Before using the Toolbox Tube Design it is recommended to prepare the machine data in the [Machine Window](#).

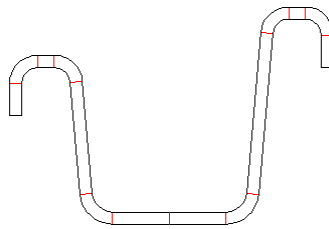


### 2.5.3 Automatic Flower Creation

If the flower design of a similar profile already exists and a new profile should be developed by the same scheme, creating the flower pattern is quite simple.



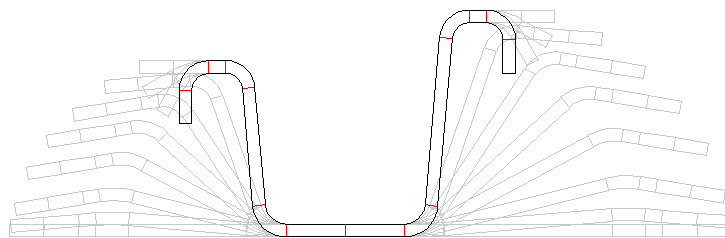
Imagine, an old profile project exists, and you know that the profile was roll formed on the machine successfully.



Now you get the task to design a flower pattern for a new similar profile which has slightly differing dimensions or angles. You surely would look into the old project and use the same forming concept, if it was successful in the past.

%	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
1	100.000														
2		100.000													
3			100.000												
4				100.000											
5					100.000										
6						100.000									
7							100.000								
8								100.000							
9									100.000						
10										100.000					
11											100.000				

This can be done easier: Call from the old project the function [Development Table](#) and select **Development Table, Create from current project**. Save the development table to a file after converting the angles to percent (function **Development Table, View Angle in %**). This is recommended in order to apply the development table to new project with various bending angles.



Now open the **new project** with the final profile cross-section (**L01**). Open the development table of the **old project** and select **Development Table, Apply and Create Flower Pattern**. The profile flower for the new project is created automatically.

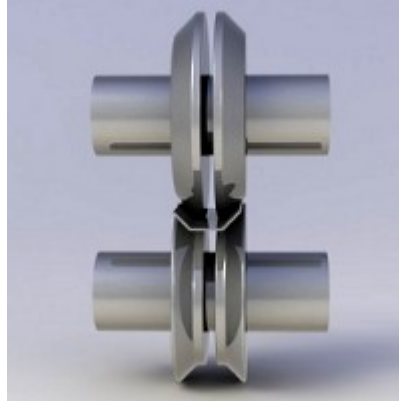
Similar means: the profile list has the same profile element sequence regarding line and arc types. If it is not, you can move certain columns (function **Development Table, Column**) or you can create and edit a new development table manually.

Toggle the development table to [Bending Methods](#) (function **Development Table, Bending Method**) and define bending methods that should deviate from the method in the target profile list. It is also possible to convert a line element (from L01) to an arc element (in Lnn).

By using this feature, in the course of time you will get a collection of development tables for various kinds of profiles, which contain your individual know-how. This enables you to quicker react on inquiries for new profiles.

## 2.6 Roll Tools

### 2.6.1 Designing the Roll Tools



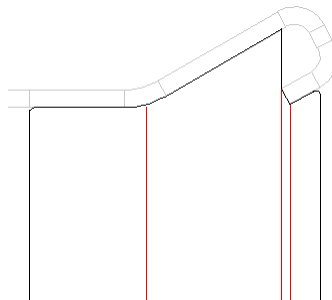
Select among 3 various methods, automatically, with the help of the CAD-system or by re-using of existing rolls from the roll stock management:

Precondition for both methods is: you have prepared a machine data in the [Machine Window](#).

- Choose the [Roll Design using the Profile Drawing](#), if PROFIL should suggest a roll for your profile pattern automatically. It is the fastest method for designing rolls.
- Choose the [Roll Design using the CAD System](#) if you want to give the rolls an individual form.
- Choose the [Search for existing Rolls](#), if you want to re-use existing rolls from the roll stock management for reducing costs.

### 2.6.2 Roll Design using the Profile Drawing

Choose this method, if PROFIL should suggest a roll for your profile pattern automatically. It is the fastest method for designing rolls.

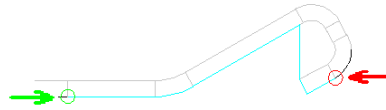


The profile drawing displayed in the [Drawing Area](#) is scanned. Only the contour of the profile and other yet existing rolls is considered that is visible from the shaft. A roll is created that touches the complete visible contour.

First open the [Machine Window](#) and enter the machine data.

Select the desired profile pass and define by using the function [Profile, Loaded](#), whether you want to design the rolls for the discharged state or the loaded state. This is useful to compensate the spring back.

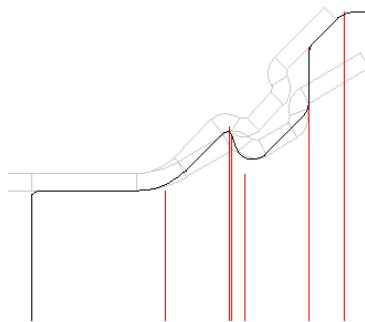
Select the drawing of the rolls by using function [View Rolls](#). Now the center lines of the shafts will be displayed. Select one of the center lines for defining the type of roll. Afterwards call function [Roll Scan Profile Drawing](#).



Now a roll is generated and displayed that touches the complete visible contour of the profile and also existing rolls on other shafts.

### 2.6.3 Roll Design using the CAD System

Choose this method especially if you want to give the rolls an individual form.



#### Preparing the roll contour inside the CAD system

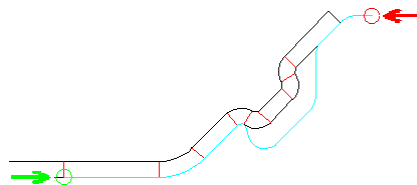
First draw the desired roll contour inside your CAD system. It is very useful, if you transfer the drawing of the profile pass to the CAD system (function [Drawing -> CAD](#)) if you use the contour of the pass for your drawing.

If you plan to design split rolls, also create a contour line for all rolls of a shaft together. You can proceed the splitting later. Do not draw the roll edges (the first and the last radial line of a roll) and the center line; these will be created automatically. Do not draw a fillet at the first and last corner; they can simply be inserted later.

#### Reading the roll contour

First open the [Machine Window](#) and enter the machine data.

Select the desired profile pass and select the drawing of the rolls by using the function [View Rolls](#). Now the center lines of the shafts will be displayed. Select one of the center lines for defining the type of roll. Afterwards call function [Roll Read CAD-Contour](#).



Now a roll can be created, which has the prepared CAD contour.

### 2.6.4 Searching for existing Rolls

Only with option Roll Stock Management.

To search for suited rolls in the roll database, select among these methods to create a filter:

- First define the contour of the desired roll by using the method [Roll Design using the Profile Drawing](#) or [Roll Design using the CAD-System](#). Function [Roll Cut](#) moves the roll to the clipboard. Now call the [Roll Stock Management](#) and create a [Filter](#) by pressing the button **Create Filter from Clipboard**. If needed, modify the filter conditions manually.
- The second method is to define the filter conditions manually. If a suited filter yet exists, it only needs to be loaded. Get more information in chapter [Filter](#).

If the filter is defined, press the button **Filter On**. Now only those rolls are visible that match to the filter conditions. If no roll is displayed, no suited roll exists or the allowance is too small.

Now browse through the rolls and select a suited one. Press the button **Copy Roll to Clipboard** and change to the main window. Select the shaft and insert the roll from the clipboard by using the function [Roll Insert](#). Move the roll to the correct position (function [Roll Move](#)). If you observe, however, that the roll is not suited, delete it again (function [Roll Delete](#)) and search for a better one.

### 2.6.5 Adapting the Roll Tools

If necessary, you can modify the roll by using the functions of the pull down menu **Roll**:

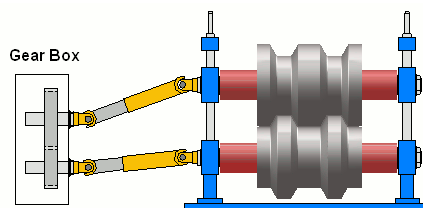
<a href="#">Conical Extension</a>	<a href="#">Cylindrical Extension</a>
<a href="#">Arched Extension</a>	<a href="#">Split at Corner</a>
<a href="#">Split between Corners</a>	<a href="#">Join</a>
<a href="#">Turn</a>	<a href="#">Move</a>
<a href="#">Mirror</a>	
<a href="#">Cut</a>	<a href="#">Copy</a>
<a href="#">Paste</a>	<a href="#">Delete</a>

To modify single roll corner points, use the [Toolbox Modify](#). After selecting the desired roll corner point, select **Width**, **Diameter** or **Radius** in the toolbox and click on the arrow buttons for stepping the value up or down.

A [Clearance Angle](#) is used to reduce wear out of rolls, if the rolls have different circumference speeds because of the different heights of the legs of the profile. By using the function [Gap](#) the clearance angle can be continued in the successive profile segments.

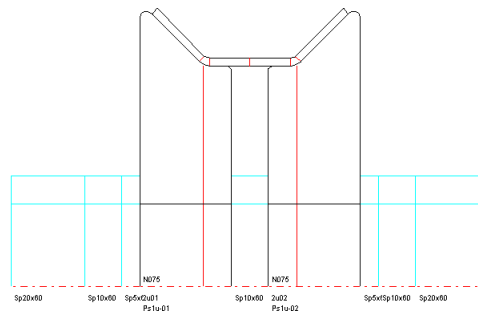
Check by using the function [View, Show, Previous Pass](#) if the profile is treated correctly by the rolls and if no conflict occurs.

### 2.6.6 Creating Spacer Rolls



Spacers (red, see picture) are bushings that are mounted on the bottom or top shaft in order to fix the horizontal position of the forming rolls. They have a smaller diameter than the forming rolls and do not touch the profile. The width of the spacers results from the leftover distance between the forming rolls and the machine stand (defined by the [Working Width](#) in the [Machine Window](#)). So

after designing the forming rolls the size of the spacers is defined. The diameter of the spacers is constant (defined by the [Spacers Ø](#) in the [Machine Window](#)).



Spacers also can be inserted between forming rolls. In this way roll tools can be used for varying profile widths only by replacing the spacers against others with varying widths.

In order to avoid to manufacture the spacers for each project newly, this method often is used: The forming rolls are furnished with extensions ([Cylindrical](#), [Conical](#), or [Arched](#)) with the option **To Width (Absolute)** and a smooth value is entered in the raster 1mm, 5mm or 10mm. This will lead to smooth spacers widths. In addition, the spacers are split to shims, e.g. in the raster 100mm, 50mm, 10mm, 5mm, 1mm. So the spacers can be combined from an existing spacer stock.

**PROFIL** provides two alternatives to insert spacers:

- **Automatic Spacers:** By checking the box **Autom. Spacers** in [Options, Drawing, Rolls](#) the empty space left and right of the forming rolls are filled with spacers automatically. The automatic spacers are transferred to CAD, and they appear in the parts list. However, because they are not objects like the forming rolls in the PROFIL object hierarchy, they cannot be selected by mouse-click and they cannot be dimensioned and split. They also do not get a designation like the forming rolls. In case a spacer is needed between forming rolls, a cylindrical roll with small spacer diameter must be inserted.
- **Spacer Rolls:** These kind of spacers are handled like all other objects in the PROFIL object hierarchy. They are created with the function [Rolls, Spacer Rolls, Create](#) and are removed by [Rolls, Spacer Rolls, Remove](#) and they can be selected by mouse-click, can be dimensioned and split, and they get roll numbers and part numbers dependant on special number keys for spacer rolls only in [Options, Spacer Rolls](#). The width adaptation is not automatic, if the width of the forming rolls is changed, the spacer rolls have to be re-created. If **Split Spacers to Shims** is checked in [Options, Spacer Rolls](#), the spacer rolls are split automatically dependant on the **Shim Width Table**. Spacer rolls also are created between forming rolls if a gap is left between them during roll design. Spacer rolls appear in the parts list and in the NC program.

For both kinds of spacers is effective: Because spacers do not contact the profile, they are not considered during the [Finite Element Simulation](#). Both kinds of spacers can be switched on and off in order e.g. to transfer the drawing to CAD or to print the drawing without spacers..

## 2.6.7 Output of the Manufacturing Data

### Profile and Roll Tool Drawings, Assembly Drawings

Use the [Toolbox Dimensioning](#) to dimension the profile and roll tool drawings either manually or automatically. Use the function [Output Drawing -> CAD](#) to transfer the desired drawings to the [CAD-System](#). After completion of the drawings use the CAD system functions to plot out or print out the drawing. Or use the function [File Print](#) to print the drawings directly. By using the function [File Plot](#) you get a standardized assembly drawing with frame and title block.

### Parts List (Sawing List)

Use the function [Output Create Parts List](#) to create a text file, which contains the parts list of all rolls of the project. You can print the text file or transfer it to a spread sheet. Or use the same function to transfer the parts list directly into a predefined work sheet of MS Excel.

### NC-Programs

Use the function [Output Create NC](#) to create a text file, which contains the NC-programs (Contour data) of all rolls of the project. You can split the file into separate files for each roll.

### NC-Drawing

If you use a NC programming system and it needs DXF-files, use the function [Output Drawing -> NC](#) to create these files.

### Programming the CNC-Lathe manually

If the machine control does not support importing external created programs, use for the manual programming the roll data from the [Roll Tool Window](#), printed with function [File Print Preview](#) and [File Print](#). The data of the [Roll Corner Points](#) are suited for entering directly into the machine control.

## 2.6.8 Saving the Rolls into the Roll Data Base

Only with option Roll Stock Management.

### Saving a single roll by using the clipboard

For saving a single roll into the roll database by using the clipboard, select the desired roll by a click in the [Drawing Area](#) and copy it by function [Roll Copy](#) to the clipboard. After you have opened the [Roll Stock Management](#), press the button **Insert Roll from Clipboard** of the [Roll Table](#).

### Saving one or more rolls from the project

Call the function [Roll Stock Management, Save](#) and select, if you want to save the previously marked roll, all rolls of the marked shaft, all rolls of the current stand or all rolls of the whole project.

### Entering a roll from a paper drawing into the database

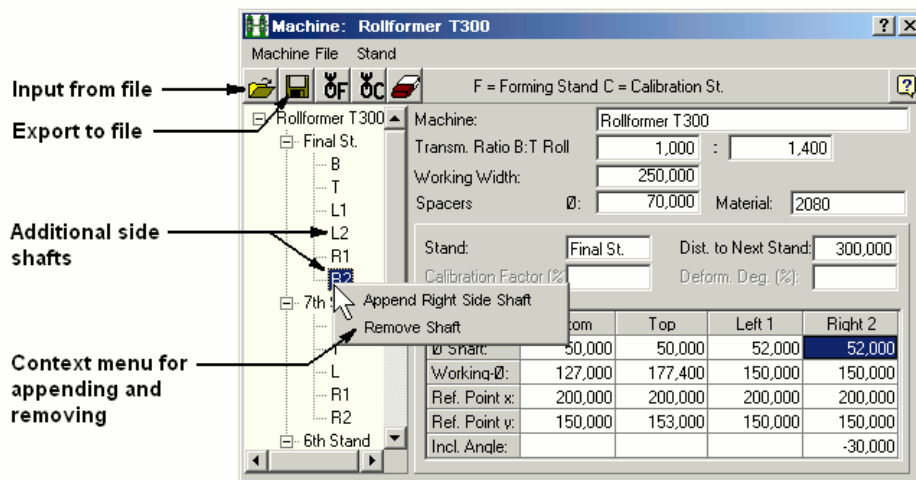
Create a new record in the [Roll Table](#) of the [Roll Stock Management](#) by using the button **Insert Roll** of the database navigator and enter the roll data into the record. Use the button **Post edit** to save the record. While the new roll is still active, for every corner of the roll create a new record in the [Roll Corner Table](#) of the [Roll Stock Management](#) by using the button **Insert Corner** of the database navigator and enter the [Roll Corner Data](#) into each record: **Width**, **Diameter** and **Radius**. The width is related to the left edge of the roll unlike to the rolls in the profile project, i.e. the width of the first corner is always 0. The field **Angle** is not an input field, the angle will be calculated automatically. Save every record by using the button **Post edit**.

### Saving a roll from a CAD-drawing

Press the button **Insert Roll from CAD** in the top bar of the [Roll Table](#).

## 2.7 Machine

### 2.7.1 New Handling Of Machine Data



#### Old handling, until PROFIL Rel. 4.6:

Machine data are saved in the [Machine File](#) \*.m01. The profile project contains a link to the machine file only. When additional machine data are needed (e.g. during designing new rolls), they are auto loaded from the machine file. If machine data are modified in the project (e.g. from the context menu by right click on the shaft centerline), the data are saved into the machine file. When machine data are modified in the [Machine Data Window](#), the project is updated by saving the machine data into the machine file.

#### Why changing this handling method?

The old handling method has the advantage that the machine data could be used for various profile projects for the same machine universally. The bad thing about it is that modifying machine data may unnoticed impact other projects unintentionally. This is why many users save the machine file in the project path. But this does not meet the idea of the universal usage of the machine file.

#### New handling, since PROFIL Rel. 4.7:

The machine data are handled in the profile project completely, and are saved into the [Project File](#) \*.pro. They are shown in the [Machine Data Window](#) that is nearly unchanged against previous releases except in two items: the load file and save file functions are replaced by import and export file functions for transferring machine data via \*.m01 files to other projects. The machine file is not saved anymore automatically, but only by calling the function consciously. In case of missing machine data (e.g. not enough stands), PROFIL shows a message in the bottom dialog line and the machine window is opened and the user is requested e.g. to append an additional stand. This also occurs if a new project is created and it still has not machine data. Then the user can enter machine data first or import an existing machine file. When machine data are modified in the machine data window, the modifications are shown in the project immediately. It is not necessary to save the machine data or to close the machine data window. Thereby interactive modifications are possible (similar to the profile list window) and the result is shown in the drawing area at once.

#### Compatibility:

Old and new [Project Files](#) are fully compatible. However, if an old project file is opened with the new PROFIL release, the machine window is filled out partly only with machine data that could be extracted from the project. Then the appropriate machine file should be imported once and the project has to be saved with the machine data. Afterwards the machine file in principle is not needed anymore for this project. The old PROFIL release also is able to open a new project file with machine data. However, the machine data from the file are ignored.

**Working Instructions:**

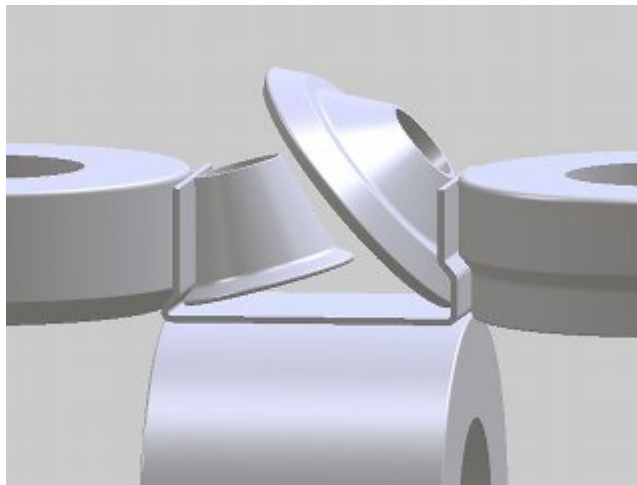
The new button „Machine“ in the top button bar opens and closes the machine data window. This window shows the machine data of the currently opened project. Entered values take effect on the project immediately. After closing the machine data window the machine data keep alive. When an older project has been opened that does not contain machine data, the machine window possibly shows not all stands of the machine, but only the stands that are used in the project. In this case the appropriate machine file should be imported first. Both the import and the export function open the file selection window and show the machine file name that belongs to the project as default file name. Further [Import](#) and [Export](#) functions for machine files are in the file menu of the PROFIL main window. However, the default file name here is the recently used file name for these functions.

The stand that is shown in the machine data window is linked up both with the stand that is selected in the explorer and that is shown in the drawing area in the main window. So the stand can be selected in any of these three items; the other two are updated. This takes effect also for the selected shaft. Because the machine data are handled within the project, also the undo and redo functions are available.

**Interactive operation:**

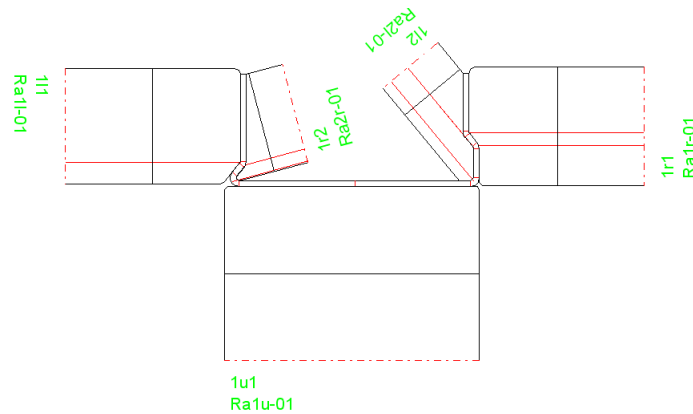
The machine window works interactively with the drawing area (similar to the profile list and roll tool window). This means the effect of data modifications are shown in the drawing immediately. Vice versa, if the user selects or modifies drawing entities, the content of the data windows is updated.

## 2.7.2 Multi Axles

**Additional Side Axles:**

In order to form more complex roll formed profiles on a roll forming machine, it often is necessary to have additional side axles with inclined angles that plunge into the open cross-section. This makes it easier to form the inner radii precisely, which are not accessible for horizontal top rolls. For this purpose additional side axles with arbitrary inclination angle can be created in the machine window.





### Operating Instructions:

After right clicking on the side axle symbol (L or R) in the machine window a context menu opens and additional side axles can be defined – and removed again in the same way. The axle can be vertical ( $0^\circ$  = normal position) or inclined with any inclination angle (positive outwards and negative inwards). Any number of additional axles can be defined. During creation of a new axle the data of the previous axle are copied. The inclination angle, however, is modified in order to have the new axle visible in the drawing area and selectable by mouse click. Afterwards the user should set the desired inclination angle.

### Roll Numbering:

In order to enable the automatic numbering of the roll number and the part number, two new variables are provided. They can be used in the number keys input fields (Options, Rolls).

**\$SA** consecutive number over all rolls of a certain stand in the order bottom-top-left-right

**\$TA** consecutive number over all rolls of a certain shaft/axle type (B, T, L, R) of a certain stand.

All left side rolls belong to the type L and all right to the type R.



## 3 Reference

### 3.1 Menu Items


#### 3.1.1 File

##### 3.1.1.1 New Project

Use this function to create a new and empty [Profile Project](#).

##### Calling the function

Call this function optionally by:

- Main menu: **File, New Profile Project**.
-  Button **New Project** in the [Button Key Bar](#).

The window **New Profile Project** appears. Profile projects have the file extension **.pro**. In the top area of the window you can change the path or drive, if needed. All existing projects of the selected path are displayed. Be sure that you enter a file name that does not exist.

##### Principle of operation

After entering a valid file name, the [Project Data Window](#) for the new profile project and the empty [Profile List Window](#) for the first bending step (finishing step) appear. Some input fields already contain default values. These values come from your last profile project. Of course you can change them.

##### Hint:


If the profile project already exists, use the function [File Open Project](#).

##### 3.1.1.2 Open Project

Use this function to open an existing [Profile Project](#).

##### Calling the function

Call this function optionally by:

- Main menu: **File, Open Profile Project**.
-  Button **Open Project** in the [Button Key Bar](#).

The window **Open Profile Project** appears and you can select the desired project file by a double click. Profile projects have the file extension **.pro**. In the top area of the window you can change the path or drive, if needed.

**Project Preview:** On the right, you see a quick preview of the profile and beneath it [Customer](#), [Description](#) and [Drawing No.](#) and on the right the count of passes.

##### Principle of operation

After selection, the [Project Data Window](#) for the selected profile project and the [Profile List Window](#) for the first bending step (finishing step) appear. The [Drawing Area](#) displays a preview of the drawing of the profile.

You get a message if you try to open a project that already is opened by another user. Then the project is opened "on approval only" (shown in the top line) and you cannot save the file. In order to edit the file reopen it after the other user has saved it. It is possible to use [Save As...](#) to save the project under a different file name.

##### Setup



While working with larger projects the screen refresh takes more time because of the refresh of

the preview. This may impede quick working. Simply switch off the preview in [Options, General](#).

**Hint:**


See also: [Partial Project Add On](#).

### 3.1.1.3 Save Project

Use this function to save the [Profile Project](#) after designing or after modification. The project is saved into the project file that is displayed in the top bar of the main window.

**Calling the function**

Call this function optionally by:

- Main menu: **File, Save Profile Project**.
-  Button **Save Project** in the [Button Key Bar](#).
- **Function key F2**. The function key assignment can be modified in [Options, Keyboard, Shortcut Key Assignment](#).

**Principle of operation**

The [Profile Project](#) is saved to the project file with the extension **.pro**. Before saving, the old **.pro** file is renamed in a backup file **.BAK**.

**Hints:**

- The function is active only, if changes have been done.
- If you forgot to save the data, PROFIL will remind you to save before exit.
- Before saving, the old **.pro** file is renamed in a backup file **.BAK** automatically. In case an error occurs during saving or you saved the file erroneously after unwanted modifications, you can restore the old content by deleting the new **.pro** file and by renaming the **.BAK** file to **.pro**.

### 3.1.1.4 Save as..

Use this function to save the [Profile Project](#) into another file.

**Calling the function**

Call this function by:

- Main menu: **File, Save as..**

The window **Profile Project Save as..** appears. Profile projects have the file extension **.pro**. In the top area of the window you can change the path or drive, if needed. After entering or selecting a file name, the project will be saved.

**Principle of operation**

The [Profile Project](#) is saved to the project file with the new name and the extension **.pro**. Furthermore the name of the current profile project is changed. The new name is displayed in the head line of the main window.

**Hint:**

See also: [Partial Project Save as..](#)

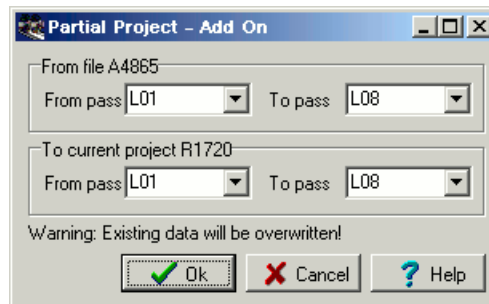
### 3.1.1.5 Partial Project Add On

A project file (**.pro**) can contain not only a complete [Profile Project](#), but also a **Partial Project**, e.g. the left or right side only, the pass only, or a subset of the stands only. By using the function **Partial Project Add On** you can add such a partial project to your currently opened project. During this operation it is essential that the objects of the added project overwrite the objects of the current project. So it is a good idea to save the project previously. The [Machine Data](#) are not added, so they have to be transferred separately.

### Calling the function

Call this function by:

- Main menu: **File, Partial Project Add On.**



After calling the function **Partial Project Add On** and after selecting the file that has to be added, the dialog window opens and the source and target passes can be defined:

**From file, From pass, To pass:** The selection boxes show the passes of the selected project file. If needed, you can select a subset.

**To current project, From pass, To pass:** Select the target position in the current project where the added passes should appear. Please consider that the new passes are not inserted or appended, however they overwrite the existing passes instead. If the new passes e.g. only contain the left side of the profile, the right side of the current project will remain unchanged. However, if you want to insert instead, please create the desired count of passes by using the function [Profile, Insert](#) or [Profile, Append](#) and overwrite these passes with **Partial Project Add On**.

### Principle of operation

The partial project is added from the selected project file by using the settings to the current project. These cases are handled specially:

- **Different sheet thickness in the current and the added project:** A dialog window appears with the question: "**Target project's sheet thickness differs from the add-on project. Adapt the target project?**" The user should decide if it makes sense to agree.
- **Rolls already exist:** For each roll a dialog window is opened with the question: "**Roll already exists. Overwrite?**" The user decides if he wants so. It is also possible to remove surplus rolls later.

These cases are not handled by PROFIL:

- **Different strip widths in certain passes,** especially if a subset of passes is added. The user should check this and correct it if indicated by modifying certain profile elements.
- **Penetrations between rolls passes.** Also this should be checked and corrected by the user.
- **Different machine data:** This causes that rolls of the added project do not touch the profile or penetrate the profile. The reason is that machine data are not added from the source to the target project. Please pay attention that the machine data fit together.

### Hints:

- In order to save a partial project to a project file, use the function [Partial Project Save as..](#).
- The partial project file has the same file format as the project file, so you can open, edit, and save it by using the functions [Open Project](#) and [Save Project](#).
- If the added project contains rolls, they will be locked against automatic renumbering. By using the function [Partial Project Add On](#) the rolls can be unlocked.

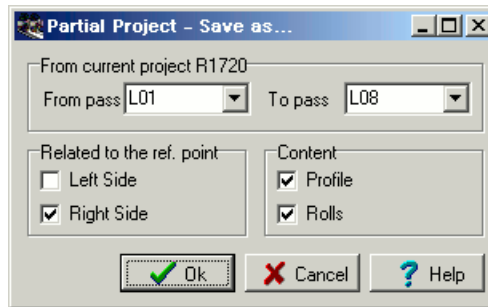
### 3.1.1.6 Partial Project Save as..

This function is needed if a certain part of a project, e.g. the left or right side only, the passes only without rolls, or a subset of the passes should be saved to a file. This can be used for combining a new project from already existing modules by the function [Partial Project Add On](#).

#### Calling the function

Call this function by:

- Main menu: **File, Partial Project Save as..**



After calling the function the dialog window is shown with these settings:

**From current project, From pass, To pass:** Select, which passes of the current project should be saved. The selection boxes first show all passes.

**Related to the ref. point, Left Side, Right Side:** Select, if the left side only, the right side only (related to the reference point), or both sides should be saved. The terms **left** and **right** are effective if the start direction of the profile is "to the right" (-90 .. +90 degree) and the rolls are defined from negative coordinates to positive coordinates (related to the reference point). Otherwise **left** and **right** are mixed.

**Content, Profile, Rolls:** Select, if the partial project should contain the profile only, the rolls only, or both.

#### Principle of operation

After pressing the Ok button, the file selection box appears for selecting the path and the file name. The partial project is saved to the project file with the new name and the extension **.pro**.

#### Hints:

- In order to add the partial project to the current project, use the function [Partial Project Add On](#).
- Because the file format of the partial project file is identical to the project file, you can open, edit, and save it by using the functions [Open Project](#) and [Save Project](#).

### 3.1.1.7 Import

This menu item contains all import functions that can be used for data transfer from other software systems. These functions also can be called from other menu items with fixed file names and input paths (pre-set in the options window). Therefore they can be used as temporary files for data transfer only. Unlike this, the import function opens a file selection window and the **file name**, **input path**, and **input format** can be set individually. So the import function is useful for importing files with frequently alternating file names.

#### Calling the function

Call this function by:

- Main menu: **File, Import**

The file selection window is opened and the **file name**, **input path**, and **input format** can be set individually. The settings are remembered when the function is recalled. So it is quite easy to import series of different files.

#### Principle of operation

Which **file formats** can be opened is dependent on the used software version. These formats are

available:

- **KTR Files PROFIL (\*.KTR):** File format defined by **UBECO**, see [Contour File \(KTR Format\)](#). Use this format if a macro in your CAD system proceeds the contour tracking and saves the drawing elements in sorted order. See also DXF Files.
- **DXF Files AutoCAD (\*.DXF):** File format defined by Autodesk. Nearly all [CAD systems](#) are able to create files in this format. Dependent on the current design stage (if [View, Pass](#) or [View, Roll Tools](#) is selected), the function [Profile, Read CAD Contour](#) or [Rolls, Read CAD Contour](#) or [Roll, Read CAD Roll](#) is called. During Import the temporal input file (pre-set in [Options, Files](#)) is not used.
- **Profile Lists PROFIL/DOS (\*.Lnn):** Use this function to import a set of [Profile Lists](#) and a set of roll tool files in order to create a new [Profile Project](#). In previous MS-DOS releases of PROFIL the data were not saved in a project file (extension **.pro**), but every profile list was saved in a profile list file (extension **.L01**, **.L02**, . .) and the roll tools of each stand were saved in a roll tool file (extension **.R01**, **.R02**, . .). You need this function, if you want to convert profile lists and roll files from previous releases to project files, for example if you obtained an update of PROFIL or if you want to exchange data to and from a subcontractor/customer who is using such a release. After importing a profile project is created with the name of the imported profile lists. If you save the project afterwards by using the function [File Save Project](#), it will be saved into the path of the profile lists.
- **Machine Files (\*.m01):** When you start with a new Project that does not contain [Machine Data](#), you can import machine data from a machine file \*.m01, which you created with the function [Export](#) from another project previously. Thus you can transfer machine data created in a previous project to a new project, in case the roll tools should be designed for the same machine.
- **FEA Result Files ABAQUS/Explicit (\*.FIL):** Use this function to display the result of the simulation of the roll forming process by [FEA - Finite Element Method](#) in the [Drawing Area](#). See also [View, FEA Result](#).

#### Hints:

- In order to import a partial project file (\*.pro), use the function [File, Partial Project Add On](#).
- In order to open a PROFIL project (\*.pro) use the function [File, Open Project](#).

### 3.1.1.8 Export

This menu item contains all export functions that can be used for data transfer to other software systems. These functions also can be called from other menu items with fixed file names and output paths (pre-set in the options window). Therefore they are overwritten each the function is called (temporary files for data transfer only). Unlike this, the export function opens a file selection window and the **file name**, **output path**, and **output format** can be set individually. So the export function is useful for exporting files with frequently alternating file names.

#### Calling the function

Call this function by:

- Main menu: **File, Export**

The file selection window and the **file name**, **output path**, and **output format** can be set individually. The settings are remembered when the function is recalled. So it is quite easy to create series of different files.

#### Principle of operation

Which **file formats** can be saved is dependent on the used software version. These formats are available:

- **DXF Files AutoCAD (\*.DXF):** File format defined by Autodesk. Nearly all CAD systems are able to read files in this format. The whole drawing which is shown in the drawing area or the selected drawing object is saved in the file. The function [Drawing -> CAD](#) does the same; the name and the path of the temporal output file, however, are taken from [Options Files](#).
- **IGES Files (\*.IGS, \*.IGES):** File format defined by the Initial Graphics Exchange Specification. See also: DXF Files.
- **MI Files CoCreate ME10 (\*.MI):** File format defined by Hewlett Packard (CoCreate, PTC) for

the CAD system ME10 (OneSpace Designer Drafting or Creo Elements/Direct Drafting respectively). See also: DXF Files.

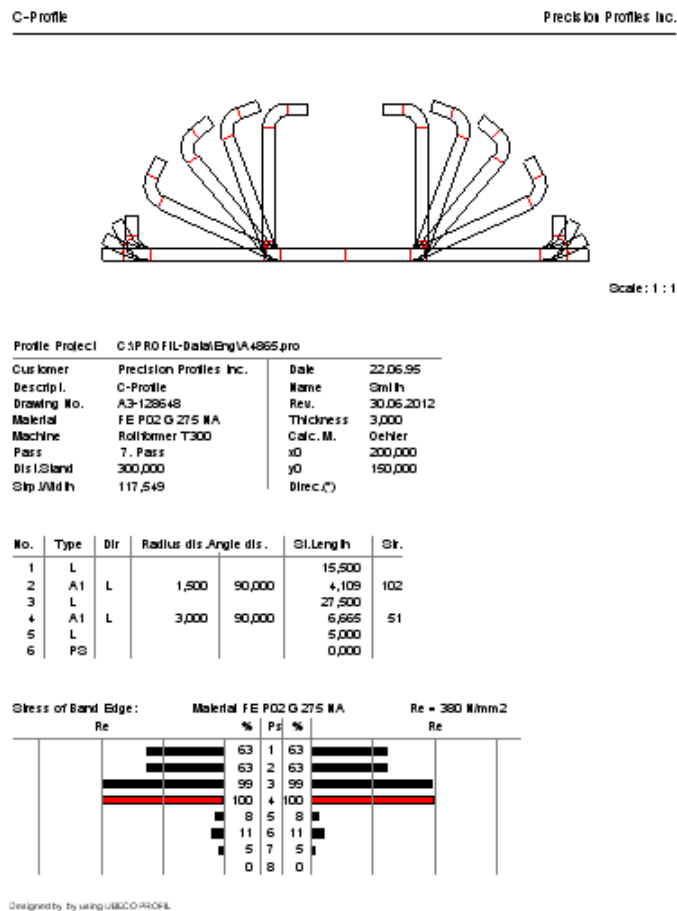
- **A11 Files PC-DRAFT (\*.A11):** File format defined by ISD for the CAD system PC-DRAFT (in the mean time discontinued). See also: DXF Files.
- **STEP AP214:** Creates the profile pass, the roll tool stand, or all stands as 3D models in STEP format in accordance with DIN ISO 10303 "Product data representation and exchange, EXPRESS language". The single stand can contain a profile pas with user defined length. See also [Options, Files](#), **Profile built from separate bodies**.
- **Profile Lists PROFIL/DOS (\*.Lnn):** Converts the current [Profile Project](#) to a set of [Profile Lists](#) with the same name. In previous MS-DOS-releases of PROFIL the data were not saved in a project file (extension **.pro**), but every profile list was saved in a profile list file (extension **.L01**, **.L02**, ...) and the roll tools of every stand were saved in a roll file (extension **.R01**, **.R02**, ...). You need this function, if you want exchange data to and from a subcontractor/customer who is using such a release. Exporting of roll data files (**.R01**, ...) is not implemented.
- **Machine Files (\*.m01):** By using this function you can transfer [Machine Data](#) created in a previous project to a new project, in case the roll tools should be designed for the same machine.
- **Parts List Files, Sawing List Files (\*.txt):** Generates a parts list (sawing list) of the rolls of your [Profile Project](#). The parts list contains all rolls of all stands of the roll forming machine. See [Options, Parts](#) List for the set-up of the parts list. The function [Output, Part List](#) does the same; the name and the path of the output file, however, are taken from the name of the current project.
- **NC Program Files, G-Codes (\*.G00):** Generates the NC programs for all rolls of your [Profile Project](#). See [Options, NC](#) for the set-up of the NC output. The function [Output, NC](#) does the same; the name and the path of the output file, however, are taken from the name of the current project.
- **Windows Bitmap Files (\*.bmp):** Copies the current drawing (shown in the [Drawing Area](#)) into the Windows clipboard and saves it to a bmp file. See [Options, General](#) for the set-up of resolution and background color. See also [Edit, Copy](#).
- **FEA Input Files ABAQUS/Explicit (\*.inp):** Creates the simulation model for the [FEA-Simulation](#), which can be loaded from the FEA system ABAQUS/Explicit. See also [Output, FEA, ABAQUS](#).
- **FEA Input Files LS-DYNA (\*.dyn):** Creates the simulation model for the [FEA-Simulation](#), which can be loaded from the FEA system LS-Dyna. See also [Output, FEA, LS-Dyna](#).

#### Hints:

- In order to import a partial project file (\*.pro), use the function [File, Partial Project Add On](#).
- In order to open a PROFIL project (\*.pro) use the function [File, Open Project](#).



### 3.1.1.9 Print Preview



Use this function to open the window **Print Preview** on the screen.

#### Calling the function

Call this function by:

- Main menu: **File, Print Preview**



The displayed sheet shows you the print-out of the drawing and the data of the profile project. If the print preview contains more than one page, you can browse by using the page selector in the top button bar.

#### Principle of operation

You can call these functions from the menu or the button bar of this window:

- **Printer Print:** The print-out is started.
- **Printer Setup:** Select the printer, the paper size and orientation.
- **Printer Font:** Select font, color and size.
- **Contents Headline:** Printing of the headline (description on the left and customer on the right) is switched on and off.
- **Contents Drawing:** Printing of the drawing that is visible in the [Drawing Area](#) is switched on and off.
- **Contents Profile/Roll List:** Printing of the current profile/roll list is switched on and off.

Whether the [Spring Back](#) or the [Holes/Cut-Outs](#) will also be printed out, you select in [Options Profile List](#). Bold print style for radius/angle marks the [loaded/discharged](#) state.

-  **Contents Statics:** Printing of the table of statics is switched on and off (only if a profile list is selected). The table is identical with the table called by the function [Calculate Statics](#).
-  **Contents Stress of Band Edge:** Printing of the diagram of stress of band edge is switched on and off (only if a profile list is selected). The table is identical with the table called by the function [Calculate Stress of Band Edge](#).
- **Contents NC program:** Printing of the NC program is switched on and off (requires [View Rolls](#) and **Contents Roll List** is switched on).
- **Page:** If the print-out has more than one page, select the desired one.
- **Scale:** The drawing can be printed out by using one of the following fixed scales:
  - 20:1
  - 10:1
  - 5:1
  - 2:1
  - 1:1
  - 1:2
  - 1:5
  - 1:10
  - 1:20

Select the scale from the drop-down-list, so that the drawing fits the sheet. If you don't find a practical scale, select **User** and enter the desired scale into the input window, e.g.

3.000 for scale 3 : 1

0.250 for scale 1 : 4

Select **Fit** if the scale should be adjusted in order to have the whole print-out on one page only. This function is limited to small table lengths. I will not work in case the table lengths need a form feed anyway.


**Hint:** All settings in this window are also effective in the function [File, Print](#).

### 3.1.1.10 Print

Use this function to print out the drawing that is displayed in the [Drawing Area](#) and the data of the selected [Profile Project](#). In order to select the desired printer, print modus, and content of the print-out, call [File, Print Preview](#) previously.

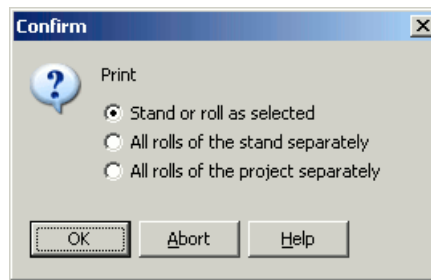
#### Calling the function

Call this function optionally by:

- Main menu: **File, Print**.
-  Button **Print** in the [Button Key Bar](#).

#### Principle of operation

If [View Pass](#), [View Statics](#), [View Flower Nested](#), [View Flower Separated](#), or [View Flower 3D](#) is the current view, the content of the [Drawing Area](#) (or what is selected) is printed. If needed, successive pages are printed.



If [View Roll Tools](#) is the current view, the dialog window appears and you can choose:

**Stand or roll as selected:** If a roll is selected in the [Drawing Area](#), only this roll is printed. If nothing is selected (function [Inspect](#)), all rolls of the current stand are printed together on one page.

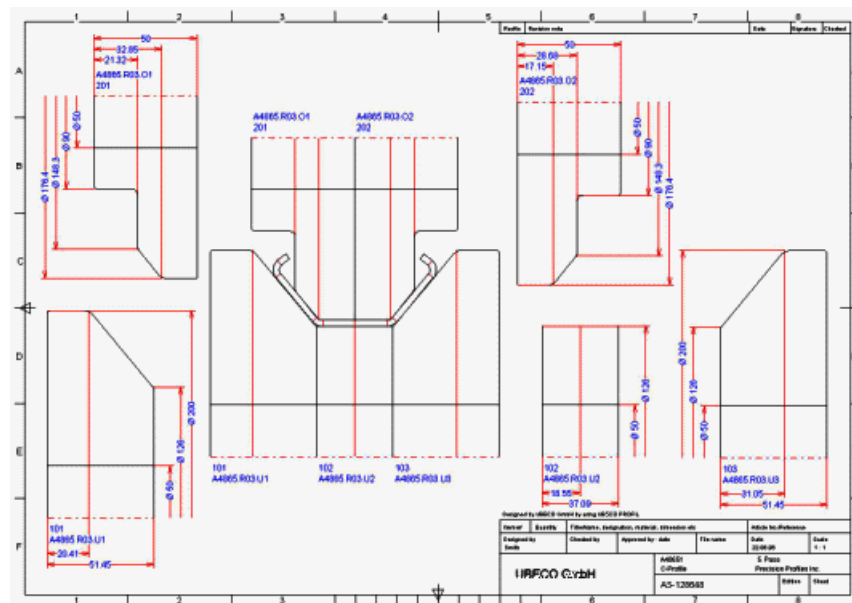
**All rolls of the stand separately:** All rolls of the current stand are printed, each roll on a separate page.

**All rolls of the project separately:** All rolls of the whole project are printed, each roll on a separate page.

#### Hints:

- Select in [File, Print Preview](#) previously what should be printed and set-up the printer and the font. The settings are used for all later print-outs with the function **Print**.
- The function [File, Plot](#) can be used to print or plot the **Roll Assembly Plan**.
- The function Edit, Copy can be used to copy the drawing that is displayed in the [Drawing Area](#) to the WINDOWS clipboard. From this, it can be pasted to any other WINDOWS program in order to print it.

#### 3.1.1.11 Plot



This function creates the **Roll Assembly Plan** that shows the mounted rolls and the separated rolls with dimensioning in addition. A prepared **Drawing Template** from a DXF file is opened and the stand drawing is created in the center. Then the rolls just can be picked up by mouse-click and copied or moved to another position and rotated or mirrored at the x- and y-axis. The dimensioning can be switched on and off as desired. The title block of the drawing is filled out automatically.

Afterwards the drawing template can be saved. This means, instead of the rolls and the title block entries **Variables** are saved as placeholders. When the template later is reopened from another stand, the variables are replaced again by the rolls and title information from the new stand. Since


the drawing template is a DXF file, it also can be modified by using any CAD system, if desired.

The created roll assembly drawing can be plotted directly without the help of a CAD system. It also can be transferred to any CAD system to plot it from there, in case further additions are necessary. A third feature is to copy the drawing to the Windows clipboard in order to paste it into any other program. When in the course of time a collection of drawing templates for different kinds of roll forming stands exists, it is very easy to create new roll assembly plans just by using the appropriate drawing template.

### Calling the function

Call this function optionally by:






- Main menu: **File, Plot.**

-  Button **Plot** in the [Button Key Bar](#).

The window **Plot Assembly Plan** is opened, it contains the last used [Drawing Template](#) and the drawing that currently is shown in the [Drawing Area](#) of the main window. If you attached dimensioning to your drawing objects, they at first do not appear in the assembly plan. The [Navigator](#) helps you to zoom, to move and to fit the drawing. Furthermore the Wheel Mouse is supported.

### Principle of operation

You can call these functions inside this window:





- **Left mouse button:** Click on an origin object, e.g. a roll in the assembly part. The roll is picked up and you can copy it to another position in order create the detailed part. At the same time the dimensioning appears, if you attached dimensioning in the [Drawing Area](#) of the main window previously. If you pick up an object, which is copied already, it is now moved in order to position it precisely.
- **Right mouse button:** The context menu drops down and you can select if all objects should be moved or how a single object should be handled: copy, move, rotate, mirror, or delete. Furthermore you can switch on and off the dimensioning of an object.
-  **Open template:** Open a DXF file that contains a [Drawing Template](#), this is a drawing frame and a title block. Use one of the existing templates or define a template by yourself by using any CAD system. If the template contains [Variables](#), they are replaced by the drawing objects or texts of the current drawing or project.
-  **Save template:** Save the current [Drawing Template](#) to a DXF file with any name. The drawing frame, the title block, and all fixed texts are saved to the file. The drawing objects and the variable texts are not saved; instead of them [Variables](#) are saved, which are replaced by the belonging drawing objects or texts when the template is reopened from another stand or project.
-  **Copy drawing to clipboard:** Use this button to copy the whole assembly drawing to the Windows clipboard (see also [Edit Copy](#)).
-  **Plot To Scale:** The plotting scale of the drawing template always is 1:1. The content of the drawing is plotted by using the selected Scale. If the plot range of your output device is smaller than the drawing, only the center part is plotted. Better use Scaled To Fit instead.
-  **Scaled To Fit:** The size of the drawing is reduced or extended in order to fit the maximum plot range of the output device.
- **Scale:** The drawing can be sent to the output device by using one of the following fixed scales:
  - 20:1
  - 10:1
  - 5:1
  - 2:1

1:1  
1:2  
1:5  
1:10  
1:20

Select the scale from the drop-down-list, so that the drawing fits the sheet. If you don't find a practical scale, select **User** and enter the desired scale into the input window, e.g.

3.000 for scale 3 : 1

0.250 for scale 1 : 4

-  **Drawing -> CAD:** Use this button to transfer the assembly drawing to the CAD system. The settings of the function [Drawing -> CAD](#) in the PROFIL main window are used as well. If you do not want to transfer a drawing template, use an empty DXF file as template.
-  **Plot:** The Windows print dialog opens and you can select the desired output device.
-  **Reset:** Use this button to undo all modifications and to start newly.
-  **Cancel:** Closes the window.

#### Hints:


- The functions [File, Print Preview](#) or [File, Print](#) can be used to print the drawing and further information.
- The function Edit, Copy can be used to copy the drawing that is displayed in the [Drawing Area](#) to the WINDOWS clipboard. From this, it can be pasted to any other WINDOWS program in order to print it.

### 3.1.1.12 Exit

Use this function to terminate **PROFIL**.

#### Calling the function

Call this function optionally by:

- Main menu: **File, Exit**.
-  Button **Exit**.

#### Principle of operation

**PROFIL** is terminated. If you forgot to save your profile project, **PROFIL** will remind you.


## 3.1.2 Edit

### 3.1.2.1 Undo

Use this function to reverse the most recent operations.

#### Calling the function

Call this function optionally by:

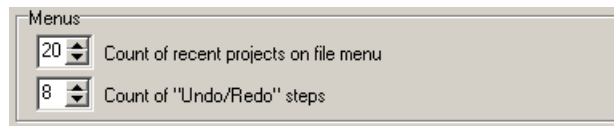
- Main menu: **Edit, Undo**.
-  Button **Undo** in the [Button Key Bar](#).
- Shortcut: **Ctrl Z** (Setup in [Options, Keyboard](#)).



Undo: Spacer Rolls Create

The menu item and the button hint shows the name of the operation that can be reversed next.

### Setup




Select in [Options, General](#), **Menus** how many steps should be available for **Undo**.

### Hint:


If you did this in error, you can  [Redo](#) the commands.

### 3.1.2.2 Redo

If you called  [Undo](#) too often in error, you can reverse the effect of the last undo command by using this function.

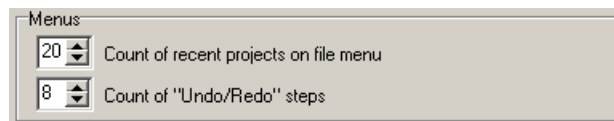
### Calling the function

Call this function optionally by:

- Main menu: **Edit, Redo**.
-  Button **Redo** in the [Button Key Bar](#).


The menu item and the button hint shows the name of the operation that can be reversed next.

### Setup



Select in [Options, General](#), **Menus** how many steps should be available for **Redo**.

### Hints:

If you did this in error, you can  [Undo](#) the commands.

### 3.1.2.3 Copy

Use this function to copy the drawing from the [Drawing Area](#) as a bitmap to the Windows clipboard. This is useful to transfer the picture to any Windows application (e.g. Word, Paint and others) by using the function **Edit, Paste** in the other application.

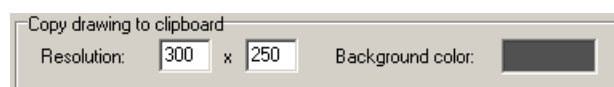
### Calling the function

Call this function optionally by:

- Main menu: **Edit, Copy**.

### Setup

Because of conversion of a vector graphic to a bitmap the width of the lines is one pixel exactly. This is why it is not recommended to zoom the picture in the target application afterwards; if you do so, you will get improper results.



Better set a practical **Resolution** in [Options General](#), **Copy Drawing to Clipboard** previously to get the desired size of the picture. The drawing will always fit to the bounds of the picture, not dependent on the zoom scale in **PROFIL**. Furthermore you can set the desired **Background**

**Color** of the picture.

**Hint:**


In order to save the drawing to a bitmap file (.bmp), use the [Export](#) function.

#### 3.1.2.4 Machine

This function opens and closes the [Machine Window](#), which shows the machine data from the current [Project](#).

**Calling the function**

Call this function optionally by:

- Main menu: **Edit, Machine**.
-  Button **Machine** in the [Button Key Bar](#).

**Hints:**


- For designing the profile and the flower pattern, the **Distances between Stands** are needed.
- For designing the roll tools, further machine data, e.g. **Working Diameter**, **Shaft Diameter** and **Working Width** are needed.

#### 3.1.2.5 Window visible

Use this function to select, if you want to see the [Project Data Window](#), the [Profile List Window](#) and the [Roll Tool Window](#) on the screen. The status of this switch is displayed by a little hook.

**Calling the function**

Call this function optionally by:

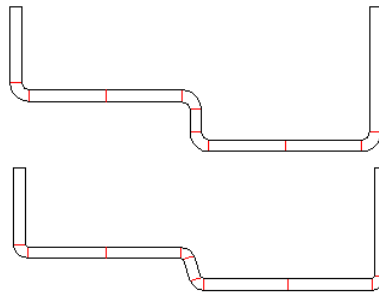
- Main menu: **Edit, Window visible**.
-  Button **Window visible** in the [Button Key Bar](#).

**Hints:**

- Decide by your own, if you prefer to work numerically in the profile list (the drawing is always displayed in the background) or graphical only in the drawing. In this case, you can switch off the windows to get an extended [Drawing Area](#).
- Preset in [Options Profile list](#), **Profile list window** how many windows you want to see.

### 3.1.2.6 Draft Modus

The **Draft Modus** switch can be used to toggle between **bending** an arc (while [Designing the Flower Pattern](#)) and **modifying** an arc (while [Designing the Profile](#)).



Example: In the draft modus angles and radii can be modified mutually independently

#### Calling the function

Call this function optionally by:

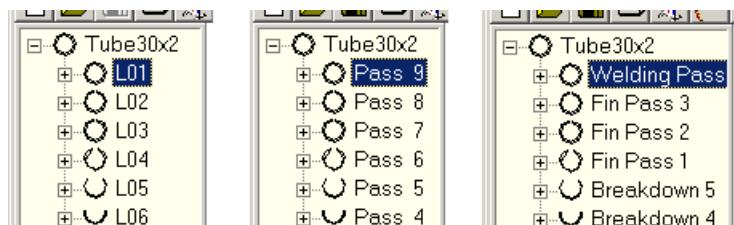
- Main menu: **Edit, Draft Modus**

#### Principle of operation

**Draft Modus checked:** Use this setting while [Designing the Profile](#), if modifications of the [Profile List](#) are necessary. You can modify angles and radii of [Arc Segments](#) without changing the corresponding other value and without changing neighboring segments (this means the [Bending Methods](#) A1..A4 are **not** considered). The straight length and with it the strip width is changed necessarily. The switched-on draft modus is displayed by a changed background color. The draft modus is switched off again automatically if you select another view or if you open another project. The picture shows an example wherein inner radii and angles has been modified mutually independently in the top cross-section. The bottom cross-section shows the result.

**Draft Modus unchecked:** Use this setting for [Designing the Flower Pattern](#) after [Designing the Profile](#) is finished. By modifying angles and radii of arc segments the arc is bent or unbent dependent on the [Bending Methods](#) A1..A4. During this operation the [Strip Width](#) (the sum of the [Straight Lengths](#) of all profile segments) keeps unchanged. The background of the [Drawing Area](#) has the color that is selected in [Options, Drawing](#), "Colors of Drawing Area, Background".

### 3.1.2.7 Explorer



Select how the profile lists in the [Explorer](#) should be called:

- **Show profile list number (default)**, e.g. L01, L02, ..., counting against the sheet running direction.
- **Show pass from profile list**, e.g. Pass 1, Pass 2, ..., counting in sheet running direction. It comes from the [Pass](#) input field of the [Profile List Windows](#).
- **Show stand from machine data**, e.g. Welding Pass, Fin pass. It comes from the [Stand Name](#) input field in the [Machine Window](#).



### Calling the function

Call this function optionally by:

- Main menu: **Edit, Explorer.**

#### Hint:


The function [Output Drawing -> CAD](#) uses the profile list name as layer name for the pass or the stand.

### 3.1.2.8 Options

Use this function to adapt PROFIL to your individual needs.

#### Calling the function

Call this function optionally by:

- Main menu: **Edit, Options.**
-  Button **Options** in the [Button Key Bar](#).

Select the following options:

- [Options General](#)
- [Options Drawing](#)
- [Options Profile list](#)
- [Options Calculate](#)
- [Options Rolls](#)
- [Options Spacer Rolls](#)
- [Options Database](#)
- [Options Parts List](#)
- [Options NC](#)
- [Options Files](#)
- [Options ActiveX](#)
- [Options PSA](#)
- [Options Keyboard](#)
- [Options Mouse](#)

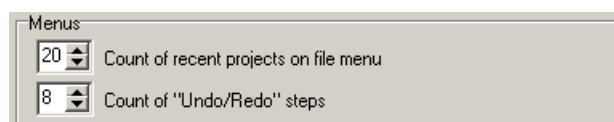
#### Principle of operation

The options are saved by pressing the Ok button. After restarting PROFIL, the options will be restored.



### 3.1.2.8.1 General

Use this function for these general settings:

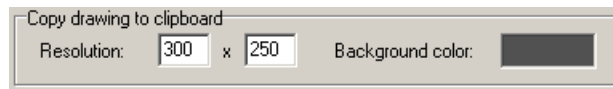
#### Menus:



**Count of recent projects on file menu:** Set the number of recently opened projects on the menu **File**. Enter a number between 0 and 20.

**Count of "Undo/Redo" steps:** Select the count of steps for the functions  [Edit, Undo](#) and  [Edit, Redo](#).

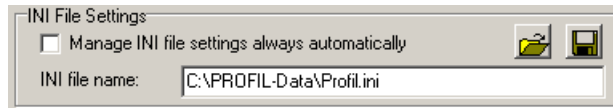
#### Copy drawing to clipboard:



**Resolution:** Set a practical resolution of the bitmap to get the desired size of the picture, if you use the function [Copy](#) to copy the drawing from the [Drawing Area](#) as a bitmap to the Windows clipboard.

**Background color:** Select the desired background color of the copied picture.

### INI File Settings:



All system and user settings are saved in the WINDOWS Registry at program exit and are loaded at program start automatically. For special purposes you can manage them in an INI file as well, e.g.:

- If you want to transfer the settings to another computer.
- If you use different computers and you always want to work with the same settings.

**Manage INI file settings always automatically:** Settings are saved in the WINDOWS Registry in any case, at program start the INI file has priority however. If the INI file is not available, e.g. in case of network, the settings from the Registry are used.

**Load/Save INI file settings once:** Buttons for loading and saving the setting from and to respectively the INI file.

**INI file name:** Select path and name of the INI file. If necessary, you can select a network path as well.

### Text Editor:



Several PROFIL function need a text editor. If you do not want to use the WINDOWS NotePad, you can set up another one. After mouse click on this input field the file select window opens and you can select the EXE file of the desired text editor. Take care that the text editor saves files in pure ASCII format.

### "Open Project" and Explorer with preview

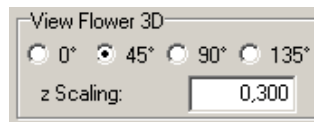


While working with larger projects the screen refresh takes more time, because refreshing the preview pictures is necessary after each modification. This may impede quick working. Simply switch off the preview.

## 3.1.2.8.2 Drawing

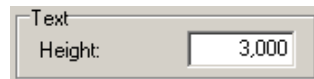
Use this function to set the parameters for views and drawings:

### View Flower 3D:



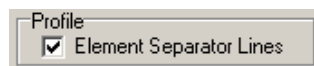
Select the angle for the [View Flower 3D](#), [View, PSA - Profile Stress Analysis](#), and [View FEA Result](#). The angle means: **0°** - to right (side view), **45°** - to upper right, **90°** - to upper, **135°** - to upper left. By using the z scaling factor you can compress the drawing in z direction (sheet running direction) for a better overview: 1.0 means not compressed, 0.2 means compressed to 20%.

#### Text Height:



Select the text height for the table of statics, the roll numbering and the dimension texts on the [Drawing Area](#).

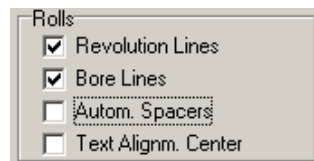
#### Profile:



Select, if the [Profile Drawing](#) should contain the following information:

- **Element Separator Lines:** mark start and end of a [Profile Element](#) (type L or A). They are shown in **Auxiliary Line Color**.

#### Rolls:

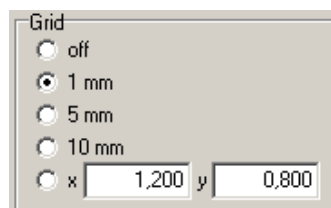


Select, if the [Roll Tool Drawing](#) should contain the following information:

- **Revolution Lines:** are the (imagined) lines to the [Roll Corner Points](#). These lines are useful for selecting a roll corner point by mouse click. They are shown in **Auxiliary Line Color**.
- **Bore Lines:** are displayed with the [Diameter Shaft](#).
- **Automatic Spacers:** are for fixing the horizontal position of the forming rolls. PROFIL creates them automatically by filling the distance between forming rolls and the [Working Width](#) of the machine.
- **Text Alignment Center:** The texts for [Roll No.](#), [Part-No.](#) and [Bore Hole](#) can be aligned center instead of left.


**Hint:** The difference between **Automatic Spacers** and **Spacer Rolls** is described in [Creating Spacers](#).

#### Grid:



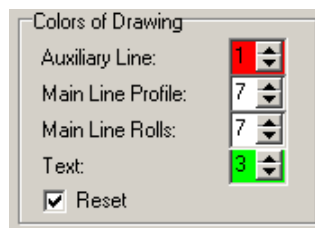
Select the grid lines distance and if the grid lines should be visible on the screen or not.

**Hints:**

- If you have selected a grid lines distance that starts with number 1, 2, or 5 (most significant digit), the grid lines with double, 5-fold, or 10-fold distance are drawn bold.
- Select the grid lines color in [Options, Colors](#).
- Disable the grid lines with [Options, Drawing, Grid](#) or  **Button Grid on-off**.

**3.1.2.8.3 Colors**

Use this function to preset the colors in the [Drawing Area](#):

**Colors of drawing:**

**Auxiliary line, main line profile, main line roll, text:** Select the colors for the drawing. The numbers of the colors are the same as in AutoCAD:

0 = by block

1 = red

2 = yellow

3 = green

4 = cyan

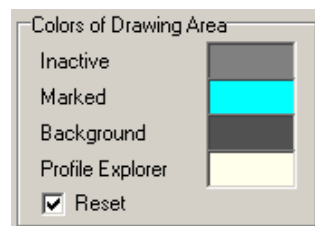
5 = blue

6 = magenta

7 = white

The selected colors will be transferred into the CAD and NC output file.

**Default:** All colors of the drawing are reset to the default colors.

**Colors of drawing area:**

By mouse-click a color selection window will appear. Select the desired colors for the [Drawing Area](#):

**Inactive:** Color of the parts of the drawing, which does not represent the active pass.

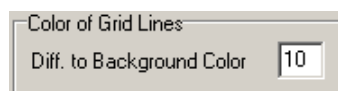
**Marked:** Color of the drawing element that you have selected by mouse-click.

**Background:** Color of the background of the drawing area.

**Profile Explorer:** Color of the background of the Profile Explorer.

The selected colors will NOT be transferred into the CAD and NC output file.

**Default:** All colors of the drawing area are reset to the default colors.

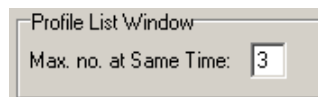
**Color of Grid lines:**

The **color of the grid lines** should differ only a little from the **background color of the drawing area**. Select how much it should differ. If the background color is dark, the grid lines are shown in a color that is a little bit lighter, otherwise darker. Recommended is a value of 6..12. A larger value causes a more brightness distance. Try a value that you just can see the grid lines, they should not interfere with the drawing.

#### 3.1.2.8.4 Profile List

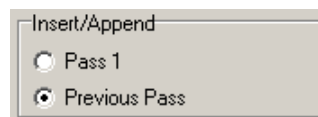
Use this function to set the parameters for the profile list:

##### Profile list window:



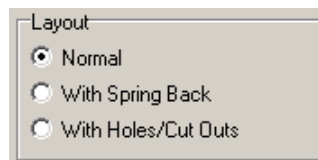
Select the **Max no. at Same Time** of [Profile List Windows](#). For example, if you select 3, the 4th window closes the 1st again. So you do not need to close superfluous windows by yourself.

##### Insert/Append



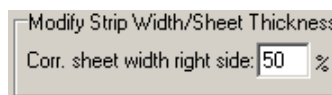
Select, if the Profile List of **Pass 1** or of the **Previous Pass** should be inserted/appended, when you call the function [Profile, Insert](#) or [Profile, Append](#).

##### Layout:



Select among 3 possible layouts of the [Profile List Window](#): **Normal** is the smallest window without spring back and holes/cut-outs. **With Spring Back** displays the [Spring Back](#), this means, besides the discharged state and the loaded state too. **With Holes/Cut-Outs** displays the [Holes/Cut-Outs](#).

##### Modify Strip Width/Sheet Thickness:



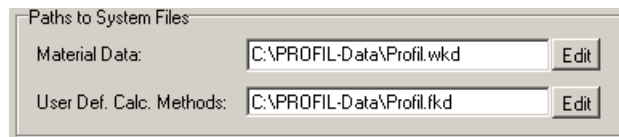
When the functions [Profile, Modify Sheet Thickness](#) and [Profile, Modify Strip Width](#) are applied on a asymmetrical profile, the partition of the strip width modification to the left and right band edge can be preset:

**Corr. sheet width right side:** Enter a value between 0 and 100 that assigns the percentage of the modification to the right band edge. The difference to 100% is assigned to the left band edge. The entry 50% means: Both band edges are modified in equal measure.

### 3.1.2.8.5 Calculate

Use this function to set the parameters for the calculation of the profile:

#### Paths to System Files:



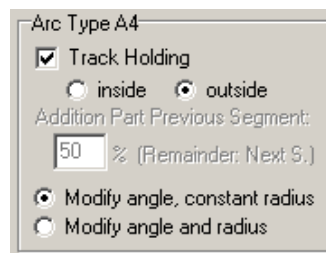
**Material data:** Use the **Edit** button to call the text editor (set in [Options General](#)) with the [Material File](#). This file contains a set of sheet materials with according data. You can modify or enhance the file to your own needs. Pay attention that all materials have an increasing number for later identification. The input field in front of the Edit button is useful for selecting the path to the material file, e.g. if the file should be installed in a network path for common access.

**User defined calculation methods:** Use the **Edit** button to call the text editor (set in [Options General](#)) with the [Factor file](#). Inside this file you can set [User Defined Calculation Methods](#), if you are not satisfied with default methods of Oehler or DIN 6935. With first installation of the system you will find two methods **User1** and **User2**. The first one is an example for a factor method and the second for an addition method. You can modify them for your own needs. The input field in front of the Edit button is useful for selecting the path to the factor file, e.g. if the file should be installed in a network path for common access.

#### Hint:

- Because the text editor is an independent program, it must be closed manually. Otherwise it will keep open in the background.

#### Arc Type A4



While bending the arc type A4 (constant radius method, see [Arc Types](#)) the residual length is added to the previous and the next segment. Select how the length should be partitioned:

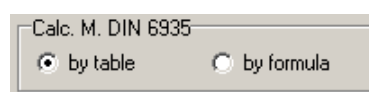
**Track Holding:** The length is partitioned automatically that the strip is guided straightaway with constant intersection point of the tangents. Select if the intersection point of the tangents **inside** or **outside** of the arc should be constant.

**Addition Part Previous Segment:** Select how much % of the length should be added to the previous segment. The remainder is added to the next segment. If "Track Holding" is set, this input field is not active.

**Modify angle, constant radius:** With this setting A4 works like A2 and A3 by keeping the radius constant if a new angle is entered.

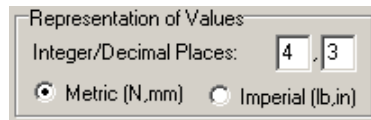
**Modify angle and radius:** For special needs an arc can be bent open by entering a new angle and a new radius, see [Arc Types](#). Case of application is to compensate the springback and to minimize the end flare effect.

#### Calc. M. DIN 6935:



In DIN 6935 a method defined by a table and a method defined by formula is described. Select if you want to use the method **by table** or **by formula**. The setting will take effect when you select the calculation method DIN 6935 in the [Project Window](#).

### Representation of values:



Representation of Values

Integer/Decimal Places:

☒ Metric (N,mm) ☐ Imperial (lb,in)

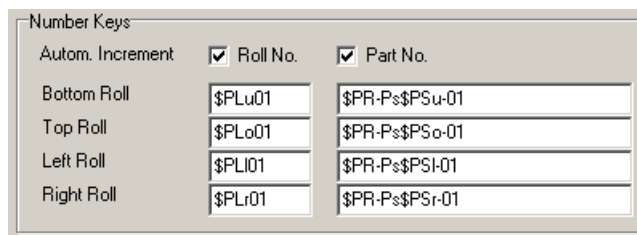
**Integer/Decimal Places:** Set the number of integer places (digits left of the decimal separator) and the decimal places (digits right of the decimal separator) for the representation of values in the data windows. For normal profile size in metric system (unit mm) use 4 integer places and 3 decimal places. If you design imperial (unit inch), use 3 integer places and 4 decimal places. For very small profiles we recommend to increase the decimal places by 1. For very large profiles it is better to decrease the decimal places by 1.

**Metric/Imperial:** Select, if PROFIL should work in the Metric System (Units mm, N) or in the Imperial System (Units in, lb). In case of change, you should set the decimal places, too.

### 3.1.2.8.6 Rolls

Use this function to set the parameters for creating forming rolls.

#### Number Keys:



Number Keys

Autom. Increment ☒ Roll No. ☒ Part No.

Bottom Roll	<input type="text" value="\$PLu01"/>	<input type="text" value="\$PR-Ps\$PSu-01"/>
Top Roll	<input type="text" value="\$PLo01"/>	<input type="text" value="\$PR-Ps\$PSo-01"/>
Left Roll	<input type="text" value="\$PLl01"/>	<input type="text" value="\$PR-Ps\$PSl-01"/>
Right Roll	<input type="text" value="\$PLr01"/>	<input type="text" value="\$PR-Ps\$PSr-01"/>

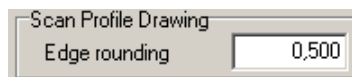
Use this function to set the keys for the automatic numbering of the roll number and the part number, when rolls are created newly. If you want to use the number keys for existing rolls, call the function [Roll Renumber](#).

**Autom. Increment:** Select, if the roll number and the part number should be incremented automatically (number +1), when you split a roll with the function [Roll Split at Corner](#) and [Roll Split between Corners](#). Precondition: the last digit must be numeric (0..9).

**Bottom Roll, Top Roll, Left Roll, Right Roll:** Select the roll and part number for the roll, which you will create with the function [Roll Read CAD-Contour](#) and [Roll Scan Profile Drawing](#). For the automatic increment it is necessary that the last digit (or last two digits) are numeric. Enter a fixed term (e.g. **b01** for the first roll of the bottom shaft) or a combination of a [Variable](#) and a fixed term (e.g. **\$PLb01**).

**Example:** If you enter **\$PSb01** into the input field **Bottom Roll, Roll No.**, the first bottom roll of the 7th pass will get the roll number 7b01.

#### Scan Profile Drawing:

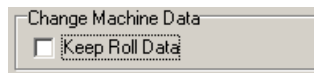


Scan Profile Drawing

Edge rounding

**Edge rounding:** Enter the radius (fillet) for the first and the last corner of a roll, when you create a roll automatically by using the function [Roll Scan Profile Drawing](#).

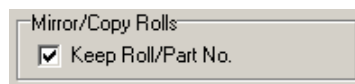
#### Change Machine Data



**Keep Roll Data:** Select what should happen with the rolls of your current project, when you modify the [Machine Data](#) (working diameter, reference point, inclination angle) or if you select another machine:

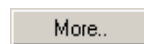
- If checked, the roll data keep unchanged, but the position of the rolls will change dependent on the new machine data. Possibly the rolls do not touch the profile anymore.
- If not checked, the working contour of the rolls will be retained, but the center line and the sides of the rolls will change. This means the rolls touch the profile at the same points as previously, but the size of the rolls is changed.

### Mirror/Copy Rolls



**Keep Roll/Part No.:** If checked, the roll/part number is not renumbered if a roll is mirrored (function [Rolls Mirror](#)) at a roll edge (bottom/top rolls) or from one side to the opposite side (side rolls). It also is not renumbered if a roll is copied to another position via the clipboard (function [Roll Copy](#) and [Roll, Paste](#)). Otherwise it get a new number dependent on the preset number keys.

### More

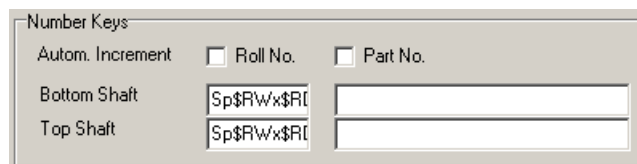


The [Expanded Roll Tool Window](#) opens. You can enter default values, which are copied into a roll dataset when you create a new roll by [Roll Read CAD-Contour](#) or [Roll Scan Profile Drawing](#).

### 3.1.2.8.7 Spacer Rolls

Use this function to set the parameters for [Creating Spacer Rolls](#).

#### Number Key



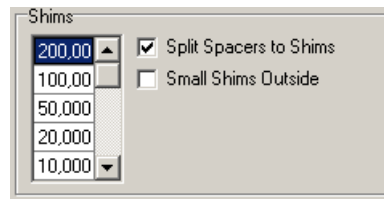
This setting enables the automatic numbering of the roll number and the part number for spacer rolls only, when spacer rolls are created newly. If you want to use the number keys for existing rolls, call the function [Roll Renumber](#).

**Autom. Increment:** Select, if the roll number and the part number should be incremented automatically (number +1), when you split a spacer roll with the function [Roll Split at Corner](#) and [Roll Split between Corners](#). Precondition: the last digit must be numeric (0..9).

**Bottom Roll, Top Roll:** Select the roll and part number for the roll, which you will create with the function [Roll Read CAD-Contour](#) and [Roll Scan Profile Drawing](#). For the automatic increment it is necessary that the last digit (or last two digits) are numeric. Enter a fixed term (e.g. **Sp01** for the first roll of the bottom shaft) or a combination of a [Variable](#) and a fixed term (e.g. **Sp\$RWx\$RD** would cause the roll number Sp50x70).



## Shims

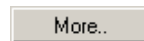


**Split Spacers to Shims:** If a spacer is one item only, it has to be manufactured for each project newly. Often spacers should be combined from an existing stock of shims, it is useful to split them into a set of shims with available sizes. If the box is checked, PROFIL splits into shims as large as possible, until a small shim with a special size remains.

**Table of Shim Sizes:** Enter the available shim width in descending order. Each width must appear only once, also if shims of this size are available multiple times. The **TAB** key creates new table entries; **0** removes an entry.

**Small Shims Outside:** Select, if the spacer splitting should start outside (at the stand) or inside (at the forming roll).

## More



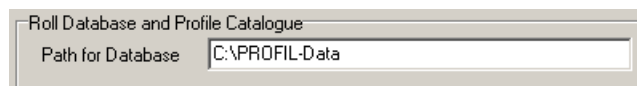
The [Expanded Roll Tool Window](#) opens. You can enter default values, which are copied into a spacer roll dataset during [Creating Spacer Rolls](#).

**Hint:** The difference between automatic spacers and spacer rolls is discussed in [Tutorial, Roll Tools, Creating Spacer Rolls](#).

### 3.1.2.8.8 Database

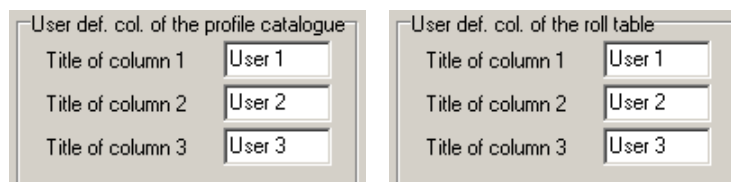
Only with option Database.

#### Profile Catalogue and Roll Database:



**Path for Database:** Enter the path to the database files. After clicking into the input field, the Explorer appears. The path must be valid, either on the local disk or on a network server. If the database files do not exist, they will be created, when you call the [Profile Catalogue](#) or the [Roll Stock Management](#) for the first time. The files are: Profiles.db and Profeles.db for the profile catalogue and Rolls.db, Corners.db and Projects.db for the roll stock management at least, further index files are created automatically, if they do not exist.

#### User defined columns of profile table and the roll table:

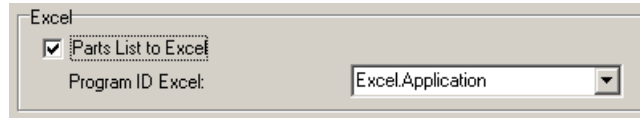


**Title of column 1..3:** Enter the titles for the last 3 columns of [Profile Table](#) and the [Roll Table](#). You can define these columns for your own needs.

### 3.1.2.8.9 Parts List

Use this function to set the parameters for the creating of the parts list of the rolls by the function [Output Create Parts List](#).

#### Excel:

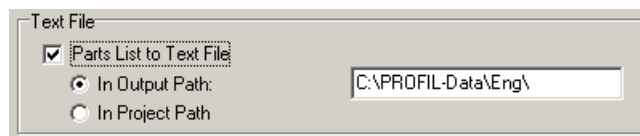


The 'Excel' dialog box contains a checked checkbox labeled 'Parts List to Excel'. Below it, the text 'Program ID Excel:' is followed by a dropdown menu currently showing 'Excel.Application'.

**Parts List to Excel:** Check here, if the parts list directly should be transferred into a predefined work sheet of MS Excel by the function [Output Create Parts List](#).

**Program Id Excel:** Select from the drop-down-list the Excel release for the ActiveX transfer. The list shows all installed Excel releases, the transfer is possible to release 8 or higher. "Excel.Application" means the topical release (last installed).

#### Text File:



The 'Text File' dialog box contains a checked checkbox labeled 'Parts List to Text File'. Below it, there are two radio buttons: 'In Output Path:' (which is selected) and 'In Project Path'. To the right of these radio buttons is a text input field containing the path 'C:\PROFIL-Data\Eng\'. There is also an 'Edit' button next to the input field.

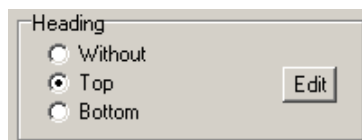
**Parts List to Text File:** Check here, if a text file should be created by the function [Output Create Parts List](#). The name of the text file is always the same as the project name, but has the extension .txt.

**In Output Path:** Enter the path of the path for the parts list text file, if necessary also with a drive letter, e.g. c:\PartsLi\

**In Project Path:** Check here, if the parts list text file should be created in the same path of the belonging project file.

#### Heading:

(for **Parts List to Text File** only)



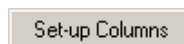
The 'Heading' dialog box contains three radio buttons: 'Without', 'Top' (which is selected), and 'Bottom'. To the right of these radio buttons is an 'Edit' button.

**Without-Top-Bottom:** Select the position of the heading top (suitable for printing) or bottom (suitable for transferring the parts list into the CAD-drawing) or without heading (for transferring into a spreadsheet or an ERP system).

**Edit:** By clicking this button the text editor (set in [Options General](#)) is opened with the file PROFIL.leg. This file contain 1 line exactly with the heading for the parts list. Modify the heading for your needs.

In case of Excel please define the heading within your initial work sheet.

#### Set-up Columns:

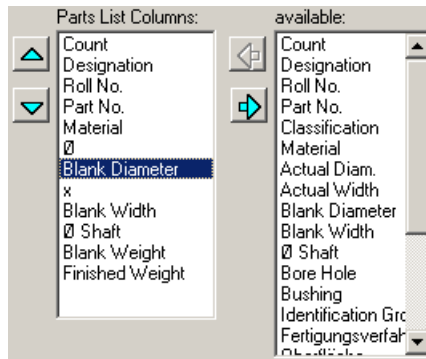


A rectangular button with the text 'Set-up Columns'.

After pressing this button the [Set-Up Parts List Columns Window](#) opens.

### 3.1.2.8.10 Set-Up Parts List Columns

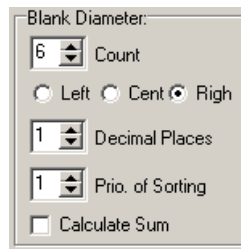
Use this window to set-up the columns of the parts list. The window appears if you press button **Set-Up Columns** in [Options Parts List](#).



**available:** This list box shows which parts list columns can be selected:

Count	count of rolls with same (visible) parts list properties.
Designation	fixed text from table <b>Designation</b>
Roll No.	from the <a href="#">Roll Tool Window</a> .
Part No.	from the <a href="#">Roll Tool Window</a> .
Classification	from the <a href="#">Roll Tool Window</a> .
Material	name of the <a href="#">Addition File</a>
Actual Diam., Width	from the <a href="#">Roll Tool Window</a> .
Blank Diam., Width	actual dimensions with additions from the <a href="#">Additions File</a> that is set in <b>Material</b> . Unit mm or inch.
Ø Shaft	from the <a href="#">Machine Data</a> .
Bore Hole	from the <a href="#">Expanded Roll Tool Window</a> .
Bushing	from the <a href="#">Expanded Roll Tool Window</a> .
Identification Groove	from the <a href="#">Expanded Roll Tool Window</a> .
Treating	from the <a href="#">Expanded Roll Tool Window</a> .
Surface	from the <a href="#">Expanded Roll Tool Window</a> .
Addition	from the <a href="#">Expanded Roll Tool Window</a> .
Remark	from the <a href="#">Expanded Roll Tool Window</a> .
Blank Weight	weight of the cylindrical tool blank with the blank size. Unit kg or lb. Multiplied with <b>Count</b> .
Finished Weight	weight of the finished roll; the outer contour and the bore hole are considered. Unit Kg or lb. Multiplied with <b>Count</b> .
Ø	diameter sign, can be inserted in front of blank or finished diameter.
x	by sign, can be inserted between diameter and width.
<User>	for any user defined characters

**Parts List Columns:** From the list **available** any entry can be copied into the **Parts List Columns** table by using the button with the left arrow. Afterwards the entry can be moved by using the buttons with the up and down arrow. To remove an entry from the list, use the button with the right arrow. The parts list is created by function [Output, Create Parts List](#) dependent on the order in the **Parts List Columns** table.



Blank Diameter:

Count: 6

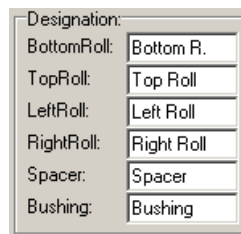
Position: ☐ Left ☐ Cent ☒ Right

Decimal Places: 1

Prio. of Sorting: 1

☐ Calculate Sum

**Column Properties:** For each entry in the **Parts List Columns** table the properties can be set: **Count** (of characters), **Position** (left, center or right), **Decimal Places**, **Priority of Sorting** and if the **Sum** should be calculated. Decimal places, sorting and sum are only possible for numerical values. Count, Position and decimal places are only possible if the parts list is created as a text file (not for transfer to Excel). **Priority of Sorting:** Enter priority 1 (highest) if this column should be sorted. If rolls exist with same values in this column, the priority 2 can be set for another column. Then rolls with same values are sorted in this column.



Designation:

BottomRoll: Bottom R.

TopRoll: Top Roll

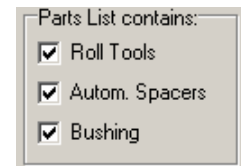
LeftRoll: Left Roll

RightRoll: Right Roll

Spacer: Spacer

Bushing: Bushing

**Designation:** The text that appears in the parts list column **Designation** can be user defined dependent on the type of roll: bottom, top, left and right roll. Furthermore the text can be defined for spacers and [Bushing](#).



Parts List contains:

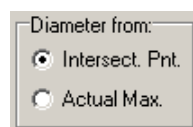
☒ Roll Tools

☒ Autom. Spacers

☒ Bushing

**Parts List contains Roll Tools/Autom. Spacers/Bushing:** Select what kind of entries the parts list should contain..

**Hint:** Select by [View, Show, Spacer Rolls](#) whether the parts list should contain spacer rolls. The difference between automatic spacers and spacer rolls is discussed in [Tutorial, Roll Tools, Creating Spacer Rolls](#).



Diameter from:

☒ Intersect. Pnt.

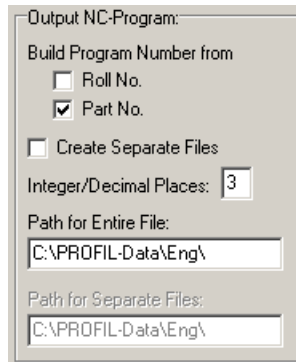
☐ Actual Max.

**Diameter from Intersection Point/Actual Maximum:** Select if the maximum roll diameter is calculated from the intersection point of the tangents ([Roll Corner Point](#)) or from the actual diameter on the fillet. The calculation of the [Additions](#) and with it the blank diameter is dependent on this selection.

### 3.1.2.8.11 NC

Use this settings for the NC output as NC program and DXF file.

#### Output NC Program:



Output NC-Program:

Build Program Number from

☐ Roll No.

☒ Part No.

☐ Create Separate Files

Integer/Decimal Places:

Path for Entire File:

Path for Separate Files:

Use the left side of this window to set the parameters for creating the NC programs of the rolls for the CNC lathe by the function [Output Create NC](#). The file is in text format; it contains NC commands according to DIN 66025. The roll contour is described in form of G01/G02/G03 commands and can be transferred to any machine control that is able to read an external program. Afterwards the program can be completed on the shop floor e.g. by rough cutting cycles, tool selection etc.

#### Build Program Number from:

Every NC program starts in the first line with **%0** and a following program number. Select if this program number should be created **from Roll No.** or **from Part No.**. If you check both boxes, Roll No. and Part No will be combined. If you do not check both, the NC programs will be numbered starting with **001**.

If you check Modus **Separate Files** the file names become the program number.

#### Create Separate Files:

The function [Output Create NC](#) creates a NC file, which contains the NC programs of all rolls of the profile project. If you check the box **Create Separate Files**, you get a set of files, each of them contains one NC program only. The file name is the same as the program number.

PROFIL avoids double file names and with it overwriting existing files by attaching **-1**, **-2** etc. to the file name, if necessary. In a first step after calling this function, all existing NC program files that belong to the project are deleted before they are created newly. So the function can be called repeatedly.

#### Decimal Places:

Set the precision of the coordinates in the NC program. The default is 3 decimal places.

#### Path for Entire File/Separate Files:

Enter the path for the NC files, if necessary also with a drive letter, e.g. c:\NC\. You can select the path **for Entire File** and **for Separate Files** separately. After clicking into the input field, the Explorer appears.

#### Output DXF-File:

Use the right side of the window to set the parameters for creating the DXF files of the rolls by function [Drawing -> NC](#), which you want to transfer to a NC programming system. The used file format is:

**DXF format rel. 12:** file format defined by Autodesk. Nearly all NC systems are able to read files in this format.

#### Whole Project in Separate Files:

If not selected, a temporary file is created, which path and name are taken from the input field **Filename for Entire File**. Temporary means: the file is overwritten at every output and should be transferred to the NC programming system afterwards. The file contains either the selected roll or the complete stand, if no roll is selected (by button [Inspect](#)). The roll contour consists of Lines and Arcs.

If **Whole Project in Separate Files** is selected, for all rolls of all stands DXF files will be created that contains one roll only. The roll contour consists of Polylines; the center line and the bore hole line are normal lines. The start point of each roll is 0,0 and all rolls are turned and displayed like bottom rolls.

PROFIL avoids double file names and with it overwriting existing files by attaching **-1**, **-2** etc. to the file name, if necessary. In a first step after calling this function, all existing NC program files that belong to the project are deleted before they are created newly. So the function can be called repeatedly.

#### Build Filename from:

Select, if the filename should be built by using the roll number or the part number or a combination of them. Pay attention that filenames are unambiguous: otherwise the files are overwritten.

#### Turn over Roll 1

In order to make visible the labeling groove, roll 1 can be turned over in the output file.

#### Filename for Entire File:

Enter path and filename for the temporary entire file. After clicking into the input field, the file select window appears.

#### Path for Separate Files:

Enter path for the separate files. The filename is built automatically by using roll number or part number. After clicking into the input field, the Explorer appears.

**Hint:** Select by [View, Show, Spacer Rolls](#) whether the NC output should contain spacer rolls.

### 3.1.2.8.12 Files

Use this function to select by mouse click the file format; the file name is always **PROFIL**. If you want to use another name, another path or another drive, enter the full path into the input field. Pay attention that the path exists and the extension is correct. If you have a network, you can enter a network path too. Also a network path to a UNIX-server is possible. If you double click into the input field, the file select window opens and you can search for the desired path.

#### Input from CAD:

Select the path, the name and the format of the input file, which your [CAD System](#) generates by transferring a profile contour or a roll tool contour to PROFIL. This file is be read by PROFIL, if you use the function [Profile, Read CAD-Contour](#) or [Roll Read CAD-Contour](#). Select among the following possibilities:

**KTR-format:** file format defined by UBECO, see [Contour File \(KTR Format\)](#). Use this format if a macro in your CAD system proceeds the contour tracking and saves the drawing elements in sorted order. Such macros currently are available for the CAD systems PC-DRAFT, AutoCAD until rel. 12, ME10, and CADDAA.. Please contact your dealer.

**DXF-format:** file format defined by Autodesk, used from nearly all CAD systems. The [Contour File \(DXF Format\)](#) can contain unsorted LINE, ARC, POLYLINE, LWPOLYLINE, and ELLIPSE-entities, the contour tracking is done within PROFIL. Use this file format for all CAD systems that do not support ActiveX.

**CAD Drawing Scale:** Enter the drawing scale that has been used in the [CAD System](#) when the file was saved. This allows **PROFIL** to recalculate the imported contour in order to have the proper dimensions in the [Profile List](#).

Settings in **from CAD** are only possible, if in [Options ActiveX](#) the ActiveX Input from CAD is disabled.

#### Output to CAD:

Select the path, the name and the format of the output file of the drawing. This temporary file is be used to transfer the drawing to your [CAD System](#). This file is generated by PROFIL if you call the function [Drawing -> CAD](#). Select among the following possibilities:

**A11-format:** file format defined by ISD for the CAD system PC-DRAFT.

**DXF format:** file format defined by Autodesk. Nearly all CAD systems are able to read files in this format.

**MI-format:** file format defined by Hewlett Packard (CoCreate) for the CAD system ME10 (OneSpace Designer Drafting or PTC Creo Elements/Direct Drafting respectively).

**IGES-format:** file format defined by the Initial Graphics Exchange Specification.

**STEP AP214:** 3D file format in accordance with DIN ISO 10303 "Product data representation and exchange, EXPRESS language".

If you have selected the DXF-format, you can preset:

**With Blocks:** Please select this option, if the objects (passes, rolls,...) should not only organized on layers, but the layers should also be converted to blocks. Hint: In AutoCAD, blocks are handled easier in all (e.g. moving the whole block). For modifying, however, blocks have to be exploded.

If you have selected the STEP format, you can preset:

**Profile built from separate bodies:** Each [Profile Element](#) becomes a separate 3D body. Otherwise the whole profile becomes one entire 3D body. The setting **Separate Bodies** is useful if the [CAD system](#) is not able to represent a profile with self-contact (i.e. one part of the profile touches another part)

If your CAD-System is running under other operating systems, Umlaute and special letters must be converted:

**WINDOWS (Latin 1)** writes all letters unchanged to the output file.

**DOS (PC-8)** converts to the PC-8 symbol set for DOS-based systems.

**UNIX (Roman-8)** converts to the Roman-8 symbol set for UNIX-based systems.

**Sheet Length 3D:** If you transfer 3D models, you can preset the sheet length (in sheet running direction).

**Use layer numbers instead of names:** Select this option if your CAD system does not support layer names, then PROFIL writes layer numbers into the output file. Press the button **Edit** to enter the numbers.

**Edit:**

Item	Layer Number
1. Pass:	150
1. Bottom Roll:	101
1. Top Roll:	201
1. Left Roll:	111
1. Right Roll:	211
Spacers Bottom:	100
Spacers Top:	200
Statics:	149
View Flower 3D:	149

Enter the layer numbers for the 1st pass and the 1st bottom, top and side roll. The next passes and rolls are numbered consecutively. Furthermore you can enter the layer numbers for the spacers, the statics and the 3D drawing.

Settings in **to CAD** are only possible, if in [Options ActiveX](#) the ActiveX Output to CAD is disabled.

**Hint:** Select by [View, Show, Spacer Rolls](#) whether the file output should contain spacer rolls.



### 3.1.2.8.13 ActiveX

ActiveX (in former times called OLE-Automation) is a very useful interface to control programs under WINDOWS with one another and to transfer data to and from other programs. The CAD system is working as ActiveX-Server and PROFIL as ActiveX-Client.

Features of this interface are:

- PROFIL controls CAD, i.e. all operations are made in PROFIL, CAD does not need to be operated.
- No special adaptation of the CAD system is necessary, the standard installation is sufficient. Any user defined adaptation is permissible.
- The drawings transferred to CAD are using layers for every part. All layers can be updated separately, i.e. before transferring the matching layers will be deleted.
- While reading contours from CAD first the entities of all layers and colors are transferred. Disrupting layers and colors can be removed subsequently.



#### Properties of the AutoCAD interface:

Since PROFIL Rel. 2.5 and AutoCAD R14 this interface has been provided to transfer profile and roll tools drawings from PROFIL to AutoCAD and to transfer profile and roll contours from AutoCAD to PROFIL.

- AutoCAD Full Version is needed, in AutoCAD LT the ActiveX interface is not enabled.
- When dimensioned drawings are transferred to AutoCAD, genuine associative AutoCAD dimensioning is created.
- PROFIL needs special line types, textfonts and dim styles. If they exist in AutoCAD, they are used. If not, they will be created automatically.

**Hints:** The ActiveX interface will only work, if AutoCAD accepts external control. It will not work, if a window is open or if AutoCAD is waiting inside a multi-stage command. Finish it or break it by using ESC.

#### Line type Dash Dot for center lines

If not existing, the line type **ACAD\_ISO10W100** (Dash Dot) is been loaded from the file ACAD.LIN.

#### Fixed font text style for the statics table

If not existing, the font MONOTXT.SHX is loaded and assigned to the text style MONO.

#### Dim style for the profile and roll tool dimensioning

If not existing, the dim style **PROFIL** is created with these properties:

- dimension line color, extension line color, text color, text height, number of decimal places = as preset in PROFIL,
- dimension unit = decimal,
- suppression of zeroes = on.

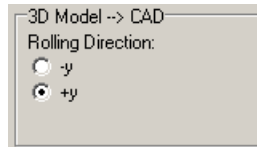
#### Dim style for the roll diameter dimensioning

If not existing, the dim style **RollDiam** is created with the same properties as PROFIL, but:

- extension line 1 = suppressed,
- arrowhead block = none.

If special modifications of the settings are necessary, simply create the style by your own, before you transfer the drawing. If the styles with the above names yet exist, they are not replaced but used.

#### Output, 3D -> CAD



The function [Output 3D Model -> CAD](#) transfers the 3D model of all stands of the whole roll tool set should be transferred to CAD. Select, if the rolling direction should be shown in positive or negative y direction.

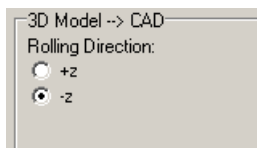


#### Properties of the SolidWorks interface:

Since PROFIL Rel. 4.0 and SolidWorks 2003 this interface has been provided to transfer profile and roll tools drawings from PROFIL to a SolidWorks drawing and to transfer profile and roll contours from a SolidWorks drawing to PROFIL.

- Before you transfer a 2D drawing by [Output Drawing -> CAD](#), select in SolidWorks a document **Drawing**. Otherwise an error messages appears.
- Before you transfer a 3D model by [Output 3D Model -> CAD](#), select in SolidWorks a document **Part**. Otherwise an error messages appears.
- Colors and line types are transferred as preset in PROFIL.
- If dimensioned drawings are transferred, in SolidWorks non-associative dimensioning is created by using the current settings.

#### Output, 3D -> CAD



The function [Output 3D Model -> CAD](#) transfers the 3D model of all stands of the whole roll tool set should be transferred to CAD. Select, if the rolling direction should be shown in positive or negative z direction.



#### Properties of the SolidEdge interface:

Since PROFIL Rel. 4.3 and SolidEdge Rel. ST7 this interface has been provided to transfer profile and roll tools drawings from PROFIL to SolidEdge and to transfer profile and roll contours from SolidEdge to PROFIL, preliminary only for 2D drawings. The extension to 3D models is planned for a later release.

- Before you transfer a 2D drawing by [Output Drawing -> CAD](#), select in SolidEdge a document **Drawing**. Otherwise an error messages appears.
- Colors and line types are transferred as preset in PROFIL.
- Dimensions currently are not transferred.



#### Properties of the BricsCAD interface:

Since PROFIL Rel. 4.3 and BricsCAD Rel. 15 this interface has been provided to transfer profile and roll tools drawings from PROFIL to BricsCAD and to transfer profile and roll contours from BricsCAD to PROFIL, preliminary only for 2D drawings. The extension to 3D models is planned for a later release.

- Before you transfer a drawing with [Output Drawing -> CAD](#), [Profile, Read CAD-Contour](#), or [Roll Read CAD-Contour](#), select in BricsCAD **COM-ACAD-Compatibility** in **Settings, Settings, Program Options, System**.

#### Program ID CAD:



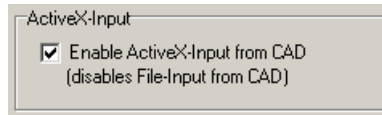
Determine the Program ID of the ActiveX target system. For explanation: The CAD system registers itself to WINDOWS with its Program ID. This information is stored in the system registry of WINDOWS. By this Program ID the connection to PROFIL is made. PROFIL notices which CAD releases are registered and displays them in a drop-down-list, when you click on the arrow symbol.

Open the drop-down-list and select one of the Program ID's:

- **AutoCAD.Application.nn:** Connects to AutoCAD Release nn. Use this Program ID, if you have installed more than one AutoCAD Release and you want to select a special one:
  - AutoCAD 14: AutoCAD.Application.14
  - AutoCAD 2000: AutoCAD.Application.15
  - AutoCAD 2004: AutoCAD.Application.16
  - AutoCAD 2005: AutoCAD.Application.16.1
  - AutoCAD 2006: AutoCAD.Application.16.2
  - AutoCAD 2007: AutoCAD.Application.17
  - AutoCAD 2008: AutoCAD.Application.17.1
  - AutoCAD 2009: AutoCAD.Application.17.2
  - AutoCAD 2010: AutoCAD.Application.18
  - AutoCAD 2011: AutoCAD.Application.18.1
  - AutoCAD 2012: AutoCAD.Application.18.2
  - AutoCAD 2013: AutoCAD.Application.19
  - AutoCAD 2014: AutoCAD.Application.19.1 etc.
- **AutoCAD.Application:** Connects to the topical AutoCAD Release, this is the last installed one.
- **SldWorks.Application.nn:** Connects to SolidWorks release nn. Use this Program ID, if you have installed more than one SolidWorks release and you want to select a special one:
  - SolidWorks 2003: SldWorks.Application.11
  - SolidWorks 2004: SldWorks.Application.12
  - SolidWorks 2005: SldWorks.Application.13
  - SolidWorks 2006: SldWorks.Application.14
  - SolidWorks 2007: SldWorks.Application.15
  - SolidWorks 2008: SldWorks.Application.16
  - SolidWorks 2009: SldWorks.Application.17
  - SolidWorks 2010: SldWorks.Application.18
  - SolidWorks 2011: SldWorks.Application.19
  - SolidWorks 2012: SldWorks.Application.20
  - SolidWorks 2013: SldWorks.Application.21
  - SolidWorks 2014: SldWorks.Application.22
  - SolidWorks 2015: SldWorks.Application.23 etc.
- **SldWorks.Application:** Connects to the current SolidWorks release, this is the last installed one.
- **SolidEdge.Application:** Connects to the current SolidEdge release, this is the last installed one.
- **BricscadApp.AcadApplication.nn:** Connects to BricscadApp release nn. Use this Program ID, if you have installed more than one BricscadApp release and you want to select a special one:
  - BricsCAD Release 15: BricscadApp.AcadApplication 15.0
- **BricscadApp.AcadApplication:** Connects to the current BricscadApp release, this is the last installed one.

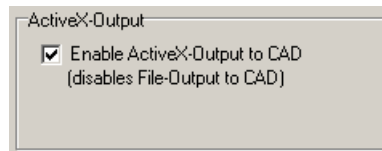
If the drop-down-list is empty, no AutoCAD or SolidWorks or BricsCAD release is installed on your computer.

### Enable ActiveX Input from CAD:



This function enables the ActiveX input and switches the function [Profile, Read CAD-Contour](#) and [Roll Read CAD-Contour](#) and the matching button on the [Button Key Bar](#) to ActiveX. At the same time the file input (see [Options Files](#)) will be disabled.

#### Enable ActiveX Output to CAD:



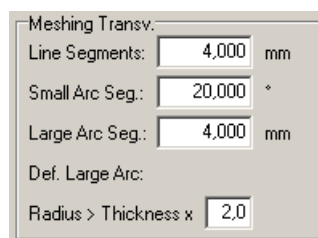
This function enables the ActiveX output and switches the function [Output Drawing -> CAD](#) and the matching button on the [Button Key Bar](#) to ActiveX. At the same time the file output (see [Options Files](#)) will be disabled.

**Hint:** Select by [View, Show, Spacer Rolls](#) whether the ActiveX output should contain spacer rolls.

### 3.1.2.8.14 PSA

Use this window to set-up the [PSA - Profile Stress Analysis](#).

#### Meshing Transv.



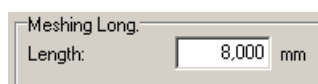
**Line Segments:** select the desired length of the shell elements for [Profile Elements](#) of type "Line". Since the divisor must be a whole number, the effective length will be approximate to the setting.

**Small Arc Seg.:** select the desired length of the shell elements for [Profile Elements](#) of type "Arc". The arc angle is divided. Regards the whole number divisor see above.

**Large Arc Seg.:** select the desired length of the shell elements for [Profile Elements](#) of type "Arc". The arc developed length is divided. Regards the whole number divisor see above.

**Def. Large Arc:** enter the limit between large and small arcs. A large arc is detected when the radius of the centerline of the sheet is larger than the sheet thickness multiplied by the preset factor.

#### Meshing Long.



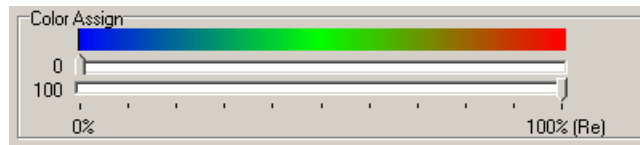
Length: select the desired length of the shell segments in longitudinal direction.

#### Initial Passes Count



Select how many passes should be displayed when the profile stress analysis is called for the first time. Later you can modify the analysis range by using the "From Pass" and "To Pass" selectors.

### Color Assign

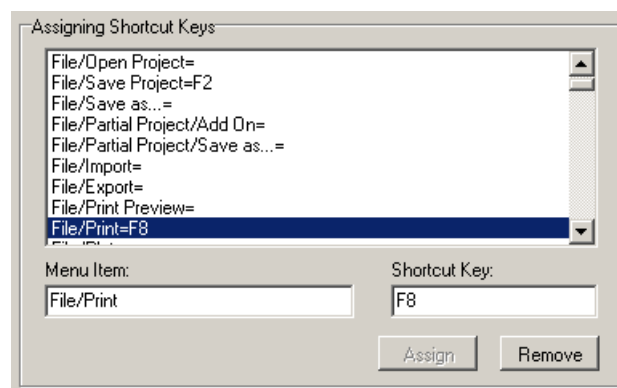


Select by moving the tabs which relative stresses should be assigned to the colors blue and red. The stresses are related to the yield point of the selected material. 0% means no stress and 100% means the yield point stress is reached.

### 3.1.2.8.15 Keyboard

Use this window to assign shortcut keys to any menu item.

#### Assigning Shortcut Keys:



All menu item are listed, behind the equal sign the currently assigned shortcut is displayed. If you select an entry, menu item and shortcut are copied to the fields below for modifying the assignment.

**Menu Item:** This field shows the selected menu item.

**Shortcut Key:** This field shows the shortcut key that is currently assigned to the selected menu item. To enter a new shortcut set the cursor into this field and press the desired shortcut on the keyboard. These shortcuts are possible:

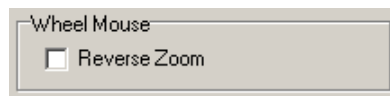
- F2 .. F12 (F1 is reserved for the context sensitive help)
- Shift + any character
- Ctrl + any character
- Alt + any character
- Alt Gr + any character

**Assign:** Assigns the new shortcut key to the selected menu item. If the shortcut is yet assigned to another menu item, a message is displayed.

**Remove:** Removes the shortcut from the selected menu item.

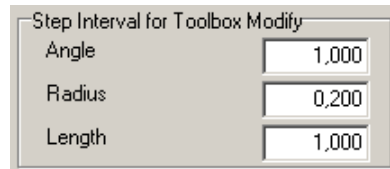
### 3.1.2.8.16 Mouse

#### Wheel Mouse



**Reverse Zoom:** Use this check box to change the zoom direction when spinning the wheel mouse. So you can adapt it to the zoom direction of your CAD system.

#### Step interval



Select the step interval for changing the values in the input fields for **Angle**, **Radius** and **Length** in the profile list and roll tool window by using the **Up/Down** buttons of the [Toolbox Modify](#) (also for the **Pg Up/Dn** keys of the keyboard ).

## 3.1.3 Profile


### 3.1.3.1 Read CAD Contour

Use this function to import a contour, which you have prepared in your CAD system, into PROFIL in order to generate a [Profile list](#). Whether a [Contour File \(KTR format\)](#) or a [Contour File \(DXF format\)](#) is being read, you select in [Options, Files, from CAD](#). If you want to read files with frequent varying file names instead of a temporary file with a constant file name, better use the [Import Function](#).

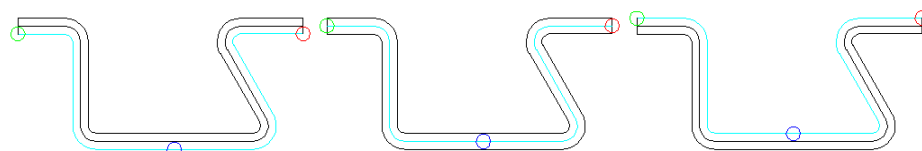
#### Calling the function

Previously select in the [Profile List Window](#) the [Profile Element](#), which should be the start point for inserting the read data. The start point normally is element 1, except you read the 2nd half of an asymmetrical profile, then the start point is the next element behind element **P**.

Call this function optionally by:

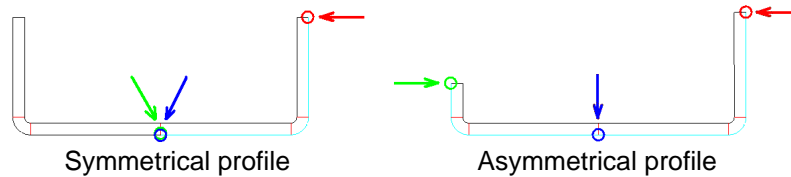
- Main menu: Profile, **Read CAD Contour**.
-  Button **Read CAD Contour** in the [Button Key Bar](#).

If you have enabled the ActiveX Input from CAD in [Options ActiveX](#), the drawing is directly read from CAD and the [Window Read CAD-Contour](#) appears. You define the desired contour inside this window. The same window appears if you have disabled ActiveX and in [Options, Files, from CAD](#), **DXF-Format** is selected.



Scanning the profile bottom side, the sheet center line, and the profile top side

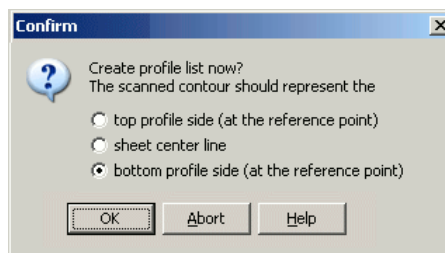
Optionally, scanning the profile bottom side, the sheet center line, or the profile top side (bottom and top related to the reference point) is possible. In case profile legs touch each other and have common lines, it can be valuable to scan the opposite side.



If you design a symmetrical profile (left picture), set both the marker **Reference Point x0/y0** (blue) and the marker **Contour Start Point** (green) to the [Reference Point X0/Y0](#) of the profile. If you design an asymmetrical profile (right picture), set the marker **Contour Start Point** (green) to the left corner of the profile and the marker **Reference Point x0/y0** (blue) to the [Reference Point X0/Y0](#) of the profile. For both set the marker **Contour End Point** (red) to the right corner of the profile. More information: [Window Read CAD-Contour](#).

By checking the color changes to light blue you can find out, if the right contour is tracked between **Contour Start Point** (green) and **Contour End Point** (red). If not, proceed like this:

- **At an intersection the contour tracking turned in the wrong direction** (it prefers the straight on direction): Click onto the path of the right direction behind the intersection.
- **Clicking onto the path of the right direction is ineffectively**: Double lines exist in the CAD drawing or there is no start point of a connection element at an end point of the previous element. Modify the CAD drawing.
- **The blue line ends before the contour end point (red) is reached**: At this position, a faulty connection between two elements exist (gap, overlap, etc.). Correct the CAD drawing.
- **Left and right profile half have common contour elements and cannot be tracked unambiguously**: Move apart the left and right profile half in the CAD system and import only the right half first. Then append a point **P** in the profile list manually, select the next empty line and import the left half separately. Or import the top profile side instead of the bottom side and vice versa.



After confirmation by pressing the **Ok** button the dialog window appears with the question **Create profile list now? The scanned contour should represent the..** Select, if you scanned the top profile side, the sheet center line, or the profile bottom side previously. It is mandatory that this selection is the same as in the previous step in order to enable **PROFIL** to create the sheet thickness on the right side.

### Principle of operation

After confirmation by pressing the **Ok** button in the [Window Read CAD-Contour](#) the profile list is created and the drawing of the profile is shown in the [Drawing Area](#). Now check:

- **Does the imported profile coincide with the CAD drawing?** Use the function [Output Drawing -> CAD](#) to check. Hint: The transferred drawing is on layer **L01**. Or dimension the profile and compare the dimensions with the dimensions in CAD.
- **Do arc elements with radius 0 and very small straight length exist in the profile list?** These are correction elements; PROFIL inserts them if in case of not tangential connections of the elements in the CAD drawing. It is strongly recommended to avoid this, because the correction elements have to be handled and developed during flower design as well. If rolls are derived from such contours, maybe they will cause grooves in the sheet surface. Correct the CAD drawing, empty the profile list, and import again the contour.

By using the switch [Profile, Loaded](#) you can define, whether the contour is read into the columns discharged or loaded of the profile list (normally discharged). The other columns are calculated dependent on the spring back of the selected material. The status of the switch is displayed by a coloured background of the table fields in the profile list window (possibly select **with spring back**

in [Options Profile List](#)).

If the CAD drawing contains partial ellipses, they are converted to arcs approximately (only [DXF input](#) and [ActiveX Input](#)). One arc with small radius is created at the major axis of the ellipse, another arc with large radius at the minor axis. Between both an arc with medium radius is created. There is exact correlation in the quadrant points of the ellipse (intersection points with major and minor axis), i.e. the contour tracking finds connection to other elements.

As a result of the approximation procedure there is small deviation between the ellipse and the arcs in the other parts except the quadrant points. This is why the contour tracking does not find the connection. We recommend to proceed like this: first, transfer the ellipse to PROFIL and afterwards transfer the created arcs back again to CAD. Then create the connection elements in CAD and transfer the whole drawing to PROFIL.

#### Hints:

Further function for creating the profile list are:

- The [Toolbox Profile Design](#) for simple profiles like U, C, Hat, etc.
- The [Numerical Method](#) for simple, rectangular profiles.

### 3.1.3.2 Empty

Use this function to empty the profile list, i.e to delete all [Profile Elements](#).

#### Calling the function

Before you call this function, select the pass whose profile list should be emptied.

Call this function optionally by:

- Main menu: **Profile, Empty**.
- Context menu (right mouse button click on any profile element of the desired pass in the [Drawing Area](#)): **Profile, Empty**.

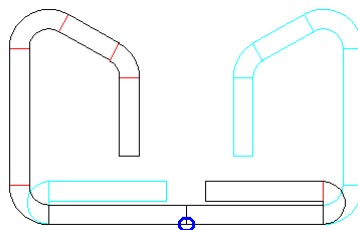
#### Principle of operation

All profile elements or the profile list are deleted. An empty profile list will remain.

#### Hints:

- To recall data in case of an error call function [Open Project](#) and answer **Cancel** to the question **Save data?**. Reopen the same profile project.
- To remove a whole profile list, use the function [Profile, Remove](#).

### 3.1.3.3 Mirror



Use this function to mirror a unsymmetrical profile list at the [Reference Point X0/Y0](#).

#### Calling the function

Before you call this function, select the pass whose profile list should be mirrored.

Call this function optionally by:

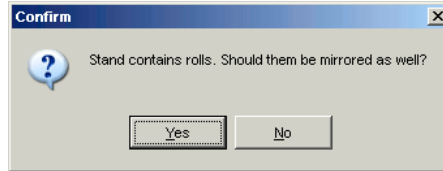
- Main menu: **Profile, Mirror**.
- Context menu (right mouse button click on any profile element of the desired pass in the



[Drawing Area](#)): **Profile, Mirror.**

### Principle of operation

The profile is mirrored at the [Reference Point X0/Y0](#). The left leg is mirrored to the right and the right is mirrored to the left. If the profile list is symmetrical, mirroring does not make sense and a message is shown.



If the stand contains rolls, they can be mirrored as well. In this case the user is asked to agree. The width of top and bottom rolls are mirrored at the [Roll Reference Point](#). Side rolls are exchanged and the width is mirrored at the roll reference point in addition.

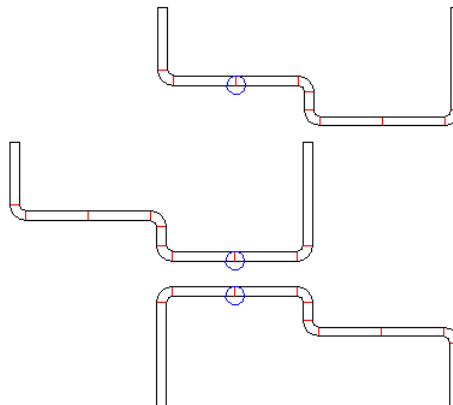
### Hints:

The function does not modify

- The start [Direction](#) of the profile list. If necessary modify it manually.
- The [Machine Data](#). If they are different in case of side rolls, the roll may not touch the profile anymore. Manual correction is necessary.

### 3.1.3.4 Modify Start Element

The first profile element of the [Profile List](#) in the [Profile List Window](#) starts at the [Reference Point X0/Y0](#). Define the reference point in the [Window Read CAD-Contour](#) by selecting the marker **Reference Point x0/y0** (blue). The reference point normally should be nearly the center of the profile's web. Because both legs turn around the reference point during forming, it is recommended to put it in such a position in order nearly to have equal leg heights on both sides, also in case of a unsymmetrical profile. Thus the [Stress of Band Edge](#) is distributed equally to both sides. This also can be optimized by selecting a suitable [Direction](#).



### Calling the function

Call this function optionally by:

- Main menu: **Profile, Start Element.**
- Context menu (right mouse button click on a profile element in the [Drawing Area](#)): **Profile, Modify Start Element.**

After importing the profile cross-section, you can use this function to modify the profile element order and to select which profile element should be the first in the profile list. Click on any drawing element of the current pass. The nearest end point is caught.

It is recommended to apply this function to the final profile (L01) in case the succeeding passes (flower pattern) do not exist. Afterwards create the flower pattern for the profile with the modified

start element.

### Principle of operation

If you selected a point on the **sheet's bottom side** (more precisely: a point on the same side as the reference point), the profile list will be re-sorted accordingly and turned if necessary. In doing so, the co-ordinates of the [Reference Point X0/Y0](#) and also the [Direction](#) keep unchanged. Afterwards the profile with the selected start point will be the first in the profile list. Thus the drawing of the profile is moved. The picture shows an example wherein the function has been applied to the top profile. The center profile shows the result.

If you selected a point on the **sheet's top side** (more precisely: a point on the opposite side of the reference point), the whole profile will be turned by 180 degree first and afterwards the profile list will be re-sorted just like above. If the profile opening was on top previously, it will be now on bottom and vice versa. In the picture the function has been applied to the center profile. The bottom profile shows the result.

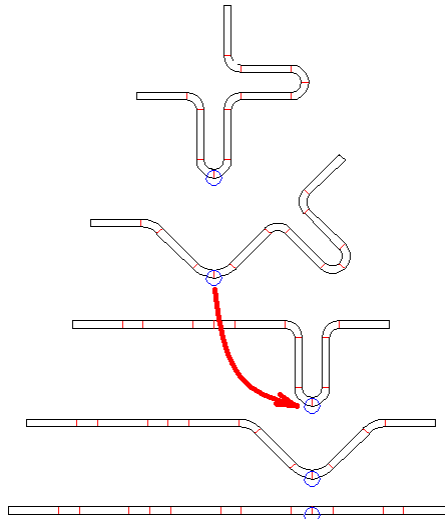
If you modify the start element of a symmetrical profile list (with **Symmetrical Point PS**), the list becomes unsymmetrical (with **Point P**).

### Hints:

- If the start element should be a part of an existing profile element, you can [Split](#) it previously.  
Example: A V-profile should be formed upright.
- If you want to move the reference point (develop point) starting from a certain pass of the flower pattern, use the function [Modify Develop Point](#).

### 3.1.3.5 Modify Develop Point

The [Reference Point X0/Y0](#) of a profile is identical to the develop point, i. e. the point the profile's legs turn around during developing the flower pattern.



Some applications require to redefine the develop point in a certain pass in order to turn the profile's legs around a different point in the next passes (see the example in the picture).

### Calling the function

Use this function in the desired pass during creation of the flower pattern. The next passes (against the sheet running direction) should not exist. If they already exist, they should be deleted.

Call this function optionally by:

- Main menu: **Profile, Develop Point**.
- Context menu (right mouse button click on a profile element in the [Drawing Area](#)): **Profile, Modify Develop Point**.

Click on any drawing element of the current pass, appropriately on the bottom side of the profile. The nearest end point is caught.

### Principle of operation

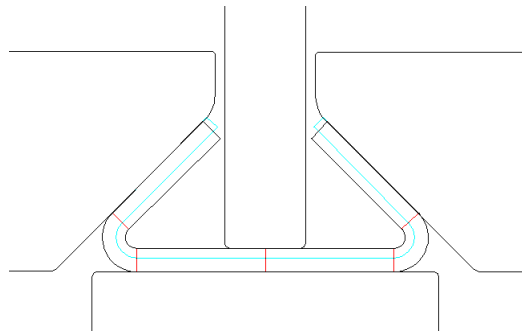
The profile list will be re-sorted (similar to the function [Modify Start Element](#)), additionally the [Reference Point X0/Y0](#) and also the [Direction](#) will be adapted in order that the position of profile in the stand is not changed. Unless you have selected a point on the sheet's top side, then the profile is turned. When you continue developing the [Flower Pattern](#), the unbending is related to the new develop point.

If you modify the start element of a symmetrical profile list (with **Symmetrical Point PS**), the list becomes unsymmetrical (with **Point P**).

### Hints:

- If needed, you can [Split](#) a profile element previously, if the new develop point should be on the split point.
- If you get an inappropriate start element after importing a CAD drawing, use the function [Modify Start Element](#) for correction.

### 3.1.3.6 Modify Sheet Thickness

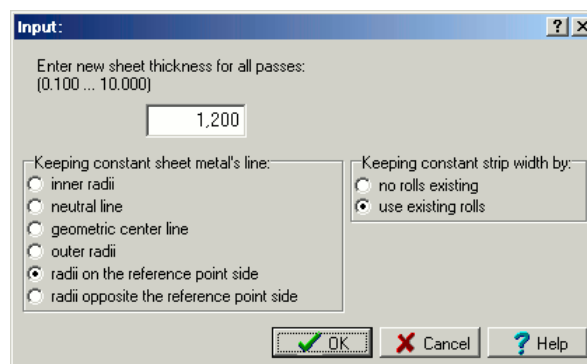


This function enables to modify the sheet thickness in the design state of the final profile, during flower pattern design, or in a finished project. This may be necessary if the roll tools should be designed for various sheet thicknesses and if the position of the sheet for the minimal and maximal thickness should be checked. Another application is: A running roll forming machine should be refitted for another sheet thickness. The task is to check if this is possible with the existing rolls plus additional spacers or if in part new rolls have to be prepared.

### Calling the function

Call this function optionally by:

- Main menu: **Profile, Modify Sheet Thickness.**
- Context menu (right mouse button click on a profile element in the [Drawing Area](#)): **Profile, Modify Sheet Thickness.**
- [Project Window](#): Enter the thickness in the input field [Thickness](#).



After calling this function, a dialog window is opened. Enter the desired new sheet thickness and select which **sheet metal's line should be kept constant** during this operation:

- **Inner radii:** The outer radii are changed dependent on the new sheet thickness, as well as the strip width, because the straight lengths of the arc elements are changed.
- **Neutral line:** The neutral line is the virtual line whose length is not changed during bending. It is moved a little bit from the geometric center line toward the inside. How much, is calculated by the preset [Calculation Method](#). Inner and outer radii are changed dependent on the new sheet thickness. The strip width (sum of all straight lengths) keeps constant.
- **Geometric center line:** This is the virtual line in the center of the sheet. Inner and outer radii are changed dependent on the new sheet thickness, as well as the straight lengths and the strip width.
- **Outer radii:** Inner radii are changed dependent on the new sheet thickness, as well as the strip width, because the straight lengths are changed.
- **Radii on the reference point side:** The [Reference Point](#) always is on the sheet metal's bottom side near the profile's web center point. By selecting this item the (inner or outer) radii keep constant that are on the same side as the reference point.
- **Radii opposite the reference point side:** By selecting this item the (inner or outer) radii keep constant that are on the opposite side as the reference point.

If the selection **Keeping constant sheet metal's line** is applied on all passes, different strip widths will result in the passes inevitably. In order to avoid this, select between two different methods for **Keeping constant strip width by:**

**No rolls existing:** Check this item, if no rolls exist or existing rolls should not be considered for the profile with new sheet thickness. The selection which line should keep constant, is applied on the final pass L01 only. All other passes are created newly by presetting the arc angles and arc bending methods from the origin flower pattern. Thus all passes get the same strip width, however, the radii in passes L02..Lnn will differ from the constant line setting above. If rolls exist, they will not fit to the new profile contour. This is why this setting makes sense only during flower design, if rolls not yet exist.

**Use existing rolls:** Select this item, if you want to check if an existing or already designed set of rolls will work with a modified sheet thickness. The selection which line should keep constant is applied on all passes. The inevitably resulting different strip width is corrected by lengthening or shortening the profile elements at the band edges. Thus a flower pattern is obtained with different allocation of the straight lengths to the profile elements. However, because profile and roll contours fit together or are parallel together, the displacement of the band edges is an indicator if the roll set is good for the new sheet thickness or if it has to be adapted.

The occurrence of a negative inner radii is not checked. Please check by yourself after proceeding the modification, if a negative radius is shown in the [Drawing Area](#) or in the column **Radius** of the [Profile List Window](#).

#### Principle of operation, selection "No rolls existing"

The settings in the dialog window have influence on the final pass L01 only. In order to have equal strip widths in all passes, the following method is applied on the passes L02 .. Lnn: In the background (i.e. not shown on the screen), the [Development Table](#) is called with the function **Development Table, Create from current project**. Then the sheet thickness is changed in the final pass L01 dependent on the selected "keeping constant" method. Afterwards the function **Development Table, Apply and Create Flower Pattern** is called. This means: The profile lists L02..Lnn are emptied (in which the rolls are preserved) and all passes L02..Lnn are created newly for the new sheet thickness from the development table. In doing so all arc angles and all arc bending methods are taken into account from the origin flower pattern in order to create the new flower pattern for the new sheet thickness. As a result the strip width in all passes is the same and all rolls from the origin project are preserved as designed for the old sheet thickness.

The described steps also can be proceeded manually by calling the development table from the profile menu and applying it to the project. This enables the user to make more modifications in the development table in addition to the thickness change.

### Principle of operation, selection "Use existing rolls"

The settings in the dialog window have influence on all passes L01..Lnn. As a result, the sheet thickness in all passes is unequal. In order to get equal strip widths in all passes, the following method is applied on the passes L02 .. Lnn: For each pass, the difference between the strip width and the one of pass L01 is determined. Then the profile elements at the band edge are corrected in the following way:

- Is the profile element at the band edge a line **L**, it is lengthened or shortened.
- Is the profile element at the band edge an arc **A**, the angle is increased or decreased by keeping the radius constant.
- Is the profile list symmetrical (i.e. it contains a symmetrical point **PS**), the band edges are modified symmetrically.
- Is the profile list asymmetrical (i.e. it contains a point **P**), the partition of the strip width modification to the left and right band edge can be preset in [Options, Profile List](#), topic **Modify Strip Width/Sheet Thickness**.

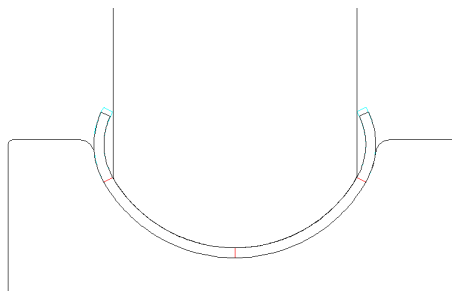
The method is useful in combination of the selection **Constant Inner Radii** or **Constant Outer Radii** for checking if existing rolls are suited for various sheet thickness. Another application is to learn how to design rolls for a sheet thickness interval. Pay attention by using this method: The profile elements in the flower pattern do not have the same developed length anymore. The possible result may be that a part of an arc is formed and in a later stand pressed to flat again.

After changing the sheet thickness the new thickness is shown in the field [Thickness](#) of the [Project Window](#).

### Restrictions - Limits of the operating method

- The operation method **No Rolls Existing** is applicable only, if no manual modification is made in the passes, e.g. inserting correction elements. These will be lost during changing the sheet thickness by using the development table.
- Both operation methods cannot be applied for shaped or round tubes in calibration stands C02 .. Cnn and in the subsequent welding station, because a closed tube is then not closed anymore. It can be used, however, if only the final shaped tube exists in the calibrating stand C01 or a round tube exists in the welding station F01.

#### 3.1.3.7 Modify Strip Width



After defining the desired [profile cross-section](#) the [strip width](#) of the flat sheet is defined, too. Sometimes the question comes up, whether an already existing strip with a differing width can be used for forming this profile or whether a strip with a standard width can be used. Another important application is: If the [roll tools](#) are already designed, the designer wants to know how a strip behaves between the rolls, in case the width allowances are at the minimal and maximal limits. For this purpose the function **Modify Strip Width** is provided. It shortens or lengthens the [profile elements](#) at the left and right band edge in order to get the desired strip width.

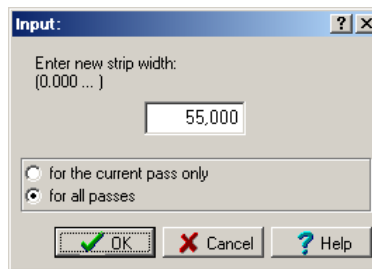
### Calling the function

Call this function optionally by:

- Main menu: **Profile, Modify Strip Width**.
- Context menu (right mouse button click on a profile element in the [Drawing Area](#)): **Profile,**

### Modify Strip Width.

- [Profile List Window](#): Enter the new width in the input field [Strip Width](#).



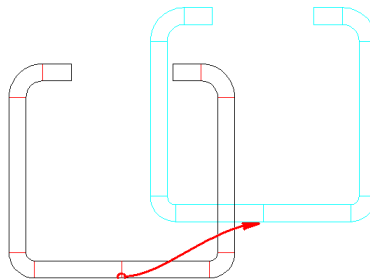
After calling this function, a dialog window is opened. Enter the desired new strip width and select, whether the new strip width should be effective on the current pass only or on all passes of the whole flower pattern.

### Principle of operation

The function **Modify Strip Width** enables to modify the strip width for the current pass or all passes. This is achieved by lengthening or shortening the profile elements at both band edges: Then the profile elements at the band edge are corrected in the following way:

- Is the profile element at the band edge a line **L**, it is lengthened or shortened.
- Is the profile element at the band edge an arc **A**, the angle is increased or decreased by keeping the radius constant.
- Is the profile list symmetrical (i.e. it contains a symmetrical point **PS**), the band edges are modified symmetrically.
- Is the profile list asymmetrical (i.e. it contains a point **P**), the partition of the strip width modification to the left and right band edge can be preset in [Options, Profile List](#), topic **Modify Strip Width/Sheet Thickness**.

### 3.1.3.8 Modify Reference Point



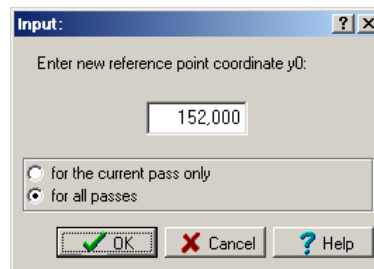
The [Reference Point](#) belongs to the [Profile List](#) data and defines the start point of the profile in the xy plane. The reference point normally should be at or near the center of the profile's web on the sheet metal's bottom side. Usually it has the same position as the [Roll Reference Point](#) in order to get roll diameters whose ratio is equal to the [Machine Transmission Ratio](#).

The function **Modify Reference Point** enables to modify the x and y coordinates of the reference point either for the current pass or for all passes.

### Calling the function

Call this function optionally by:

- Main menu: **Profile, Modify Reference Point, x0 or y0.**
- Context menu (right mouse button click on a profile element in the [Drawing Area](#)): **Profile, Modify Reference Point, x0 or y0.**
- [Profile List Window](#): Enter the value in the input field [x0/y0](#).



After calling this function, a dialog window is opened. Enter the desired new reference point coordinate and select, whether the modification should be effective on the current pass only or on all passes of the whole flower pattern.

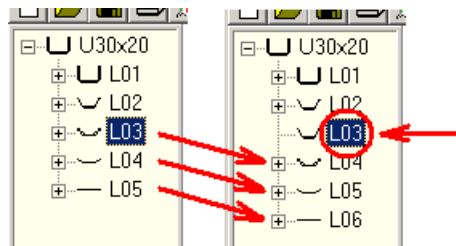
### Principle of operation

The function **Modify Reference Point** enables to modify the x and y coordinate of the reference point mutually independently. Afterwards the drawing is moved to the new position.

### Hints:

- In case the drawing of the profile has disappeared from the view port of the [Drawing Area](#), you can make it visible again by the button **Fit** of the [Navigator](#).
- Select [View, Flower Nested](#) in order to check all passes for proper position.

### 3.1.3.9 Insert



Use this function to insert a new [Profile List](#) at the current position. This means to insert a stand.

### Calling the function

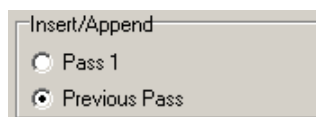
Before you call this function, select the pass at whose position a new profile list should be inserted. Call this function optionally by:

- Main menu: **Profile, Insert**.
- Context menu (right mouse button click on any profile element of the desired pass in the [Drawing Area](#)): **Profile, Insert**.

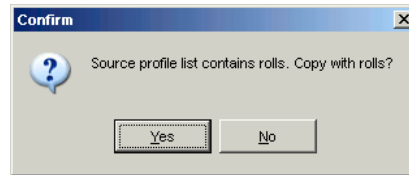
### Principle of operation

A new profile list is inserted. The list numbers of the current and all following lists will be increased by 1. The content of an existing profile list is copied to the new one.

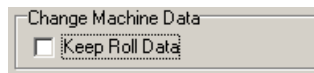
### Setup



Select in [Options Profile List](#), **Insert/Append** if the contents of **Pass 1** or of the **Previous Pass** will be copied into the inserted list.



If the pass that is copied already contains rolls, you are asked **Source profile list contains rolls. Copy with rolls?** Please consider that rolls are renumbered when copied into another stand (dependent on the number keys in [Options Rolls](#)) and that rolls are modified dependent on the data of the shafts of the new stand (in the [Machine Window](#)).

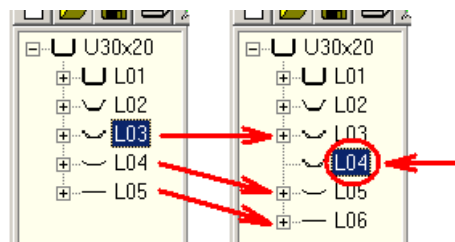


Furthermore the setting **Keep Roll Data** in [Options Rolls](#) determines how the rolls are treated by changed machine data.

#### Hints:

- It is not possible to insert a list before the first list. Use [Profile, Append](#) instead.
- To remove a list, use the function [Profile, Remove](#).

### 3.1.3.10 Append




Use this function to append a [Profile List](#) behind the current one.

#### Calling the function

Before you call this function, select the pass behind whose position a new profile list should be appended.

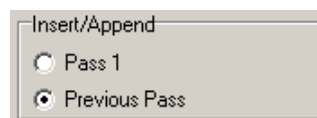
Call this function optionally by:

- Main menu: **Profile, Append**.
- Context menu (right mouse button click on any profile element of the desired pass in the [Drawing Area](#)): **Profile, Append**.
-  Button **Append Profile List** in the [Button Key Bar](#).

#### Principle of operation

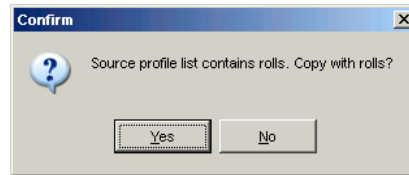
A new profile list is appended. The list numbers of all following lists will be increased by 1. The content of an existing profile list is copied to the new one.

#### Setup

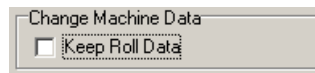


Select in [Options Profile List](#), **Insert/Append** if the contents of pass 1 or of the previous pass will be copied into the appended list.





If the pass to be copied contains rolls yet, you are asked **Source profile list contains rolls. Copy with rolls?** Please consider that rolls are renumbered when copied into another stand (dependent on the number keys in [Options Rolls](#)) and that rolls are modified dependent on the data of the shafts of the new stand (in the [Machine Window](#)).

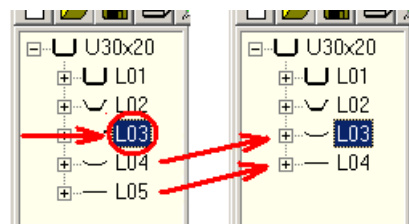


Furthermore the setting **Keep Roll Data** in [Options Rolls](#) determines how the rolls are treated by changed machine data.

#### Hints:

- To insert a list before the current one, use [Profile, Insert](#).
- To remove a list, use the function [Profile, Remove](#).

### 3.1.3.11 Remove



Use this function to remove a [Profile List](#) from the set of lists.

#### Calling the function

Before you call this function, select the pass that should be removed.

Call this function optionally by:

- Main menu: **Profile, Remove**.
- Context menu (right mouse button click on any profile element of the desired pass in the [Drawing Area](#)): **Profile, Remove**.

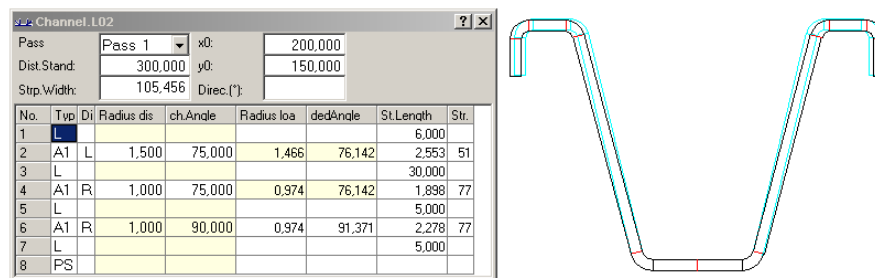
#### Principle of operation

The profile list is removed irretrievably. The numbers of all following lists will be decreased by 1.

#### Hints:

- To insert or append a profile list, use the functions [Profile, Insert](#) and [Profile, Append](#).
- To delete all profile elements without removing the profile list, use the function [Profile, Empty](#).

## 3.1.3.12 Loaded



The fields with the yellow background in the profile list window (left) cause if in the drawing (right) the discharged or loaded state (blue) is shown.

Whether the pass is drawn discharged or loaded, you define by setting this toggle switch. The result of the switching is dependent on the selection before:

**Discharged state:** Cross-section of the profile without consideration of the spring-back, also the cross-section of the final product with the desired dimensions.

**Loaded state:** Cross-section of the profile with over bent arcs in order to compensate the spring-back. After leaving the last stand, the profile springs back and has the desired cross-section of the discharged state afterwards. Each arc has to be over bent at least in one stand. In most cases this has to be considered in the last (final) stand. In some cases the arc has to be over bent in a previous stand, e.g. for complicated profiles, if the rolls of the last stand cannot attack the arc.

### Calling the function

Before you call this function, select the arc that should be toggled. Or deselect all (function [Inspect](#)) in order to toggle all profile elements.

Call this function optionally by:

- Main menu: **Profile, Loaded.**
- Context menu (right mouse button click on the arc that should be toggled in the [Drawing Area](#)): **Profile, Loaded.**

### Principle of operation

If a single arc of a pass is selected (by clicking of the arc in the [Drawing Area](#) or in the [Profile Window](#)) (thus, the arc is drawn in the mark color), only the loading of this arc is switched on and off. If no arc is selected (by button [Inspect](#) or by the right mouse button), the loading of all arcs of the current profile list is switched on and off. The status of the switch is displayed by a yellow coloured background of the table fields in the profile list window (to watch this, select **with spring back** in [Options Profile List](#)).

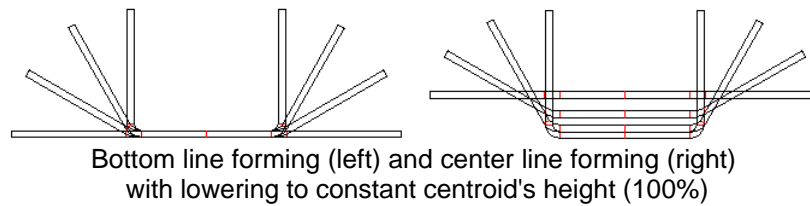
The selected status of the switch is saved into the file of the [Profile Project](#).

The pass that is displayed in the drawing of the [Roll Tools](#) and the calculation of the [Stress of Band Edge](#) is also dependent on the status of the switch.

### Hints:

- The [Statics](#) is calculated for the discharged state only.
- Furthermore you can select, if you want to design a profile pass or the roll tools discharged or loaded. Set the switch to the desired status before you use the functions [Profile, Read CAD-Contour](#) or [Roll Scan Profile Drawing](#).

### 3.1.3.13 Center Line Forming



**Center line forming** also is called **Downhill Forming** or **Natural Forming**.

In order to decrease the way of the band edge and with it to decrease the stress of band edge it is recommended to lower the profile web from pass to pass. This means the flat strip enters into the machine in a higher position than the web of the final profile leaves the machine. This can be done by modifying the Y0 value of the profile [Reference Point](#). In order to keep the circumference speed of both top and bottom roll constant, adjustable top and bottom shafts are needed (not possible by all machines).

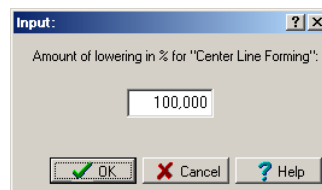
**Bottom Line Forming** means in contrast to **Center Line Forming**: The coming flat strip has the same height as the leaving web of the final profile.

In principle, any amount of lowering is possible. Lowering to a constant height of the [Centroid](#) is often used. This enables regular distribution to the whole profile cross section.

#### Calling the function

Call this function optionally by:

- Main menu: **Profile, Center Line Forming**.
- Context menu (right mouse button click on any profile element of any pass in the [Drawing Area](#)): **Profile, Center Line Forming**.



After calling this function, a dialog window asks **Amount of lowering in % for Center Line Forming?** Enter:

- 100%, if you want all centroids to have the same height.
- 0%, if you want no lowering or if you want to reset to **Bottom Line Forming**.
- A negative value, if you want the strip moving upwards (for special application, e.g. if the profile is open at the bottom side).
- A value greater than 100%, if extreme bottom line forming is needed (for special application, e.g. if the distance between stands is extreme small).

#### Principle of operation

The Y coordinates of the [Reference Point](#) of all passes except L01 are modified dependant on the entered percentage. Open the window [Stress of Band Edge](#) or toggle to [PSA - Profile Stress Analysis](#) to view the effect of lowering to the profile stress.

#### Hints:

- In order to undo **Center Line Forming**, call the same function again and enter the amount of lowering 0%.
- If the automatic assignment of lowering does not fit your needs, you can set the Y0 coordinate of the [Reference Point](#) in the [Profile List Window](#) manually.

### 3.1.3.14 Development Table

%	1	L	2	A1	3	L	4	A1	5	L	6	A1	7	L	8	P	9	L	10	A1	11	L	12	A1	13	L	14	A1	15	L
1			100.000				100.000				100.000								100.000				100.000							
2			100.000				100.000				66.667								100.000				100.000					66.667		
3			100.000				100.000				33.333								100.000				100.000					33.333		
4			100.000				100.000												100.000				100.000							
5			83.333				88.889												83.333				88.889							
6			66.667				77.778												66.667				77.778							
7			50.000				61.111												50.000				61.111							
8			33.333				44.444												33.333				44.444							
9			16.667				27.778												16.667				27.778							
10			5.556				11.111												5.556				11.111							
11																														

The development table is the representation of the bending angle sequence of the flower pattern in tabular form. It is created automatically from an existing flower pattern or can be created and edited manually. It can be saved to a file and can be reopened again. The development table can be used for the [Automatic Flower Creation](#) for a similar profile.

#### Calling the function

Call this function optionally by:

- Main menu: **Profile, Development Table.**
- Context menu (right mouse button click on any profile element of any pass in the [Drawing Area](#)): **Profile, Development Table.**

After calling the function the window **Development Table** appears with an empty table first. The window contains:

**Editor for your remarks (on top):** Add your own remarks or edit and modify the automatic created remarks from the source profile. Example: kind of profile the development table applies to.

**Bending angle table and bending method table (at bottom):** The header contains the profile element numbers and types of the currently opened profile list L01, this means each column represents a profile element. The left column shows the pass number (1=L01 etc.), this means the table rows represent the forming passes (stands). The table fields show the [Angles](#) in degree or percent (toggle by **Development Table, View Angle in %**) or the [Bending Methods](#) (toggle by **Development Table, View Bending Method**).

#### Menu Functions

**File New, Open, Save, Save as...:** Development Tables are saved to a file with the file extension **.dtf**. By using these functions you can manage the existing development tables in files and reuse them later for new projects. If a file is opened that was created from another project, possibly bending angles appear in a column whose header entry is not an arc. This is why the structure of the current project (shown in the header row) does not fit to the structure of the development table from the file (shown by the bending angle sequence in the table). In this case it is necessary to adapt the structure of the table by using the functions **Development Table, Column**.

**Development Table, Create from current project:** The [Angle](#) sequence of the currently opened project is inserted to the table, the angles are in degree first. Furthermore the [Bending Methods](#) are inserted (toggle by **Development Table, View Bending Method**). Thus, the table length is adapted, the row count is the number of profile lists (passes, stands). Name and path of the project file is inserted to the remarks editor, furthermore customer, designation and drawing number.

**Development Table, Apply and Create Flower Pattern:** From the development table the [Automatic Flower](#) is created for the currently opened Profile (L01), including:

- The target profile list structure of the new profile must be the same as the source structure, this means the angles of the development table must be in the angle columns (A1..A4 in the table header). If it isn't, you can adapt the table by using the function **Development Table, Columns**.
- If an angle entry is below a line header entry (L), the line is converted to an arc.
- If an angle entry is below a point header entry (P or PS), it is ignored.

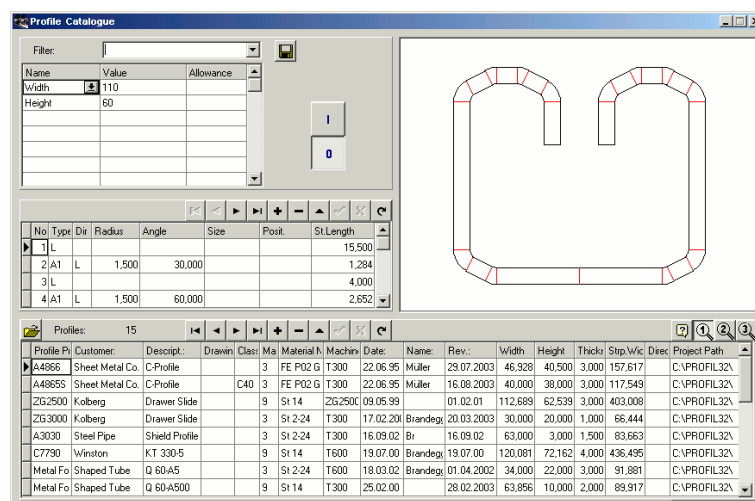
- If the development table has more columns than the target profile list's count of profile elements (if empty header entries exist), the surplus columns are ignored.
- The existing target profile list L01 keeps unchanged, this means the first development table row is ineffective.
- If profile lists L02..Lnn exist, they are emptied.
- The profile list L01 is copied and appended regards the count of rows of the development table and the table's angles and bending methods are set to the profile lists. Angles in degree are set directly and angles in percent are transformed to degree related on the final angle in L01 of the target profile list. Appending profile list works dependent on the switch Insert/Append in Options Profile List.

**Development Table, Bending Method:** The development table is able to show the [Angles](#) or [Bending Methods](#) as desired. Toggle between both by using this function. During automatic flower creation bending method entries are needed only, if the method should deviate from the method in the target profile list L01 or if a line should be converted to an arc. Therefore, if the development table does not contain a bending method, the bending method from L01 is taken. If there is no bending method in L01, A1 is taken.

**Development Table, Angel in %:** To use the development table most universally also if the angles of the target profile list differ from the source, it is recommended to convert the angles from degree to percent related to the final angle. The top left table field shows the setting, furthermore the setting is saved to the development table file. Before applying the development table to a new profile the angles are converted to degree again, this time related to the final angle in the target profile list.

**Development Table, Row/Column:** By using these functions you can adapt and modify the development table for best applying to a new profile. By using **Insert/Append** a new empty row/column is created. **Paste from clipboard** replaces the entries. All functions **Column..** only modify the content of a development table, not the header, which is created from the profile element types of the current project. Thus you can adapt the development table from another project to the element type sequence of the current project, in case the angles are not in an appropriate arc column.

### 3.1.3.15 Profile Catalogue



Only with option Database.


While dealing with an inquiry the designer needs calculation data from similar profiles that are produced in the past. Designing new roll form projects should consider experience from earlier projects. The profile catalogue gives a quick overview of all produced roll formed parts. Any filters

can be defined, named, saved and reloaded by the user for selecting the desired profiles.

The profile catalogue contains the [Profile List](#) of the final pass L01 each, and the project data of the [Profile Project](#). The belonging drawing of the final pass is quickly generated and displayed while browsing through the database. A special button in the profile window enables quick access to the project file. 3 different user defined views with desired columns are displayed by key-stroke.

### Calling the function

Before you call this function, select one of the views [View Pass](#), [View Statics](#), [View Flower Nested](#), [View Flower Separated](#), [View Flower 3D](#). This prepares opening the profile catalogue for the button in the top button bar. Call this function optionally by:

- Main menu: **Profile, Profile Catalogue**.
-  Button **Profile Catalogue** in the [Button Key Bar](#).

### Profile Catalogue Open

After calling this function, the window **Profile Catalogue** appears, which contains 4 areas:

- [Profile Table](#) (lower area): This is the real profile catalogue, each row displays the project data of one profile.
- [Profile Element Table](#) (in the middle, left). This area displays the profile list L01 (final pass) data of the profile selected in the profile table.
- [Drawing Area](#) (top, on the right): This area displays the drawing of the profile selected in the profile table.
- [Filter](#) (top, on the left): Any filters can be defined for reducing the amount of displayed profiles.

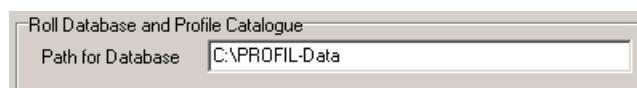
### Profile Catalogue Save

Use this function to store the final pass of one or more projects into the profile catalogue.

- **Current Project:** The currently opened project is stored.
- **All Projects of a Path:** The path selection window appears and all projects of the selected path are stored.

If the window Profile Catalogue is not yet open, it will be opened after calling one of these functions.

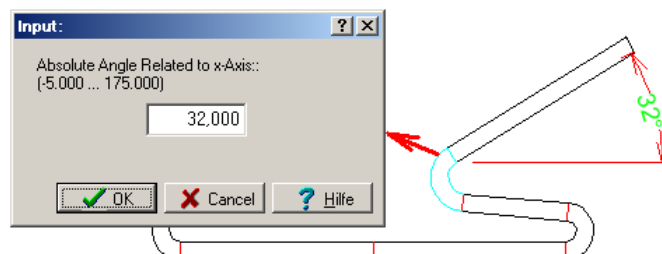
### Setup



Set the path to the database in [Options Database](#).

## 3.1.3.16 Element

### 3.1.3.16.1 Abs. Angle



The absolute angle is the angle of any [Profile Element](#) related to the x-axis. Use this function to display the absolute angle of a profile element. Furthermore you can bend the profile by modifying the absolute angle.

### Application spectrum

- Checking if a certain profile flank fits to the requirement, e.g. a surface should be precisely horizontal.
- For the flower pattern standard rolls with given flank angles should be used.

### Calling the function

Before you call this function, select the profile element in the column **angle discharged** or **angle loaded** inside the [Profile List Window](#).

Call this function optionally by:

- Main menu: **Profile, Element, Abs. Angle**.
- Context menu (right mouse button click on the profile element in the [Drawing Area](#)): **Profile, Element, Abs. Angle**.

### Principle of operation

If you have selected a profile element **Line**, the absolute angle is displayed only without edit feature.

If you have selected a profile element type **Arc**, you can modify the absolute angle between given limits. Valid limits are corresponding to a relative arc angle from 0..180°. The displayed absolute angle is always the angle at the exit of the arc.

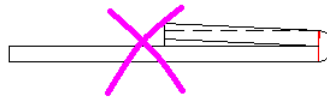
#### Remark:

The column **Angle** of the [Profile List](#) is the relative bending angle related to the previous profile list entity. In order to determine the absolute bending angle, the relative bending angles up to this point must be added regarding its sign. This simplifies the function **Absolute Angle**.

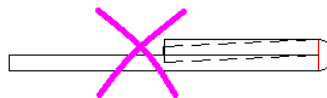
#### 3.1.3.16.2 Open Fold



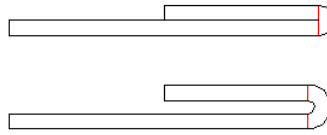
**Problem:** The band edge should be folded to a hem by 180 degree with inner radius zero. The sheet surfaces should touch closely and should not spring up.



**No solution:** If the fold is roll formed until the sheet surfaces touch, the spring back of the material causes the fold to spring up. This is not the desired result.

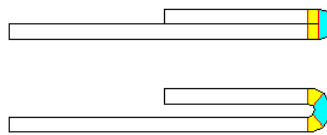


**Not possible:** Usually a bending zone is over bent in order to compensate the spring back. If this is tried with the 180 degree bend, the sheet would penetrate itself. This is not possible.



**Solution:** In the previous pass, the fold is opened to a small inner radius by keeping the angle of 180 degree. Because the developed length of the arc increases by this operation, the adjoining line segments have to be decreased by half of the arc lengthening each in order to keep constant the whole sheet width. Afterwards the new arc is developed as usual to the flat sheet (in design direction, i.e. against the sheet running direction). In sheet running direction's view, an open fold is roll formed first. Then it is compressed by a roll pair.

**Why doesn't this fold spring up?**

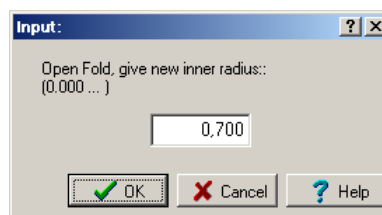


**Explanation:** Imagine, the new arc (with the larger inner radius) consists of three parts. The center part has exactly the developed length of the final 180 degree bend with inner radius zero. The two small parts have the developed length that is subtracted from the adjoining line segments. When the fold is compressed this happens: the large arc is bent down (to inner radius zero) and springs up after leaving the roll forming stand. The two small arcs, however, are bent up (to flat sheet) and spring back to the closed state. So the spring back of the small arcs have an affect in the opposite direction to the spring back of the large arc. By choosing the correct inner radius it is possible that the sum of all spring forces is zero and the 180 degree fold keeps closed. In practice, an inner radius in the range 0.4-0.8 x sheet thickness is used. A larger inner radius gives more effect on the small arcs and creates a permanent force that holds the fold closed.

### Calling the function

Before you call this function, select the profile element with the 180 degree arc which should be opened. Call this function optionally by:

- Main menu: **Profile, Element, Open Fold.**
- Context menu (right mouse button click on the arc element for opening in the [Drawing Area](#)): **Profile, Element, Open Fold.**

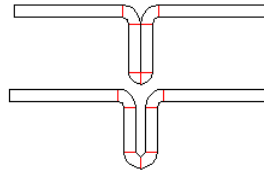


The input window appears with the question **Open fold, give new inner radius.** Enter the new radius that should be in the range 0.4-0.8 x sheet thickness (empirical value).

### Principle of operation

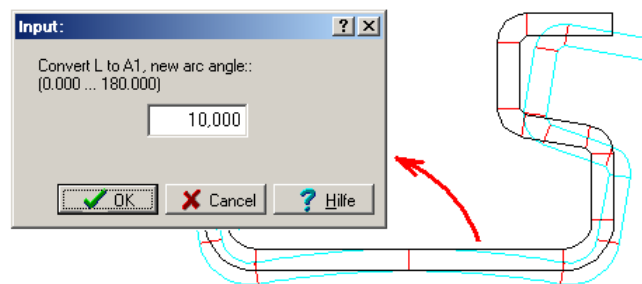
After confirming with the **Ok** button, the fold is opened as described. Precondition is: The previous and the next elements are lines and they have sufficient material that they can be shortened to move the material into the arc.





Any arc angle is possible. So the function also can be used e.g. if a symmetrical profile starts with a 90 degree arc.

### 3.1.3.16.3 Convert L to A1



Use this function to convert a profile element of type **L** (Line) to a profile element of type **A1** with the same developed length. This is used if first an arc is formed into the flat strip, which is pressed to flat again in one of the last stands.

#### Application spectrum

- Compensating the spring back in case the top rolls do not reach the bottom web of the profile anymore.
- Complicated shapes of the profile flanks can be better roll formed, if the flanks are in a more suitable horizontal position.

#### Calling the function

Before you call this function, select the line element **L**, that should be converted to an arc element. Call this function optionally by:

- Main menu: **Profile, Element, Convert L to A1**.
- Context menu (right mouse button click on the arc element in the [Drawing Area](#)): **Profile, Element, Convert L to A1**.

The input window appears with the question **Convert L to A1, new angle?**. Enter the desired angle in degree.

#### Principle of operation

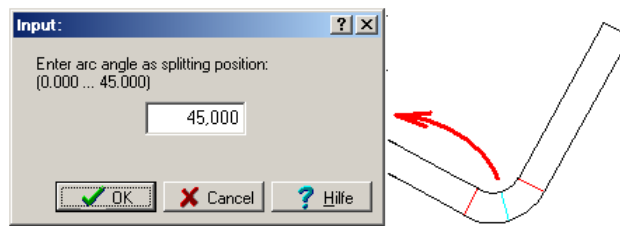
After confirming with the **Ok** button, an arc is created with these properties:

- **Developed Length:** the same as the line's length.
- **Radius:** is calculated from the given arc angle and the developed length.
- **Type:** always A1. If needed, the arc type can be modified in the [Profile List Window](#).
- **Direction:** always downwards or to the outer side. If needed, the direction can be modified by toggling the direction **L** against **R** in the [Profile List Window](#).

#### Hint:

In order to convert an arc to a line you only need to enter the angle 0 in the profile list window.

### 3.1.3.16.4 Split



This function splits the marked [Profile Element](#) of type Line (B) or Arc (A1..4) at a desired position in two parts.

#### Application spectrum

- The [Reference Point](#) should be set to a certain point of the profile in order to form the profile in a position as symmetrical as possible or the deformation energy should be as equal as possible on the left and right side. If there is no splitting point at this position, it can be created by this function.
- A part of an arc should be developed by using another [Bending Method](#), because the design of the roll tools requires it.
- Two or more [Holes/Cut-Outs](#) should be added to a line segment. It is necessary to split the line.

#### Calling the function

Before you call this function, select the profile element type **L** (line) or **A** (arc) that should be split. Call this function optionally by:

- Main menu: **Profile, Element, Split.**
- Context menu (right mouse button click on the profile element in the [Drawing Area](#)): **Profile, Element, Split.**

The input window appears with the question **Enter arc angle as splitting position** or **Enter line length as splitting position**. Enter the desired value.

#### Principle of operation

After confirming with the **Ok** button, the profile element will be split as follows:

**Type L (Line):** The length of the marked line will be modified to the given length. Behind the marked item a new line will be appended with the residual length. The [Strip Width](#), i.e. the sum of all straight lengths will keep constant during this operation.

**Type A1..4 (Arc):** The angle of the marked arc will be modified. Behind the marked item a new arc will be appended with the residual angle. The [Strip Width](#) will keep constant if the [Calculation Method](#) DIN 3965 is selected. It will change marginally, if Oehler is selected, because the straight length calculated by the Oehler method is dependent on the arc angle.

#### Hint:

Redo splitting by the function [Profile, Element, Join](#).

### 3.1.3.16.5 Join

This function joins the marked [Profile Elements](#) of type Line (L) or Arc (A1..4) together with the next profile element in the [Profile List](#).

#### Calling the function

Before you call this function, select the profile element type **L** (line) or **A** (arc) that should be joined with the next one:

- If the marked element is a **Line** (type L), the next element must be a line, too.
- If the marked element is an **Arc** (type A1..4), the next element must be of the same [Arc Type](#) and must have the same [Arc Direction](#) and [Radius](#).

Call this function optionally by:

- Main menu: **Profile, Element, Join.**

- Context menu (right mouse button click on the profile element in the [Drawing Area](#)): **Profile, Element, Join.**

### Principle of operation

The selected profile element is joined with the next profile element. For arcs is effective: The [Strip Width](#) will keep constant if the [Calculation Method](#) DIN 3965 is selected. It will change marginally, if Oehler is selected, because the straight length calculated by the Oehler method is dependent on the arc angle.

### Hint:

Redo joining by the function [Profile, Element, Split.](#)

### 3.1.3.16.6 Insert

Nr.	Typ	Ri	Radius entl.	Winkel entl.	qestr. L.	Bel.
1	S				11,000	
2	B1	L	2,000	90,000	4,444	61
3	S				16,000	
4	PS					

Use this function to insert a new, empty [Profile Element](#) or a set of profile elements from the clipboard at the current position. Numbers of the following elements will be increased by the count of inserted profile elements.

### Calling the function

Before you call this function, select the profile element. Call this function optionally by:

- Main menu: **Profile, Element, Insert.**
- Context menu (right mouse button click on the profile element in the [Drawing Area](#)): **Profile, Element, Insert.**

### Principle of operation

If you call this function and the clipboard contains profile elements, you are asked **Clipboard contains profile element(s). Paste?** and in the next line a list of [element types](#) are shown, e.g. L - A1 - L. If you press **Yes**, all profile elements from the clipboard are pasted. Otherwise an empty line is inserted.

### Hints:

- To append a profile element behind the current one, use the function [Element Append.](#)
- To remove a profile element from the profile list, use the function [Element Remove.](#)
- To copy profile elements to the clipboard, use the function [Element Copy.](#)

### 3.1.3.16.7 Append

No.	Typ	Di	Radius dis.	Angle dis.	St.Length	Str.
1	L				11,000	
2	A3	L	2,000	90,000	4,444	61
3	L				16,000	
4	PS					

Use this function to append a new, empty [Profile Element](#) or a set of profile elements from the clipboard behind the current position. Numbers of the following elements will be increased by the count of appended profile elements.

### Calling the function

Before you call this function, select the profile element. Call this function optionally by:

- Main menu: **Profile, Element, Append.**
- Context menu (right mouse button click on the profile element in the [Drawing Area](#)): **Profile, Element, Append.**

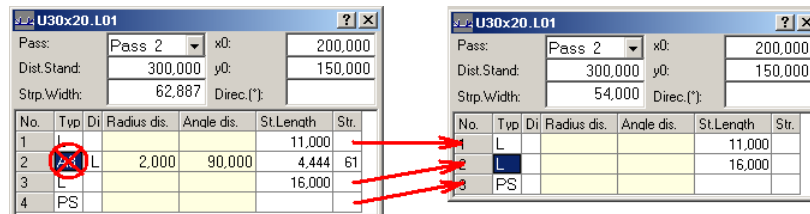
### Principle of operation

If you call this function and the clipboard contains profile elements, you are asked **Clipboard contains profile element(s). Paste?** and in the next line a list of [element types](#) are shown, e.g. L - A1 - L. If you press **Yes**, all profile elements from the clipboard are pasted. Otherwise an empty line is appended.

### Hints:

- To insert a profile element at the current position, use the function [Element Insert](#).
- To remove a profile element from the profile list, use the function [Element Remove](#).
- To copy profile elements to the clipboard, use the function [Element Copy](#).

#### 3.1.3.16.8 Remove



Use this function to remove the current [Profile Element](#) from the profile list. Numbers of the following elements will be decreased by 1.

### Calling the function

Before you call this function, select the profile element. Call this function optionally by:

- Main menu: **Profile, Element, Remove.**
- Context menu (right mouse button click on the profile element in the [Drawing Area](#)): **Profile, Element, Remove.**

### Principle of operation

The profile element is removed irretrievably. The numbers of all following elements will be decreased by 1.

### Hints:

- To insert a profile element at the current position, use the function [Element Insert](#).
- To append a profile element behind the current one, use the function [Element Append](#).

#### 3.1.3.16.9 Copy

Use this function to copy the topical [profile element](#) or the marked set of profile elements to the clipboard. This enables to paste them at another position in the same [Profile List](#), in another profile list of the same [profile project](#), or in another profile project. Use the function [Element Insert](#) or [Element Append](#). to paste the profile elements.

### Calling the function

Before you call this function, select one or more profile elements. Call this function optionally by:

- Main menu: **Profile, Element, Copy.**
- Context menu (right mouse button click on the profile element in the [Drawing Area](#)): **Profile,**

### Element, Copy.

To select a single profile element, click on the desired object, either in the [Drawing Area](#) or in the [profile list window](#).

No.	Typ	Di	Radius dis	ch.Angle	St.Length	Str.
1	L				11,000	
2	A3	2,000	90,000		4,444	61
3	L				16,000	
4	PS					

In order to mark a set of profile elements, click on the first profile element in the profile list window, then press and hold the **Shift Key**, and click on the last profile element. Or press the **Arrow Down Key** while the **Shift Key** is pressed. It is sufficient, if the row [Type](#) is marked.

### Principle of operation

The marked profile elements are copied to the clipboard can be inserted or appended inside **PROFIL** at any other position. The content of the clipboard keeps available until **PROFIL** is terminated or you copy a roll to the clipboard (function [Roll Copy](#))

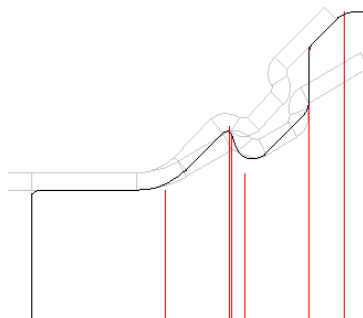
#### Hint:

This function does not allow to transfer profile elements to other Windows programs. For this, use the function [Edit Copy](#).

## 3.1.4 Roll

### 3.1.4.1 Read CAD Contour

Use this function to read a contour, which you have prepared in your [CAD system](#), into **PROFIL** to generate a roll. Whether a [Contour File \(KTR format\)](#) or a [Contour File \(DXF Format\)](#) is being read, you select in [Options Files from CAD](#). If you have enabled the ActiveX Input from CAD in [Options ActiveX](#), the drawing is directly read from the CAD system. If you want to read files with frequent varying file names instead of a temporary file with a constant file name, better use the [Import Function](#).



Use this function in the roll designing stage, if the function [Scan Profile Drawing](#) cannot produce the desired result and you prepared the roll contour in CAD, because the roll should get a contour that differs from the profile contour considerably. Only the working contour (without side flanks) should be imported, working diameter, reference point, and shaft diameter are taken from the [Machine Data](#).

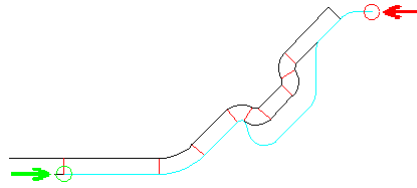
### Calling the function

Before reading the contour, select one of the center lines for defining whether a bottom, a top or a side roll should be created. The center lines are visible in the [Drawing Area](#), if you have selected [View Rolls](#) and the [Machine Window](#) contains machine data. In case of rolls already exist on the shaft, select any roll in order to mark the shaft.

Call this function optionally by:

- Main menu: **Rolls, Read CAD Contour.**

-  Button **Read CAD Contour** in the [Button Key Bar](#).



The [Window Read CAD-Contour](#) appears and you define the **Contour Start Point** (green) and **Contour End Point** (red) of the contour. Do not include the side flanks (vertical lines to the shaft) in the contour.

By checking the color changes to light blue you can find out, if the right contour is tracked between **Contour Start Point** (green) and **Contour End Point** (red). If not, proceed like this:

- **At an intersection the contour tracking turned in the wrong direction** (it prefers the straight on direction): Click onto the path of the right direction behind the intersection.
- **Clicking onto the path of the right direction is ineffectively**: Double lines exist in the CAD drawing or there is no start point of a connection element at an end point of the previous element. Modify the CAD drawing.
- **The blue line ends before the contour end point (red) is reached**: At this position, a faulty connection between two elements exist (gap, overlap, etc.). Correct the CAD drawing.

#### Principle of operation

After confirmation by pressing the **Ok** button in the [Window Read CAD-Contour](#) the position on the shaft is checked first:

- The position is free: The new roll is inserted at the defined position. The user has to check if there are remaining gaps between the rolls.
- A roll yet exists with same width and position: The existing roll is replaced by the new one if the user agrees.
- An overlap occurs between an existing roll and the new one: The existing rolls are moved by the overlap width if the user agrees.

Now a roll is created and displayed, which has the same contour as the one prepared in the CAD system. Afterwards you can modify the roll for your own needs by using the functions of the menu **Roll**.

#### Hints:

Further function for creating rolls are:

- Automatic creation by [Scan Profile Drawing](#).
- Import an existing roll from CAD by [Roll Read CAD Roll](#).
- Import roll from the [Roll Data Base](#).

#### 3.1.4.2 Read CAD Roll

Use this function to import from the [CAD System](#) a roll that already exists but still is not available in a PROFIL Project File. This is useful in case you want to import rolls from third party systems or if you want to reuse old rolls. An important property of this function is that the roll keeps unchanged during positioning on the shaft centerline (in contrast to the function [Roll Read CAD Contour](#) that adapts the diameter dependent on the machine data). The consequence is that the imported roll possibly does not touch the profile contour or even penetrates it. If it does, you have to modify the [Machine Data](#) (working diameter or reference point) afterwards in order to get contact.

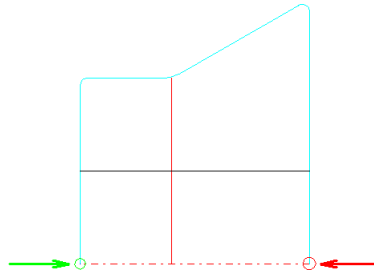
Whether a [Contour File \(KTR format\)](#) or a [Contour File \(DXF Format\)](#) is being read, you select in [Options Files from CAD](#). If you have enabled the ActiveX Input from CAD in [Options ActiveX](#), the drawing is directly read from the CAD system. If you want to read files with frequent varying file names instead of a temporary file with a constant file name, better use the [Import Function](#).

### Calling the function

Before reading the contour, select one of the center lines for defining whether a bottom, a top or a side roll should be created. The center lines are visible in the [Drawing Area](#), if you have selected [View Rolls](#) and the [Machine Window](#) contains machine data. In case of rolls already exist on the shaft, select any roll in order to mark the shaft.

Call this function by:

- Main menu: **Rolls, Read CAD Roll.**



The [Window Read CAD-Contour](#) appears and you define the start (green) and end point (red) of the contour. Because the whole (half) roll should be imported, set both points on the centerline, where left and right side flank intersect the centerline. The blue line of the contour tracking must follow the outer roll contour. If it doesn't, click on the correct line near the start point to redirect the start direction.

### Principle of operation

After confirmation by pressing the **Ok** button the position on the shaft is checked first:

- The position is free: the new roll is inserted at the defined position. The user has to check if there are remaining gaps between the rolls.
- A roll already exists with same width and position: the existing roll is replaced by the new one if the user agrees.
- An overlap occurs between an existing roll and the new one: the existing rolls are moved by the overlap width if the user agrees.

The imported roll appears at the selected position of the roll set. Check if the roll touches the profile correctly, otherwise modify the [Working Diameter](#) in the [Machine Window](#).

### Hints:

Further function for creating rolls are:

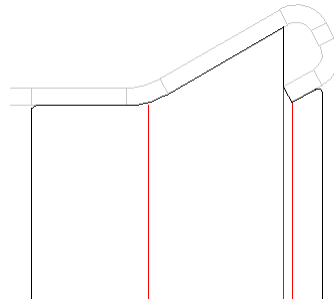
- Automatic creation by [Scan Profile Drawing](#)
- Import contour from CAD by [Roll Read CAD Contour](#)
- Import roll from the [Roll Data Base](#)

#### 3.1.4.3 Scan Profile Drawing

Use this function to scan the profile drawing in the [Drawing Area](#) and to generate a roll for the desired shaft automatically. Only the contour of the profile and other already existing rolls are considered that are visible from the shaft.



Undercuts are considered.




A roll is created that touches the complete visible contour.

By using the switch [Profile, Loaded](#) you can define, whether the discharged or loaded state of the profile should be used. The status of the switch is displayed by a coloured background of the table fields in the [Profile List Window](#) (possibly select **with spring back** in [Options Profile List](#)).

### Calling the function

Before reading the contour, select one of the center lines for defining whether a bottom, a top or a side roll should be created. The center lines are visible in the [Drawing Area](#), if you have selected [View Rolls](#) and the [Machine Window](#) contains machine data. In case of rolls already exist on the shaft, select any roll in order to mark the shaft.

Call this function optionally by:

- Main menu: **Rolls, Scan Profile Drawing**.
-  Button **Scan Profile Drawing** in the [Button Key Bar](#).



The [Window Scan Profile Drawing](#) appears and you define the start (green) and end point (red) of the contour.

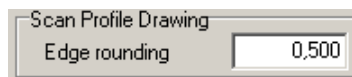
### Principle of operation

After confirmation by pressing the **Ok** button in the [Window Scan Profile Drawing](#) the position on the shaft is checked first.

- The position is free: the new roll is inserted at the defined position. The user has to check if there are remaining gaps between the rolls.
- A roll yet exists with same width and position: the existing roll is replaced by The new one if the user agrees.
- An overlap occurs between an existing roll and the new one: The existing rolls are moved by the overlap width if the user agrees.

A roll is created that matches the scanned profile contour. Afterwards you can modify the roll for your own needs by using the functions of the menu **Roll**.

### Setup



Select in [Options Rolls, Scan Profile Drawing](#) the fillet radius of the automatic edge rounding at the left and right edge of each roll.

### Hints:

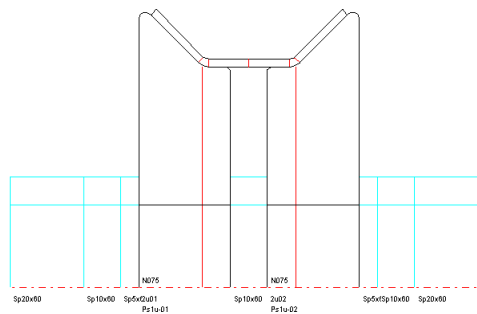
Further function for creating rolls are:

- Import contour from CAD by [Roll Read CAD Contour](#)
- Import an existing roll from CAD by [Roll Read CAD Roll](#)
- Import roll from the [Roll Data Base](#)



### 3.1.4.4 Spacer Rolls

#### 3.1.4.4.1 Create Spacer Rolls



Spacer rolls are needed to fix the horizontal position of the forming rolls on the bottom and top shaft. They have a smaller diameter than the forming rolls and do not touch the profile. After designing the forming rolls, the widths of the spacer rolls result from the residual space between the forming rolls and the machine stand.

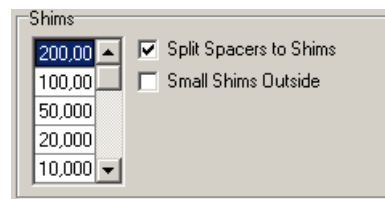
#### Calling the function

Call this function optionally by:

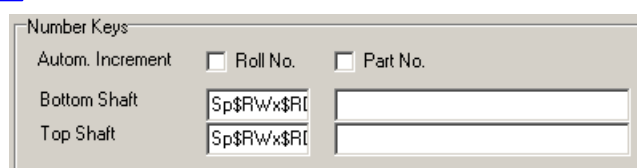
- Main menu: **Rolls, Spacer Rolls, Create.**
  - Context menu (right mouse button click on a roll in the [Drawing Area](#)): **Spacer Rolls, Create.**
- If the bottom or top shaft is selected by mouse click on an existing roll, the spacer rolls will be created for the selected shaft only. Otherwise, if nothing is selected (button **Inspect**), both bottom and top shaft will be furnished with spacer rolls. If spacer rolls already exist, a dialog box asks if they should be removed and replaced. As a distinctive mark to forming rolls the box **Spacer Roll** in the header of the [Roll Tool Window](#) is checked.

#### Principle of operation

Available space between forming roll flanks and stands and between rolls among themselves are filled with spacer rolls.



- **Width:** The width of the spacer rolls results from the available space between the forming rolls and the machine stand (defined by the [Working Width](#) in the [Machine Window](#)). If **Split Spacers to Shims** is checked in [Options, Spacer Rolls](#), the spacer is split into shims dependant on the **Table of Shims**. The checkbox **Small Shims Outside** causes if the spacer splitting should start outside (at the stand) or inside (at the forming roll).
- **Diameter:** The diameter of the spacer rolls is obtained from the input field [Spacers Ø](#) in the [Machine Window](#). It is possible to change it later by changing the two corner point diameter (just like at forming rolls).
- **Material:** The material of the spacer rolls is obtained from the selection box **Spacers Material** in the [Machine Window](#).



- **Designation:** Roll number and part number are created dependant on **Number Keys** in [Options, Spacer Rolls](#). For spacer rolls, the variables \$RW (roll width) and \$RD (roll diameter) are of interest. Example: The number key Sp\$RWx\$RD creates the designation Sp50x70.

Spacer rolls are handled like all other objects in the **PROFIL** object hierarchy. They can be selected by mouse-click and can be dimensioned. All roll modify functions can be applied to spacer rolls as well. Also the [Expanded Roll Tool Window](#) is available.

#### Hints:

- Remove spacer rolls with [Rolls, Spacer Rolls, Remove](#).
- Spacer rolls' width is not adapted automatically after changing forming rolls' width. The spacer rolls must be created newly.
- Spacer rolls can be converted to forming rolls by unchecking the box **Spacer Roll** in the header of the [Roll Tool Window](#). As a result the **Number Keys** in [Options, Rolls](#) are used during [Renumbering](#) and the roll is handled like a forming roll during all output functions.
- Spacer rolls can be switched on and off by using the function [View, Show, Spacer Rolls](#). This takes effect for the [Drawing Area](#) and for the output to [CAD](#), [Printer](#), [Plotter](#), [Parts List](#) and [NC-Program](#). If the spacer rolls are switched off, they are switched on automatically by calling the create function.
- The difference between automatic spacers and spacer rolls is discussed in [Tutorial, Roll Tools, Creating Spacer Rolls](#).

#### 3.1.4.4.2 Remove Spacer Rolls

This function removes spacer rolls. The box **Spacer Roll** in the header of the [Roll Tool Window](#) determines which rolls are spacer rolls.

#### Calling the function

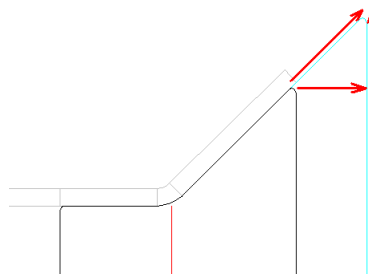
Call this function optionally by:

- Main menu: **Rolls, Spacer Rolls, Remove**.
  - Context menu (right mouse button click on a roll in the [Drawing Area](#)): **Spacer Rolls, Remove**.
- If the bottom or top shaft is selected by mouse click on an existing roll, the spacer rolls will be removed from the selected shaft only. Otherwise, if nothing is selected (button **Inspect**), the spacer rolls from both bottom and top shafts are removed.

#### Hints:

- In order to remove one certain roll only, use the function [Roll, Delete](#).
- If spacer rolls should be created newly (e.g. after changing the width of forming rolls), they do not need to be removed. Use [Rolls, Spacer Rolls, Create](#) and reply **Yes** to the question **Warning: Spacer rolls already exist on the shaft. Remove?**.

#### 3.1.4.5 Conical Extension



Use this function to add a conical extension to the beginning or the end of a roll. This will be regularly necessary, if you created the roll with function [Scan Profile Drawing](#) and the roll set should have more width than the profile itself. The extension can be negative in order to get a smaller roll.

Conical means that start and end of the contour will be lengthened with the same angle. You define the amount either - or

- **by width** (relative)
- **to width** (absolute)
- **to diameter** (absolute)

### Calling the function

Before you call this function, activate either the left or the right corner of the roll to define, whether the extension should be left or right.

Call this function optionally by:

- Main menu: **Rolls, Conical Extension.**
- Context menu (right mouse button click on the left or right corner of the desired roll in the [Drawing Area](#)): **Conical Extension.**

The input window appears and asks **Give width for the conical extension** or **Give diameter for the conical extension**. In case of **Width** you can select between relative **By Width** and absolute **To Width**. In case of **By Width** the input can be negative in order to get a smaller roll.

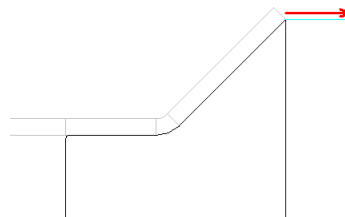
### Principle of operation

The conical extension will be attached to the selected roll corner. If further rolls exist, they will be moved in case there is not enough space between the rolls. The new coordinates of the [Corner Point](#) will be calculated.

#### Hint:

Alternatively you can append a [Cylindrical Extension](#) or an [Arched Extension](#).

### 3.1.4.6 Cylindrical Extension



Use this function to add a cylindrical extension to the beginning or the end of a roll. This will be regularly necessary, if you created the roll with function [Scan Profile Drawing](#) and the roll set should have more width than the profile itself. The extension can be negative in order to get a smaller roll.

Cylindrical means that start and end of the contour will be lengthened parallel to the axis. You define the amount either - or

- **by width** (relative)
- **to width** (absolute)

### Calling the function

Before you call this function, activate either the left or the right corner of the roll to define, whether the extension should be left or right.

Call this function optionally by:

- Main menu: **Rolls, Cylindrical Extension.**
- Context menu (right mouse button click on the left or right corner of the desired roll in the [Drawing Area](#)): **Cylindrical Extension.**

The input window appears and asks **Give width for the cylindrical extension**. Select between relative **By Width** and absolute **To Width**. In case of **By Width** the input can be negative in order to get a smaller roll.

### Principle of operation

The cylindrical extension will be attaches to the selected roll corner. If the roll is not cylindrical

before or behind the selected [Corner Point](#), a new corner point will be appended, in the other case the selected corner point will be moved. If further rolls exist, they will be moved in case there is not enough space between the rolls. The new coordinates of the [Corner Point](#) will be calculated.

**Hint:**

Alternatively you can append a [Conical Extension](#) or an [Arched Extension](#).

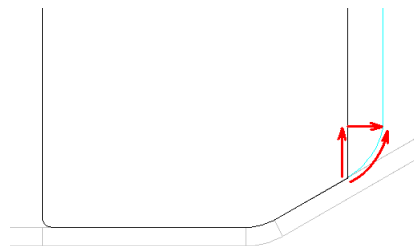
### 3.1.4.7 Arched Extension



If a roll has an arc with a radius at the beginning or the end, it may be necessary to lengthen the arc instead of appending a conical extension. A reason for it may be: The surface of the sheet should not be damaged by a roll corner. This means, the center point and the radius keeps constant, the angle will be increased or decreased. The extension can be negative in order to get a smaller roll.

You define the amount either - or

- **by width** (relative)
- **to width** (absolute)
- **to diameter** (absolute)



If the roll ends with a line, an arched extension can be added with a selectable radius. In this case the arched extension always is convex, this means it points to the centerline of the roll.

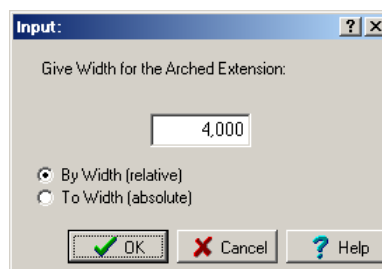
#### Calling the function

Before you call this function, activate either the left or the right corner of the roll to define, whether the extension should be left or right.

Call this function optionally by:

- Main menu: **Rolls, Arched Extension.**
- Context menu (right mouse button click on the left or right corner of the desired roll in the [Drawing Area](#)): **Arched Extension.**

The input window appears and asks **Give width for the arched extension** or **Give diameter for the arched extension**. In case of **Width** you can select between relative **By Width** and absolute **To Width**. In case of **By Width** the input can be negative in order to get a smaller roll.



If the roll ends with a line, an additional input window appears and asks **Give radius for the**

**arched extension.**

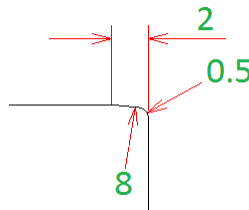
### Principle of operation

The arched extension will be attached to the selected roll corner. If further rolls exist, they will be moved in case there is not enough space between the rolls. The new coordinates of the [Corner Point](#) will be calculated.

### Hint:

Alternatively you can append a [Cylindrical Extension](#) or a [Conical Extension](#).

### 3.1.4.8 Double Fillet



This function creates a double fillet at a roll corner with a larger and a smaller radius. Both arcs have tangential connections.

### Application spectrum

- Avoiding track markers on the sheet surface caused by production tolerances of the rolls, e.g. of split rolls (large radius).
- Avoiding damage of the roll corners during handling operations (small radius).
- Avoiding operator's hands injury during handling operations (small radius).

### Calling the function

Before you call this function, identify the [Roll Corner Point](#) that should be furnished with a double fillet.

Call this function optionally by:

- Main menu: **Rolls, Double Fillet.**
- Context menu (right mouse button click on the corner of the desired roll in the [Drawing Area](#)): **Double Fillet.**



The input window appears and asks **Double Fillet Width**. Enter the desired width between the start point of the large arc and the corner point of the roll's flange face.

### Principle of operation

Two fillet radii with tangential connections are applied to the selected [Roll Corner Point](#). The radii are dependent on the selected width:

- Width  $\geq 1.5\text{mm}$  (0,075in): Large radius = 8mm (0,4in), small radius = 0.5mm (0.025in).
- Width  $< 1.5\text{mm}$  (0,075in): Large radius = 5mm (0,25in), small radius = 0.3mm (0.015in).

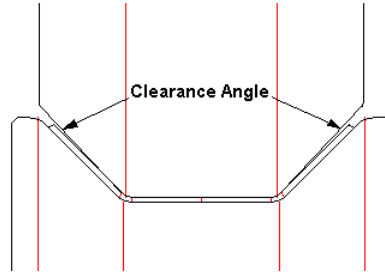
### Hints:

If the message appears **Double fillet not possible for this roll geometry**, the reason is:

- The distance to the neighboring roll corner point or to arc start point of this corner point is too small.

- In case of conical rolls the angle from the selected to the neighboring roll corner point is too large.

#### 3.1.4.9 Clearance Angle



##### What is a clearance angle?

Clearance angle means, the inclined flank of a roll (top roll in the picture above) is rotated by a small angle (normally 1-2 degree) around the [Roll Corner Point](#). Thereby an acute gap opens between roll and sheet. The more the circumferential speeds of top and bottom rolls differ, the larger is the gap width.

##### What a clearance angle is required for?

There are two reasons why a clearance angle should be provided:

- Balancing of different circumferential speeds in case of the roll diameter ratio does not fit to the gear transmission ratio.
- Avoiding wedging, thereby better “flowing” (transversal moving) of the profile legs into the stand.

We will discuss these two reasons now.

##### Balancing of different circumferential speeds

Each roll has a [Working Diameter](#) (pitch diameter), this is the diameter at the profile web (the horizontal part of the profile, see picture above). If the [Gear Transmission Ratio](#) of the machine is 1:1, top and bottom rolls should have the same working diameter in order to have equal circumferential speeds and the profile is moved safely and free of slip. Some machines have the transmission ratio 1:1.4, this enables forming open profiles with higher legs. If e.g. the transmission ratio is 1:1.4, equal circumferential speeds are possible if the working diameter of the top roll is set to 1.4 times the working diameter of the bottom roll. But equal circumferential speeds only can be achieved at the web of the profile. At the profile legs always friction arises, enhanced roll wearing, and sheet surface damaging will be incurred. In order to minimize this, experienced designers use a clearance angle.

##### Better “flowing” of the profile legs into the stand

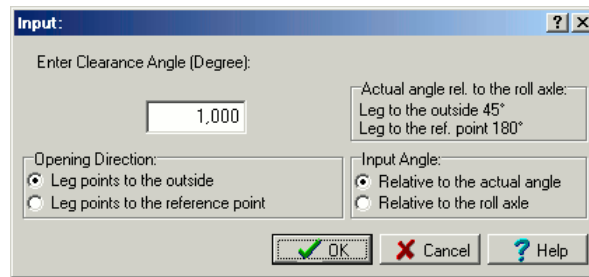
Imagine, the flat sheets run into a stand like in the picture above. First it will get contact with the roll surfaces that have the largest diameters. The top roll has its largest diameter at the center cylindrical part, whereas the bottom roll has it at the side shoulders. By turning on the top roll presses the sheet into the trapezoidal slot of the bottom roll and the profile legs are drawn over the shoulder edges of the bottom roll. The shoulder edges should have a large fillet radius as shown in the picture in order to prevent the sheet surface from damage. In doing so, the legs must be able to move horizontally, they say, they must “flow”. This can be achieved by using a clearance angle. Without it, the legs would wedge and the sheet would be stretched in transversal direction. Very important are clearance angles for profiles with more than one bending zone per side and per stand. Example: [Trapezoidal Profiles](#). Often a full trapezoid that consists of four angles is bent per side.

##### Calling the function

Before you call this function, select the vertex of the clearance angle; in the example the [Roll Corner Point](#) of the fillet of the top roll.

Call this function optionally by:

- Main menu: **Rolls, Clearance Angle**.
- Context menu (right mouse button click on the left or right corner of the desired roll in the [Drawing Area](#)): **Clearance Angle**.



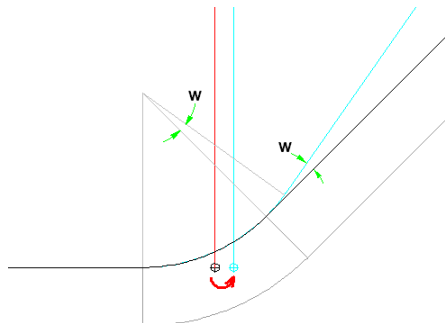
The input window appears and asks **Enter Clearance Angle (Degree)**.

**Opening Direction:** In most cases the legs of the clearance angle point to the outside direction (view from the reference point). This direction **Legs point to the outside** is predefined in **PROFIL**. If in special cases the legs should point to the inside (e.g. for trapezoid profiles), check the box **Leg points to the reference point** before you press **Ok**.

**Input Angle:** Enter the angle **Relative to the actual angle** or **Relative to the roll axle** as desired. In the first case, you get the desired opening angle. In the second case, you can define a desired (rounded) roll angle. If you select **Relative to the actual angle**, the clearance angle will be increased after entering a positive value and decreased after entering a negative value.

**Actual angle rel. to the roll axle:** If you selected **Input Angle Relative to the roll axle**, pay attention on the box **Actual angle rel. to the roll axle**, that shows the angles **Leg to the outside** and **Leg to the ref. point**. This helps defining the right value for the input field. Pay attention on the sign, it must be entered, too.

### Principle of operation



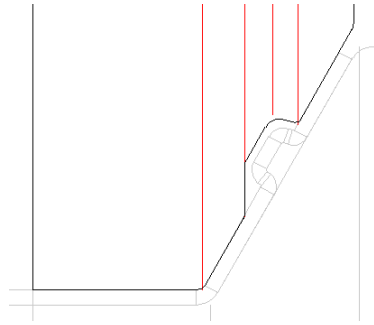
In order that the fillet of the roll clings to the profile, **PROFIL** keeps the center of the arc constant; the angle of the arc is extended or reduced by the clearance angle **w**. The [Roll Corner Point](#) (intersection point of the tangents) moves accordingly.

A positive clearance angle creates or extends the gap; a negative reduces the gap. Please check after creating the clearance angle if the line-arc transition is correct at the next roll corner point.

### Hints:

- If a clearance angle should be continued over multiple roll corner points (e.g. for trapezoidal profiles), the designer should attach a clearance angle for the first point. For the rest a gap should be created by the function [Roll, Gap](#).
- Alternatively, the angle to the next roll corner point can be modified by direct input in the [Angle](#) input field of the [Roll Tool Window](#). In doing so, the coordinates of the roll corner point keep unchanged.

### 3.1.4.10 Gap



Use this function to create or modify a parallel gap between roll and profile. The gap can be created between two [Roll Corner Points](#) or for the whole roll.

#### Application spectrum

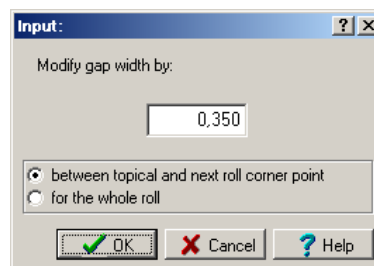
- Coated sheet metal should be processed.
- Raised cut-outs should not be deformed.
- A horizontal sheet guide should have a gap in order to avoid pinching in case of sheet width allowances.
- A [Clearance Angle](#) should be continued in the successive profile segments, e.g. for forming trapezoidal profiles.
- Already finished parts of the profile should not be touched by rolls anymore (see picture).

#### Calling the function

Before you call this function, identify the previous [Roll Corner Point](#) of the two that should be treated. If the gap should be applied to the whole roll, select any corner point of the roll.

Call this function optionally by:

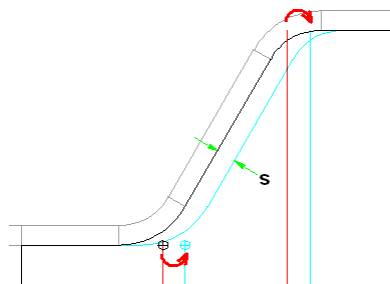
- Main menu: **Rolls, Gap**.
- Context menu (right mouse button click on the corner of the desired roll in the [Drawing Area](#)): **Gap**.



The input window appears and asks **Modify gap width by**. Enter the desired value related to the topic gap width. A positive value enlarges the gap and a negative reduces it.

- **Between topical and next roll corner point:** Select this option if the gap should be created between two corner points.
- **For the whole roll:** In this case the whole roll gets a gap to the profile.

#### Principle of operation





If you selected **Between topical and next roll corner point**, the connection line between the points is moved in parallel by the gap width **s**. If the connection line does not exist, i.e. if one arc is connected to the next tangentially, the connection point is moved. Thus no new corner points are created, but the existing corner points are moved on the outer tangents (similar to the [Clearance Angle](#) function). By selecting **For the whole roll** the parallel gap is created for the whole roll.

### 3.1.4.11 Renumber

Use this function to renumber [Roll Number](#) and [Part Number](#) of all [Forming Rolls](#) and [Spacer Rolls](#) of a stand, e.g. if you modified the number keys.

#### Calling the function

Call this function optionally by:

- Main menu: **Rolls, Renumber**.
- Context menu (right mouse button click on the left or right corner of the desired roll in the [Drawing Area](#)): **Renumber**.

#### Principle of operation

Renumbering works dependant on the **Number Keys**. For the forming rolls, the number keys in [Options Rolls](#) are used:

Number Keys		
Autom. Increment	<input checked="" type="checkbox"/> Roll No.	<input checked="" type="checkbox"/> Part No.
Bottom Roll	\$PLu01	\$PR-Ps\$PSu-01
Top Roll	\$PLo01	\$PR-Ps\$PSo-01
Left Roll	\$PLl01	\$PR-Ps\$PSl-01
Right Roll	\$PLr01	\$PR-Ps\$PSr-01

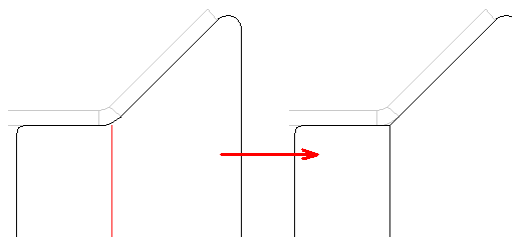
For the spacer rolls, the number keys in [Options Spacer Rolls](#) are used:

Number Keys		
Autom. Increment	<input type="checkbox"/> Roll No.	<input type="checkbox"/> Part No.
Bottom Shaft	Sp\$RWx\$RI	
Top Shaft	Sp\$RWx\$RI	

During this operation, the [Variables](#) are replaced by the corresponding project or roll data.

If single rolls are locked against number changes (e.g. because they came from the roll database or from a partial project or numbers are modified manually) you are asked **One ore more rolls are locked against number change. Unlock and renumber?** You can decide how the rolls should be treated.

### 3.1.4.12 Split at Corner



Use this function to split a roll at a [Roll Corner Point](#). Precondition: The roll has 3 or more corner points.

### Calling the function

Before you call this function, identify the desired split [Roll Corner Point](#).

Call this function optionally by:

- Main menu: **Rolls, Split at Corner**.
- Context menu (right mouse button click on the left or right corner of the desired roll in the [Drawing Area](#)): **Split at Corner**.

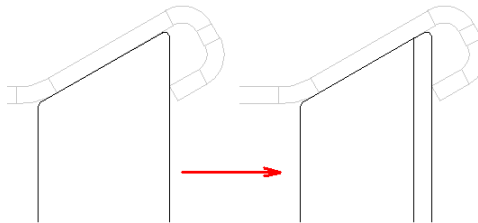
### Principle of operation

The roll is split at the selected roll corner point in two partial rolls. If the selected corner point has a [Fillet Radius](#), the radius will be set to zero.

#### Hints:

- If you want to undo splitting, use the function [Join](#).
- To split a roll between two corners, use the function [Split between Corners](#).

#### 3.1.4.13 Split between Corners



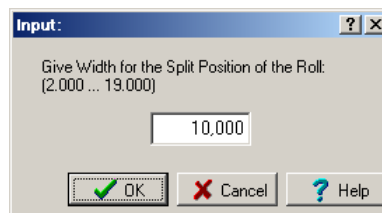
Use this function to split a roll between two [Roll Corner Points](#).

### Calling the function

Before you call this function, identify the previous [Roll Corner Point](#).

Call this function optionally by:

- Main menu: **Rolls, Split between Corners**.
- Context menu (right mouse button click on the left or right corner of the desired roll in the [Drawing Area](#)): **Split between Corners**.



The input window appears with the question **Give width for the split position of the roll** and an admissible range of values for the input, which begins at the end of the last fillet and ends at the beginning of the next fillet. The width is related to the [Roll Reference Point](#).

In the input field appears the rounded mean value as a proposal. You can modify this proposal or overwrite it. If two fillets touch one another, the roll will be split at the point of tangency and the input window will not appear.

### Principle of operation

The roll is split at the entered width behind the selected roll corner point in two partial rolls. Both partial rolls get the same roll corner point.

#### Hints:

- If you want to undo splitting, use the function [Join](#).
- To split a roll at a corner, use the function [Split at Corner](#).

### 3.1.4.14 Join

Use this function to join two rolls to one roll.

#### Calling the function

Before you call this function, identify the last [Roll Corner Point](#) of a roll, which should be joined to the next one. Or identify the first corner point of a roll, which should be joined to the previous one. Call this function optionally by:

- Main menu: **Rolls, Join**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Join**.

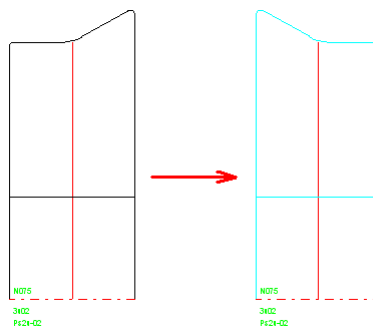
#### Principle of operation

Two rolls with the same roll corner point are joined together to one roll.

#### Hints:

- If after joining a surplus roll corner point exists, remove it with the function [Roll, Corner, Remove](#).
- If you want to undo the joining, use the function [Split at Corner](#).

### 3.1.4.15 Turn



Use this function to turn a roll. This means, you take off a roll from the shaft and install it inverted.

#### Calling the function

Before you call this function, identify a roll by clicking any [Roll Corner Point](#) of the roll. Call this function optionally by:

- Main menu: **Rolls, Turn**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Turn**.

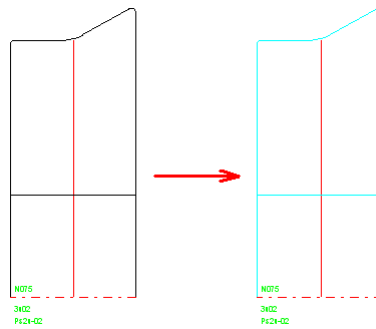
#### Principle of operation

The roll is turned by changing the order of the [Roll Corner Points](#) in the [Roll Tool Window](#). The position of the roll on the shaft keeps unchanged.

#### Hint:

If you want to undo the turning, turn the roll again with the same function.

### 3.1.4.16 Move



Use this function to move a certain roll or all rolls on the shaft.

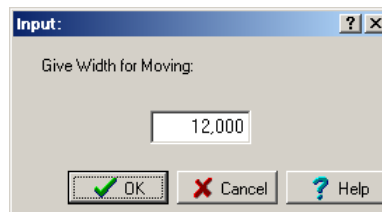
#### Calling the function

**Marked:** Mark the desired roll by marking any [Roll Corner Point](#).

**All of shaft:** Mark the desired shaft by marking any [Roll Corner Point](#) of any roll.

Call this function optionally by:

- Main menu: **Rolls, Move**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Move**.



The input window appears with the question **Give width for moving**. Enter the value for moving. Entering a positive value moves the rolls into the direction to the last roll, a negative value moves the rolls into the direction to the first roll. To obtain the desired amount of moving width, the function [Measure](#) is helpful.

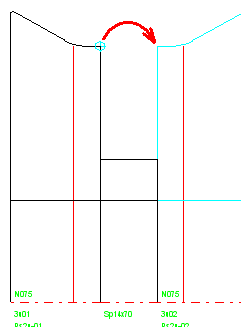
#### Principle of operation

The roll is moved by adding the given width to the [Width](#) of each [Roll Corner Point](#) in the [Roll Tool Window](#). If a gap appears after moving a single roll the user has to fill out the gap manually. If a neighboring roll exists, it will be moved in case there is not enough space between the rolls

#### Hints:

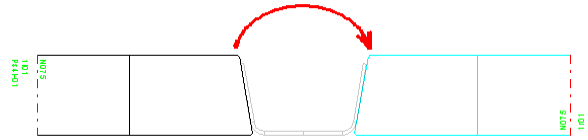
- The function [Rolls, Spacer Rolls, Create](#) fills a gap between rolls automatically with a spacer roll.
- If you want to undo a moving, move again in the opposite direction.

### 3.1.4.17 Mirror



**Top/bottom roll:**

Use this function to mirror a roll at the right or left outer edge of a roll set. This is useful for easier designing of rolls for symmetrical profiles: create the rolls for one half only and mirror them afterwards.

**Side roll:**

This function creates a mirrored side roll on the opposite side. This means, a left side roll will be mirrored to a right side roll and vice versa. This simplifies designing rolls for symmetrical profiles.

**Calling the function**

**Top/bottom roll:** Before you call this function, identify the roll by clicking a [Roll Corner Point](#) of the roll that should be mirrored. It is important which corner is selected: the mirrored roll will be created at the right or left outer edge of the roll set dependent on if you select a corner on the right or left.

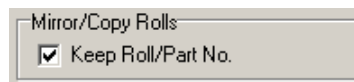
**Side roll:** Before you call this function, identify the roll by clicking any corner of the roll that should be mirrored. Check previously, if no roll exist on the opposite side in the corresponding height. If more than one roll should be mirrored, use this function for each roll successively.

Call this function optionally by:

- Main menu: **Rolls, Mirror.**
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Mirror.**

**Principle of operation**

The new roll with same size is created dependant on the specification. In case of top/bottom rolls the order of the [Roll Corner Points](#) in the [Roll Tool Window](#) is changed, in case of side rolls preserved.



In [Options, Rolls](#), **Mirror Rolls** can be preset if the [Roll Number](#) and [Part Number](#) should be kept during mirroring. Otherwise they are renumbered dependent on the defined **Number Keys**.

**3.1.4.18 Cut**

Use this function to cut a roll. This means, the roll will be removed from the set of rolls and will be transferred to the clipboard.

**Calling the function**

Before you call this function, identify the roll by clicking any [Roll Corner Point](#) of the roll.

Call this function optionally by:

- Main menu: **Rolls, Cut.**
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Cut.**

**Principle of operation**

The roll is moved to the clipboard. From the clipboard, you can insert the roll at any position, also to another shaft or to another stand, by using the function [Roll, Paste](#). In this way you also can duplicate a roll. Furthermore, in this way you can transfer a roll into the [Roll Stock Management](#).

**Hints:**

- If you want to undo the cutting, call function [Roll, Paste](#) at the same position.
- Transferring the roll to another Windows programs is not possible, use the function [Edit, Copy](#) instead.

### 3.1.4.19 Copy

Use this function to copy a roll to the clipboard. The origin roll remains unchanged.

#### Calling the function

Before you call this function, identify the roll by clicking any [Roll Corner Point](#) of the roll.

Call this function optionally by:

- Main menu: **Rolls, Copy**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Copy**.

#### Principle of operation

The roll is copied to the clipboard. From the clipboard, you can insert the roll at any position, also to another shaft or to another stand, by using the function [Roll, Paste](#). In this way you also can duplicate a roll. Furthermore, in this way you can transfer a roll into the [Roll Stock Management](#).

#### Hint:

- Transferring the roll to another Windows programs is not possible, use the function [Edit, Copy](#) instead.

### 3.1.4.20 Paste

Use this function to insert a roll from the clipboard into a set of rolls.

Precondition: You have stored a roll to the clipboard by using function [Roll Cut](#) or [Roll Copy](#).

Furthermore, in this way you can mount a roll from the [Roll Stock Management](#) into your current project.

This description assumes that all your bottom and top rolls have the first [Roll Corner Point](#) at the left side. To reach this during creating the rolls in the [Window Read Contour](#), set the green start point to the left and the red end point to the right contour point. Otherwise, if you built the rolls in inverse direction, replace "left" by "right". For side rolls, replace "left" by "smallest width coordinate" (see [Roll Tool Window](#)).

#### Calling the function

Before you use this function, mark the position for inserting by clicking to the left or right [Roll Corner Point](#). If no roll exists, mark the shaft.

Call this function optionally by:

- Main menu: **Rolls, Paste**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Paste**.

#### Principle of operation

If you marked the left corner point of an existing roll, the new roll is inserted at the left of the existing roll. Further existing rolls on the left are moved to the left if the user agrees. The marked roll and further rolls on the right are not moved. Otherwise, if you marked the right side, the new roll is inserted at the right analogously.

If a gap exists beside the marked roll (e.g. after deleting a roll), the width is checked: if the new roll fits into the gap exactly or the new roll is smaller than the gap, the roll is inserted. A remaining gap has to be filled by the user. If the roll is wider than the gap, further rolls are moved if the user agrees; the marked corner point determines the direction of movement.

If no roll exists on the marked shaft, the new roll is inserted at 0 width position.

#### Hints:

- The function [Rolls, Spacer Rolls, Create](#) fills a gap between rolls automatically with a spacer roll.
- If you want to undo the paste function, use the function [Roll Delete](#).

### 3.1.4.21 Delete

Use this function to delete a roll.

#### Calling the function

Before you call this function, identify the roll by clicking any [Roll Corner Point](#) of the roll.

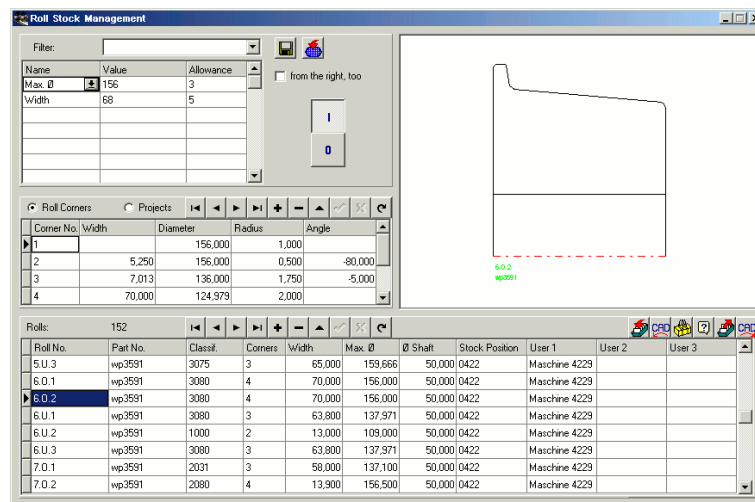
Call this function optionally by:

- Main menu: **Rolls, Delete.**
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Delete.**

#### Principle of operation

The roll is deleted irretrievably.

### 3.1.4.22 Stock Management



Only with option Database.


Afterwards the manufacturing of a profile has been finished, the roll tools are removed from the roll former and are taken to the roll stock, so the roll former can be set up for the next project. This is the right time to transfer the rolls from the project to the roll database.

The roll database gives information about the rolls in the roll stock. This helps while designing a new profile project, if you want to re-use existing rolls for reducing the costs. The roll database performs quick filter and search functions.

To transfer the rolls from and to the database, the clipboard can be used. All rolls of a shaft, a stand or of the complete project can be stored directly. Also a roll drawn in CAD can be stored. Rolls from the database can be transferred directly to CAD. If only a paper drawing of old rolls is available, the data can be entered directly into the database.

#### Calling the function

Before you call this function, select the [View Roll Tools](#). This prepares opening the roll stock management for the button in the top button bar. Call this function optionally by:

- Main menu: **Rolls, Stock Management.**
-  Button **Roll Stock Management** in the [Button Key Bar](#).

#### Stock Management Open

After calling this function, the window **Roll Stock Management** appears, which contains 4 areas:

- [Roll Table](#) (lower area): This is the real roll database, each row displays the data of one roll.

- [Roll Corner Table](#) (in the middle, left), can be switched to the [Project Table](#): This area displays the data of the roll selected in the roll table.
- [Drawing Area](#) (top, on the right): This area displays the drawing of the roll selected in the roll table.
- [Filter](#) (top, on the left): Any filters can be defined for reducing the amount of displayed rolls.

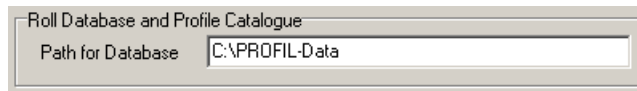
### Stock Management Save

Use this function to store selected rolls into the roll database. Which rolls are stored, you define by selecting one of the following sub functions and by marking a roll in the [Drawing Area](#) respectively:

- **Roll:** The marked roll is stored.
- **Shaft:** All rolls of a shaft are stored. Define the shaft by marking any roll of the shaft.
- **Stand:** All rolls of all shafts of one stand are stored.
- **Project:** All rolls of all stands of the whole project are stored.

If the window Roll Stock Management is not yet open, it will be opened after calling one of these functions.

### Setup



Set the path to the roll database in [Options Stock Management](#).

## 3.1.4.23 Corner

### 3.1.4.23.1 Append

Rolls are built of a count of [Roll Corner Points](#), that are shown in the [Roll Tool Window](#). Use this function to append a corner point to a roll, e.g. if you want to modify the contour of a roll.

### Calling the function

Before you call this function, identify the roll by clicking the previous [Roll Corner Point](#) of the roll. Relevant is the order in the [Roll Tool Window](#).

Call this function optionally by:

- Main menu: **Rolls, Corner Append**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Corner Append**.

### Principle of operation



The input window appears with the question **Give width for the new corner** and an admissible range of values for the input, which begins at the end of the last fillet and ends at the beginning of the next fillet. The width is related to the [Roll Reference Point](#).

In the input field appears the rounded mean value as a proposal. You can modify this proposal or overwrite it.

### Principle of operation

The new roll corner point is appended at the given position. If two fillets touch one another, the new corner point will be inserted at the point of tangency and the input window will not appear.



**Hint:**

- If you want to undo a this function, use the function [Corner Remove](#).

**3.1.4.23.2 Remove**

Use this function to remove a [Roll Corner Point](#).

**Calling the function**

Before you call this function, identify the [Roll Corner Point](#) for removing.  
Call this function optionally by:

- Main menu: **Rolls, Corner Remove**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the [Drawing Area](#)): **Corner Remove**.

**Principle of operation**

The roll corner point is removed. After removing, the previous and next corner points will be connected.

**3.1.5 Calculate****3.1.5.1 Statics**


A4865.L01				
Statics:	Related to:	x	y	
Centroid of an Area	Ref. Point x0/y0	0,000	16,176	mm
Shear Center Point	Ref. Point x0/y0	0,000	-21,426	mm
Moment of Inertia	x/y in Centroid	6,259	8,738	cm4
Moment of Inertia	in Principal Axes	8,738	6,259	cm4
Moment of Resistan.	x/y in Centroid	2,868	4,369	cm3
Moment of Resistan.	In Principal Axes	4,369	2,868	cm3
Max. Distance	In Principal Axes	2,182	2,000	cm
Inertial Radius	In Principal Axes	1,574	1,332	cm
Area of Cross Sec.		3,526		cm2
Weight without Holes		2,768		kg/m
Princip. Axes Angle	x-Axis in Centroid	90,000		deg
Vault Resistance	Shear Center Point	31,713		cm6
Torsion Moment	Centroid	0,106		cm4

Use this function to calculate the statics of the profile list, which is showed in the current [Profile List window](#). The statics parameters are displayed in the **Statics Window**. They are calculated by using the discharged columns of the profile list only; they are not dependent on the status of the menu switch [Profile, Loaded](#).

[Holes/Cut-Outs](#) will weaken the area of cross of the profile. The calculation of statics considers this. Exception: for the [Weight](#) holes/cut-outs are not considered since input of count and length is not supported.

**Calling the function**

Before you call this function, select the pass that should be calculated. Call this function optionally by:

- Main menu: **Calculate, Statics**.
-  Button **Statics** in the [Button Key Bar](#).

**Principle of operation**

The **Statics Window** is opened and shows these statics parameters:

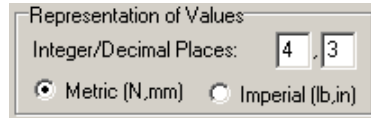
[Reference Point](#)  
[In Principal Axes](#)

[Centroid](#)  
[Shear Center Point](#)

[Moments of Inertia](#)  
[Max. Distance](#)  
[Area of Cross](#)  
[Principal Axes Angle](#)  
[Torsion Moment](#)

[Moments of Resistance](#)  
[Inertial Radii](#)  
[Weight](#)  
[Vault Resistance](#)

### Setup



Select in [Options Calculate](#), **Representation of Values** whether the statics should be displayed in the **Metric System** (mm, N) or the **Imperial System** (in, lb.). Set the count of **Integer/Decimal Places** as well.

#### Hints:

- Use the functions [File, Print Preview](#) or [File, Print](#) to print the statics.
- Use the function [View Statics](#) to display the statics in the [Drawing Area](#) and to transfer them to the [CAD System](#).

#### 3.1.5.1.1 Reference Point

The reference point is the point inside your drawing that you have defined as your [Reference Point X0/Y0](#) (i.e. start point) of the profile. Furthermore it is identical with the symmetrical point of [Profile Element PS](#) of symmetrical profiles or with the point **P** of an asymmetrical profile, which is the beginning of the second half.

#### Hints:

- Modify the [Reference Point X0/Y0](#) by using the function [Modify Reference Point](#).
- Some [Statics](#) are related on the reference point.

#### 3.1.5.1.2 Centroid

The centroid is the centroid of the area of the profile. Inside the drawing, the centroid is the origin of the principal axes (red axis system).

Cold roll former often lower the web of the profile from one stand to another to reduce the stress of the band edge ([Center Line Forming](#), also called downhill forming or natural forming). Usually this is done by placing the centroid of all passes to the same level.

See also: [Statics](#)

#### 3.1.5.1.3 In Principal Axes

The **In principal axes** is an axis system for maximum (bend round the X-main axis) and minimum (bend round the Y-main axis) load of the profile. The origin of axis system is the [Centroid](#).

Inside the drawing principal axes are displayed in red color. The angle between the X-axis of principal axes and the horizontal X-axis is the [Principal Axes Angle](#).

See also: [Statics](#)

#### 3.1.5.1.4 Shear Center Point

The shear center point is the optimal point of application of load in case of bending without shearing force, i.e. bending stress without torsion.

Inside the drawing the shear center point is displayed as a small red cross.

For U-type profiles the shear center point will be mostly outside the profile. Then a welded on bracket may allow application of load in the shear center point.

See also: [Statics](#)

#### 3.1.5.1.5 Moments of Inertia

The moments of inertia will be calculated as well as for the coordinate system of the drawing as for the [In principal Axes](#) system. Moments of inertia in the **In principal axes** system represent the maximum (x) and the minimum (y) moment.

See also: [Statics](#)

#### 3.1.5.1.6 Moments of Resistance

By using of moments of resistance it is possible to calculate the effect of stress in the material based on the cause of bending stress (applied torque):

$$\sigma_b = \frac{M_b}{W}$$

The profile will not be deformed if appearing stress is lower than safe bending load (get from material tables).

Based on the moments of inertia the moments of resistance are calculated from the outside distance. For the worst case (lowest moments of resistance) use the maximum distance:

$$W = \frac{J}{e_{\max}}$$

PROFIL calculates the moments of resistance related on the [In Principal Axes](#) system; which are the axes of minimum and maximum loading.

See also: [Statics](#)

#### 3.1.5.1.7 Max. Distance

Because bending will cause maximum stress at the outside of strip, the maximum distance in x- and y-direction is important. It will be used to calculate the moments of resistance based on the moments of inertia.

The drawing displays the maximum distance as the length of the red axis system.

See also: [Statics](#)

### 3.1.5.1.8 Inertial Radii

The inertial radius is the radius of a (imaginary) round tube, that has the same moments of inertia as the designed profile with any cross-section.

The inertial radius is displayed separately for the X- and Y-direction; the lower value is used to calculate the slenderness ratio of profiles that are loaded by longitudinal compression:

$$\varrho = \sqrt{\frac{i_{\min}}{A}}$$

$i_{\min}$  = minimum inertial radius  
A = Area of cross

See also: [Statics](#)

### 3.1.5.1.9 Area of Cross

The area of cross is calculated by multiplication of strip width (=sum of all straight lengths) and sheet thickness.

See also: [Statics](#)

### 3.1.5.1.10 Weight

The weight per meter is calculated by multiplication of the [Area of Cross](#) and density of the selected material (for example steel 7.85 kg/dm<sup>3</sup>).

The calculation needs a valid material selection inside the [Project Data Window](#). In other case you will get a null weight. Holes/cut-outs are not considered since input of count and length is not supported.

See also: [Statics](#)

### 3.1.5.1.11 Principal Axes Angle

The principal axes' angle is the angle between the x-main axis (= axis of maximal loading) and the horizontal x-axis of the drawing.

For symmetrical profiles the principal axes' angle is either 0° or 90°, dependent on the maximum length in x or y direction. For asymmetrical profiles the principal axes' angle can be any value between -45° and 135°.

See also: [Statics](#)

### 3.1.5.1.12 Vault resistance

If the load is not applied in the shear center point, a vault of the profile will appear and this causes an additional shearing stress of the profile. Based on the vault resistance it is possible to calculate this composite stress.

See also: [Statics](#)

### 3.1.5.1.13 Torsion moment

The torsion moment has for torsion stress a similar meaning as the moment of inertia for bending. It allows to calculate the effect of torsion (shear stress) from the cause of torsion (applying torque):

$$\tau_t = \frac{M_t}{W_t} = \frac{M_t \cdot e_{\max}}{J_t}$$

$M_t$  = applied torque,

$J_t$  = torsion moment,

$e_{\max}$  = maximum distance

The profile will not be deformed if the appearing shearing stress is lower than safe working stress for torsion (get values from material tables).

For exact interpretation the torsion moment is only defined for circular or similar profiles. For thin walled profiles with any design the torsion moment can be replaced with proper accuracy by the St. Venant's drill resistance. PROFIL does it, provided that the profile is open.

See also: [Statics](#)


### 3.1.5.2 Stress of Edge

Stress of Band Edge:	Material:	St	%
<input type="checkbox"/> Center Line Forming	3 FE P02 G 275 NA	Re = 380 N/mm2	

Calculation of stress of edge is step 1 of the three step concept for quality management. This function calculates approximately the strain of the band edge and the resulting relative stress related to the yield point of the used material. Thus you can check quickly whether the edge keeps or exceeds the dangerous yield point.

#### Calling the function

Before you call this function, select the pass that should be calculated. Call this function optionally by:

- Main menu: **Calculate, Stress of Edge.**
-  Button **Stress of Edge** in the [Button Key Bar](#).

#### Principle of operation

The window **Stress of Band Edge** is opened. The strains are calculated by using an approach model that depends on an empirical analysis and the results converted to relative stress related to the yield point are displayed in the window. The first row shows the [Material](#) entered in the [Project Data Window](#) and the stress of the yield point for this material (Re).

Each row in the table represents a stand. The column **St** displays the number of the stand (1=finishing stand, numbering opposite the sheet running direction).

The columns % display the relative stress of the band edge related to the yield point. A value of 100 means: the stress will presumably reach the yield point. Because the maximum stress occurs shortly in front of a stand, it is allocated to the stand. Example: In the line of stand 1 the stress is displayed that occurs in front of stand 1 (seen from stand 2).

The two bar diagrams show the relative stress for the left and the right side of the profile in a clearly laid out manner. So it is easy to check if the stresses of all stands are below the yield point and are distributed uniformly. If necessary, you can modify and optimize the bending angles.

The following rules are important:

- avoid increasing the stress of the edge to or over the limit of the yield point,
- all stresses at the edge should have nearly the same amount (with exception of the first and the last bending step),
- in the first and the last passes the stress should be smaller than in the other passes.

You define by using the switch [Profile, Loaded](#), whether the stress of the band edge is calculated for the discharged or the loaded state of the profile. The status of the switch is displayed by a coloured background of the table fields in the profile list window.

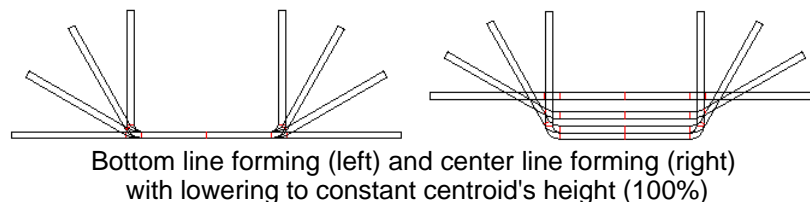
**Center Line Forming** (also called downhill forming or natural forming) is often used to reduce the strain at the edge. In this case, decrease the position of the web of the profile from pass to pass in order to reduce the ways of the band edges. If the centroids of all profile cross sections have the same height, the stress is partitioned to the whole profile more regular.

After checking the box [Center Line Forming](#) the stresses are calculated for constant height of centroids and you can see the effect of lowering to the stress. If this solves the stress problem you can decide if you want to use this method. Proceed the lowering by using the function [Profile, Center Line Forming](#).

#### Hints:

- Use the functions [File, Print Preview](#) or [File, Print](#) to print the table of stress.
- In order to get the stresses in the whole profile (i.e. not only at the band edge), use the function [View, PSA - Profile-Stress-Analysis](#).

#### 3.1.5.2.1 Center Line Forming



**Center line forming** also is called **Downhill Forming** or **Natural Forming**.

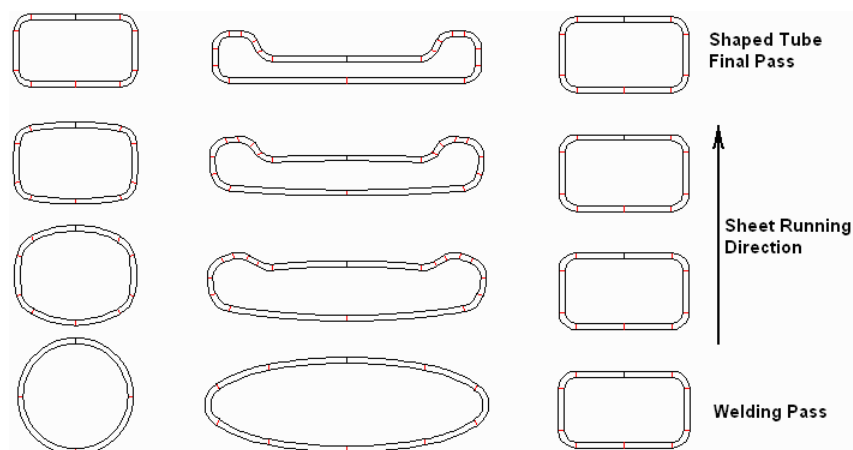
**Center Line Forming** is a checkbox in the window [Calculate, Stress of Edge](#). For a preview, switch the calculation method for the stress of the band edge from bottom line forming, i.e. constant level of the web of the profile, to center line forming and back again. So you quickly can check, if center line forming solves a problem of excessive stress or if your machine has not enough stands.

**Center line forming** means to keep the [Centroids of an Area](#) of all passes on a constant level. This is caused by decreasing the web from one stand to another. And also the stress of the band edge normally will decrease.

This enables you to decide, if center line forming solves a problem. But remember that your machine only allows center line forming, if the shafts are adjustable. If you want to use this method, proceed the final lowering by calling the function [Center Line Forming](#).

See also: [Calculate, Stress of Edge](#).

### 3.1.5.3 Shaped Tube Calibration



Shaped tube calibration from a round tube (left), from an elliptical tube (center), and by keeping the cross-section unchanged (right)

This function is a part of the [Toolbox Tube Design](#) and creates the calibrating stands for a shaped tube. It also can be used to calibrate a round tube.


A shaped tube is a tube with any symmetrical or unsymmetrical but closed cross- section. It is manufactured by a welded tube or a welded elliptic tube that is gradually formed by a certain count of calibrating stands. A [Calibrating Factor](#) defines how much the developed length of the tube decreases in each stand. Furthermore a [Deformation Degree](#) enables to partition the 100% of the forming between the welded tube and the shaped tube to each stand.

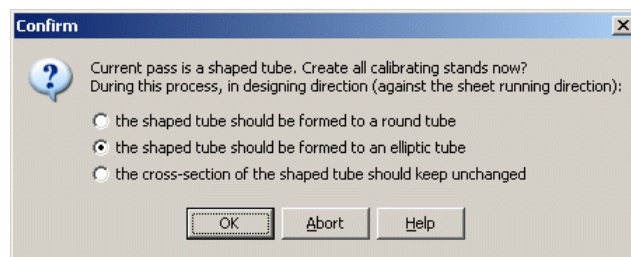
#### Calling the function

Before you call this function, use the [Toolbox Profile Design](#), the [Graphical Method](#), or the [Numerical Method](#) to define the cross-section of the shaped tube. It can be symmetrical or unsymmetrical with any count of profile elements. Take care that both ends touch together, both at the outer and inner side of the sheet. If the final product should be a calibrated round tube, use the function [Welding Pass](#) to define the final tube and enter 0 for the addition for welding.

Select a [Machine](#) that contains calibrating stands. For each calibrating stand define the [Calibrating Factor](#) and the [Deformation Degree](#).

Call this function optionally by:

- Main menu: **Calculate, Shaped Tube Calibration.**
-  Button **Shaped Tube Calibration** in the [Toolbox Tube Design](#).



After checking if the given cross-section is a closed shaped tube, a dialog window is opened with the question: **Current pass is a shaped tube. Create all calibrating passes now? During this process, in designing direction (against the sheet running direction):**

You can select between three items (see also the drawing above):

- **the shaped tube should be formed to a round tube.** The calibrating passes are created dependent on calibration factor and deformation degree. Finally, a round tube is created at the stand that has the deformation degree zero for the first occurrence (normally the welding station, a calibrating stand is possible, however). Because there are many possible cross

section patterns for the calibrating stands, PROFIL has a built-in random generator that calculates 10 various solutions for the problem. 9 of them are thrown away and the best with the minimum horizontal deviation of centroid of the area is kept. So you will get another result every time you call the function for the same shaped tube repeatedly.

- **the shaped tube should be formed to a elliptic tube.** This method should be used if the shaped tube is either quite large and flat or quite high and narrow, this means it differs extensively from the square form (with same width and height). High deformation would be required to calibrate this kind of shaped tube from a round tube. Better is to calibrate it from an elliptic tube. The input window **Aspect ratio major/minor axis of the ellipse** requests to enter the desired ratio. The value must be between 1.1 and 16.0. PROFIL decides by itself dependent on the position of the final shaped tube whether a flat or upright ellipse is created. Get further details from the selection "to a round tube".
- **the cross-section of the shaped tube should keep unchanged.** The deformation degree is ignored and dependent on the calibrating factor the straight length of each profile element is increased.

### Principle of operation

Starting with the current profile list (normally L01) the passes for all calibrating stands are created automatically, calibrating factor and deformation degree are considered. If profile lists with a number higher than the current yet exist, you are asked if they can be deleted.

### Modify the reference point that the welding seam is at the desired position?

This question appears if you have defined an unsymmetrical shaped tube and the developed lengths differ on the left and right side. Because the welding seam always is at the top of the round tube in the welding station, it is not possible both to keep the reference point and to set the welding seam to the point where the left and the right side touch themselves. If you answer "Yes" the reference point is moved (and the shaped tube is rotated if necessary) in order to get the welding seam at the desired position. If you answer "No" the reference point is kept and the position of the welding seam is set to the point that has the same developed length on the left and right from the reference point. If you later want to use [PSA - Profile Stress Analysis](#) or [FEA - Finite-Element-Analysis](#) you have to answer "Yes" to this question.

### Shaped Tube Calibration: No valid solution found!

If this message is shown, PROFIL was not able to convert the shaped tube cross-section into an elliptic cross-section. The reason: Each side of the ellipse is represented by a sequence of 5 arc elements: the first (in case of a flat ellipse) is an arc with a large radius, then comes a transition with a medium radius and then at the right or left an arc with a small radius. Afterwards again a medium radius and a large radius at the top. If the shaped tube has element lengths that not at all fit to the 5 ellipse elements, PROFIL cannot convert them correctly. Workaround: Split the much too large elements of the shaped tube into two parts with the same sum of developed lengths.

If a round tube is created in the welding station, use the function [Welding Pass](#) in order to attach the addition for welding. Otherwise, increase the ends of the profile by modifying the profile elements in the profile list. Also the function [Profile, Modify Strip Width](#) can be used for this purpose.

The [Reference Points \(x0/y0\)](#) both for the shaped tube and for all calibration passes are modified in this way that the centroids of the area are in the center of the round tube of the welding station.

### Hints:

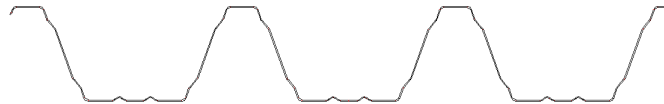
- If you created a round tube, use for the tube forming the functions:  
[Fin Pass](#)  
[Break Down Pass](#)  
[Break Down Pass, W-Forming](#)
- Create the rolls for the calibrating stands by using the functions:  
[Roll, Scan Profile Drawing](#)  
[Roll Read CAD-Contour](#)
- Create the rolls for the tube forming passes by using the functions:  
[Fin Pass, Top Roll](#)  
[Fin Pass, Bottom Roll](#)



[Break Down Pass, Top Roll](#)  
[Break Down Pass, Bottom Roll](#)  
[Fin Pass, Side Rolls](#)  
[Break Down Pass, Side Rolls](#)

- Another method is to handle the shaped tube as an open profile. This means, it becomes the desired final shape already in the welding station. Afterwards it can be calibrated in order to get a shaped tube with small allowances.

#### 3.1.5.4 Trapezoidal Profile Forming



When wide sheet metal strips are roll formed to a trapezoidal profile, the course of the band edge has a crucial influence on the quality of the product. On the one hand the course must be as short as possible in order to form the band edge elastically only. The shortest connection between two points is the straight line - but the band edge is creased both at the machine entrance and exit. This is why the second important point of view for choosing the right band edge course is the tangential transition at entrance and exit. In practise, a linear function with fillet radii is often used or a cosine function. **PROFIL** is able to create the flower pattern for trapezoidal profile automatically.

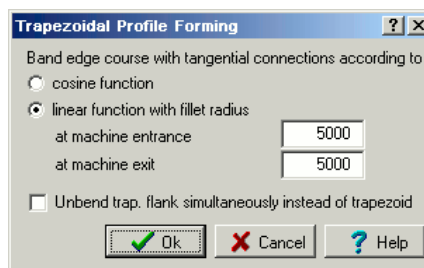


The function can be used for the forming of corrugated sheet, too (see Hint).

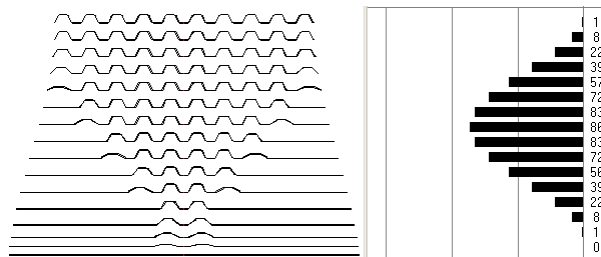
##### Calling the function

Before you call this function, define the final trapezoidal profile by the function [Trapezoidal Profile](#) from the [Toolbox Profile Design](#) or by CAD drawing, see [Graphical Method](#). Select the pass that should be handled as the final pass, normally L01. Call this function by:

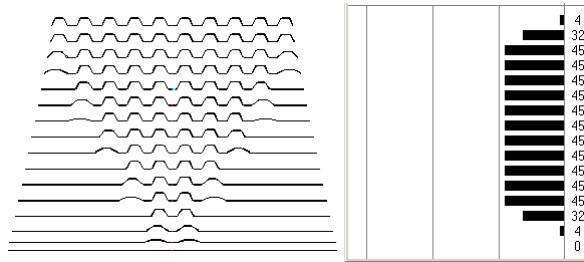
- Main menu: **Calculate, Trapezoidal Profile Forming.**



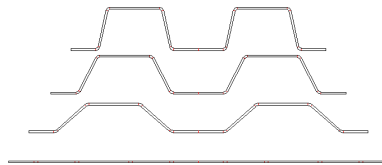
A dialog window is opened with the question: **Band edge course with tangential connections according to:**



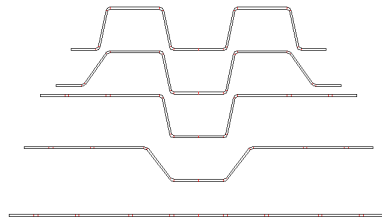
- **Cosine function:** The **cosine band edge course** (left) creates a **sine course of the stress of band edge** (right). The advantage of this method is the very smooth transition both from the entering flat sheet and to the ready product leaving the machine.



- **Linear function with fillet radius:** The **linear band edge course** (left) creates a **linear course of the stress of band edge** (right) and with it the smallest count of roll forming stands. Fillet radii at the machine entrance and exit prevent creases.
- **Unbend trap. flank simultaneously instead of trapezoid:** (can be selected both for cosine and linear function) The roll forming process always starts with the inner trapezoids. This is why the material should be able to "flow" to the inside during bending. The user can select if the entire trapezoid or the trapezoid's flank is formed simultaneously.



The **trapezoid is formed simultaneously**: fewer stands are needed. Problem: when the material is impeded to "flow" to the inside, this may cause deep drawing effects.

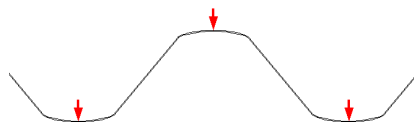


The **trapezoid's flank is formed simultaneously**: more stands are necessary, the material can better "flow" to the inside. Problem: the band edge moves up and down, this causes higher stress. The stress can be reduced by [Center Line Forming](#).

### Principle of operation

After checking, if the given cross-section is a trapezoidal profile (i.e. it has horizontal top and bottom lines), the function creates the flower pattern for a trapezoidal profile automatically, either with cosine band edge course or linear course with user defined fillet radii as desired. If passes already exist after the selected pass, a dialog box asks if they can be removed

### Hints:



- In order to enable **PROFIL** to recognize the trapezoids correctly, it is necessary that the trapezoids have a horizontal line or a tangential connection between two arcs at the top and the bottom of each trapezoid. If they have not (e.g. corrugated sheet), simply split the arcs at the top and bottom point (quad point 90° and 270°). You can do this either in CAD before [Reading the CAD Contour](#) or in PROFIL by the function [Profile, Element, Split](#).
- **Sharp edges**, i.e. arcs with inner radius 0, should be avoided. This is why the automatic trapezoidal profil forming works numerically and cannot find a proper solution if the function is

discontinuous. Use radii not less than 0.2 mm inner radius instead.

### 3.1.5.5 Required Count of Stands

During processing a quotation and a cost assessment the cost of a roll tool set or a roll forming machine must be estimated roughly. The expected count of passes or stands for a given profile cross section is needed for this operation.

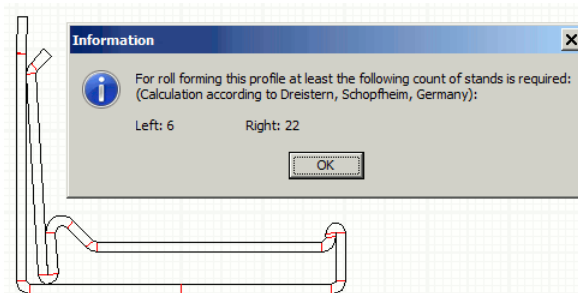
#### Calling the function

Define as pass **L01** the profile that the estimated count of stands is needed for. Use the [Toolbox Profile Design](#), the [Graphical Method](#) or the [Numerical Method](#) for it. Select pass **L01**.

Call this function by:

- Main menu: **Calculate, Required Count of Stands**.

#### Principle of operation



A window opens and shows the estimated count of stands for the left and right half of the profile. Left side is the side left of the [Reference Point X0/Y0](#), provided that the [Start Direction](#) points to the right. The calculation is according the method of the company **Dreistern, Germany**.

#### Hint:

- If you have not selected the final stand **L01** previously, but another stand of a already (partly) existing flower pattern, a message is shown. If you confirm, the estimated count of stands from the selected stand to the flat sheet is calculated and displayed.

### 3.1.5.6 Plausibility Check

This function is useful to check the whole project on:

- Errors in the logical structure of the profile list, e.g.: empty profile elements, negative radii, existence of P or PS, missing distance between stands.
- Overlap of roll contour arcs.

#### Calling the function

Call this function by:

- Main menu: **Calculate, Plausibility Check**.

#### Principle of operation

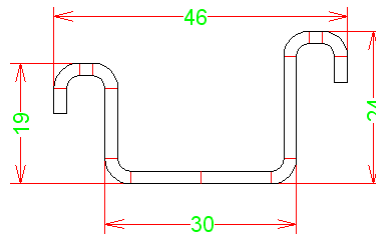
All passes and all rolls of the project are checked. When the first error is found, the program breaks and shows a message in the bottom dialog line at the bottom of the screen. At the same time the faulty pass or roll corner point is shown in the drawing area. After correction the plausibility can be restarted in order to find more errors.

#### Hint:

- It is strongly recommended to check the project before creating the NC data and before starting the FEA simulation. Otherwise implausible NC data could be created or the FEA simulation stops because solid bodies cannot be created from the transferred roll contours.

### 3.1.6 View


#### 3.1.6.1 Pass



Use this function to display the drawing of the profile in the [Drawing Area](#).

#### Calling the function

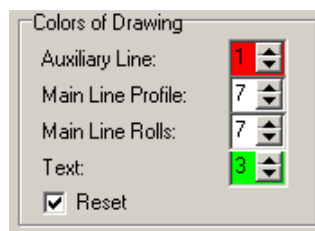
Select the pass that should be displayed. Call this function optionally by:

- Main menu: **View, Pass**.
-  Button **View Pass** in the [Button Key Bar](#).

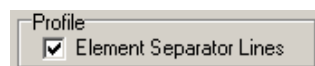
#### Principle of operation

The pass is displayed graphically in the [Drawing Area](#). The switch [Profile, Loaded](#) determines whether the pass is displayed discharged or loaded. If you have entered [Holes/Cut-Outs](#) into the profile list, they will be displayed too.

#### Setup



Select the **Auxiliary Line Color** and the **Main Line Color Profile** in [Options Drawing](#).

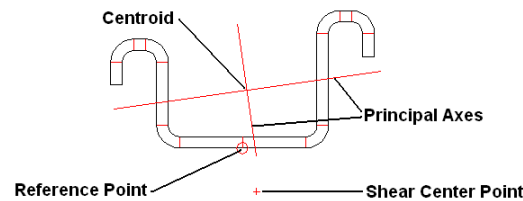


In [Options Drawing](#), **Profile** you can switch off the **Element Separator Lines** (between line and arc segments).

#### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- Use the [Toolbox Modify](#) to bend the profile up and down and to modify the size of the profile.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).

### 3.1.6.2 Statics




Statics:	Related to:	x	y	
Centroid of an Area	Ref. Point x0/y0	0.799	10.138	mm
Shear Center Point	Ref. Point x0/y0	2.469	-7.581	mm
Moment of Inertia	x/y in Centroid	1.078	3.751	cm4
Moment of Inertia	in Principal Axes	3.806	1.023	cm4
Moment of Resistan.	x/y in Centroid	0.777	1.576	cm3
Moment of Resistan.	In Principal Axes	1.624	0.882	cm3
Max. Distance	In Principal Axes	1.159	2.344	cm
Inertial Radius	In Principal Axes	1.473	0.764	cm
Area of Cross Sec.		1.753		cm2
Weight without Hole		1.376		kg/m
Princip. Axes Angle	x-Axis in Centroid	98.074		deg
Vault Resistance	Shear Center Point	1.662		cm6
Torsion Moment	Centroid	0.023		cm4

Use this function to display the profile discharged in the [Drawing Area](#). In addition the drawing will be enhanced with the static parameters. Thus the they can be transferred to the [CAD System](#) by using the function [Drawing -> CAD](#).

#### Calling the function

Before you call this function, select the pass that should be calculated. Call this function optionally by:

- Main menu: **View, Statics**.
-  Button **View Statics** in the [Button Key Bar](#).

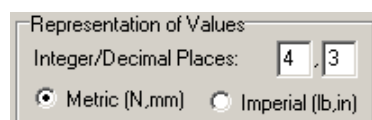
#### Principle of operation

In the [Drawing Area](#) the profile pass, the statics table, and additionally all static parameters that can be drawn graphically, are displayed in the drawing:

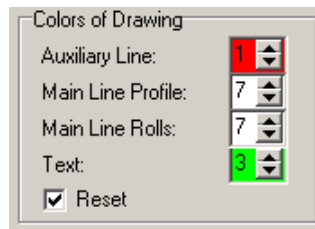
- [Centroid](#) = Origin of the **Principal Axes**
- [In Principal Axes](#) = Large axes system
- [Max. Distance](#) = Lengths of the **Principal Axes**
- [Principal Axes Angle](#) = Angle of the larger axis of the **Principal Axes** against the horizontal line
- [Shear Center Point](#) = small cross
- [Reference Point](#) = small circle

The statics are calculated by using the discharged columns of the profile list only; they are not dependent on the status of the menu switch [Profile, Loaded](#). [Holes/Cut-Outs](#) will weaken the area of cross of the profile. The calculation of statics considers this and they will be shown in the drawing. Exception: for the [Weight](#) holes/cut-outs are not considered since input of count and length is not supported.

#### Setup



Select in [Options Calculate](#), **Representation of Values** whether the statics should be displayed in the **Metric System** (mm, N) or the **Imperial System** (in, lb.). Set the count of **Integer/Decimal Places** as well.



Select the **Auxiliary Line Color**, **Main Line Color Profile**, and **Text Color** in [Options Drawing](#).

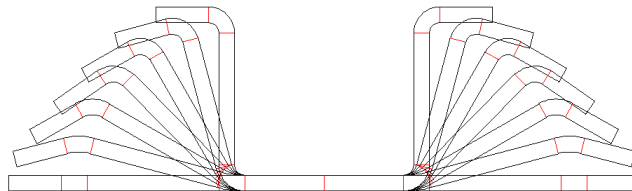


Preset the text height for the table of statics in [Options Drawing](#), **Text**, **Height**.

#### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).

### 3.1.6.3 Flower Nested



Use this function to display the flower pattern nested of the profile project in the [Drawing Area](#). Nested means, the webs of all passes have the same y-coordinate.

#### Calling the function

Call this function optionally by:

- Main menu: **View, Flower, Nested**.
- Button **View Flower Nested** in the [Button Key Bar](#).

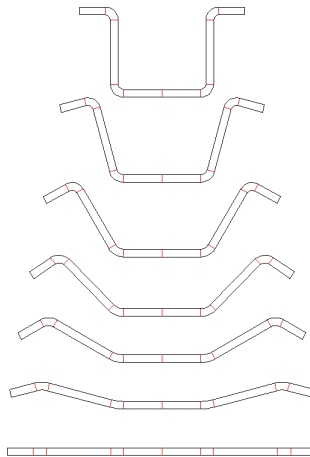
#### Principle of operation

The flower pattern nested of the project is displayed in the [Drawing Area](#). Settings are the same as described in [View Pass](#).

#### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- Use the [Toolbox Modify](#) to bend the profile up and down and to modify the size of the profile.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).
- If the drawing is not clear (e.g. in case of complicated profiles) use the function [View, Flower, separated](#).


### 3.1.6.4 Flower Separated



Use this function to display the flower pattern separated of the profile project in the [Drawing Area](#). Separated means, the webs of all passes have different y-coordinates.

#### Calling the function

Call this function optionally by:

- Main menu: **View, Flower, Separated**.
-  Button **View Flower Separated** in the [Button Key Bar](#).

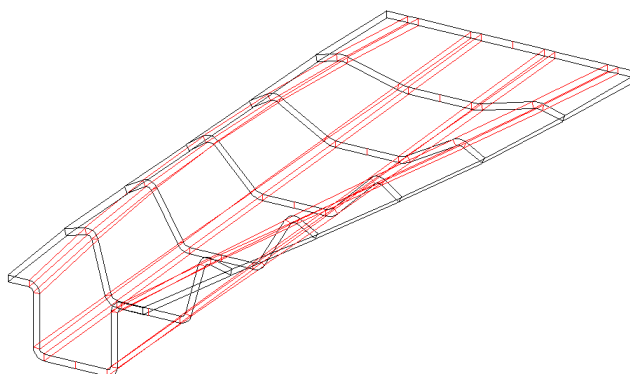
#### Principle of operation

The flower pattern of the project is displayed separately in the [Drawing Area](#). This kind of display has the advantage that the flower pattern is shown more clearly in case of complicated profiles. Settings are the same as described in [View Pass](#).

#### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- Use the [Toolbox Modify](#) to bend the profile up and down and to modify the size of the profile.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).
- Another kind of displaying the flower pattern is [View Flower Nested](#), [View Flower 3D](#).


### 3.1.6.5 Flower 3D



Use this function to display the flower pattern 3D of the profile project in the [Drawing Area](#). The passes are displayed in a perspective view. So you are able to examine, if the course of the band edge is smoothly without detours. This is important in order to minimize the unwanted stress of band edge.

### Calling the function

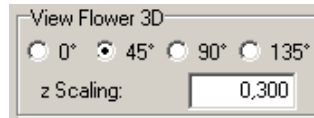
Call this function optionally by:

- Main menu: **View, Flower, 3D.**
-  Button **View Flower 3D** in the [Button Key Bar](#).

### Principle of operation

The flower pattern 3D of the project is displayed in the [Drawing Area](#). Settings are the same as described in [View Pass](#).

### Setup

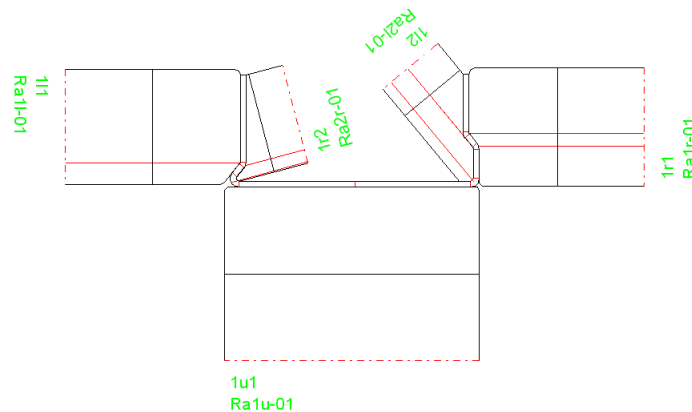


Select the view port angle, and the scaling factor of the z axis in [Options Drawing](#). The settings also can be modified in the context menu (right mouse button).

### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- By using the [Navigator 3D](#) you can rotate the drawing and toggle to a 2D view.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).
- Another kind of displaying the flower pattern is [View Flower Nested](#), [View Flower Separated](#)

## 3.1.6.6 Roll Tools



Use this function to display the roll tools of a stand in the [Drawing Area](#).

### Calling the function

Select the roll stand that should be displayed. Call this function optionally by:

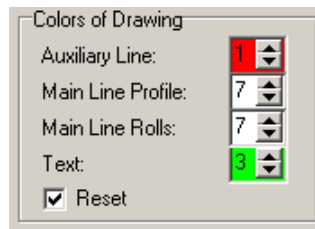
- Main menu: **View, Roll Tools.**
-  Button **View Roll Tools** in the [Button Key Bar](#).

### Principle of operation

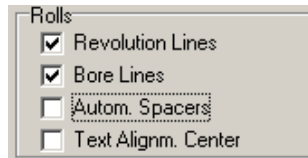
The roll tools of the selected stand are displayed in the [Drawing Area](#). Furthermore the profile pass is shown between the rolls. You define by setting the menu switch [Profile, Loaded](#) whether the pass inside the roll tool drawing is displayed discharged or loaded.



## Setup



Select the **Auxiliary Line Color**, the **Main Line Color** Rolls and the **Text Color** in [Options Drawing](#).



In the same dialog, you can define if the roll drawing should contain **Revolution Lines**, **Bore Lines**, and **Automatic Spacers**. **Text Alignment Center** positions roll number and part number in the center of a roll instead on the left corner.

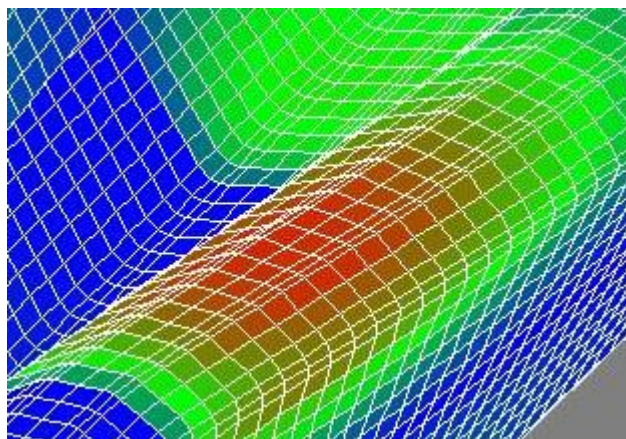


Preset the text height for the roll number and part number in [Options Drawing](#), **Text**, **Height**.

### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- Use the [Toolbox Modify](#) to modify the width, diameter or radius of a corner point of the roll.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).

### 3.1.6.7 PSA - Profile Stress Analysis



The Profile Stress Analysis is the second step of the three step quality concept for [Quality Management](#). The surface of the strip is divided into small rectangular shell elements. While running through the roll forming machine, the shell elements are deformed, this means the lengths of the edges are changed. From these changes, the strains and stresses in longitudinal direction are calculated and displayed by colored faces in the [Drawing Area](#). The calculation works quickly and approximately and does not need FEA (Finite Element Analysis).

Compared with step 1 ([Stress of Edge](#)) the Profile Stress Analysis has the advantage that the longitudinal stresses are not only calculated at the edge but within the whole profile. This is

important when the maximum stress is not at the edge, e.g. when edges are folded and the folds are bent.

Compared with step 3 ([FEA - Finite Element Analysis](#)) the Profile Stress Analysis has the advantage that the method is very quick and can be used while designing. On the other hand, FEA calculates much more precisely and not only stress and strain, but also the final profile pattern is calculated.

### Calling the function

Select the start pass for the calculation. Call this function optionally by:

- Main menu: **View, PSA - Profile Stress Analysis**.
-  Button **PSA - Profile Stress Analysis** in the [Button Key Bar](#).

### Principle of operation

A 3D drawing is created in the [Drawing Area](#) that shows the strip while running through the roll forming machine.

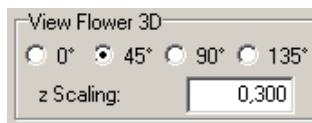


You can modify the analysis range by using the **From Pass** and **To Pass** selectors.

Stresses are calculated by colored faces. To assign the colors to the stresses and to set-up the width and length of the shell segments and for furthermore settings, call [Options PSA](#).

The bottom status bar shows the maximum relative stress in % related to the yield point. The analysis range **From Pass, To Pass** is considered. By moving the range limits you can explore points of high stress.

### Setup



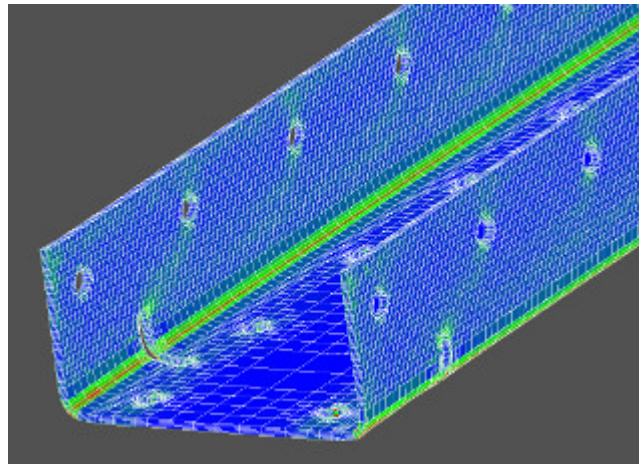
Select the initial view port angle in [Options Drawing](#) or modify the angle by using the context menu (right mouse button).

### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- By using the [Navigator 3D](#) you can rotate the drawing and toggle to a 2D view.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).

## 3.1.6.8 FEA-Result

### 3.1.6.8.1 LS-Dyna




Result of an FEA simulation with **LS-Dyna**, shown in the **PROFIL** drawing area

Use this function to display the result of the roll forming simulation with **LS-Dyna** by [FEA - Finite Element Method](#) in the [Drawing Area](#).

This function currently is possible only if the element type **Shell** is selected, see [Output, FEA, LS-Dyna, Profile](#). In a later version this function will be available for element type **Solid**, too.

#### Calling the function

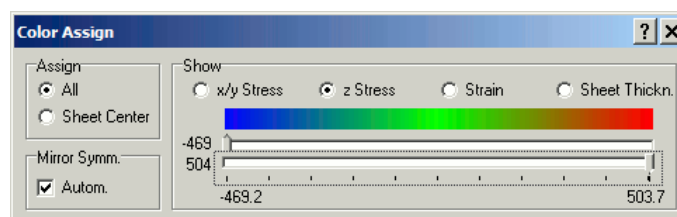
Before you call this function, proceed the [FEA Simulation](#). The **LS-Dyna** Solver creates a set of **.d3plot** files (one for each stand) that is opened with **View, FEA Result**. Open the project file **.pro** that contains the belonging project. Call this function optionally by:


- Main menu: **View, FEA Result LS-Dyna**.
-  Button **View FEA Result** in the [Button Key Bar](#).

#### Principle of operation

The FEA result is displayed as a 3D graphic. Select the initial view port angle in [Options Drawing](#) or modify the angle by using the context menu (right mouse button).

The button bar on the right enables you to modify the view.



 **Color Assign:** By pressing this button the window Color Assign is opened. By moving the tabs you can assign the colors blue and red to any strain, stress, and sheet thickness. Furthermore you can select whether all 5 layers within the sheet thickness should get different colors (**Assign All**) or if the color of all layers should be assigned to the stress/strain/thickness of the centerline of the sheet (**Assign Sheet Center**). The last one is useful if you do not want to see the (desirable) stress and strain on the sheet top or bottom surface, but only the (undesirable) deformation of the sheet center. In case of a symmetrical profile only one half is simulated for optimizing the calculation time. Select if the result should be mirrored automatically (**Mirror Symm. Autom.**).



**From Layer, To Layer:** In [Output FEA, LS-Dyna](#), page "Profile, Meshing in Sheet Running Direction, Analysis Count" is defined, how many layers in longitudinal direction are simulated.

Select which layers should be displayed.



**Time Interval:** In [Output FEA, LS-Dyna](#), page "Meshing Long., Temporally, Intervals Count" is defined, how many time intervals should be saved into the results file. Select which time interval should be displayed.



**2D Rolls on/off:** This button can be used to switch on/off the 2D drawing of the designed rolls within the 3D drawing of the FEA result.



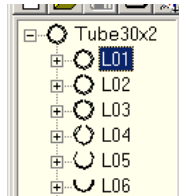
**3D Rolls on/off:** This button can be used to switch on/off the 3D models of the designed rolls within the 3D drawing of the FEA result.



**One layer only:** Use this button to reduce the FEA result to one layer only (see also **From Layer, To Layer**). Either layer 1 is shown or - if you have selected a stand in the [Profile Explorer](#) previously - the layer that is between the rolls of the selected stand (see also ). Press again this button to show all layers again.



**Graph:** This button opens the [Graph](#) window that shows the stress, strain, and thickness dependant on the position of the sheet in the machine in sheet running direction.



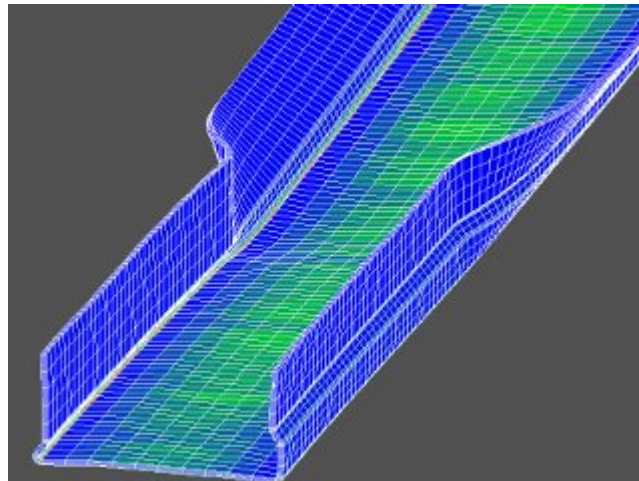
#### Show layer between the rolls:

By selecting a stand entry in the [Profile Explorer](#) the selector **From Layer, To Layer** is set automatically to the layer that is between the rolls of the selected stand (Precondition: the layer exists in the FEA result). To reset the selection click on the center button of the [Navigator 3D](#). or on the button **One layer only**.

#### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- By using the [Navigator 3D](#) you can rotate the drawing and toggle to a 2D view.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).

### 3.1.6.8.2 ABAQUS




Result of an FEA simulation with ABAQUS/Explicit, shown in the PROFIL drawing area

Use this function to display the result of the roll forming simulation with **ABAQUS/Explicit** by [FEA - Finite Element Method](#) in the [Drawing Area](#).

#### Calling the function

Before you call this function, proceed the [FEA Simulation](#). The **ABAQUS** Solver creates a **.FIL** file that is opened with **View, FEA Result**. Open the project file **.pro** that contains the belonging project. Call this function optionally by:

- Main menu: **View, FEA Result, ABAQUS**.
-  Button **View FEA Result** in the [Button Key Bar](#).

Select in [Output FEA](#), page **Files** whether the FEA result of the currently opened project should be displayed or if you want to select the desired FEA result within the file selection window. The file selection window appears as well, if the FEA result file of the current project does not exist. If you have moved the result file into a new path, you can save the new path by function [Save Project](#) into the project file afterwards.

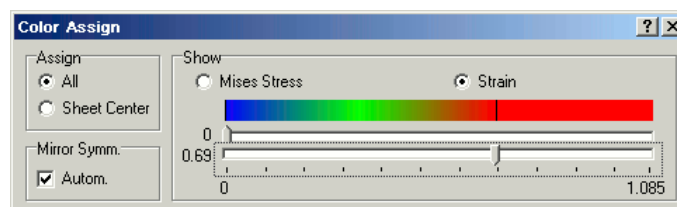
#### Principle of operation

The FEA result is displayed as a 3D graphic. Select the initial view port angle in [Options Drawing](#) or modify the angle by using the context menu (right mouse button).

The button bar on the right enables you to modify the view.



**Mean Energy Ratio:** If the time increment for the mass scaling (selected in [Output FEA](#), page "Others") is too large, the FEA result becomes invalid. To check the validity the mean energy ratio is displayed. For a valid result the mean energy ratio should be  $< 0.025$ .



**Color Assign:** By pressing this button the window Color Assign is opened. By moving the tabs you can assign the colors blue and red to any strain and stress. Furthermore you can select whether all 5 layers within the sheet thickness should get different colors (**Assign All**) or if the color of all layers should be assigned to the stress/strain of the centerline of the strip (**Assign Sheet Center**). The last one is useful if you do not want to see the (desirable) stress and strain on

the sheet top or bottom surface, but only the (undesirable) deformation of the sheet center. In case of a symmetrical profile only one half is simulated for optimizing the calculation time. Select if the result should be mirrored automatically (**Mirror Symm. Autom.**).



**From Layer, To Layer:** In [Output FEA, ABAQUS](#), page "Meshing Long., Analysis Count" is defined, how many layers in longitudinal direction are simulated. Select which layers should be displayed.



**Time Interval:** In [Output FEA, ABAQUS](#), page "Meshing Long., Temporally, Intervals Count" is defined, how many time intervals should be saved into the results file. Select which time interval should be displayed.



**2D Rolls on/off:** This button can be used to switch on/off the 2D drawing of the designed rolls within the 3D drawing of the FEA result.



**AVI-File:** like **Video Start/Stop**, but in addition the video is saved into an AVI file. Afterwards the WINDOWS Media Player is started to run the video. Remark: Both for creating and for running the video a compression method is used that is available in newer WINDOWS versions only!



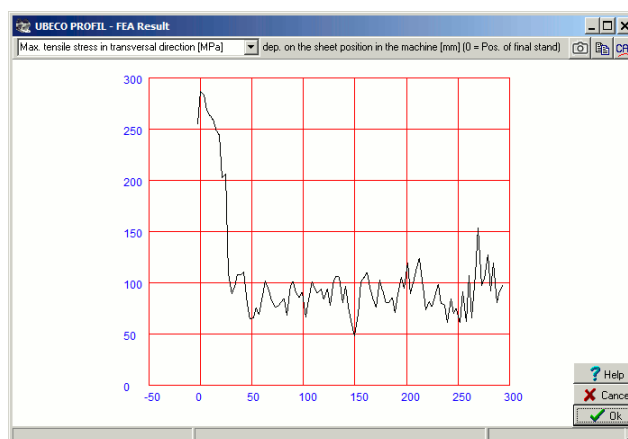
**Video Start/Stop:** The time intervals are displayed sequential, this means the FEA simulation result is shown like a video, as desired with or without rolls. After a short waiting period the video is repeated automatically until the stop button is pressed.

By selecting a stand entry in the [Profile Explorer](#) the selector **From Layer, To Layer** is set automatically to the layer that currently is within the rolls of the selected stand (Precondition: the layer exists in the FEA result). To reset the selection click on the center button of the [Navigator 3D](#).

#### Hints:

- The [Navigator](#) helps you to zoom, to move and to fit the drawing.
- By using the [Navigator 3D](#) you can rotate the drawing and toggle to a 2D view.
- With the function [Output, Drawing -> CAD](#) you can transfer the drawing to your [CAD System](#).

### 3.1.6.8.3 Graph



The graph shows the stress, strain, and thickness dependant on the position of the sheet in the machine in sheet running direction (for **LS-Dyna** simulation result only).

#### Calling the function






**Graph:** This button is visible at the right corner of the **PROFIL** main screen after selecting a



LS-Dyna simulation result by using the function [View FEA Result LS-Dyna](#). The window **Graph** opens and shows stress, strain, or thickness of the FEA result.

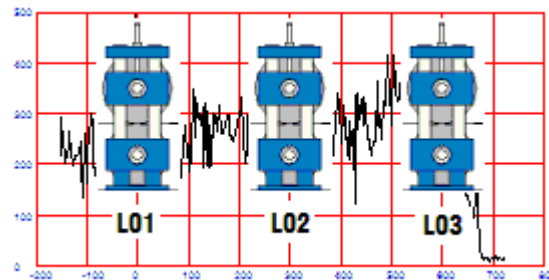
### Modifying the graph

The graph drawing can be modified by:

- **Selection:** Select if you want to see the **Max. tensile stress in transversal direction [MPa]**, the **Max. tensile stress in sheet running direction [MPa]**, the **Max. tensile strain [%]**, or the **Min. sheet thickness [mm]**, each dependant on the sheet position in the machine [mm] (0 = Pos. of final stand).
-  **Copy Snapshot to Assembly Window:** Opens the new window **FEA Result Assembly**, into which all values are copied and saved, see also **Hints**.
-  **Copy drawing to clipboard:** Use this button to copy the graph as pixel drawing to the Windows clipboard (see also [Edit Copy](#)).
-  **Drawing -> CAD:** Use this button to transfer the graph as vector drawing to the CAD system. The settings of the function [Drawing -> CAD](#) in the PROFIL main window are used as well.

### Principle of operation

The graph shows the course of the stress, strain, and thickness of the current visible FEA result dependant on the position of the sheet in the machine in sheet running direction. The sheet position is:

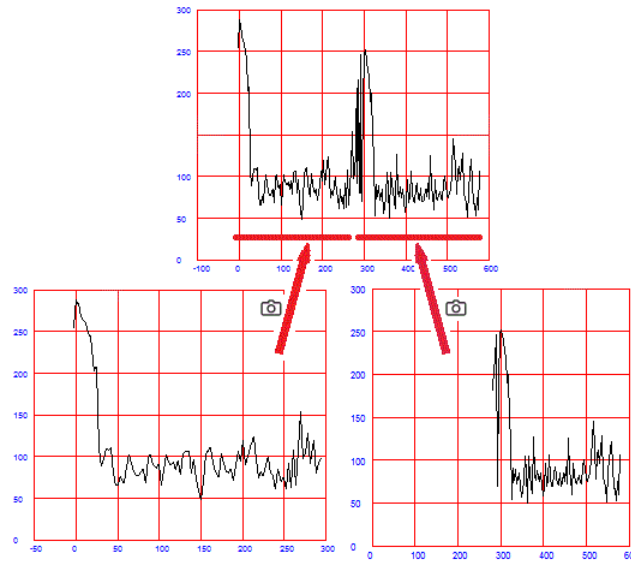


Example: 3 stands with distance 300mm each






- **< 0:** Region after leaving the final stand (L01)
- **= 0:** Position of the final stand (L01)
- **> 0:** Range within the machine and region before entering the first stand. Stand position at  $n \cdot \text{distance between stands}$  ( $n = \text{no. of the stand}$ )

### Hints:

The graph always shows the FEA result values of one time interval. If you need the graph displayed for a larger range or for the whole machine plus the region after the machine exit, proceed like this:



How to use the  **Copy Snapshot to Assembly** function

- Select a time interval and open the  **Graph** (pic. bottom left).
- Press  **Copy Snapshot to Assembly Window**. The **FEA Result Assembly** window opens (top pic.) that first is identical with the graph window.
- Select another time interval (e.g. via  **Time Interval** or or via  **View, FEA Result**. The graph window shows the values of the new selected time interval (pic. bottom right). Press again  **Copy Snapshot to Assembly Window**. The assembly (top picture) is extended by the new values. In case values already exist (at the same sheet position), the medium value is calculated.
- Repeat this procedure for other time intervals until the desired region is covered.

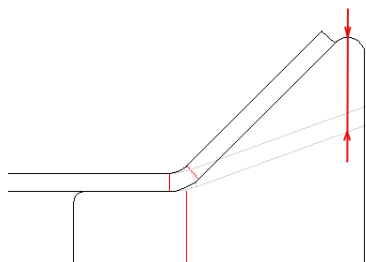


- In case the sheet front or back-end has unwanted deformation that should not be displayed in the assembly, reduce the length of the evaluated sheet section by using the selection boxes **From Layer, To Layer**. Then copy the snapshot to the assembly window.

### 3.1.6.9 Show

#### 3.1.6.9.1 Previous/Next Pass

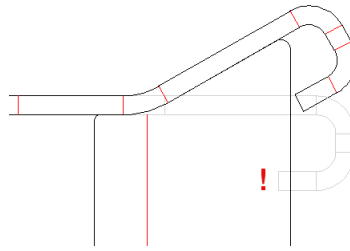
For the design of a roll tool set for a stand not only the form of the profile within the stand is important (= **current pass**), but also the form of the incoming profile from the previous stand (= **previous pass**) and the outgoing profile to the next stand (= **next pass**).



If the previous pass is shown, it is easy to find out which point of the running-in profile touches first which roll of the current stand. It is a good idea to provide a large radius instead of a sharp corner



to avoid scratches and to enable the profile to flow smoothly into the rolls.



Also risk of collision can be noticed easily, if the running-in profile is shown.

### Calling the function

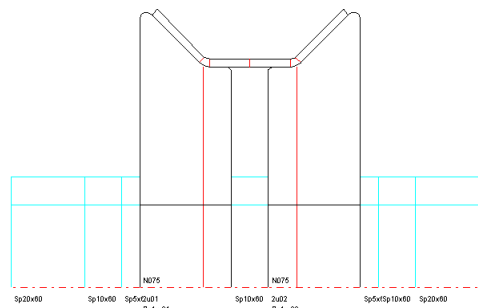
Call this function optionally by:

- Main menu: **View, Show, Previous Pass** or **View, Show, Next Pass**.
- **Function key F6** (for previous pass). The function key assignment can be modified in [Options, Keyboard, Shortcut Key Assignment](#).

### Principle of operation

By using this function the drawing of the previous and next pass can be inserted in the drawing of the [Roll Tools](#) of the current stand or in the drawing of the current [Pass](#). The previous and next passes always are displayed in the **Inactive Color** (Setup in [Options, Drawing, Colors of drawing area](#)). Printing and transfer to CAD is not possible.

### 3.1.6.9.2 Spacer Rolls



Spacer rolls are objects in the **PROFIL** object hierarchy like forming rolls and can be output, dimensioned, and modified as well. They are necessary to fix the horizontal position of the forming rolls in case the whole working width of the machine is not used for forming rolls. Because spacer rolls often are not manufactured but taken from a spacer stock, the spacer rolls can be switched off for output as desired.

### Calling the function

Call this function by:

- Main menu: **View, Show, Spacer Rolls**.

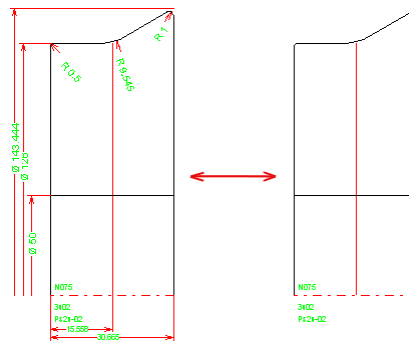
### Principle of operation

By using this switch you can switch off the spacer rolls temporarily in order not to display them in the [Drawing Area](#) and not to output them to [CAD](#), [Printer](#), [Plotter](#), [Parts List](#) and [NC-Program](#).

If the spacer rolls are switched off, they are switched on automatically by [Creating Spacer Rolls](#).

**Hint:** The difference between automatic spacers and spacer rolls is discussed in [Tutorial, Roll Tools, Creating Spacer Rolls](#).

### 3.1.6.9.3 Dimensioning



In case the dimensioning should be not visible, it can be switched off.

#### Calling the function

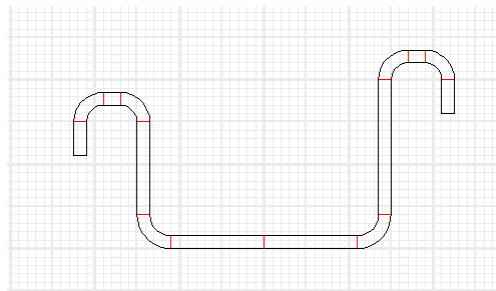
Call this function by:

- Main menu: **View, Show, Dimensioning.**

#### Principle of operation

This function switches all dimensions temporarily off and on again. In position **switched off** the dimensions are not displayed in the [Drawing Area](#), not transferred to CAD ([Drawing -> CAD](#)), not printed ([Print](#)), and not plotted ([Plot](#)).


### 3.1.6.9.4 Grid



In order to have a better overview of the size proportion during profile and roll design it is helpful to have a graph paper in the background of the [Drawing Area](#). Example: While choosing the width of side extensions at rolls. Grid lines distance and color can be preset by the user.

#### Calling the function

Call this function by:

- Main menu: **View, Show, Grid.**
-  Button **Grid on-off** in the [Button Key Bar](#).

#### Principle of operation

This function switches the grid lines on and off. After switching on, the dialog line at the bottom of the screen shows, which grid lines distance currently is preset.

#### Hints:

- Preset in [Options, Drawing](#) the desired grid lines distance.
- Preset in [Options, Colors](#) the desired color of the grid lines.
- In order to get the precise dimensions of the drawing objects, use the [Toolbox Dimensioning](#).


### 3.1.7 Toolboxes

#### 3.1.7.1 Profile Design

Use the Toolbox Profile Design for quick and easy design of simple or standard profiles without CAD.

##### Calling the function

Call this function optionally by:

- Main menu: **Toolbox, Profile Design**.
-  Button **Toolbox Profile Design** in the [Button Key Bar](#).

##### Content

**Section 1** contains cross-sectional elements for continuing the design, these elements can be mounted together to the final profile.



[Line](#)



[Arc](#)



[Partial Ellipse](#)



[Arc < 90° - Line](#)



[Arc > 90° - Line](#)



[Stage](#)



[Trapeziodal Profile](#)

**Section 2** contains basic profiles that can be extended afterwards by elements from section 1.



[U-Profile](#)



[C-Profile](#)

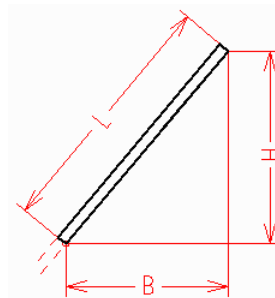


[Hat-Profile](#)



[Z-Profile](#)


##### 3.1.7.1.1 Line



This function is a part of the [Toolbox Profile Design](#) and generates a new element of type line.

#### Calling the function

Call this function by:

-  Button **Line** from the [Toolbox Profile Design](#).

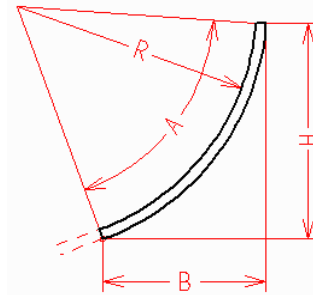
The window **Line** is opened. If you enter one of the three variables **Length**, **Width** and **Height**, the other two will be calculated. The length is always positive.

The insert point is the start point of the line element. Width and height are measured in the x/y-coordinates of the drawing, enter the correct sign dependent on the direction of the previous element.

#### Principle of operation

If you have selected a [Profile Element](#) previously, the new profile element will be appended to the selected one. Otherwise it will be appended to the end of the profile.


### 3.1.7.1.2 Arc



This function is a part of the [Toolbox Profile Design](#) and generates a new element of type arc.

#### Calling the function

Call this function by:

-  Button **Arc** from the [Toolbox Profile Design](#).

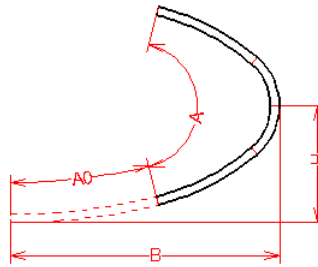
The window **Arc** is opened. If you enter **Angle** and **Radius**, width and height will be calculated. If the angle is negative, an arc to the right will be generated. If you enter **Width** and **Height**, angle and radius (and direction) will be calculated. The length is always positive.

The insert point is the intersection point of the tangents to the arc, this means the length of the previous element will be shortened. Width and height are measured in the x/y-coordinates of the drawing, enter the correct sign dependent on the direction of the previous element.

#### Principle of operation

If you have selected a [Profile Element](#) previously, the new profile element will be appended to the selected one. Otherwise it will be appended to the end of the profile.


### 3.1.7.1.3 Partial Ellipse



This function is a part of the [Toolbox Profile Design](#) and generates a set of arc elements that approximate a partial ellipse.

#### Calling the function

Call this function by:

-  Button **Partial Ellipse** from the [Toolbox Profile Design](#).

The window **Partial Ellipse** is opened.

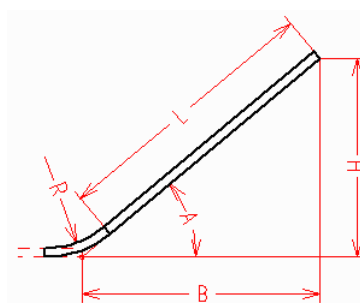
Enter the half **Major Axis Length B** and the half **Minor Axis Length H**. The major axis length always must be larger than the minor axis length. In case of  $H > 0$ : A counter clockwise partial ellipse is created.  $H < 0$ : A clockwise partial ellipse is created.

Enter the **Start Angle A0** and the **Apex Angle A** of the partial ellipse. In case of  $A0 = 0^\circ$  or  $180^\circ$ : The partial ellipse starts at the minor axis.  $A0 = 90^\circ$  or  $270^\circ$ : The partial ellipse starts at the major axis.

#### Principle of operation

If you have selected a [Profile Element](#) previously, the new profile element will be appended to the selected one. Otherwise it will be appended to the end of the profile.


### 3.1.7.1.4 Arc <90° - Line



This function is a part of the [Toolbox Profile Design](#) and generates a new element of type arc and a new element of type line. The dimension reference point is the intersection point of the tangents to the arc, so this function is useful for arcs until  $90^\circ$ .

#### Calling the function

Call this function by:

-  Button **Arc <90° - Line** from the [Toolbox Profile Design](#).

The window **Arc <90° - Line** is opened.

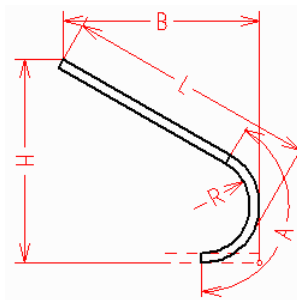
If you enter **Angle** and **Length**, width and height will be calculated. If you enter **Angle** and **Width**, length and height will be calculated. If you enter **Angle** and **Height**, length and width will be calculated. If you enter two of the three variables length, width and height, all other variables will be calculated. If the angle is negative, an arc to the right will be generated. If width and/or height are negative, the matching direction will be calculated.

The insert point is the intersection point of the tangents to the arc, this means the length of the previous element will be shortened. The width is measured in the direction of the previous element; the height perpendicular to it.

#### Principle of operation

If you have selected a [Profile Element](#) previously, the new profile elements will be appended to the selected one. Otherwise they will be appended to the end of the profile.


#### 3.1.7.1.5 Arc >90° - Line



This function is a part of the [Toolbox Profile Design](#) and generates a new element of type arc and a new element of type line. The dimension reference point is the maximum point of the arc, so this function is useful for arcs larger than 90°.

#### Calling the function

Call this function by:

-  Button **Arc >90° - Line** from the [Toolbox Profile Design](#).

The window **Arc >90° - Line** is opened.

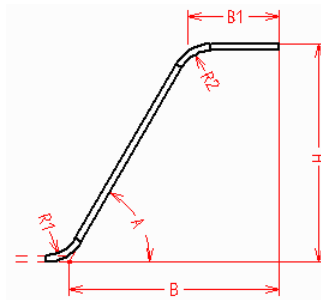
If you enter **Angle** and **Length**, width and height will be calculated. If you enter **Angle** and **Width**, length and height will be calculated. If you enter **Angle** and **Height**, length and width will be calculated. If you enter two of the three variables length, width and height, all other variables will be calculated. If the angle is negative, an arc to the right will be generated. If width and/or height are negative, the matching direction will be calculated.

The insert point is the tangential point of the arc (see drawing), this means the length of the previous element will be shortened. The width is measured in the direction of the previous element; the height is perpendicular to it.

#### Principle of operation

If you have selected a [Profile Element](#) previously, the new profile elements will be appended to the selected one. Otherwise they will be appended to the end of the profile.


### 3.1.7.1.6 Stage



This function is a part of the [Toolbox Profile Design](#) and generates three new elements: arc, line and arc, which represent a stage. The dimension reference point is the intersection point of the tangents to the arc.

#### Calling the function

Call this function by:

-  Button **Stage** from the [Toolbox Profile Design](#).

The window **Stage** is opened.

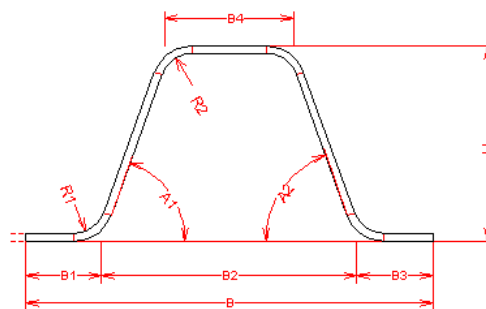
If you select **Angle** and **lower Width**, the height will be calculated. If you select **Angle** and **Height**, the lower width will be calculated. If you select **Width** and **Height**, the angle will be calculated. Angle and height can be negative too; in this case first an arc to the right and afterward to the left will be created. Please enter the upper width first!

The insert point is the tangential point of the arc (see drawing), this means the length of the previous element will be shortened. The width is measured in the direction of the previous element; the height is perpendicular to it.

#### Principle of operation

If you have selected a [Profile Element](#) previously, the new profile elements will be appended to the selected one. Otherwise they will be appended to the end of the profile.

### 3.1.7.1.7 Trapezoidal Profile



This function is a part of the [Toolbox Profile Design](#) and generates a trapezoidal profile. The opening is on the bottom or on the top as desired.

#### Calling the function

Call this function by:

-  Button **Trapezoidal Profile** from the [Toolbox Profile Design](#).

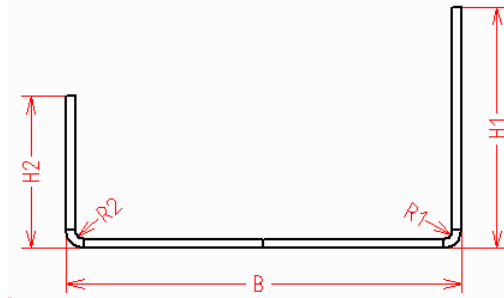
The window **Trapezoidal Profile** is opened. Enter the dimensions in any sequence. When enough

parameters are entered, the remaining dimensions are calculated automatically and the **Ok** button is enabled. If the trapezoid should open on the bottom, enter negative angles A1 and A2. The height H always is positive.

#### Principle of operation

If you have selected a [Profile Element](#) previously, the new profile elements will be appended to the selected one. Otherwise they will be appended to the end of the profile. Thus trapezoidal profiles with several trapezoids can be created.


#### 3.1.7.1.8 U-Profile



This function is a part of the [Toolbox Profile Design](#) and generates a simple, rectangular U-profile, either symmetrical or asymmetrical.

#### Calling the function

Call this function by:

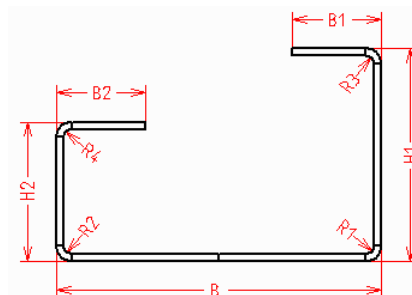
-  Button **U-Profile** from the [Toolbox Profile Design](#).

The window **U-Profile** is opened. Enter **Width**, **Height** on the right and on the left and - if you want - the **Radii**. If you enter equal heights and radii, a symmetrical profile will be created, otherwise an asymmetrical.

#### Principle of operation

If the profile list already contains profile elements, they will be deleted.


#### 3.1.7.1.9 C-Profile



This function is a part of the [Toolbox Profile Design](#) and generates a simple, rectangular C-profile, either symmetrical or asymmetrical.

#### Calling the function

Call this function by:

-  Button **C-Profile** from the [Toolbox Profile Design](#).

The window **C-Profile** is opened. Enter **Width**, **Height** on the right and on the left and the upper

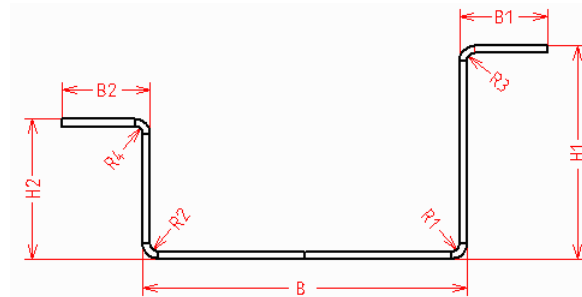


web **Widths**. If you enter equal values on the right and the left, a symmetrical profile will be created, otherwise an asymmetrical.

#### Principle of operation

If the profile list already contains profile elements, they will be deleted.

#### 3.1.7.1.10 Hat-Profile



This function is a part of the [Toolbox Profile Design](#) and generates a simple, rectangular hat-profile, either symmetrical or asymmetrical.

#### Calling the function

Call this function by:

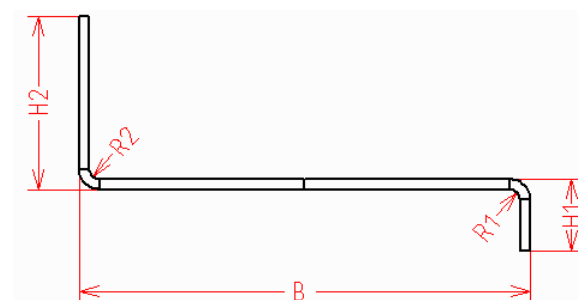
-  Button **Hat-Profile** from the [Toolbox Profile Design](#).

The window **Hat-Profile** is opened. Enter **Width**, **Height** on the right and on the left and the upper web **Widths**. If you enter equal values on the right and the left, a symmetrical profile will be created, otherwise an asymmetrical.

#### Principle of operation

If the profile list already contains profile elements, they will be deleted.


#### 3.1.7.1.11 Z-Profile



This function is a part of the [Toolbox Profile Design](#) and generates a simple, rectangular Z-profile, either symmetrical or asymmetrical.

#### Calling the function

Call this function by:

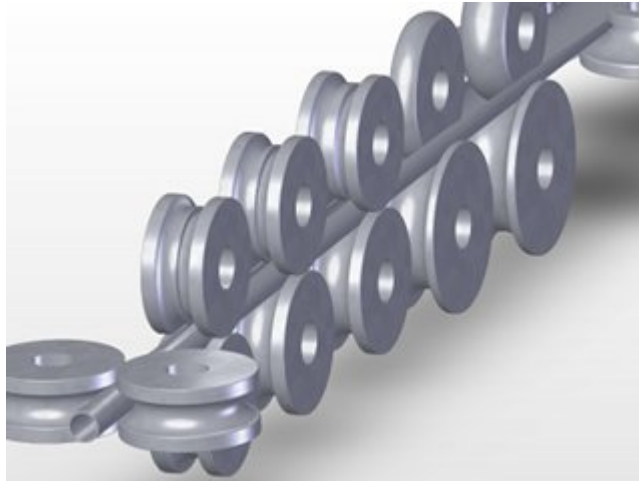
-  Button **Z-Profile** from the [Toolbox Profile Design](#).

The window **Z-Profile** is opened. Enter **Width**, **Height** on the right and on the left and - if you want - the **Radii**.

### Principle of operation

If the profile list already contains profile elements, they will be deleted.


#### 3.1.7.2 Tube Design



By using the Toolbox Tube Design you quickly can design the flower pattern for seamed tubes and the roll tools without a CAD system.

### Calling the function

Call this function optionally by:

- Main menu: **Toolbox, Tube Design**.
-  Button **Toolbox Tube Design** in the [Button Key Bar](#).

### Content

**Section 1** contains functions for creating the different passes for tube forming and for tube calibrating:



[Shaped Tube Calibration](#)



[Welding Pass](#)



[Fin Pass](#)



[Break Down Pass](#)



[Break Down Pass, W-Forming](#)

**Section 2** contains functions for generating the rolls for tube forming:



[Fin Pass, Top Roll](#)



[Fin Pass, Bottom Roll](#)



[Break Down Pass, Top Roll](#)



[Break Down Pass, Bottom Roll](#)



[Fin Pass, Side Rolls](#)



[Break Down Pass, Side Rolls](#)

### Principle of operation

- **Preparing:** Create a new profile project by using [File New](#). Open the [Machine Window](#) and enter the machine data or import a [Machine File](#) that you exported from a previous project. If the welded tube should be formed to a shaped tube, the machine must contain calibrating stands.
- **Defining the tube dimensions (round tube):** Call the function [Welding Pass](#) of the Toolbox Tube Design and enter the diameter, the sheet thickness and the necessary addition for welding.
- **Defining the tube dimensions (shaped tube):** Use the [Toolbox Profile Design](#), the [Graphical Method](#), or the [Numerical Method](#) to define the cross-section of the shaped tube. Afterwards call the function [Shaped Tubes Calibration](#) of the Toolbox Tube Design. The cross section patterns for the calibrating stands and the welding stand are created automatically. By using the function [Welding Pass](#) you can attach the addition for welding.
- **Creating the passes:** For each stand create a pass by using the function [Profile, Append](#) one after another and call in each pass one of the functions [Fin Pass](#), [Break Down Pass](#) or [Break Down Pass, W-Forming](#), dependent on whether it is a fin pass or a break down pass. All these functions bend open the existing profile dependent on the entered parameters as desired.
- **Generating the roll tools:** For each pass call the functions [Fin Pass, Top Roll](#), [Fin Pass, Bottom Roll](#), [Break Down Pass, Top Roll](#), [Break Down Pass, Bottom Roll](#), [Fin Pass, Side Rolls](#) or [Break Down Pass, Side Rolls](#), dependent on whether it is a fin pass or a break down pass or whether you want to create top, bottom or side rolls. The rolls for the calibrating stands can be created by using the function [Roll, Scan Profile Drawing](#).

### Properties

Except of the function **Welding Pass** (which is called in a new project with an empty profile list) all other functions use the pass in which they are called, i.e. the functions **Fin Pass** and **Break Down Pass** bend the tube profile dependent on the entered parameters (This is why you should have called the function [Profile, Append](#) previously). The functions **Top Roll**, **Bottom Roll** and **Side Rolls** generate rolls for the current pass.

All functions of the Toolbox Tube Design (except **Shaped Tube Calibration**) are designed for symmetrical tubes with two arc segments on each side, i.e. the profile list belonging to it **must** look like this:

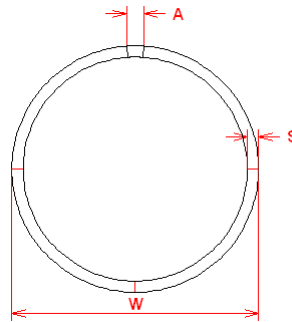
A1  
A1  
PS

With other kinds of profile lists the Toolbox Tube Design will not work. Modifications by hand or by the [Toolbox Modify](#) are allowed, however.

- The function **Shaped Tube Calibration** needs a closed cross section pattern, which can be symmetrical or unsymmetrical with any count and type of profile elements.

Before using the Toolbox Tube Design it is recommended to prepare the machine data in the [Machine Window](#).

### 3.1.7.2.1 Welding Pass



This function is a part of the [Toolbox Tube Design](#) and creates the tube profile for the welding pass.

#### Calling the function

Call this function by:

-  Button **Welding Pass** from the [Toolbox Tube Design](#).

The window **Welding Pass** is opened. Enter these parameters into the input fields:

- **tube diameter D**,
- **sheet thickness s**,
- **addition for welding A**.

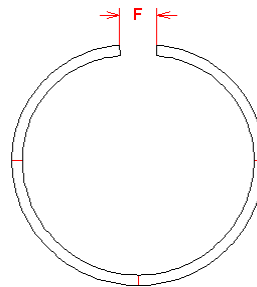
#### Principle of operation

After pressing the **Ok**-button the tube profile for the welding pass is entered into the profile list. If the profile was not empty, the contents will be replaced.

#### Hint:

- For creating the rolls for the welding pass, use the function [Fin Pass, Side Rolls](#).

### 3.1.7.2.2 Fin Pass




This function is a part of the [Toolbox Tube Design](#) and creates the tube profile for a Fin Pass.

#### Calling the function

Before calling this function you should have created a Welding Pass and afterwards by using the function Profile, Append have appended a profile list for the new fin pass.

Call this function by:

-  Button **Fin Pass** from the [Toolbox Tube Design](#).

The window **Fin Pass** is opened. Enter the **Fin Width F** into the input field or confirm the default value.

### Principle of operation

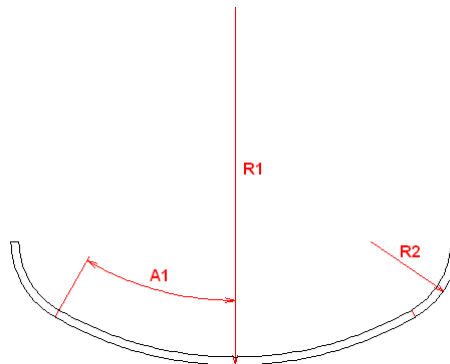
After pressing the **Ok**-button the tube profile for the fin pass is entered into the profile list. The first arc segment (at the bottom) will be bent open by increasing the radius. Then the angle of this arc is set to 90°, thus the top and bottom rolls later will get one single radius only. The second arc segment (at the top) keeps its radius constant (the same radius as for the previous pass). The angle however is modified in the way that the sheet width (sum of all developed lengths) is constant.

If the machine has more than one fin pass, use this function for each fin pass. For generating the rolls for the fin passes use the functions [Fin Pass, Bottom Roll](#) and [Fin Pass, Top Roll](#).

#### Hint:

- For creating the rolls for the fin pass, use the function [Fin Pass, Top Roll](#) and [Fin Pass, Bottom Roll](#).

### 3.1.7.2.3 Break Down Pass



This function is a part of the [Toolbox Tube Design](#) and creates the tube profile for a break down pass.

#### Calling the function

Before calling this function you should have created a Welding Pass and have created one or more Fin Passes. Afterward use the function Profile, Append for creating a profile list for the new break down pass.

Call this function by:

-  Button **Break Down Pass** from the [Toolbox Tube Design](#).

The window **Break Down Pass** is opened. Enter the desired **Angle** and **Radius** for the first arc segment and the **Radius** for the second arc segment into the input fields. If the message appears **Incorrect Input**, the entered parameters are not suited for creating a break down pass, the developed length of which is the same as the one of the previously appended profile list.

#### Principle of operation

After pressing the **Ok**-button the tube profile for the break down pass is entered into the profile list. The first arc segment (at the bottom) has the desired angle and radius. The second one (at the outside) has the desired radius and its angle is calculated on this way that the developed length of the whole profile is the same as the one of the previously appended profile list.

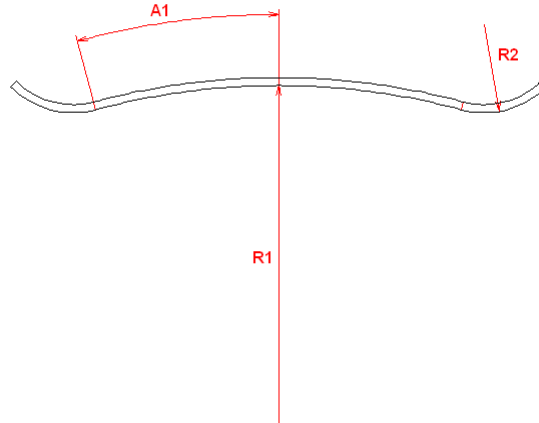
Call this function for each break down pass one after another. For generating the rolls for the break down passes use the functions [Break Down Pass, Bottom Roll](#) and [Break Down Pass, Top Roll](#).

#### Hint:

- For creating the rolls for the break down pass, use the function [Break Down Pass, Top Roll](#) and

[Break Down Pass, Bottom Roll.](#)

### 3.1.7.2.4 Break Down Pass W-Forming



This function is a part of the [Toolbox Tube Design](#) and creates the tube profile for a break down pass. W-Forming is a special forming method which raises the middle of the profile first while the finished radius is formed at the outsides.

#### Calling the function

Before calling this function you should created already some [Break Down Passes](#) Afterward use the function [Profile, Append](#) for creating a profile list for the new break down pass. Call this function by:

-  Button **Break Down Pass, W-Forming** from the [Toolbox Tube Design](#).

The window **Break Down Pass, W-Forming** is opened. Enter the desired **Angle** and **Radius** for the first arc segment and the **Radius** for the second arc segment into the input fields.

If the message appears **Incorrect Input**, the entered parameters are not suited for creating a break down pass, the developed length of which is the same as the one of the previously appended profile list.

#### Principle of operation

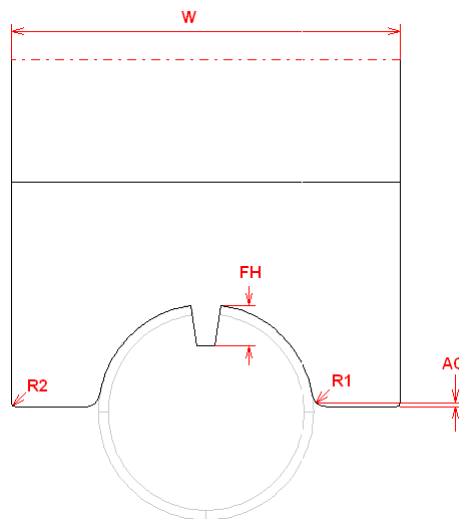
After pressing the **Ok**-button the tube profile for the break down pass is entered into the profile list. The first arc segment (at the bottom) has the desired angle and radius. The second one (at the outside) has the desired radius and its angle is calculated on this way that the developed length of the whole profile is the same as the one of the previously appended profile list.

For generating the rolls for the break down passes use the functions [Break Down Pass, Bottom Roll](#) and [Break Down Pass, Top Roll](#).

#### Hint:

- For creating the rolls for the break down pass, use the function [Break Down Pass, Top Roll](#) and [Break Down Pass, Bottom Roll](#).

### 3.1.7.2.5 Fin Pass Top Roll



This function is a part of the [Toolbox Tube Design](#) and is used for generating the top roll for a fin pass.

#### Calling the function

Before calling this function you should have created a fin pass by using the function [Fin Pass](#). If you created the fin pass in another way, pay attention that the profile list contains exactly 3 entries: A1, A1, PS. Only for such a tube profile a roll can be generated automatically. Call this function by:

-  Button **Fin Pass, Top Roll** from the [Toolbox Tube Design](#).

The window **Fin Pass, Top Roll** is opened.

Enter these parameters:

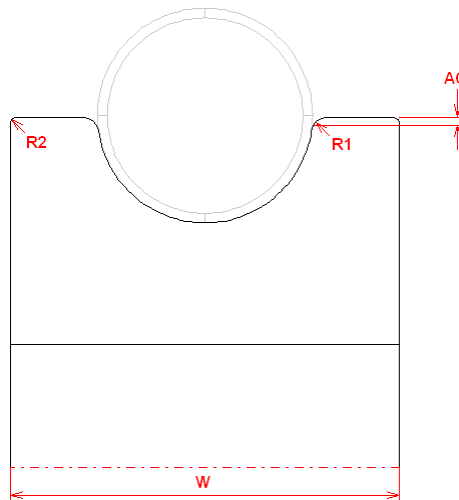
- **Total Width W:** If your entered value is too small, the width is set automatically to a minimum width that has enough space for all fillet radii. The result is rounded to 10 mm or 1 in.
- **Fillet Radii R1, R2:** Select a fillet radius at the inside (at the tube) and a fillet radius at the outside (at the roll edges).
- **Fin Height FH:** Enter the fin height (Remark: The fin width is set in [Fin Pass](#)). If the opening in the tube is less than 1 mm or 0.1 in, no fin is created.
- **Half Air Gap AG:** Enter the desired vertical air gap between the roll and the center of the tube (more exact: center of the first arc segment of the tube profile).

Working diameter and reference point are taken from the [Machine Data](#).

#### Principle of operation

After pressing the **Ok**-button the roll is generated. If a roll is yet existing, it will be replaced. If the fin should be a simple disc, you simply can produce it by using the [Split at Corner](#) function at each edge of the fin.

### 3.1.7.2.6 Fin Pass Bottom Roll



This function is a part of the [Toolbox Tube Design](#) and is used for generating the bottom roll for a fin pass.

#### Calling the function

Before calling this function you should have created a fin pass by using the function [Fin Pass](#). If you created the fin pass in another way, pay attention that the profile list contains exactly 3 entries: A1, A1, PS. Only for such a tube profile a roll can be generated automatically.

Call this function by:

-  Button **Fin Pass, Bottom Roll** from the [Toolbox Tube Design](#).

The window **Fin Pass, Bottom Roll** is opened.

Enter these parameters:

- **Total Width W:** If your entered value is too small, the width is set automatically to a minimum width that has enough space for all fillet radii. The result is rounded to 10 mm or 1 in.
- **Fillet Radii R1, R2:** Select a fillet radius at the inside (at the tube) and a fillet radius at the outside (at the roll edges).
- **Half Air Gap AG:** Enter the desired vertical air gap between the roll and the center of the tube (more exact: center of the first arc segment of the tube profile).

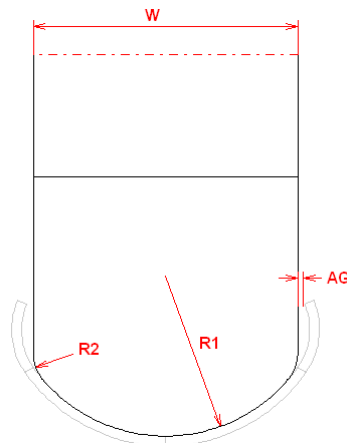
Working diameter and reference point are taken from the [Machine Data](#).

#### Principle of operation

After pressing the **Ok**-button the roll is generated. If a roll is yet existing, it will be replaced.



### 3.1.7.2.7 Break Down Pass Top Roll



This function is a part of the [Toolbox Tube Design](#) and is used for generating the top roll for a break down pass.

#### Calling the function

Before calling this function you should have created a break down pass by using the function [Break Down Pass](#). If you created the break down pass in another way, pay attention that the profile list contains exactly 3 entries: A1, A1, PS. Only for such a tube profile a roll can be generated automatically.

Call this function by:

-  Button **Break Down Pass, Top Roll** from the [Toolbox Tube Design](#).

The window **Break Down Pass, Top Roll** is opened.

Enter these parameters:

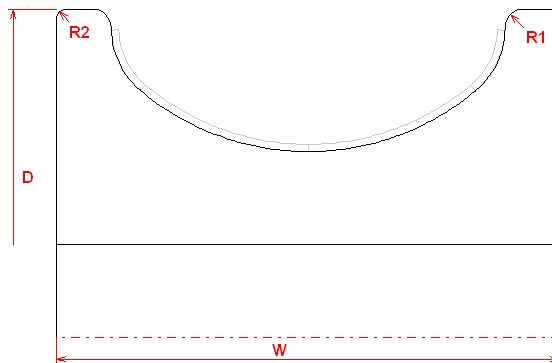
- **Total Width W:** This input is only necessary, if the roll should become smaller than the opening of the tube. If your entered value too big, your input is ignored and the width is set automatically to a maximum width by considering the desired air gaps at both sides (if the edges of the tube are bent inside) or the upper arc segment is lengthened to the vertical (if the edges of the tube are bent outside).
- **Fillet Radii R2:** Select a fillet radius at the roll edges (only if the edges of the tube are bent inside). In the other case your input is ignored and the radius of the upper arc segment is taken.
- **Air Gap AG:** Enter the desired horizontal air gap between the roll and the edge of the tube edges (only if the edges of the tube are bent inside).
- **Lower Roll Radius R1:** The inner radius of the first arc segment is proposed in the input field. If necessary, you can lower the radius if the roll should touch the profile at the bottom part only, e.g. if you want to design a roll set for various sheet thickness.

Working diameter and reference point are taken from the [Machine Data](#).

#### Principle of operation

After pressing the **Ok**-button the roll is generated. If a roll is yet existing, it will be replaced.

### 3.1.7.2.8 Break Down Pass Bottom Roll



This function is a part of the [Toolbox Tube Design](#) and is used for generating the bottom roll for a break down pass.

#### Calling the function

Before calling this function you should have created a break down pass by using the function [Break Down Pass](#). If you created the break down pass in another way, pay attention that the profile list contains exactly 3 entries: A1, A1, PS. Only for such a tube profile a roll can be generated automatically.

Call this function by:

-  Button **Break Down Pass, Bottom Roll** from the [Toolbox Tube Design](#).

The window **Break Down Pass, Bottom Roll** is opened.

Enter these parameters:

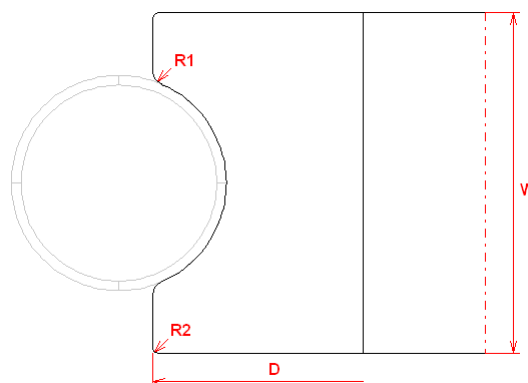
- **Roll Diameter D:** If your entered diameter is too small, that the roll cannot touch the tube, an error message appears.
- **Total Width W:** If your entered value is too small, the width is set automatically to a minimum width that has enough space for all fillet radii. The result is rounded to 10 mm or 1 in.
- **Fillet Radii R1, R2:** Select a fillet radius at the inside (at the tube) and a fillet radius at the outside (at the roll edges).

Working diameter and reference point are taken from the [Machine Data](#).

#### Principle of operation

After pressing the **Ok**-button the roll is generated. If a roll is yet existing, it will be replaced.

### 3.1.7.2.9 Fin Pass Side Rolls



This function is a part of the [Toolbox Tube Design](#) and is used for generating the side rolls for a fin pass without a shoulder.

### Calling the function

Before calling this function you should have created a fin pass by using the function [Fin Pass](#). If you created the fin pass in another way, pay attention that the profile list contains exactly 3 entries: A1, A1, PS. Only for such a tube profile rolls can be generated automatically. Call this function by:

-  Button **Fin Pass, Side Rolls** from the [Toolbox Tube Design](#).

The window **Fin Pass, Side Rolls** is opened.

Enter these parameters:

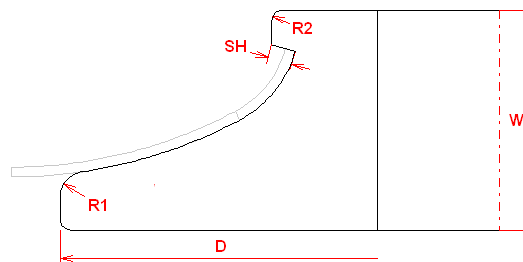
- **Roll Diameter D:** If your entered diameter is too small, that the roll cannot touch the tube, an error message appears. If it is too big, the diameter is set automatically that both side rolls touch together.
- **Total Width W:** If your entered value is too small, the width is set automatically to a minimum width that has enough space for all fillet radii. The result is rounded to 10 mm or 1 in.
- **Fillet Radii R1, R2:** Select a fillet radius at the inside (at the tube) and a fillet radius at the outside (at the roll edges).

Working diameter and reference point are taken from the [Machine Data](#).

### Principle of operation

After pressing the **Ok**-button the roll is generated. If a roll is yet existing, it will be replaced.

#### 3.1.7.2.10 Break Down Pass Side Rolls



This function is a part of the [Toolbox Tube Design](#) and is used for generating the side rolls for a break down pass. The shoulder is suited for leading the edge of the sheet to the correct position.

### Calling the function

Before calling this function you should have created a break down pass by using the [Break Down Pass](#). If you created the break down pass in another way, pay attention that the profile list contains exactly 3 entries: A1, A1, PS. Only for such a tube profile rolls can be generated automatically. Call this function by:

-  Button **Break Down Pass, Side Rolls** from the [Toolbox Tube Design](#).

The window **Break Down Pass, Side Rolls** is opened.

Enter these parameters:

- **Roll Diameter D:** If your entered diameter is too small, that the roll cannot touch the tube, an error message appears. If it is too big, the diameter is set automatically that both side rolls touch together.
- **Total Width W:** If your entered value is too small, the width is set automatically to a minimum width that has enough space for all fillet radii. The result is rounded to 10 mm or 1 in.
- **Fillet Radii R1, R2:** Select a fillet radius at the inside (at the tube) and a fillet radius at the outside (at the roll edges).
- **Shoulder Height SH:** Enter the height of the shoulder, which should lead the edge of the sheet.

Working diameter and reference point are taken from the [Machine Data](#).

### Principle of operation


After pressing the **Ok**-button the roll is generated. If a roll is yet existing, it will be replaced.

#### 3.1.7.3 Modify

Use the Toolbox Modify to bend the profile up and down, to modify the size of the profile, and to modify the corner points of the rolls.

#### Calling the function

Call this function optionally by:

- Main menu: **Toolbox, Modify**.
-  Button **Toolbox Modify** in the [Button Key Bar](#).



One of these buttons are shown at the right screen edge, dependant on you just are designing the profile, the flower pattern, or the rolls.

#### Content



#### Angle/Radius/Length

These switches are visible in the left column of the Toolbox Modify, if you have selected a single pass or a pass of the flower. You can modify the **Angle**, the **Radius** or the **Length** of a [Profile Element](#).

To modify an angle or radius, you must have selected an arc, either by clicking into the [Drawing Area](#) or in the [Profile List Window](#). The profile element will be bent in accordance with the [Arc Type](#). You define by setting the menu switch [Profile List Loaded](#), if the discharged or loaded values should be modified.

Notice:

- For the loaded state only the angle, not the radius can be modified.
- For the arc type A2/A3/A4 only the angle (loaded or discharged) can be modified.

To modify a length, you must have selected a line.



#### Width/Diameter/Radius

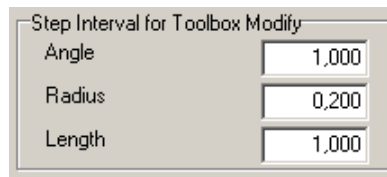
These switches are visible in the left column of the Toolbox Modify, if you have called [View Roll Tools](#) and you have selected a [Roll Corner Point](#), either by clicking into the [Drawing Area](#) or into the [Roll Tool Window](#). You can modify the **Width**, the **Diameter** or the **Radius**.



#### 10x larger/larger/smaller/10x smaller

Use these functions in the right column of the Toolbox Modify to proceed the modification, which you have selected in the left column. Both single arrow buttons in the middle will modify by using the predefined step interval. The double arrow buttons will use the tenfold step interval.

## Setup



In [Options Mouse](#), **Step Interval** you can preset the step interval for the single arrow buttons.

## Principle of operation


While modifying both the drawing and the data displayed in the different windows will be updated.

### 3.1.7.4 Dimensioning

Use this toolbox for measuring of distances and angles inside the drawing and for creating various kinds of dimensioning.

#### Calling the function

Call this function optionally by:

- Main menu: **Toolbox, Dimensioning**.
-  Button **Toolbox Dimensioning** in the [Button Key Bar](#).

#### Content



[Measure](#)



[Horizontal Dimension](#)



[Vertical Dimension](#)



[Parallel Dimension](#)



[Diameter Dimension](#)



[Radius Dimension](#)



[Angle Dimension](#)



[Automatic Roll Dimension](#)

And you can modify dimensionings:



[Move Dimension](#)



[Delete Dimension](#)

#### Drag dimensioning:

When the dimension points are defined, the complete dimensioning is already visible. The dimension text sticks to the cursor, can be moved across the drawing and positioned as desired. Thus, dimensioning is most comfortable.

### Associatively:

Dimension points are tied to the drawing elements. When you modify the drawing elements, the dimensioning will be modified automatically, too. Because of the associatively dimensioning cannot be positioned freely.

### CAD transfer:

When dimensioned drawings are transferred to AutoCAD via the [ActiveX interface](#), genuine associative AutoCAD dimensioning is created. By ActiveX transfer to SolidWorks non-associative dimensioning is created. While transferring drawings by [Output Drawing -> CAD](#) via other interface files (DXF, MI, IGES) the dimensioning is transferred, too, though it consist of lines and texts; these are not associative anymore.

### Transparency:

Concerning the zoom and move functions of the [Navigator](#) (not zoom window, however) the dimensioning functions are transparent, i.e. you can select the view port while dimensioning.

### Input:

After calling the function of this toolbox you are asked in the dialog line at the bottom of the screen to enter some inputs one after the other:

- **1./2. dimension point or 1./2. dimension element?** - Select a drawing element by mouse click; one of the end points is caught on principle. By selecting a roll corner revolution line (see [Options Drawing](#), Rolls, Revolution Lines) the intersection points of the tangents at a roll corner can be dimensioned. In some cases you can open a context menu by using the right mouse button that contains further catch points. Because dimensioning is associative, it is not possible to create dimensioning where no drawing elements are.
- **Text Position?** - If you are asked for the text position, you can position the text sticking to the mouse cursor as desired.
- **Dimension:** - Click on the text of an existing dimension, is used for identifying a dimensioning.

### Repetition:

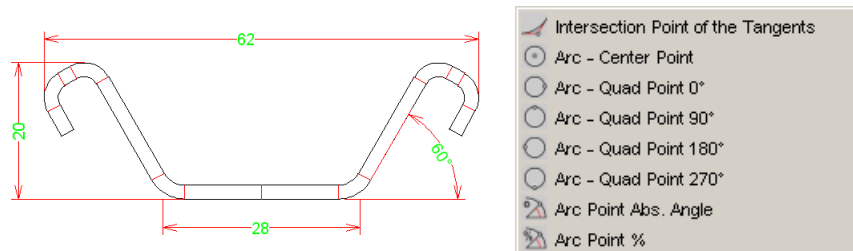
After you created a dimensioning, the dimensioning function starts again. This is useful for creating series of dimensioning of the same kind. For breaking off the function use the Esc key or select another dimensioning function.

### Visibility:

While dimensioning pay attention if a roll is selected or not:






- **A roll is selected:** The selected roll can be dimensioned only, because the drawing elements of all other rolls cannot be identified. The created dimensioning is visible only, when the dimensioned roll is selected or when no roll is selected. It is not visible, when another roll is selected.
- **No roll is selected** (Function [Inspect](#)): Now you can dimension all drawing elements, therefore dimensioning across roll borders is possible (e.g. distance between shafts). The created dimensioning is visible only, when no roll is selected.

### Context menu:

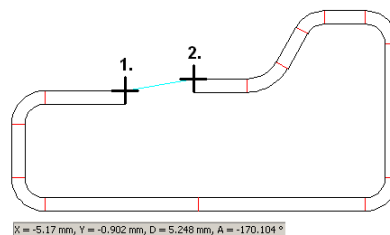


Examples for dimensioning by using the context menu

Call the context menu by using the right mouse button, when you are asked: **1./2. dimension point?**. Without using the context menu, always one of the end points is caught. If you call the context menu, you can use further extended catch points:

-  Intersection point of the tangents
-  Arc - center point
-  Arc - quad point 0°, 90°, 180°, 270°
-  Arc point abs. angle: Enter the absolute angle related to the horizontal x-axes in the following input window
-  Arc point %: Enter the relative angle in % in the following input window (0 = arc start point, 100 = arc end point)

### 3.1.7.4.1 Measure




This function is a part of the [Toolbox Dimensioning](#) and can be used for measuring of

- distances in x- and y-direction
- distances diagonal
- angles

#### Calling the function

Call this function by:

-  Button **Measure** from the [Toolbox Dimensioning](#).

After calling this function you are asked to identify two points. Click on an element of the drawing and the next end point of the element will be identified. Between the identified points a temporal line in the predefined mark color is drawn.

#### Principle of operation

The results of measuring are displayed in the bottom status bar in the form:

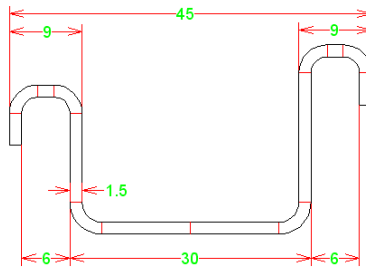
**x = ... y = ... d = ... w = ...°**

Between the identified points is

- **x** the horizontal x distance,
- **y** the vertical y distance,
- **d** the diagonal distance,
- **w** the angle between the connecting line and the horizontal x axis.

The temporal line disappears again, when you call another function.

### 3.1.7.4.2 Horizontal Dimension



This function is a part of the [Toolbox Dimensioning](#) and can be used for a horizontal dimensioning between two dimensioning points.

#### Calling the function

Call this function by:

-  Button **Horizontal Dimension** from the [Toolbox Dimensioning](#).

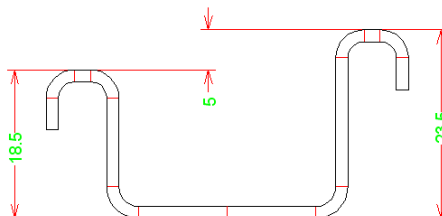
After calling this function you are asked to enter:

- **1. Dimension Point?** - afterwards a drag line appears between the selected point and the cursor position.
- **2. Dimension Point?** - afterwards a drag dimensioning appears between both selected dimensioning points and the cursor position as dimension text position.
- **Dimension Text Position?** - Afterwards the dimensioning appears.

#### Hint:

- See [Toolbox Dimensioning](#) for further information.

### 3.1.7.4.3 Vertical Dimension



This function is a part of the [Toolbox Dimensioning](#) and can be used for a vertical dimensioning between two dimensioning points.

#### Calling the function

Call this function by:

-  Button **Vertical Dimension** from the [Toolbox Dimensioning](#).

After calling this function you are asked to enter:

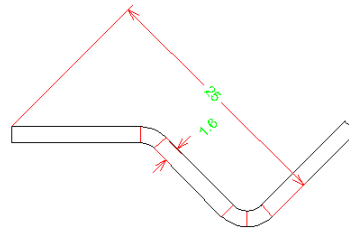
- **1. Dimension Point?** - afterwards a drag line appears between the selected point and the cursor position.
- **2. Dimension Point?** - afterwards a drag dimensioning appears between both selected dimensioning points and the cursor position as dimension text position.
- **Dimension Text Position?** - Afterward the dimensioning appears.

#### Hint:

- See [Toolbox Dimensioning](#) for further information.



### 3.1.7.4.4 Parallel Dimension



This function is a part of the [Toolbox Dimensioning](#) and can be used for a parallel dimensioning between two dimensioning points parallel to any drawing element (for profile dimensioning only).

#### Calling the function

Call this function by:

-  Button **Parallel Dimension** from the [Toolbox Dimensioning](#).

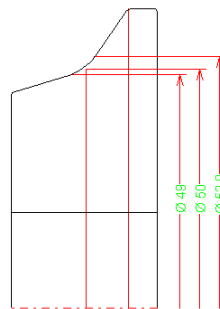
After calling this function you are asked to enter:

- **1. Dimension Point?** - afterwards a drag line appears between the selected point and the cursor position.
- **2. Dimension Point?** - afterwards a drag dimensioning appears between both selected dimensioning points and the cursor position as dimension text position.
- **Parallel to?** - identify any drawing element, which the dimensioning should be parallel to or click to an empty area of the drawing, as a result of it the dimensioning will be created parallel to the dimension points.
- **Dimension Text Position?** - Afterward the dimensioning appears.

#### Hint:

- See [Toolbox Dimensioning](#) for further information.

### 3.1.7.4.5 Diameter Dimension



This function is a part of the [Toolbox Dimensioning](#) and can be used for a diameter dimensioning (for roll tools only).

Because roll tools are drawn in a half intersection manner, the 2nd dimension point doesn't exist. Because of that a radius dimensioning is created, the dimension text however is the diameter with the sign Ø.

#### Calling the function

Call this function by:

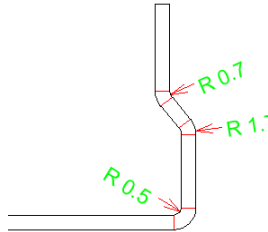
-  Button **Diameter Dimension** from the [Toolbox Dimensioning](#).

After calling this function you are asked to enter:

- **Dimension Point?** - afterwards a drag dimensioning appears.
- **Dimension Text Position?** - Afterward the dimensioning appears.

**Hint:**

- See [Toolbox Dimensioning](#) for further information.

**3.1.7.4.6 Radius Dimension**

This function is a part of the [Toolbox Dimensioning](#) and can be used for a radius dimensioning.

**Calling the function**

Call this function by:

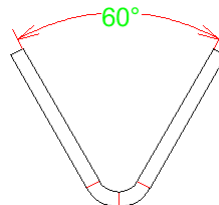
-  Button **Radius Dimension** from the [Toolbox Dimensioning](#).

After calling this function you are asked to enter:

- **Dimension Point?** - identify an arc and a drag dimensioning appears afterwards.
- **Dimension Text Position?** - Afterward the dimensioning appears. You can position it inside and outside the arc.

**Hint:**

- See [Toolbox Dimensioning](#) for further information.

**3.1.7.4.7 Angle Dimension**

This function is a part of the [Toolbox Dimensioning](#) and can be used for angle dimensioning between two drawing elements.

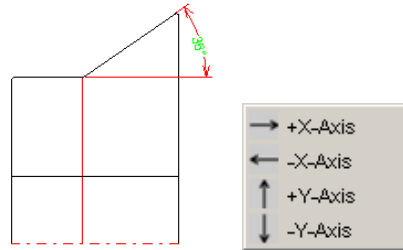
**Calling the function**

Call this function by:

-  Button **Angle Dimension** from the [Toolbox Dimensioning](#).

After calling this function you are asked to enter:

- **1. Dimension Element?** - afterwards a drag line appears between the selected element end and the cursor position.
- **2. Dimension Element?** - afterwards a drag dimensioning appears between both selected element points and the cursor position as dimension text position.
- **Dimension Text Position?** - Afterwards the dimensioning appears.



Angle Dimensions also can be set related to any horizontal or vertical axis. When you are asked **1./2. Dimension Point?**, press the right mouse button and select the desired axis from the context menu:

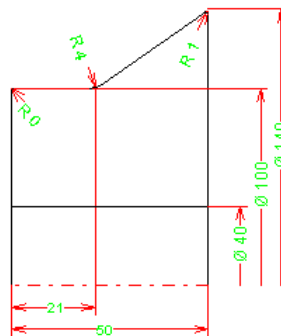
- +x - Axis
- -x - Axis
- +y - Axis
- -y - Axis

Afterwards click on the end point of any drawing element for defining the origin of the axis.

**Hint:**

- See [Toolbox Dimensioning](#) for further information.

### 3.1.7.4.8 Automatic Roll Dimensioning

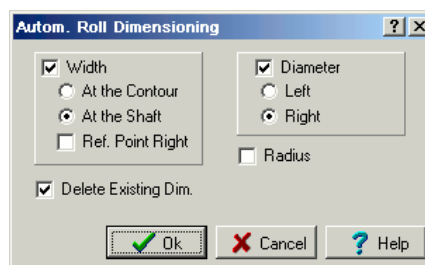


This function is a part of the [Toolbox Dimensioning](#) and can be used for automatic roll dimensioning (for roll tools only).

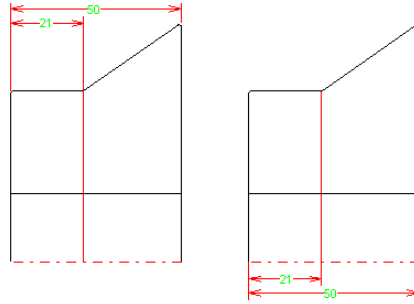
#### Calling the function

Call this function by:

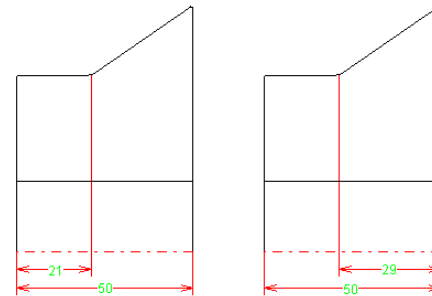
- Button **Automatic Roll Dimension** from the [Toolbox Dimensioning](#).



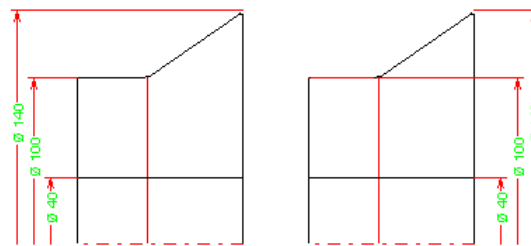
After calling this function you are asked to enter:



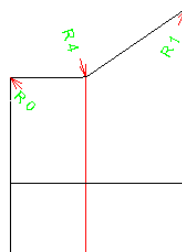
- **Width at Contour/at Shaft:** Select if the roll width should be dimensioned and whether the dimensioning should appear at the roll contour or at the shaft.



- **Ref.-Point Right:** Select if the width dimensioning should be related on the left or right side of the roll.



- **Diameter Left/Right:** Select if the roll diameter should be dimensioned and whether it should appear on the left or on the right side of the roll.



- **Radius:** Select if the radii of the roll should be dimensioned.
- **Delete Existing Dimensions:** Select if the new dimensioning should be attached to an existing dimensioning or if the existing one should be deleted previously.

### Principle of operation

After pressing the Ok key the roll will be dimensioned automatically. If necessary, use the function [Move Dimension](#) afterwards for exactly positioning the dimensioning. Certain items can be removed by the function [Delete Dimension](#).

### Hint:

- See [Toolbox Dimensioning](#) for further information.

### 3.1.7.4.9 Move Dimension

This function is a part of the [Toolbox Dimensioning](#) and can be used for exactly positioning the dimensioning.

#### Calling the function

Call this function by:

-  Button **Move Dimension** from the [Toolbox Dimensioning](#).

After calling this function you are asked to enter:

- **Dimension?** - identify a dimension text and the identified dimensioning appears as a drag dimensioning.
- **Dimension Text Position?** - Afterward the dimensioning is positioned as desired.

**Hint:**

- See [Toolbox Dimensioning](#) for further information.

### 3.1.7.4.10 Delete Dimension

This function is a part of the [Toolbox Dimensioning](#) and can be used for deleting the a dimensioning.

#### Calling the function

Call this function by:

-  Button **Delete Dimension** from the [Toolbox Dimensioning](#).

After calling this function you are asked to enter:

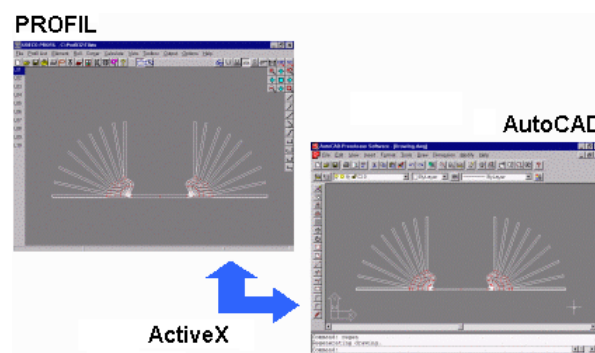
- **Dimension?** - identify a dimension text and the identified dimensioning will be deleted.

**Hint:**

- See [Toolbox Dimensioning](#) for further information.

## 3.1.8 Output

### 3.1.8.1 Drawing -> CAD




Use this function to transfer the drawing, which is currently displayed in the [Drawing Area](#), to your [CAD System](#).

#### Calling the function

Select from the menu **View**, what should be transferred. Select by mouse-click, if a certain pass or a certain roll or if all (button [Inspect](#)) should be transferred.

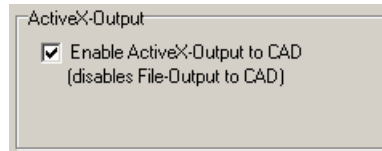
Call this function optionally by:

- Main menu: **Output, Drawing -> CAD.**
-  Button **Drawing -> CAD** in the [Button Key Bar](#).

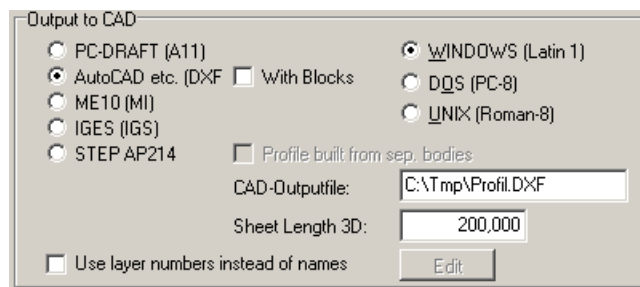
### Principle of operation

The drawing, which is currently displayed in the [Drawing Area](#), is transferred [CAD System](#)., either directly or by interface file.

### Setup



If you have enabled the **ActiveX Output** to CAD in [Options ActiveX](#), the drawing will directly be transferred to CAD (AutoCAD or SolidWorks or SolidEdge or BricsCAD) and the correct view port and zoom window is set automatically. If layers yet exist, they will be deleted before updating.



Otherwise, if ActiveX is disabled and a **Output File to CAD** (file format and file name) is selected in [Options, Files](#), the (temporal) file will be created. Afterwards, you have to open the file in your [CAD System](#).

### Hints:

- In order to create a set of different files sequentially, it is recommended to use the function [File, Export](#).
- Select by [View, Show, Spacer Rolls](#) whether the CAD output should contain spacer rolls.

### 3.1.8.2 Drawing -> NC


Use this function to transfer a single roll or all rolls of the stand, which is currently displayed in the [Drawing Area](#), or all rolls of the [Profile Project](#) to your NC system. The file format is **DXF**.

### Calling the function

Select [View Rolls](#). If a drawing object is selected in the drawing area, only this one is transferred. If you want to transfer the whole drawing, call the function [Inspect](#) previously. If you want to transfer all rolls of the project, select this in [Options NC](#) previously.

Before you create NC data you should use the function [Calculate, Plausibility Check](#) to check the whole profile project on errors.

Call this function optionally by:

- Main menu: **Output, Drawing -> NC.**
-  Button **Drawing -> NC** in the [Button Key Bar](#).

### Principle of operation

If the box **Separate Files** in [Options NC](#) is not checked, always the same (temporal) file is created. Otherwise a file with all rolls of a stand is created. Afterwards you have to load the file into your NC system.

## Setup

Output DXF-File:

☐ Whole Project in Sep. Files

Build Filename from

☐ Roll No.

☒ Part No.

☐ Turn over Roll 1

Filename for Entire File:

C:\PROFIL-Data\Eng\Profil.DXF

Path for Separate Files:

C:\PROFIL-Data\Eng\

Which filename and which path are to be used, select in [Options NC](#).

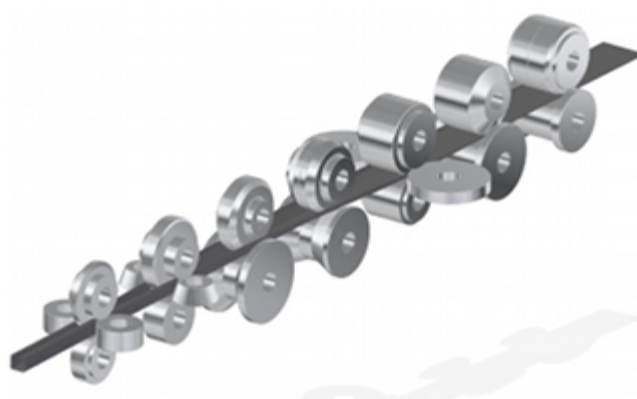
### Hints:

- In order to create a set of different files sequentially, better use the [Export](#) function.
- Select by [View, Show, Spacer Rolls](#) whether the NC output should contain spacer rolls.

### 3.1.8.3 3D Model -> CAD



Example 3D model in SolidWorks, transfer by ActiveX



Example 3D model in SolidEdge, transfer by STEP file

This function creates the 3D model of the current profile pass, the roll of the stand that is shown in

the [Drawing Area](#) or of all stands of the whole project. The model is transferred to AutoCAD or SolidWorks via ActiveX. The function needs AutoCAD R14 or higher or SolidWorks 2003 or higher respectively. The ActiveX interface must be enabled in [Options, ActiveX](#).

Alternatively, a STEP file in accordance with DIN ISO 10303 "Product data representation and exchange, EXPRESS language" can be created. So the pass, the rolls of a stand or all stands can be transferred to any 3D CAD system with STEP interface. In order to set-up this interface, open [Options, Files](#) and select **Output to CAD, STEP AP214**. If you want to use this function occasionally only and you do not want to change the setting every time, it would be better to use the [Export](#) function.

This function is useful e.g. for designing the side roll support. If the machine drawing already exists in CAD, the rolls simply can be inserted. Another application is to create photo-realistic images for

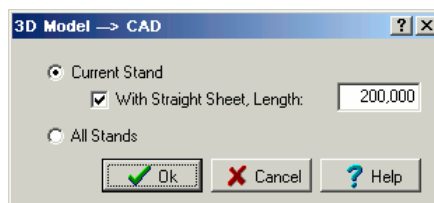
- **Advertising:** clear view of your products e.g. for info brochures, exhibitions, websites etc.
- **Presenting:** introduction of your company and your products e.g. by using PowerPoint.
- **Offers:** if the decision maker of your customer is a non-technician, clear views are more expressive than technical drawings.

### Calling the function

Select [View Pass](#) or [View Rolls](#) if you want to transfer a straight piece of profile or the roll tools of a stand.

Call this function by:

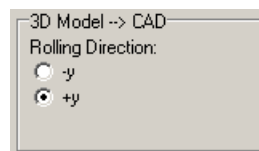
- Main menu: **Output, 3D Model -> CAD.**



In case of [View Rolls](#), a dialog window is opened for selecting:

After calling this function, select in the dialog window:

- **Current stand.** If needed, a straight sheet with desired length can be inserted between the rolls.
- **All stands** of the whole profile project.



If you select **All stands** you can preset the rolling direction in [Options, ActiveX](#) (ActiveX output only).

### Principle of operation



**AutoCAD:** You get the 3D wire frame model of the rolls. Select the view port by rotating the model (function **3D Orbit**). Call **View, Render, Render** in AutoCAD. In the dialog box, select **Photo Raytrace** and set e.g. the background color, quality and resolution, anti-aliasing. Furthermore lights can be set. You can copy the created image into the Windows clipboard and export it to other programs. Get further information from the AutoCAD manual.



**SolidWorks:** Open a **Part Document** first. After transfer, you get the 3D model of the selected stand.



**SolidEdge:** Preliminary the transfer is possible for 2D drawings only. The extension to 3D models is planned for a later release.



**BricsCAD:** Preliminary the transfer is possible for 2D drawings only. The extension to 3D



models is planned for a later release.

**STEP AP214:** You get the 3D model of your selection. The file can be opened in any 3D CAD system with STEP interface.

#### Hints:

- If the outer contour of a roll is not unambiguous (e.g. if lines intersect themselves), the 3D model cannot be created. In this case please check the roll contour by function Calculate, Plausibility Check and correct the roll data.
- Currently it is not possible to transfer all stands in conjunction with the formed sheet between all rolls. This function is planned for future enhancement. Use [View PSA](#) and [Drawing -> CAD](#) instead. This function transfers the sheet without thickness.
- Select by [View, Show, Spacer Rolls](#) whether the 3D output should contain spacer rolls.

### 3.1.8.4 Create Parts List

No	Designation	Roll No.	Part No.	Mat.	Blank	Size	Bore	Gross/Fin.	Weight
2	Bushing			2080	Ø 76,0	x 28,0	50,0	1,0	0,4
2	Bushing			2080	Ø 76,0	x 56,0	50,0	2,0	1,0
1	Bushing			2080	Ø 76,0	x 65,0	50,0	2,3	1,2
2	Spacer			2080	Ø 81,0	x 175,0	50,0	7,1	4,1
2	Spacer			2080	Ø 81,0	x 185,0	50,0	7,5	4,3
4	Spacer			2080	Ø 81,0	x 190,0	50,0	7,7	4,4
2	Spacer			2080	Ø 81,0	x 195,0	50,0	7,9	4,6
4	Spacer			2080	Ø 81,0	x 205,0	50,0	8,3	4,8
2	Spacer			2080	Ø 81,0	x 223,0	50,0	9,0	5,2
4	Spacer			2080	Ø 81,0	x 225,0	50,0	9,1	5,3
1	TopRoll	201	A4865.R02.O1	2080	Ø 101,5	x 36,0	50,0	2,3	1,4
1	TopRoll	203	A4865.R02.O3	2080	Ø 101,5	x 36,0	50,0	2,3	1,4
1	TopRoll	203	A4865.R01.O3	2080	Ø 111,5	x 28,0	50,0	2,1	0,8
1	TopRoll	201	A4865.R01.O1	2080	Ø 111,5	x 28,0	50,0	2,1	0,8
1	LeftRoll	301	A4865.R01.L1	2080	Ø 111,5	x 56,0	52,0	4,3	2,0
1	RightRoll	401	A4865.R01.R1	2080	Ø 111,5	x 56,0	52,0	4,3	2,0
1	RightRoll	401	A4865.R02.R1	2080	Ø 116,5	x 47,0	52,0	3,9	1,7
1	LeftRoll	301	A4865.R02.L1	2080	Ø 116,5	x 47,0	52,0	3,9	1,7
1	BottomRoll	102	A4865.R04.U2	2080	Ø 126,5	x 41,0	50,0	4,0	2,8
1	BottomRoll	101	A4865.R05.U1	2080	Ø 126,5	x 70,0	50,0	6,9	4,7
1	BottomRoll	102	A4865.R05.U2	2080	Ø 126,5	x 70,0	50,0	6,9	4,7
1	BottomRoll	102	A4865.R03.U2	2080	Ø 131,5	x 43,0	50,0	4,6	3,1
1	BottomRoll	101	A4865.R01.U1	2080	Ø 131,5	x 65,0	50,0	6,9	3,1
1	BottomRoll	101	A4865.R02.U1	2080	Ø 131,5	x 65,0	50,0	6,9	5,0
1	TopRoll	202	A4865.R01.O2	2080	Ø 162,0	x 23,0	50,0	3,7	2,5
1	BottomRoll	103	A4865.R04.U3	2080	Ø 172,0	x 68,0	50,0	12,4	7,9
1	BottomRoll	101	A4865.R04.U1	2080	Ø 172,0	x 68,0	50,0	12,4	7,9
1	TopRoll	202	A4865.R02.O2	2080	Ø 182,0	x 44,0	50,0	9,0	6,7
1	TopRoll	201	A4865.R03.O1	2080	Ø 182,0	x 55,0	50,0	11,2	5,5
1	TopRoll	202	A4865.R03.O2	2080	Ø 182,0	x 55,0	50,0	11,2	5,5
1	TopRoll	202	A4865.R04.O2	2080	Ø 182,0	x 65,0	50,0	13,3	7,4
1	TopRoll	201	A4865.R04.O1	2080	Ø 182,0	x 65,0	50,0	13,3	7,4
1	TopRoll	201	A4865.R05.O1	2080	Ø 182,0	x 70,0	50,0	14,3	9,9
1	TopRoll	202	A4865.R05.O2	2080	Ø 182,0	x 70,0	50,0	14,3	9,9
1	LeftRoll	301	A5865.R05.L1	2080	Ø 202,0	x 36,0	52,0	9,1	6,5
1	RightRoll	401	A5865.R05.R1	2080	Ø 202,0	x 36,0	52,0	9,1	6,5
1	BottomRoll	101	A4865.R03.U1	2080	Ø 202,0	x 57,0	50,0	14,3	9,4
1	BottomRoll	103	A4865.R03.U3	2080	Ø 202,0	x 57,0	50,0	14,3	9,4
---									-----
53									395,1 236,5

Use this function to generate a parts list (sawing list) of the rolls of your [Profile Project](#). The parts list contains all rolls of all stands of the roll forming machine.

#### Calling the function

Select [View Rolls](#).

Call this function by:

- Main menu: **Output, Create Parts List.**

#### Principle of operation

The parts list is created, either by **Text File** or within **MS Excel**. After creating the parts list as **Text File**, you can view it, change it if you want, and print it, if you confirm **Yes** to the question **Parts list has been created. View now?**. The text editor (set in [Options General](#)) will be opened with the parts list. You can do the same with the function [Edit Parts List](#).

The parts list file contains blanks as separators between the values. So it is easy to export this file into a spreadsheet or an ERP system.

The file name of the parts list file is the same as the name of the project file, the file extension is

.txt, however.

### Setup

Preset it in [Options Parts List](#) previously, whether a **Text File** is created or the part list is transferred to **MS Excel**.

In case of **Excel** we recommend to use a pre-defined work sheet and to set the cursor to the row/column where the parts list should start. Numerical cells should be formatted as numerical with the desired decimal points. Sum cells are created automatically by **PROFIL**, the sum formula is transferred and the summarization is done by Excel.

The composition of the parts list is fully user defined in the window [Parts List Columns](#), call it by the button **Set-Up Columns** in [Options Parts List](#).

Equal parts list rows are combined to one single row. In this row, count, gross and final weight are summed up. Parts list rows are equal, if all visible entries are equal.

Select in [Options Calculate](#) whether the representation of the values should be **metric** (mm, Kg) or **imperial** (in, lb). You can preset the **Integer/Decimal Places** as well.

In case of **Text File** the Heading can be attached.

### Hints:

- Select by [View, Show, Spacer Rolls](#) whether the parts list output should contain spacer rolls.
- In order to save the parts list to a file with a different name, use the [Export](#) function.

### 3.1.8.5 Edit Parts List

Use this function to view the file based parts list, to change it and to print it.

#### Calling the function

Precondition: You have generated the parts list by using the function [Output Create Parts List](#) and in [Options Parts List](#) is checked **Parts List to Text File**

Select [View Rolls](#). Call this function by:

- Main menu: **Output, Edit Parts List.**

#### Principle of operation

The text editor (set in [Options General](#)) will be opened with the parts list.

#### Hint:

- Because the text editor is an independent program, it must be closed manually. Otherwise it will keep open in the background.

### 3.1.8.6 Create NC

```
%0A4865.R01.U1
N10 G71
N20 G01 X0 Z0
N30 G01 X98 Z0
N40 G03 X100 Z-1 I0 K-1
N50 G01 X100 Z-9.5
N60 G02 X101 Z-10 I0.5 K0
N70 G01 X125 Z-10
N80 G03 X127 Z-11 I0 K-1
N90 G01 X127 Z-49
N100 G03 X125 Z-50 I-1 K0
N110 G01 X101 Z-50
N120 G02 X100 Z-50.5 I0 K-0.5
N130 G01 X100 Z-59
N140 G03 X98 Z-60 I-1 K0
N150 G01 X0 Z-60
N160 M30

%0A4865.R01.O1
N10 G71
N20 G01 X0 Z0
N30 G01 X105.4 Z0
N40 G03 X107.4 Z-1 I0 K-1
N50 G01 X107.4 Z-22
N60 G02 X107.4 Z-23 I0 K0.414
N70 G01 X0 Z-23
N80 M30
```

Use this function to generate the NC programs for all rolls of your [Profile Project](#).

#### Calling the function

Before you create NC data you should use the function [Calculate, Plausibility Check](#) to check the whole profile project on errors.

Select [View Rolls](#). Call this function by:

- Main menu: **Output, Create NC.**

#### Principle of operation

A text file will be created with the name of the profile project, but the file extension **.G00**. The file contains the data of the geometry of the rolls in form of G01-, G02- and G03 commands by DIN 66025 for the machine control.

Afterwards, you can view it and append further machine commands to it, if you confirm **Yes** to the question **NC program has been created. View now?**. The text editor (set in [Options General](#)) will be opened with the NC file. You can do the same with the function [Edit NC](#).

## Setup

Define in [Options NC](#) the path for the file.

The file contains the NC programs of all rolls of the profile project. If you want to generate a separate file for each roll, you can select it in [Options NC](#).

Select in [Options Calculate](#) whether the representation of the values should be **metric** (mm, Kg) or **imperial** (in, lb). You can preset the **Integer/Decimal Places** as well.

### Hints:

- Select by [View, Show, Spacer Rolls](#) whether the NC output should contain spacer rolls.
- In order to create a set of different files sequentially, use the [Export](#) function.
- Function [File, Print](#) enables printing the NC program in combination with the roll drawing. Preset is in [File, Print Preview](#).

### 3.1.8.7 Edit NC

Use this function to view the NC program and to append further machine commands to it.

#### Calling the function

Precondition: You have generated the NC program by using the function [Output Create NC](#). Select [View Rolls](#). Call this function by:

- Main menu: **Output, Edit NC**.

#### Principle of operation

The text editor (set in [Options General](#)) will be opened with the NC file.

### Hint:

- Because the text editor is an independent program, it must be closed manually. Otherwise it will keep open in the background.

### 3.1.8.8 FEA

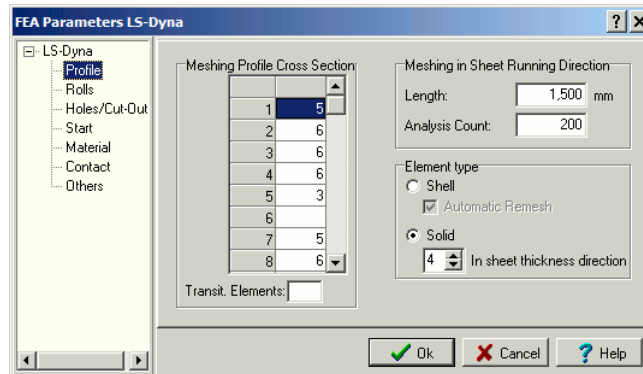
#### 3.1.8.8.1 LS-Dyna

Use this function to create the simulation model for the [FEA-Simulation](#), which can be processed by the FEA system **LS-Dyna**.

#### Calling the function

Precondition: You have created roll tools for all stands (see [How to Work](#)). L01 contains the final profile and the last pass Lnn contains the flat sheet (normally without rolls).  
Call this function by:

- Main menu: **Output, FEA, LS-Dyna**.



In the dialog window **FEA-Parameters LS-Dyna** is opened. These settings are necessary for the FEA simulation:

- [Profile](#): Meshing in profile cross section and in in sheet running direction, element type
- [Rolls](#): Meshing rolls axial and radial
- [Holes/Cut-Outs](#): Meshing the roll tools and holes/cut-outs
- [Start](#): FEA input/output path, start and restart, start position
- [Material](#): Material properties, stress-strain-curve, import
- [Contact](#): Roll tool contact, scale factor for sliding interface penalties
- [Others](#): Time step size mass scaling, guiding

#### Principle of operation

After pressing the **Ok** button, these file are created:

```
<project name>.dyn Main file that is opened by the LS-Dyna solver
<project name>.trm Trimming file, contains holes/cut-outs (if defined)
<project name>_op05.dyn File for trimming the sheet
<project name>_op10.dyn File for the first stand (in sheet running direction)
<project name>_op10.bnd File for guiding the first row of nodes (if parameterized)
<project name>_op20.dyn File for the second stand (in sheet running direction)
<project name>_op20.bnd File for guiding the first row of nodes (if parameterized)
...
etc.
<project name>.blk Geometry of the blank sheet
<project name>.mod Geometry of the rolls and the support
<project name>.dxf contains the blank sheet for defining the holes/cut-outs
<project name>.txt for documentation, contains the project data, the FEA parameterization and
messages
```

<project name> is the FEA Project Name defined in [FEA, LS-Dyna, Start](#).

Afterwards you can save all settings of the FEA parameters window into the project file by using the function [Save Project](#). If you later call **Output FEA** again, all settings are displayed again and can be modified easily.

#### Starting the simulation

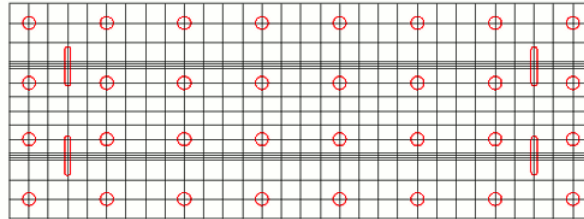
- Run the **LS-Dyna Program Manager**
- Call **Solver Start LS-Dyna Analysis**
- Enter **Input File** <project name>.dyn

- Check: **CASE run multiply load cases.**
- **Run**

### Defining prepunched holes/cut-outs

This function requires **Automatic remesh** and is currently possible only if the element type **Shell** is selected, see [Profile](#). In a later version this function will be available for element type **Solid**, too. If prepunched sheet should be roll formed, the holes/cut-outs simply can be defined in a 2D CAD drawing with any shape and position. The simulation result shows if the shape and position is as desired in the final profile after leaving the roll forming machine.

After pressing the **Ok** button in the **Output FEA LS-Dyna** window for the first time, the **DXF** file <project name>.dxf is created that contains the drawing of the flat sheet with the preset meshing. Open this file by using your CAD system and attach the desired holes/cut-outs pattern:



Please note:

- Define holes as **CIRCLES** in color **RED** (or in the color that is preset in **Holes/Cut-Outs, Colors DXF file, Holes/Cut-Outs** respectively).
- Define cut-outs as closed **POLYLINES** in color **RED** (or in the color that is preset in **Holes/Cut-Outs, Colors DXF file, Holes/Cut-Outs** respectively). The polylines can consist of line and (bulged) arc segments.
- Position the circles and polylines by orienting to the sheet corners or the mesh lines. Another method: Define [Holes/Cut-outs](#) in the [Profile List L01](#) of the [Profile List Window](#). (Previously extend the profile list window in [Options, Profile List](#) to layout **With Holes/Cut-Outs**). The center lines in the developed sheet in the DXF file are useful for positioning.
- Save the drawing in **AutoCAD 2000 DXF** format.

Press the **Ok** button again. **PROFIL** extracts the attached holes and cut-outs from the **DXF** file (only **circles** and **polylines** in color **RED**) and copies them to the trimming file <project name>.trm.

The meshing of the holes/cut-outs is done automatically by the solver (see [Profile](#), **Adaptive refinement, Automatic remesh**). In order to enable the Solver's Automesh function to do this correctly, select [Profile](#), **Meshing in Sheet Running Direction** with an aspect ratio of 1 preferably, at the utmost 5.

Start the solver and the profile is simulated with holes/cut-outs.

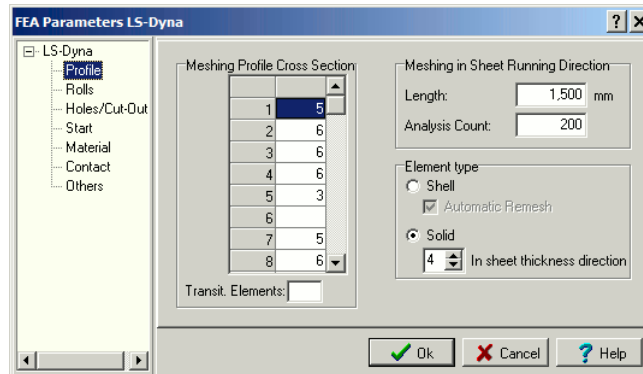
### Hints:

- Before you create the FEA simulation model you should use the function [Calculate, Plausibility Check](#) to check the whole profile project on errors.
- In order to get a good simulation result, the maximum aspect ratio of the shell or solid elements of the flat sheet should preferably not exceed 3. After pressing the **Ok** button, the status bar at the bottom of the main window shows the maximum aspect ratio of the profile and the rolls; the aspect ratio is saved in the <project name>.txt file, too. If the aspect ratio exceed 10, a message window opens and shows the corresponding profile element number. In case the sheet has bore holes/cut-outs, the maximum aspect ratio should not exceed 5. This is why the automesh function of the solver should be able to remesh the surrounding sheet mesh correctly.
- The maximum aspect ratio of the shell elements of the rolls should not exceed 20. Also this ratio is shown in the status bar and is saved in the <project name>.txt file.
- Rolls should not have sharp edges, i.e. in the cross-section should not touch two contour lines each other with an angle. The reason is: The surface normals of two adjoining surfaces change the direction abruptly. Because of the discontinuous function the numeric solver cannot consider the boundary conditions correctly. It is recommended to create a small fillet radius as in reality. The same applies to cut-outs in the sheet; it is also recommended to round out sharp corners.
- The file <project name>.txt contains the project data, the FEA parameterization and the messages **Max. aspect ratio of mesh elements Profile, Max. aspect ratio of mesh elements**

**Rolls, and Displacement guide curves,** Furthermore the number of the profile element or the width position of the roll respectively where the maximum riation occurred.

### 3.1.8.8.1.1 Profile

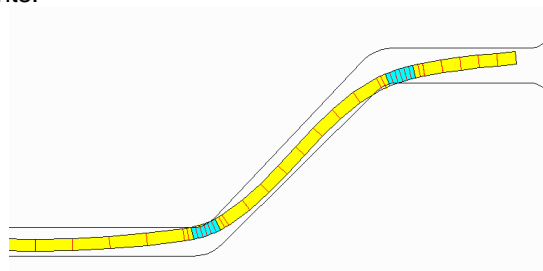
This dialog window is called by [Output, FEA, LS-Dyna](#) is used for parameterizing the meshing in profile cross section and in in sheet running direction.



Meshing means the strip is divided into small rectangular faces (shell or solid elements). The smaller the faces are, the more precise the calculation is, but the more calculation time is needed. So select the meshing as fine as necessary. Possibly select a more coarse meshing in profile parts that are less interesting. **LS-Dyna** is able to correct in case the faces are too large. Check the box **Adaptive refinement, Automatic remesh** and **LS-Dyna** will remesh automatically.

**Meshing Profile Cross Section:** Enter the count of FEA elements (right column) for each [Profile Element](#) (left column) of the [Profile List](#) individually. (L and A.. elements only, P and PS elements are ignored). The row numbering is the same as in the [Profile List Window](#). In case the project file still does not contain saved FEA parameter, a proposed meshing is shown in the table. You can modify it for your own needs.

**Transit. Elements:** possibly the FEA system wants to place an arc to a different position than you have planned it in your flower design. To enable this, transition elements can be set. When a line segment touches an arc segment, the beginning or the end of the line segment is divided in smaller items (with the same shell element length as the neighbouring arc segment). Select the count of transition elements.



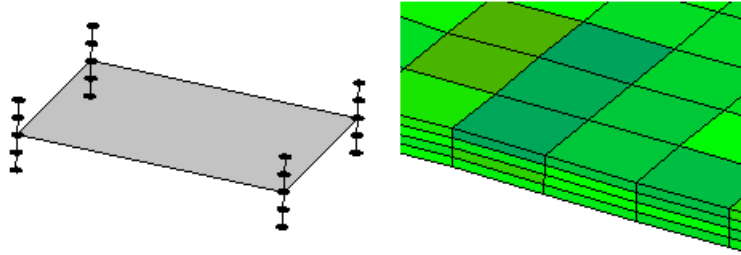
The picture shows a snap-shot while simulating a hat profile. When the sheet enters a roll stand, not only the bending zones are bent (blue, correspondents to the profile elements "Arc"), but also the parts of the profile that properly should keep unbent (yellow, correspondents to the elements "Line"). The example shows in the top right bending zone the roll attacking point actually is outside the planned bending zone. In order to enable the FEA system to calculate properly, in this example **Transit. Elements** = 2 was selected (small yellow items left and right of each bending zone).

**Meshing in Sheet Running Direction:** Select how precise the meshing should be in longitudinal direction.

- **Length:** Select the desired length a shell segments in longitudinal direction.
- **Analysis Count:** Select how many shell elements should appear in the FEA result. The

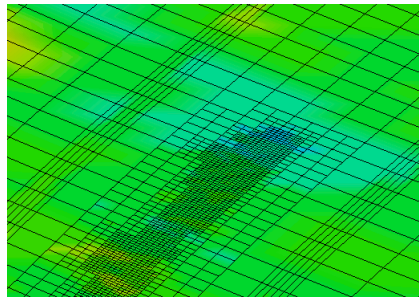
**Analysis Length** = **Analysis Count** multiplied by **Length** should be the distance between stands at minimum, better 2x the distance.

- **Element type:** Select the element type that should be used for the simulation.

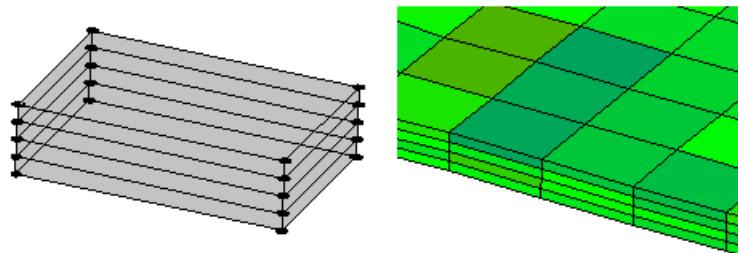


Shell element type with 5 integration points in thickness direction

- **Shell:** The shell model is adequate in case pure bending is used. This means, no deep drawing effects occur, the sheet thickness is not modified by the rolls, and 180° folds (hems) are not used. The shell is positioned in the center of the sheet. 5 integration points are calculated in sheet thickness direction: besides the shell itself 2 points above and 2 points below the sheet center. The calculation is faster than by using solid models.



**Automatic remesh:** The FEA solver is able to remesh automatically. This means, if the preset meshing is too large, the simulation does not break, but the FEA solver refines the meshing during the simulation automatically. This causes a higher stability and robustness against incorrect parameterization. Prepunched holes and cut-outs are meshed automatically as well. Check this box to enable remesh. Automatic refinement currently is possible for shell elements only.



Solid element type with 4 solids in thickness direction

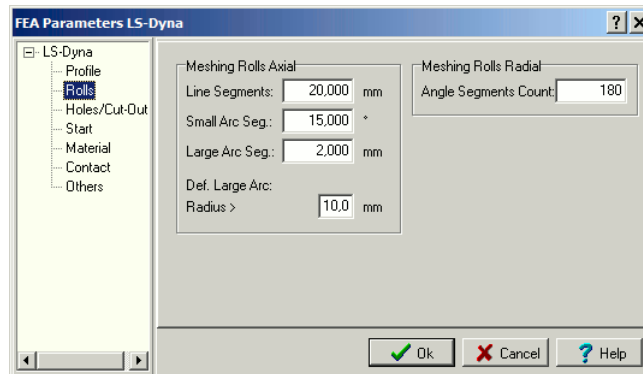
- **Solid:** The solid model is adequate for the forging process. In a roll forming process this arises during deep drawing, in case of intended sheet thickness modification, and during bending 180° folds (hems). It needs larger calculation time than by using the shell model. Automatic remesh and with it setting of bore holes and cut-outs is currently not possible; this is planned for a later release.

**In sheet thickness direction:** Select the count of solid elements in sheet thickness direction that should be used for the simulation. The count must be even; the minimum count is 2.



### 3.1.8.8.1.2 Rolls

This dialog window is called by [Output, FEA, LS-Dyna](#) is used for parameterizing the meshing of the rolls in axial and radial direction.



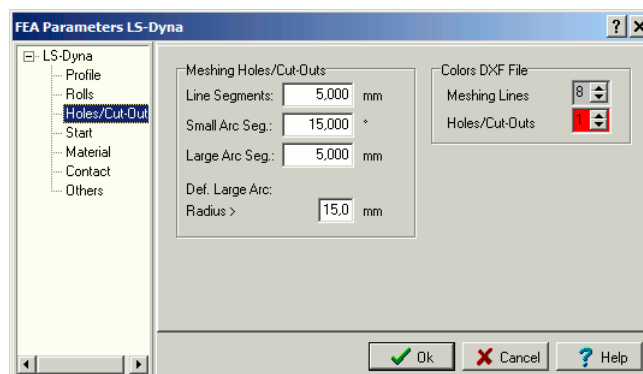
**Meshing Rolls Axial:** Line and arc segments are meshed as follows:

- **Line Segments:** Select the desired length of the shell elements for line segments. Since the divisor must be a whole number, the effective length will be approximate to the setting.
- **Small Arc Seg.:** Select the desired length of the shell elements for arc segments. The arc angle is divided. Regards the whole number divisor see above.
- **Large Arc Seg.:** Select the desired length of the shell elements for arc segments. The arc developed length id divided. Regards the whole number divisor see above.
- **Def. Large Arc:** Enter the limit between large and small arcs. A large arc is detected when the radius of the arc segment is larger than the preset limit.

**Meshing Rolls Radial:** Select in how many angle segments a roll should be split for meshing. E.g. the input **Angle Segment Count = 90** means, the rolls are split radially in  $360^\circ / 90 = 4^\circ$ -Segments.

### 3.1.8.8.1.3 Holes/Cut-Outs

This dialog window is called by [Output, FEA, LS-Dyna](#) is used for parameterizing the meshing of the roll tools and holes/cut-outs.



**Meshing Holes/Cut-outs:** Line and arc segments are meshed as follows: How to define holes/cut-outs in the flat sheet, read in the chapter **Defining prepunched holes/cut-outs** (later in this topic).

- **Line Segments:** Select the desired length of the shell elements for line segments. Since the divisor must be a whole number, the effective length will be approximate to the setting.
- **Small Arc Seg.:** Select the desired length of the shell elements for arc segments. The arc angle is divided. Regards the whole number divisor see above.
- **Large Arc Seg.:** Select the desired length of the shell elements for arc segments. The arc developed length id divided. Regards the whole number divisor see above.
- **Def. Large Arc:** Enter the limit between large and small arcs. A large arc is detected when the

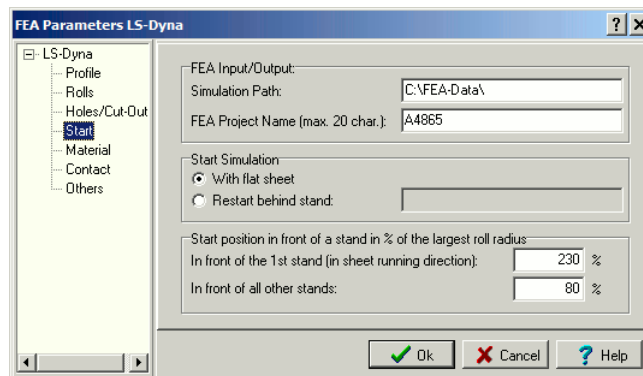
radius of the arc segment is larger than the preset limit.

**Colors DXF File:** After pressing the **Ok** button for the first time, **PROFIL** creates a DXF file with the meshed flat sheet. After pressing the **Ok** button again, the holes/cut-outs (inserted from the user) are extracted from the file and prepared for the simulation. Get more details from the chapter **Defining prepunched holes/cut-outs** (later in this topic).

- **Meshing Lines:** Modify the color for the meshing lines. This is only necessary in case the CAD system is not able to display the default color grey.
- **Holes/Cut-Outs:** In order to enable **PROFIL** to extract the holes/cut-outs entities from the file, they must be drawn in the preset color (default color red).

#### 3.1.8.8.1.4 Start

This dialog window is called by [Output, FEA, LS-Dyna](#) is used for set-up the FEA input/output path, start and restart, and the start position.

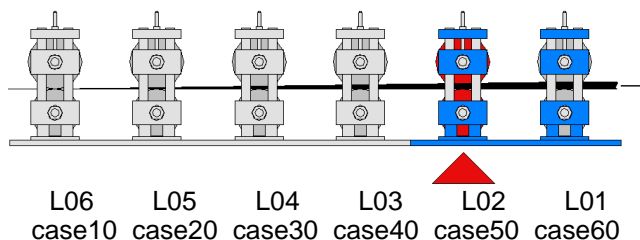


**FEA Input/Output, Simulation Path:** Enter the network path for saving the output files of the FEA simulation model and for loading the file of the FEA result. Hint: Since the FEA solver needs 100% of computing power, the FEA system **LS-Dyna** should run on a separate computer, not on the PROFIL/CAD workstation.

**FEA Input/Output, FEA Project Name:** The files of the simulation model are saved under this name. By default, the input field shows the [Profile Project](#) name. However, it can be modified in case e.g. several simulations should be made for the same project with different settings. Hint: the length of the input name should not exceed 20. If it does, the background color becomes red and the name is shortened to 20 characters. The name must not contain umlauts, blanks, and other special characters; they will be replaced by substitute characters.

#### Start Simulation:

- **With flat sheet:** Select this option, if you want to analyse a project for the first time, i.e. a result from earlier simulation does not exist.
- **Restart behind stand:** If you detect after proceeding the simulation that a roll stand does not produce the desired result, you can restart the simulation after modifying the rolls based on the result of the correct predecessor stand. This saves computation time, because successful simulated stands do not need to be repeated.



**Example:** Your project **Example.pro** consists of 6 stands **L01 .. L06** (and **L07**=flat sheet additionally). The first simulation, starting with the flat sheet, produces the files **Example\_case10.dynain** (corresponds to **L06**) until **Example\_case60.dynain**

(correspondents to **L01**). Hint: In PROFIL, the stands are numbered in designing direction, in LS-Dyna in sheet running direction. Now you detect an error in stand **L02**, i.e. the simulation result **Example\_case50.dynain** and all successive have to be repeated after modifying the roll of **L02**. Select in the file selection field **Example\_case40.dynain**, i.e. the simulation result, that still was correct. The following simulation will base on this result.

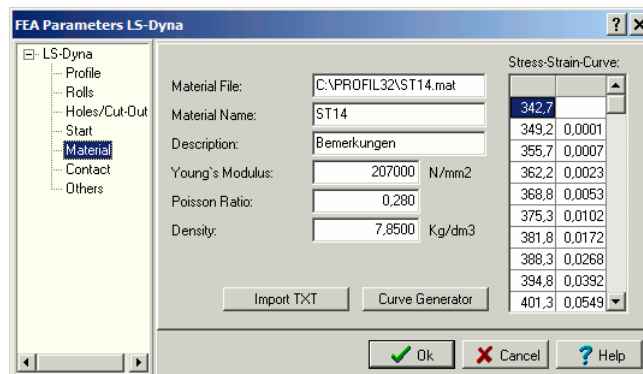
#### Start position in front of a stand in % of the largest roll radius:

The simulation of a stand is finished if the front end of the sheet is just before the next stand. This is the start position for the simulation of the next stand as well. Because the contact definitions for the next stand still are not effective, the profile can penetrate the rolls is the front end is too close to the rolls and the simulation cannot be continued. Enter the start position in % of the maximum roll diameter. 100% means maximum safety, the front end never can touch any roll. The simulation time, however, increases, because for the first interval a lower velocity is set.

- **In front of the 1st stand (in sheet running direction):** In case **Guiding the nodes of the profile reference point** is selected (see [Others](#)), the sheet can start closer, recommended is 50%. In case **Guiding the node at the profile lead end** is selected (see [Others](#)), a larger distance must be selected, e.g. 200%. This is why the guide curves must start at zero displacement.
- **In front of all other stands:** Recommended is 80% that prevents penetration. If **Guiding the first node row at the profile front** (see [Others](#)) is used, the profile can start closer to the rolls, e.g. at 30%.

#### 3.1.8.8.1.5 Material

This dialog window is called by [Output, FEA, LS-Dyna](#) is used for parameterizing the material properties, the stress-strain-curve, and for importing curves.



**Material File:** Select the name of the material file. All material data of this tab window are read from this file and are saved into this file. The file name extension is .mat.

**Material Name, Description:** for describing the material.

**Young's Modulus, Poisson Ratio, Density:** Enter the material properties.

**Stress-Strain-Curve:** Enter the points of the stress-strain-curve. The left column "Stress" should contain the true stress related to the current area, not to the original area of the tensile test specimen. The right column "Strain" contains the true (logarithmic) plastic strain, this means the true strain minus the elastic strain. Get more detailed information from the **LS-DYNA KEYWORD USER'S MANUAL VOLUME II Material Models**.



**Import TXT:** Imports a stress-strain-curve from a text file, e.g. from a tensile test according DIN standard. Each line of the file must contain a pair of values, either strain stress or stress strain of one curve point. Either decimal numbers are possible with dot or comma as decimal separator or numbers in exponential notation. The delimiter between the values can be space or tabulator. After

importing check the stress-strain-curve graphically by opening the **Curve Generator**.

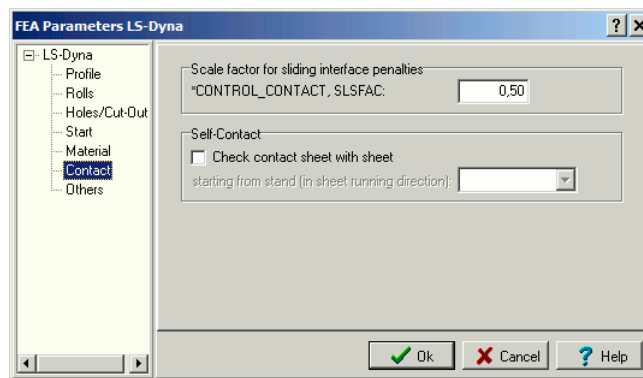
**Curve Generator:** In case the stress-strain-curve of the used sheet material is not available and you nevertheless want to proceed a FEA simulation with approximate values, you quickly can create a curve by the [Curve Generator](#) by defining three characteristic curve points.

#### Hints:

- In a roll forming process, also larger strain sometimes occurs, either desirably or undesirably. During simulating also for larger strain the belonging stress must be found. So it is recommended to extend the stress-strain-curve to strain 1 at least, better strain 2.
- Take care that the stress-strain-curve increases continuously at the end, this means the stress must be related the current area, not to the original area of the tensile test specimen.

#### 3.1.8.8.1.6 Contact

This dialog window is called by [Output, FEA, LS-Dyna](#) is used for set-up of the roll tool contact and the scale factor for sliding interface penalties.



**Scale factor for sliding interface penalties, \*CONTROL\_CONTACT, SLSFAC:** This factor determines how deep the sheet is allowed to penetrate the rolls. Recommended is 0.05 for element type **Shell** and 0.5 for element type **Solid**, see [Profile](#).

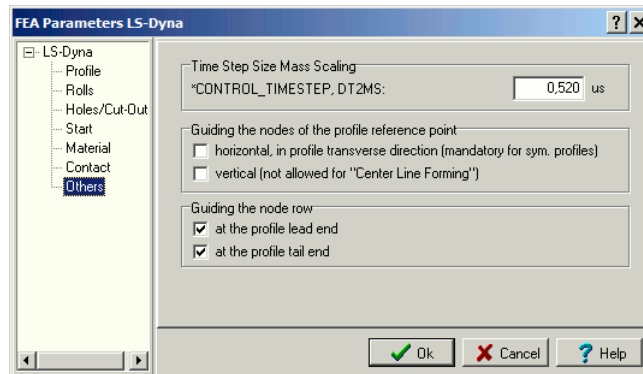
#### Self-Contact

**Check contact sheet with sheet:** Check this box, if LS-Dyna should avoid penetration in case the sheet touches itself. Self-contact is possible e.g. if the profile has a [Hem](#) or a tube enters the [Welding Station](#). Considering self-contact needs more simulation time. So it is recommended to check this box only if self-contact is possible due to the profile geometry.

**Starting from stand (in sheet running direction):** Select the stand from the drop-down-list to define the start of the self-contact check. This saves simulation time, because normally the first stands are without self-contact.

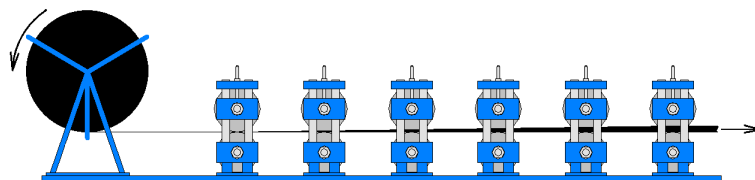
### 3.1.8.8.1.7 Others

This dialog window is called by [Output, FEA, LS-Dyna](#) is used for set-up the time step size mass scaling and the guiding.

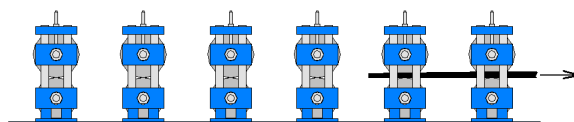


**Time Step Size Mass Scaling, \*CONTROL\_TIMESTEP, DT2MS:** Mass Scaling is useful for speeding up the FEA simulation. If, however, the time interval is too large, the FEA result may become invalid. Select the desired time interval. The recommended value for element type **Shell** is 1.2 us and for element type **Solid** 0.52 us, see [Profile](#).

**Why the profile must be guided?**



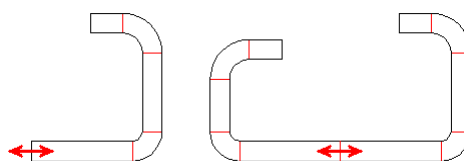
In reality, the sheet metal strip has a quasi-infinite length, because it is decoiled from a coil. However, a sheet with an infinite length cannot be simulated, because it would take infinite computing power or infinite time.



So for the simulation, a section with a finite length is used instead (setting in [Profile](#), **Meshing in Sheet Running Direction**). The picture shows a length of about 2x distance between stands. However, the behaviour of this section is different to a sheet with infinite length. Notably the front may push against a roll, this causes unrealistic deformation of the front that do not appear in case of a sheet with infinite length. That is why the tractive (the "guidance") of the sheet in front of the section is absent. In order to clone the reality at the best, it is a good idea to replace the absent sheet by a guidance, in the FEA nomenclature called a boundary condition. Exception: Blank manufacturing, when cut and clipped sections should be roll formed. In this special case the guidance can be switched off in order to simulate if the blank is fed correctly. Depending on the application, the user can select between 3 different methods of guidance, also a combination is possible:

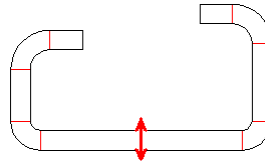
#### Guiding the nodes of the profile reference point

The node row (in sheet running direction) at the [Reference Point X0/Y0](#) of the profile is guided



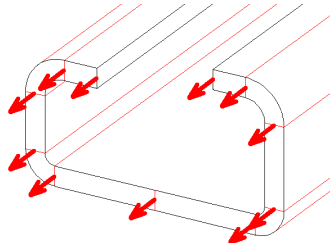
Guiding the profile reference point edge horizontally  
for symmetrical (left) and asymmetrical profiles (right)

**Horizontal:** Checking this box causes no horizontal movement of the profile reference point. This is mandatory for symmetrical profiles of which only one half is simulated (the right half) because of saving computing time. In order to avoid horizontal movement caused by the absent left half, it is necessary to guide the reference point edge (the center line edge) horizontally (see left picture). In case of asymmetrical profiles (right picture) the guidance can be enabled if needed, e.g. a very short strip length causes unwanted horizontal movement.



Guiding the profile reference point vertically

**Vertical:** After leaving a stand often the section drifts downwards, if at the top edges plastic strain occurs. In reality, the tractive of the sheet in front of the section avoids the movement. Checking this box avoids unrealistic vertical movement also for simulation. However, the guidance only is allowed if the [Reference Points X0/Y0](#) of all profile lists have the same height (bottom line forming). Otherwise, in case of [Center Line Forming](#), the vertical guidance should not be enabled.



Guiding the first row of nodes at the profile lead end

**Guiding the node at the profile lead end:** This method forces safe threading of the profile into the next roller stands and prevents pushing against the rollers. This also is assured if the [Reference Points X0/Y0](#) have different height, see [Center Line Forming](#). The designed flower pattern defines the target profile cross-section within the stands; between stands a model of cubic B-spines is used for approximately calculating the molding geometry. This clones the tractive of the sheet in front of the section and approximates the reality at the best. In other respects

**Guiding the node at the profile tail end:** This method prevents up and down oscillation of the profile tail end while the lead end enters the next stand. In other respects the mode of operation is like in **Guiding the node at the profile lead end**.

However, the guidance can be wrong. This occurs if the designed flower pattern differs from the cross-section calculated by the FEA system. This is shown by deformations of the lead or tail end during evaluating the FEA result. In order to mask this affect out during evaluation, a selectable count of rows can be cut off at the lead and tail end (see [View FEA Result](#)),

### 3.1.8.8.2 ABAQUS

Use this function to create the simulation model for the [FEA-Simulation](#), which can be processed by the FEA system **ABAQUS/Explicit**.

#### Calling the function

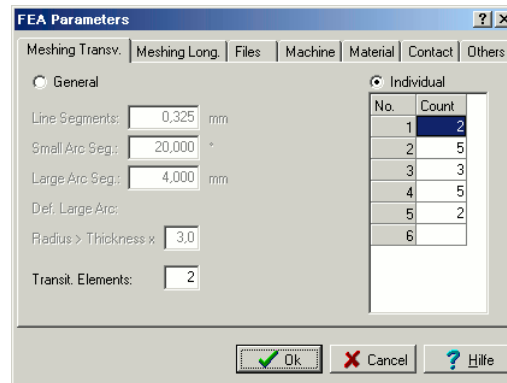
Precondition: You have created roll tools for all stands (see [How to Work](#)). L01 contains the final profile and the last pass Lnn contains the flat sheet (normally without rolls). Furthermore important for the FEA simulation: Set in [Expanded Roll Tool Window](#) the check box [Touching the Profile at](#). Call this function by:

- Main menu: **Output, FEA, ABAQUS.**

### Principle of operation

In the dialog window **FEA-Parameters** is opened. These settings are necessary for the FEA simulation:

### Meshing Transv.



Meshing means the strip is divided into small rectangular faces (shell elements). The smaller the faces are, the more precise the calculation is, but the more calculation time is needed. So select the meshing as fine as necessary. Possibly select a more coarse meshing in profile parts that are less interesting. Select if you want to define the meshing **General** (lengths and angles for lines and arcs) or **Individual** (count of shell elements per profile element).

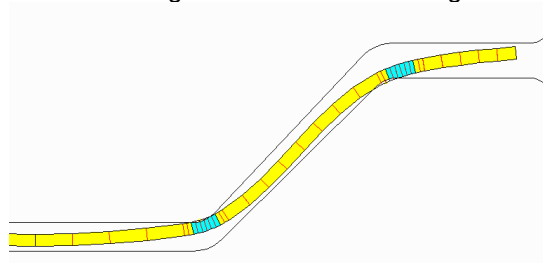
**Line Segments:** select the desired length of the shell elements for [Profile Elements](#) of type "Line". Since the divisor must be a whole number, the effective length will be approximate to the setting.

**Small Arc Seg.:** select the desired length of the shell elements for [Profile Elements](#) of type "Arc". The arc angle is divided. Regards the whole number divisor see above.

**Large Arc Seg.:** select the desired length of the shell elements for [Profile Elements](#) of type "Arc". The arc developed length is divided. Regards the whole number divisor see above.

**Def. Large Arc:** enter the limit between large and small arcs. A large arc is detected when the radius of the centerline of the sheet is larger than the sheet thickness multiplied by the preset factor.

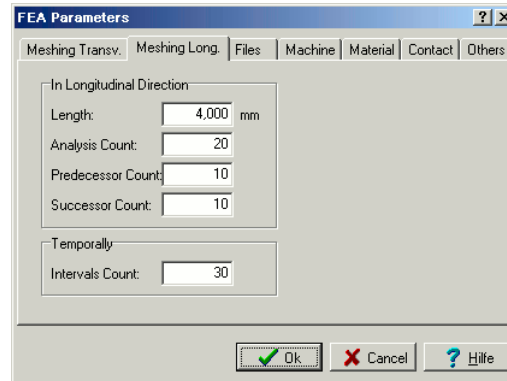
**Transit. Elements:** possibly the FEA system wants to place an arc to a different position than you have planned it in your flower design. To enable this, transition elements can be set. When a line segment touches an arc segment, the beginning or the end of the line segment is divided in smaller items (with the same shell element length as the neighbouring arc segment). Select the count of transition elements. This setting is effective for meshing **General** and **Individual**.



The picture shows a snap-shot while simulating a hat profile. When the sheet enters a roll stand, not only the bending zones are bent (blue, corresponds to the profile elements "Arc"), but also the parts of the profile that properly should keep unbent (yellow, corresponds to the elements "Line"). The example shows in the top right bending zone the roll attacking point actually is outside the planned bending zone. In order to enable the FEA system to calculate properly, in this example **Transit. Elements = 2** was selected (small yellow items left and right of each bending zone).

**Individual:** Enter the count of shell elements for each [Profile Element](#) of the [Profile List](#) individually. (L and A.. elements only, P and PS elements are ignored). The row numbering is the same as in the [Profile List Window](#). In case the project file still does not contain saved FEA parameter, a proposed meshing is shown in the table. You can modify it for your own needs.

### Meshing Long.



Set the meshing in longitudinal direction.

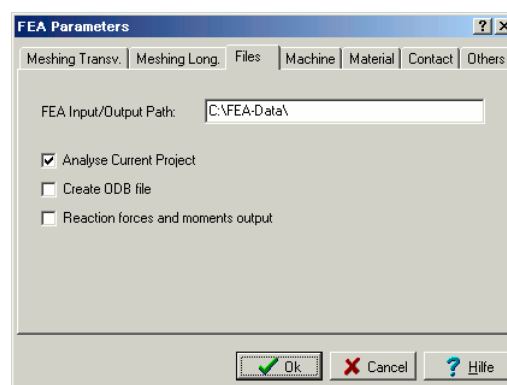
**Length:** select the desired length of the shell segments in longitudinal direction.

**Analysis Count:** Select how many shell elements should appear in the FEA result. The **Analysis Length = Analysis Count** multiplied by **Length** should be the distance between stands at minimum, better 2x the distance.

**Predecessor Count, Successor Count:** In front of the **Analysis Length** predecessor and successor elements are set that should take over the oscillation effects at the front and at the end of the sheet during the simulation. Later the predecessor and successor elements will be cut off and they will not be visible in the FEA result. Set the count of predecessor and successor elements, recommended is 10..20.

**Temporally, Intervals Count:** Select how many time intervals should appear in the FEA result. Recommended is 1..2x count of roll forming stands.

### Files



**FEA Input/Output Path:** Enter the network path for saving the output files of the FEA simulation model and for loading the file of the FEA result. Since the FEA solver needs 100% of computing power, the FEA system ABAQUS/Explicit should run on a separate computer, not on the PROFIL/CAD workstation.

**Analyze Current Project:** If the box is checked, PROFIL opens the FEA result with the same project name as the current project, when you call the function [View, FEA Result](#). Otherwise a file selection window appears and you can select any of the available FEA results file. If the FEA result

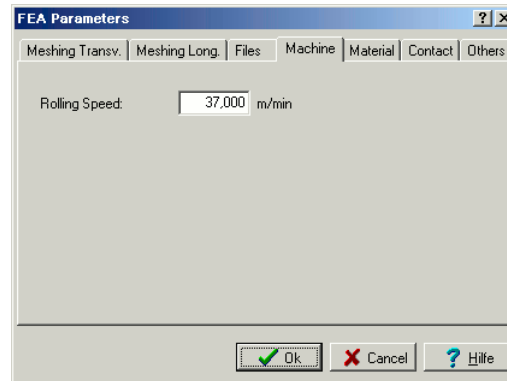


file cannot be found, the file selection window appears as well.

**Create ODB file:** If not only the ABAQUS/Explicit Solver, but also the ABAQUS/CAE (the user interface of ABAQUS) is installed, you can cause the solver to create an ODB file for **ABAQUS/CAE** (additional to the FIL file for PROFIL).

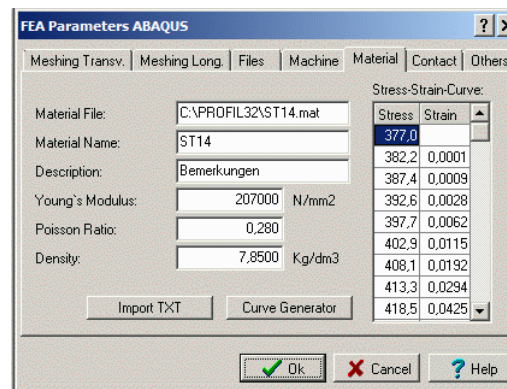
**Reaction forces and moments output:** If this option is checked (together with **Create ODB file**), the data are written into the ODB file and can be evaluated by **ABAQUS/CAE**.

### Machine



**Rolling Speed:** Select the strip speed of the roll forming machine.

### Material



**Material File:** Select the name of the material file. All material data of this tab window are read from this file and are saved into this file. The file name extension is .mat.

**Material Name, Description:** for describing the material.

**Young's Modulus, Poisson Ratio, Density:** Enter the material properties.

**Stress-Strain-Curve:** Enter the points of the stress-strain-curve. The left column "Stress" should contain the true stress related to the current area, not to the original area. The right column "Strain" contains the true (logarithmic) plastic strain, this means the true strain minus the elastic strain. Get more detailed information from the ABAQUS User Manual, Chapter "Getting Started", Topic "Materials, Plasticity in ductile metals".



**Import TXT:** Imports a stress-strain-curve from a text file, e.g. from a tensile test according DIN standard. Each line of the file must contain a pair of values, either strain stress or stress strain of one curve point. Either decimal numbers are possible with dot or comma as decimal separator or numbers in exponential notation. The delimiter between the values can be space or tabulator. After

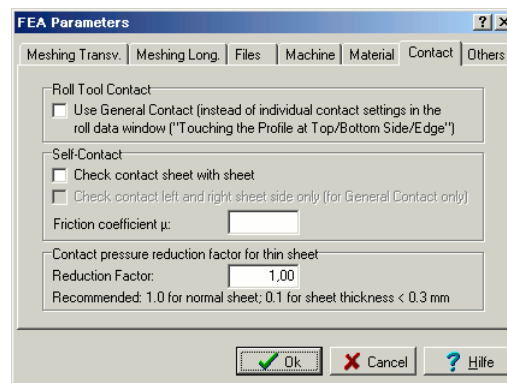
importing check the stress-strain-curve graphically by opening the **Curve Generator**.

**Curve Generator:** In case the stress-strain-curve of the used sheet material is not available and you nevertheless want to proceed a FEA simulation with approximate values, you quickly can create a curve by the [Curve Generator](#) by defining three characteristic curve points.

#### Hints:

- In a roll forming process, also larger strain sometimes occurs, either desirably or undesirably. During simulating also for larger strain the belonging stress must be found. So it is recommended to extend the stress-strain-curve to strain 1 at least, better strain 2.
- Take care that the stress-strain-curve increases continuously at the end, this means the stress must be related the current area, not to the original area of the tensile test specimen.

#### Contact



**Roll Tool Contact, Use General Contact:** The General Contact is an improved, time-saving alternative to the individual contact definition in the [Expanded Roll Tool Window, Touching the Profile at](#). ABAQUS Rel. 6.8 is necessary for this option. When this box is checked, the settings in the roll tool window are ignored.

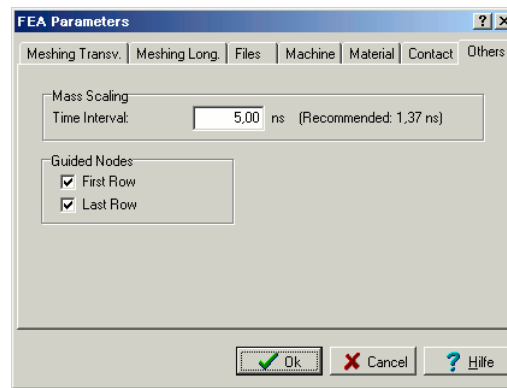
**Self-Contact, Check contact sheet with sheet:** If the profile touches itself, e.g. if edges are fold, this option can be set. This avoids unwanted self-penetration of the sheet. If no self-contact is possible, the checkbox should keep unchecked in order to reduce the computing time.

**Check contact left and right sheet side only:** This option can be set alternatively for e.g. tubes or tubular profiles. This also reduces the computing time. For this option, **General Contact** is necessary.

**Friction coefficient  $\mu$ :** To consider the friction between sheet and sheet, the friction coefficient can be set.

**Contact pressure reduction factor for thin sheet:** Contact pressure is set in order to have necessary pressure between the contact surfaces. The preset pressure is optimized for normal sheet thickness of 0.5-5mm. Select the **Reduction Factor** = 1.0 for normal sheet thickness. For thin sheet the preset pressure possibly is too big. If you see shooting out nodes in the FEA result caused by too high pressure, decrease the pressure by using the reduction factor. Recommended value is 0.1 for sheet thickness < 0.3 mm.

#### Others



**Mass Scaling, Time Interval:** Mass Scaling is useful for speeding up the FEA simulation. If, however, the time interval is too large, the FEA result may become invalid (see [View FEA-Result](#), Mean Energy Ratio). Select the desired time interval. The recommended value (behind the input field) is calculated dependent on the smallest shell element edge length and the material data.

**Guided Nodes, First Row, Last Row:** To consider the real environment (infinite long strip) while the FEA simulation (with a relative short strip), the first row of nodes of the predecessor elements and the last row of nodes of the successor elements are guided. In special cases, e.g. if cut strips are roll formed, it may be useful to switch off this guiding. Please use this checkbox with care, without guiding it may be possible that the strip is not threaded correctly (like on a real roll former).

After pressing the Ok button, these file are created:

```
<project name>.inp
<project name>.in1
<project name>.in2
<project name>.in3
<project name>.in4
<project name>.in5 (only if Check contact left and right sheet side only is set)
<project name>.txt
FE.bat
```

The first 5 files contain the FEA simulation model. The txt-file contains the documentation of the FEA-settings. FE.bat is the batch file to start the ABAQUS/Explicit solver.

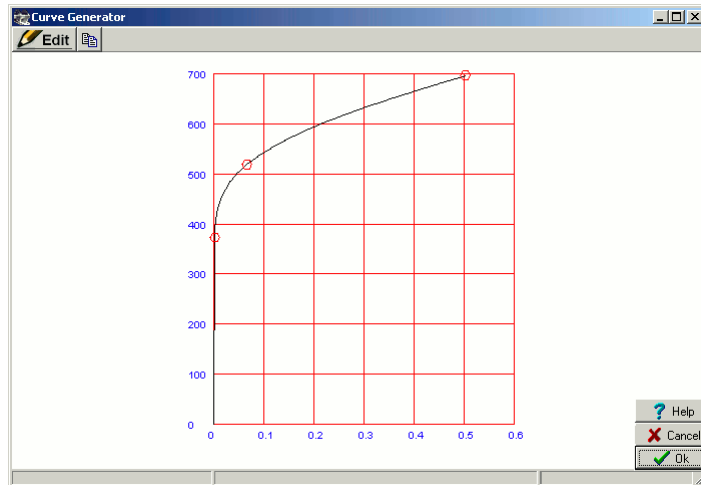
Afterwards you can save all settings of the FEA parameters window into the project file by using the function [Save Project](#). If you later call **Output FEA** again, all settings are displayed again and can be modified easily.

**Hint:**

- Before you create the FEA simulation model you should use the function [Calculate, Plausibility Check](#) to check the whole profile project on errors.

### 3.1.8.8.3 Curve Generator

Often the exact stress-strain-curve of the used material is not available and cannot be obtained shortly. Nevertheless the designer wants to proceed a [FEA simulation](#) with approximate values. The **Curve Generator** is a handy tool for creating a stress-strain-curve quickly by defining three characteristic curve points. The three points are: Yield point, tensile breaking stress point, and a medium point that controls the bulge of the curve. The curve has a tangential connection to the Hookean line (defined by the Young's modulus) and is continuous rising with rising strain.






### Calling the function

- Main menu: **Output, FEA, LS-Dyna** or **Output, FEA, ABAQUS**. In the tab **Material** press the button **Curve Generator**.

The window **Curve Generator** opens and first shows the stress-strain-curve that is selected in **Material, Material File**. If no material file is selected, an example curve is shown.

### Modifying the stress-strain-curve

You can modify the curve in this way:

-  **Edit** Press the **Edit** button. 3 curve points are shown.
- Select the **Yield Point** (left). Move the point up and down in order to modify the yield stress. Or enter the desired yield stress in the input field **Stress** in the header bar. The belonging elastic strain is calculated dependant on the Young's modulus.
- Select the **Tensile Breaking Stress Point** (right). Move the point in any direction in order to modify stress and strain at the breaking point. Or enter the desired value in the input field **Stress** or **Strain** in the header bar.
- Select the medium point. Move the point left and right in order to modify the bulge of the curve. Or use the arrow buttons in the header bar for modifying.
- Select **Interpolation Points** in the header bar in order to get a continuous curve without kinks.
-  **Copy drawing to clipboard:** Use this button to copy the stress-strain-curve as pixel drawing to the Windows clipboard (see also [Edit Copy](#)).
-  **Drawing -> CAD:** Use this button to transfer the stress-strain-curve as vector drawing to the CAD system. The settings of the function [Drawing -> CAD](#) in the PROFIL main window are used as well.

### Principle of operation

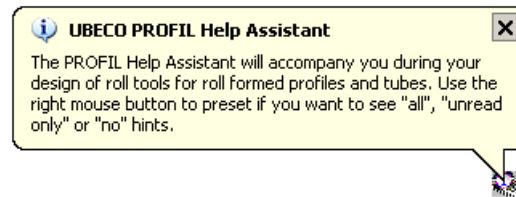
The stress-strain-curve is rebuilt permanently after each modification. A new curve appears with these properties:

- Tangential connection to the Hookean line.
- Continuous rising with rising strain (true strain for FEA, not nominal strain from the tensile test).

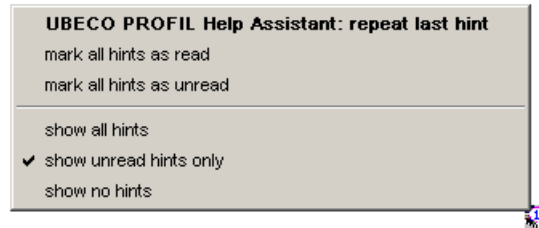
After pressing the **Ok** button the new stress-strain-curve is entered into the **Stress Strain Curve Table** in **Output, FEA**.

### 3.1.9 Help

#### 3.1.9.1 Assistant

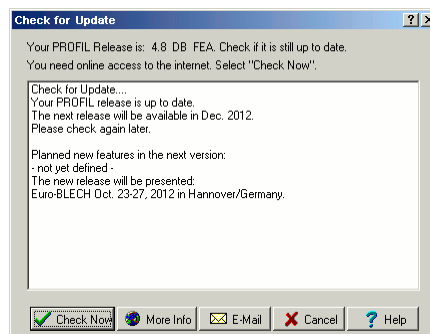


The help assistant accompanies you during your design work by showing helpful hints in a speech balloon. In case the display time was too short, click on the **PROFIL** symbol in the right corner and the hint will appear again.



By right click on the **PROFIL** symbol the assistant menu is shown. Select, if you want to see all or no hints or if only unread hints should be shown. In the last case each hint appears only once. Furthermore you can mark all hints as read or unread.

#### 3.1.9.2 Check for Update



Use this function to check whether your PROFIL release still is up to date or if a new release is available in the mean time. For this, you need online access to the internet.

#### Calling the function

Call this function by:

- Main menu: **Help, Check for Update.**

#### Principle of operation

The window **Check for Update** is opened and you can select:

**Check Now:** PROFIL checks the UBECO website if an update is available.























**More Info:** The Web browser opens the web site <http://www.ubeco.com>.

**E-Mail:** A new mail is created and the body contains the serial and release number of your PROFIL version. You can use it to ask for the price of the update.

## 3.2 Buttons

### 3.2.1 Button Key Bar

Use the button key bar for quicker working. So you do not need to open the pull down menus:

-  [New Project](#)
-  [Open Project](#)
-  [Save Project](#)
-  [Print](#)
-  [Profile, Read CAD-Contour](#) or [Roll, Read CAD-Contour](#)
-  [Roll Scan Profile Drawing](#)
-  [Profile, Append](#)
-  [Machine](#)
-  [Window Visible/Hidden](#)
-  [Statics](#)
-  [Stress of Edge](#)
-  [Profile Catalogue](#) or [Roll Stock Management](#)
-  [Undo](#)
-  [Redo](#)
-  [Options](#)
-  [Help](#)
-  [Toolbox Profile Design](#)
-  [Toolbox Tube Design](#)
-  [Toolbox Modify](#)
-  [Toolbox Dimensioning](#)
-  [Grid on-off](#)
-  [Dimensions on-off](#)


[Inspect](#)[View Pass](#)[View Statics](#)[View Flower Nested](#)[View Flower Separated](#)[View Flower 3D](#)[View Roll Tools](#)[PSA - Profile Stress Analysis](#)[View FEA Result](#)[Drawing -> CAD](#)[Drawing -> NC](#)

### 3.2.2 Dimensions on-off

This function switches all dimensions temporarily off and on again.

#### Calling the function

Call this function by:

- Main menu: **Output, Drawing -> CAD**.
-  Button **Dimensions on-off** in the [Button Key Bar](#).

#### Principle of operation


In position **switched off** the dimensions are not displayed in the [Drawing Area](#), not transferred to CAD ([Drawing -> CAD](#)) and not printed ([Print](#)).

### 3.2.3 Inspect

If you have selected a pass or a roll in the [Drawing Area](#) by mouse click, all other parts of the drawing appear in the inactive color and are not transferred to CAD by [Drawing -> CAD](#). The selection can be suspended by **Inspect**.

#### Calling the function

Call this function optionally by:

- **Right Mouse Button**, click into the background of the [Drawing Area](#).
-  Button **Inspect** in the [Button Key Bar](#).

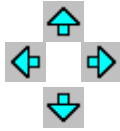
#### Principle of operation

Use the function **Inspect** to deselect the pass or the roll again and to display the whole drawing in its default colors for better inspecting.

### 3.2.4 Navigator

Use the Navigator in the upper right corner to modify the view port of the [Drawing Area](#). It contains the following buttons:

#### Left/Right/Up/Down



Use these buttons to move the view port left, right, up and down. If you use a wheel mouse, you can move the view port by moving the mouse, while the wheel key is pressed. If you use a space mouse (from 3DConnexion), press the cap to the left or right or to the front or back.



**Fit**

Use this button to fit the drawing into the view port. In case of a space mouse, use function key 1.



**Zoom +/-**

Use these buttons to zoom the view port. If you use a wheel mouse, you can zoom by turning the wheel. If you use a space mouse (from 3DConnexion), press the cap down or draw it upwards.



**Zoom Window**

Use this function to zoom by clicking two opposite corners of a rectangle for the new view port. After clicking the first point a drag box is displayed in the predefined color for marked elements.



**Previous Zoom**

Use this function to recall the previous zoom. You can repeat it 10 times.

### 3.2.5 Navigator 3D



The **Navigator 3D** appears on the right side, if you call [View, PSA - Profile Stress Analysis](#) or [View, FEA Result](#). Use the arrow buttons to rotate the 3D view port. The center button resets to the initial view port (Setting in [Options Drawing](#), View Port Angle). By clicking on the background area you can switch to the 2D view.

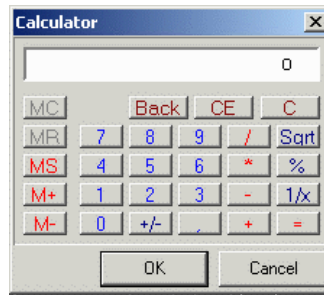


**Space-Mouse:** If you use a space mouse from [3DConnexion](#), the motion controller allows pan, zoom, and rotate the model simultaneously. The function keys are allocated by

- Key 1: Fit ([Navigator](#))
- Key 2: Reset the initial view port (Navigator 3D)
- Key 3: Front view
- Key 4: Not allocated
- Key 5: Top view
- Key 6: Side view from the right
- Key 7: Side view from the left



### 3.2.6 Calculator



The calculator can be opened from any input field's context menu.

#### Calling the function

Call this function by:

- **Right Mouse Button, Calculator** in any numerical input field

In case of table fields please double click to change to the **Edit**odus.

#### Principle of operation

The current value of the input field is copied to the calculator's display.

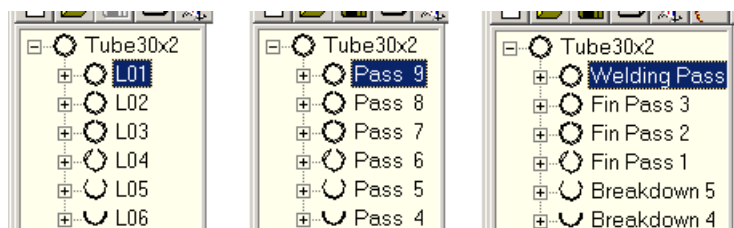
These basic arithmetical operations are possible: addition, subtraction, multiplication, and division. Furthermore: inversion, percentage calculation, square root, and memory functions.

After pressing the **OK** button the result from the display is copied into the input field the calculator was opened from. Hit **Enter** or **Tab** and the new value will take effect.

**Global Memory Functions:** If you save a value to the memory (function MS, M+, M-) you can recall it for later calculations (function MR). This also is possible if you closed the calculator and reopen it again in another input field, except you deleted the memory (function MC). Thus the content of several input fields can e.g. be multiplied by the same constant value.

## 3.3 Windows

### 3.3.1 Profile Explorer



The **Profile Explorer** on the left side of the screen clearly shows the passes, stands, and rolls of a profile project. It can be used for quick navigating between all objects of the project. The tree diagram contains the project name, the pass numbers (L01 .. etc.), the roll number, and a small icon that shows the preview of the object.

#### Principle of operation

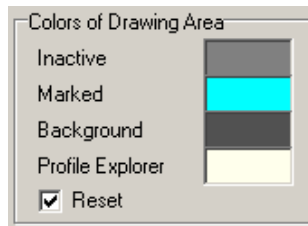
Clicking the **+** sign opens the tree node and the objects of the node become visible. The **-** sign closes the node. By clicking on an object the belonging drawing is shown in the drawing area. If necessary, the view changes automatically between flower, pass, stand, and roll. If another object is selected in the drawing area, the belonging explorer entry is marked.

### Setup

Function [Edit, Explorer](#) toggles the Explorer between **Profile List Number** (counting against the sheet running direction, left Pict.) and **Pass Number** (counting in sheet running direction, center Pict.) and **Stand Name** (right Pict.).



While working with larger projects the screen refresh takes more time, because refreshing the preview pictures is necessary after each modification. This may impede quick working. Simply switch off the preview in [Options, General](#).



The background color can be preset in [Options Drawing](#). By moving the vertical split line between explorer and drawing area the width of the explorer can be modified. If more explorer entries exist than the window is able to display, a vertical scroll bar appears.

#### Hint:

- If needed, the profile list number can be used as variable **\$PL** for the automatic roll numbering, see [Options Rolls](#).

### 3.3.2 Project Data

In this window the project data of a [Profile project](#) are shown and you can edit them.

The project data are:

<a href="#">Customer</a>	<a href="#">Date</a>
<a href="#">Description</a>	<a href="#">Name</a>
<a href="#">Drawing No.</a>	<a href="#">Revision</a>
<a href="#">Material</a>	<a href="#">Thickness</a>
<a href="#">Machine</a>	<a href="#">Calculation Methods</a>

If you want to enter a new customer or a new name, you can select it from the list box, which shows you the older names. To delete an older name from the list box, use the backspace key.

See also: [Profile List Window](#)

### 3.3.2.1 Customer

The customer's name belongs to the project data.

Enter the name of your customer, who has ordered the profile. PROFIL saves the name in the project file for your information and for easier selection.

See also: [Profile Project](#)

### 3.3.2.2 Description

The description belongs to the project data.

Enter any further information that allows easy identification. PROFIL saves this information in the project file. If needed, the description can be used as variable \$PD for the automatic roll numbering, see [Options Rolls](#).

See also: [Profile Project](#)

### 3.3.2.3 Drawing Number

The drawing number belongs to the project data.

Enter the drawing number. PROFIL saves the name in the project file for your information and for easier selection. If needed, the drawing number can be used as variable \$DR for the automatic roll numbering, see [Options Rolls](#).

See also: [Profile Project](#)

### 3.3.2.4 Material

The name of the material belongs to the project data.

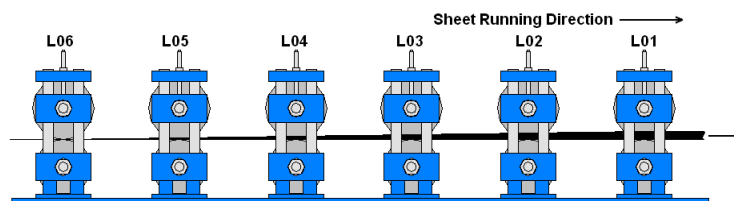
To select the material from the [Material File](#), click on the arrow button on the right and the table of materials drops down. Select the desired material.

Dependent on material are the calculations of the [Spring Back](#), the [Stress of Edge](#), and [PSA - Profile Stress Analysis](#).

To edit or to enhance the material file use the function [Options Calculate](#).

See also: [Profile Project](#)

### 3.3.2.5 Machine



This field shows the name of the machine. You cannot modify it.

In order to modify the name, click on this field and the [Machine Window](#) opens. Modify the content

of the [Machine](#) input field or load another machine file by using the import button.

See also: [Profile Project](#)

#### 3.3.2.6 Date

The date belongs to the machine data.

If you create a new project by using the function [New Project](#), the input field is automatically set with the current date.

See also: [Profile Project](#)

#### 3.3.2.7 Name

The project author's name belongs to the project data.

Enter your name the first time you create a new project. Later your name is automatically set.

See also: [Profile Project](#)

#### 3.3.2.8 Revision

The revision date belongs to the project data.

If you save the profile project by using the function [Save Project](#), the revision date is automatically set with the current date.

See also: [Profile Project](#)

#### 3.3.2.9 Thickness

The sheet thickness belongs to the project data.

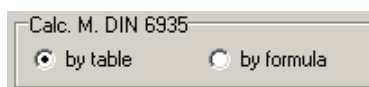
Enter the sheet thickness before you use the function [Profile, Read CAD-Contour](#) or before opening a basic profile from the [Toolbox Profile Design](#). If you want to modify the thickness afterwards, call the function [Modify Sheet Thickness](#) or enter the new thickness directly in the input field **Thickness**. In both cases a dialog window opens and you can select, which radius, should be kept constant during this operation. See also: [Modify Sheet Thickness](#).

See also: [Profile Project](#)

#### 3.3.2.10 Calculation Method

The calculation method belongs to the project data.

Select, which [Calculation Method](#) should be used for the calculation of the straight length of the arc elements and the strip width of the whole profile. The Oehler and the DIN 6935 methods have been implemented.



In [Options Calculate](#), **Calc. M. DIN 6935** you can select if the length should be calculated by **by**

### Table or by Formula.

Furthermore you can select [User Defined Calculation Methods](#). We recommend to you to use the Oehler method, because it is most precise.

If you click on the arrow at the right side of the input field, a box drops down and shows you the calculation methods. Select the desired method by clicking

If you modify the calculation method, the straight length of the arc elements will change as well as the [Strip Width](#).

See also: [Profile Project](#)

### 3.3.3 Profile List

No.	Typ	Di	Radius dis	ch.Angle	St.Length	Str.
1	L				15,500	
2	A1	L	1,500	90,000	4,109	102
3	L				27,500	
4	A1	L	3,000	90,000	6,665	51
5	L				5,000	
6	PS					

In this window, the [Profile List](#) is shown and you can edit it.

Select in [Options Calculate](#) the number of decimal places and the unit mm or inch.

Furthermore select in [Options Profile List](#), **Layout** if the window should show not only radius/angle discharged, but also radius/angle loaded (**With [Spring Back](#)**) or if also [Holes/Cut](#) Outs should be visible.

Select in [Options Profile List](#), **Profile List Window** how many windows should be visible at the same time.

The coloured background in the columns Radius/Angle shows the status of the menu switch [Profile, Loaded](#).

A zero is not displayed; empty numerical fields contain 0.

To modify a value in an input field, activate it and press the Pg Up/Dn keys on the keyboard. If you have opened further windows, e.g. statics window or stress of band edge window, the contents will simultaneously be updated. Also the drawing will be updated. Set the step interval for the modification in [Options Mouse](#).

Click into a row of the table, and the corresponding profile element in the [Drawing Area](#) will be highlighted in the predefined mark color. Vice versa, if you identify a roll corner point in the drawing, the corresponding row of the table in the profile list window will be activated.

All values are shown in mm, if you have selected in [Options Profile List](#) the representation of values **Metric**. If you have selected **Imperial**, they are shown in inch.

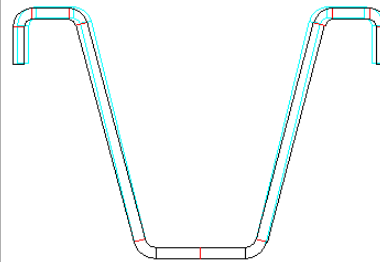
#### Hint:

In certain input fields further functions are available from the context menu (right mouse button):

- **Cut, Copy, Paste, Delete:** Enables transfer of parameters via the clipboard.
- **Copy To All:** Copies the selected parameter to the corresponding input field in all other profile lists.
- **Calculator:** Opens the [Calculator](#) and copies the selected parameter to it's display. After closing the calculator the result is transferred back to the input field. Press the **Enter** or **Tab** key afterwards.

### 3.3.4 Spring Back

Channel_L02									
Pass	Pass 1		x0:	200,000					
Dist.Stand:	300,000		y0:	150,000					
Strp.Width:	105,456		Direc.(°):						
No.	Typ	Di	Radius dis	ch.Angle	Radius loa	ded.Angle	St.Length	Str.	
1	L						6,000		
2	A1	L	1,500	75,000	1,466	76,142	2,553	51	
3	L						30,000		
4	A1	R	1,000	75,000	0,974	76,142	1,898	77	
5	L						5,000		
6	A1	R	1,000	90,000	0,974	91,371	2,278	77	
7	L						5,000		
8	PS								



For designing the roll tools for the finishing stand, you have to use the geometry of the profile in the loaded state.

The spring back of a profile is automatically calculated, if you define a profile by function [Profile, Read CAD-Contour](#) or you modify a pass in the [Profile List Window](#) or in the drawing.

Precondition is that you have selected a material in the [Project Data Window](#). The calculation of the spring back uses the factors K1 and K10 of the [Material File](#). These two factors are used as supporting values for an e-function for interpolation.

The spring back is shown, if you select **With Spring Back** in [Options Profile List](#).

You can define by using the menu switch [Profile, Loaded](#), whether the discharged or the loaded state should be used for display, design and for calculation.

PROFIL uses the Oehler method for getting the over bend angle. This method is valid from the ratio radius/thickness = 1 until 100. If in case of very large radii the ratio 100 is exceeded, a warning message comes: "Calculation of spring back not possible at ratio radius/thickness > 100". Otherwise, a loaded angle is suggested and displayed nevertheless. This value comes from extrapolating the e-curve over the limit of 100. But the value is not safe, so you must use it at your own risk. It is a good idea to provide for an adjusting mechanism in the machine in order to compensate the springback in case of very large radii.

### 3.3.5 Roll Tool

No.	Width	Diameter	Radius	Angle	PE
1	-30,000	100,000	1,000		
2	-20,000	100,000	0,500	90,000	
3	-20,000	127,000	1,000		
4	20,000	127,000	1,000	270,000	
5	20,000	100,000	0,500		
6	30,000	100,000	1,000		

In this window, the data of both a forming roll and a spacer roll are displayed and you can edit them.

The window contains in the top part the following data:

[Roll Number](#)      [Width](#)  
[Part Number](#)    [Max. Diameter](#)  
[Classification](#)   [Spacer Roll](#)

More..

Button **More** opens the [Expanded Roll Tool Window](#)

In the lower part there is the list of the [Roll Corner Points](#):

[Width](#)                      [Diameter](#)  
[Radius](#)                  [Angle](#)  
[PE](#)

Click into a row of the table, and the corresponding roll corner point in the [Drawing Area](#) will be highlighted by a little cross-hair in the predefined mark color. Vice versa, if you identify a roll corner point in the drawing, the corresponding row of the table in the roll tool window will be activated.

All values are shown in mm, if you have selected in [Options Profile List](#) the representation of values **Metric**. If you have selected **Imperial**, they are shown in inch.

### 3.3.6 Expanded Roll Tool Window

This window appears if you press the key **More** in the [Roll Tool Window](#). It is effective both for forming rolls and for spacer rolls. For each roll you can enter these additional properties:

[Diameter Shaft](#)  
[Driven](#)  
[Bore Hole](#)  
[Bushing](#)

[Identification Groove](#)  
[Material](#)  
[Treating, Surface, Addition, Remark](#)  
[Touching the Profile at](#)

The entries in the fields **Bore Hole** .. **Material** are file names. After mouse click in one of these fields the file select window opens and you can select a file name.

The keys **Edit** call the text editor (set in [Options General](#)) that opens the file with the file name in the belonging input field.

The entries **Treating**, **Surface**, **Addition** and **Remark** are user defined concerning field name and field content. The drop-down-boxes (opened by a mouse click on the arrow symbol) contains recent user inputs. This makes new input easier.

The same window appears too, when you press the button **More** in [Options Rolls](#) or in [Options, Spacer Rolls](#). You can enter default values that are copied into a roll dataset when you create a new roll by [Roll Read CAD-Contour](#) or [Roll Scan Profile Drawing](#) or a spacer roll by [Roll, Create Spacer Roll](#). Furthermore you can preset the field names of **Treating**, **Surface**, **Addition** and **Remark** in this window. If the window is opened from [Options, Spacer Rolls](#), the field **Material** is inactive, because the material for spacer rolls is preset in the [Machine Window](#). The same also applies to the field [Diameter Shaft](#) for forming rolls and spacer rolls.

### 3.3.6.1 Diameter Shaft

The diameter shaft belongs to the roll data and is displayed in the header of the [Expanded Roll Tool Window](#).

It is read out of the [Machine Data](#), when a roll is created, and can be changed subsequently. The bore hole can be displayed in the roll tool drawing, select this in [Options Drawing](#).

### 3.3.6.2 Driven

Driven belongs to the roll data and is displayed in the [Expanded Roll Tool Window](#).

A roll is driven if it has form fit with the shaft, e.g. by a key. Driven rolls move the sheet through the machine. Otherwise it is an idle (free rotational) roll, the speed of which is caused by the friction with the sheet.

When creating bottom and top rolls they are marked as driven and side rolls are marked as not driven first. These settings can be reversed manually if necessary. If a [Bushing](#) is selected for a roll, the mark **Driven** is removed.

### 3.3.6.3 Bore Hole

Bore Hole belongs to the roll data and is displayed in the [Expanded Roll Tool Window](#).

Bore Hole is the name of a file that e.g. contains these data:

@BOREFILE	# Bore Hole
75.000	# Diameter
H7	# Fit
16.000	# Key, Width
10.000	# Key, Height
6.000	# Key, Depth of keyway in shaft
3.400	# Key, Depth of keyway in hub

To assign a bore hole to a roll, click with the mouse cursor into the input field. Select the desired bore hole file within the file select window.



If you want to create new bore hole files, press **Edit** in the [Expanded Roll Tool Window](#) and select "File, Save as..".

The data in the file currently are not yet utilized. The bore hole (file name) can be used as a column of the [Roll Parts List](#). Furthermore the name is displayed in the [Roll Tool Drawing](#) within the bore hole.

#### 3.3.6.4 Bushing

Bushing belongs to the roll data and is displayed in the [Expanded Roll Tool Window](#).

Bushing is the name of a file that e.g. contains these data:

@BUSHFILE	# Bushing
80.000	# Outer Diameter
H7	# Fit Roll
f7	# Fit Bushing

To assign a bushing to a roll, click with the mouse cursor into the input field. Select the desired bushing file within the file select window. Bushing is often used for idle rolls, this means the roll is not driven. This is why the mark [Driven](#) is removed when you select a bushing.

If you want to create new bushing files, press **Edit** in the [Expanded Roll Tool Window](#) and select "File, Save as..".

Bushing are separate entries in the [Roll Parts List](#), if this is set in [Options Parts List](#). Furthermore belonging bushing can be displayed in a separate column of the rolls.

#### 3.3.6.5 Identification Groove

Identification groove belongs to the roll data and is displayed in the [Expanded Roll Tool Window](#).

Identification groove is the name of a file that e.g. contains these data:

@GROOVEFILE	# Identification Groove
80.000	# Diameter
8.000	# Groove With
2.000	# Groove Depth
80.000	# Shoulder Angle

To assign an identification groove to a roll, click with the mouse cursor into the input field. Select the desired identification groove file within the file select window.

If you want to create new identification groove files, press **Edit** in the [Expanded Roll Tool Window](#) and select "File, Save as..".

The data in the file currently are not yet utilized. The identification groove (file name) can be used as a column of the [Roll Parts List](#).

#### 3.3.6.6 Material

Material belongs to the roll data and is displayed in the [Expanded Roll Tool Window](#).

Material is the name of an [Additions File](#), which is used to calculate the blank size from the finished size of a roll.

To assign a material to a roll, click with the mouse cursor into the input field. Select the desired additions file within the file select window.

If you want to create new additions files, press **Edit** in the [Expanded Roll Tool Window](#) and select

"File, Save as..".

If you called the [Expanded Roll Tool Window](#) from [Options, Spacer Rolls](#) (button **More**), is the input field **Material** inactive. Enter the material in then [Machine Window](#), [Spacers](#) instead.

The material (file name) can be used as a column of the [Roll Parts List](#).

### 3.3.6.7 Treating, Surface, Addition, Remark

Treating, Surface, Addition, Remark are additional data, the field names of which can be defined by the user in [Options Rolls](#), Button **More**. They belong to the roll data and are displayed in the [Expanded Roll Tool Window](#).

PROFIL remembers new entries into these data fields, later you can recall them from a drop-down-box by mouse click on the arrow symbol. If you want to remove an entry from the drop-down-box, recall the entry and press **Del** on the keyboard.

### 3.3.6.8 Touching the Profile at

This switch belongs to the roll data and is displayed in the [Expanded Roll Tool Window](#).

The information is needed when you call the FEA simulation by using the function [Output FEA](#) and is used for optimizing the computation time of the simulation.

**Top/Bottom Side:** If this switch is set, the FEA system checks the contact between the roll and the top/bottom side of the strip.

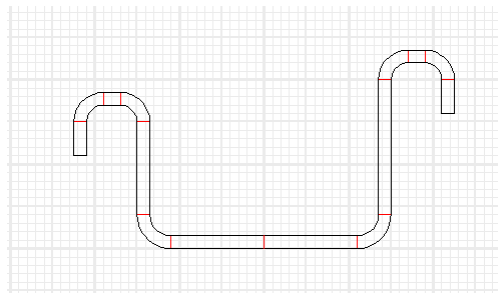
**Edge:** With this information the FEA system checks the contact with the edge of the strip (sheet thickness).

If there is contact at the top/bottom side AND the edge, both switches can be set in principle. To optimize the computing time it is a good idea to split the roll in that way, that each partial roll has either contact at top/bottom side OR at edge.

If **General Contact** is set in [Output FEA](#), **Touching the Profile at** is ignored.

## 3.3.7 Drawing Area

The drawing area in the background shows permanently the drawing of the profile, the flower pattern or the roll tools. Use the [Navigator](#) in the upper right corner to select the view port.



In order to have a better general overview of the proportions of the objects you can attach [Grid Lines](#) to the drawing background.

Select, what you want to see:

[View Pass](#)

[View Flower Nested](#)

[View Flower 3D](#)

[View Statics](#)

[View Flower Separated](#)

[View Roll Tools](#)

View PSA - Profile Stress Analysis

[View FEA Result](#)

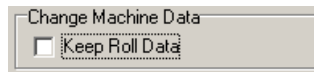
If you select **Flower**, only the active pass is shown in its defined colors. To activate another pass:

- Select the pass in the [Profile Explorer](#);
- Activate the [Profile list Window](#) (if opened);
- Click into the drawing and identify a line element (not arc element!) of the desired pass.

Inside an active pass, you can identify line and arc elements; the color changes. The data of the selected profile element are displayed in the bottom status bar. Simultaneously the corresponding [Profile Element](#) in the [Profile List Window](#) will be activated, if the window is open.

Inside the drawing of the roll tools, you can identify rolls and roll corner points. The data of the selected roll corner point are displayed in the bottom status bar. Simultaneously the corresponding [Roll Corner Point](#) in the [Roll Tool Window](#) will be activated, if the window is open.

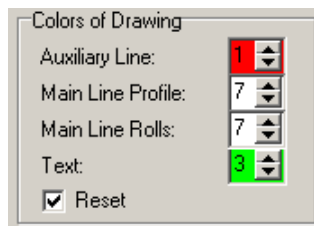
A right mouse button click on a drawing element opens the context menu **Profile Design** or **Roll Design**, dependent on the kind of drawing. Using this feature speeds up the design significantly. By using the context menu **Machine Parameters** (right mouse click on a shaft or roll centerline) the working diameter, reference point, and the inclination angle can be modified.



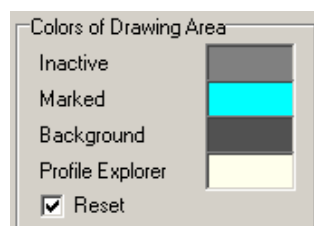
In [Options Rolls](#), **Keep Roll Data** select if the roll properties should be retained or if they should be adapted by retaining the roll contour only.

Use the function [View, Show, Dimensions](#) to switch all dimensions temporarily off and on again. Use the function [Inspect](#) to display the whole drawing in its defined colors. A right mouse button click to the background causes this, too.

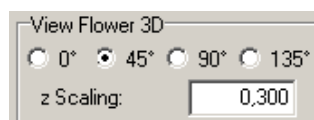
If you want to expand the drawing area, use the function [Window Visible](#) to switch off the windows.



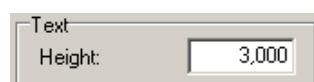
Select the drawing entity colors in [Options Drawing](#), **Colors of Drawing**.



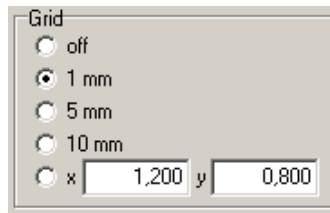
Select the drawing area colors in [Options Drawing](#), **Colors of Drawing Area**.



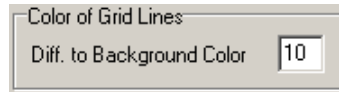
In case of a 3D drawing, select the 3D view port angle and the z scaling in [Options Drawing](#), **View Flower 3D**.



Select the text height in [Options Drawing](#), **Text**.



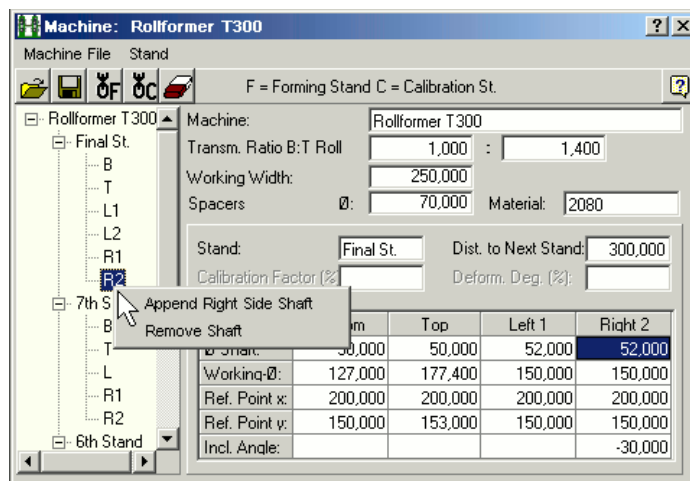
Select in [Options Drawing](#), **Grid** the grid lines distance and if the grid lines should be visible on the screen or not.




Select in [Options Drawing](#), how much the **color of the grid lines** should differ from the **background color**.

Use the function [Drawing CAD](#) to transfer the drawing, which is displayed in the [Drawing Area](#), to your [CAD System](#).

### 3.3.8 Machine



The machine window shows the machine data of the current [Project](#). It is opened (and closed again) by the menu function [Edit Machine](#) or with the button  in the top [Button Key Bar](#).

The machine window works together with the [Drawing Area](#) and the [Profile Explorer](#) fully interactively. This means, after clicking on an item the corresponding objects in the other two partitions are activated.

The left part of the window contains the [Machine Explorer](#).

The top part of the window contains the following data:

[Machine](#)      [Transmission Ratio](#)  
[Working Width](#)      [Spacers](#)

The bottom part of the window contains the following data:

[Stand Name](#)      [Distance to Next Stand](#)

(for calibrating stands only:)

[Calibrating Factor](#)      [Deformation Degree](#)

and the table with the data of the shafts/axes:

[Diameter Shaft](#)    [Working Diameter](#)  
[Reference Point](#)   [Inclination Angle](#)

The data of the shafts/axles for each stand are available separately. By clicking on an item in the [Machine Explorer](#) the corresponding stand data are selected. First, the standard shaft/axle types **B** (Bottom), **T** (Top), **L** (Left) and **R** (Right) are shown. If a stand has additional side axles, you see the identifier e.g. **L1**, **L2** and so on. By clicking on this identifier the corresponding column of the table of shaft/axle data changes to e.g. **Left 2**. and at the same time the corresponding axle in the [Drawing Area](#) is activated. Thus interactive working is possible.

After **right-click** on a **stand symbol** of the [Machine Explorer](#) the context menu opens and forming stands and calibrating stands can be attached and removed again. By **right-click** on a **side axle symbol** further side axles can be attached and removed again.

By calling the function **Append Forming Stand** or **Append Calibrating Stand** the shaft data are copied. So it is recommended to fill out the shaft data previously.

By appending side axles the data of the previous axle are copied. The inclination angle, however, is modified in order to have the axle visible in the drawing area and selectable by mouse click. Afterwards the user should set the desired inclination angle.

The drop-down menu in the top and the buttons of the button bar can be used for::



**Import a [Machine File](#).** When you start with a new [Project](#) that does not contain machine data, you can import machine data from a machine file \*.m01 that you created with the function **Export** from another project previously.



**Export a [Machine File](#).** The machine data from the current project are saved into a file \*.m01



**Append Forming Stand:** Behind the current stand (marked in the [Machine Explorer](#)) a new forming stand is created.

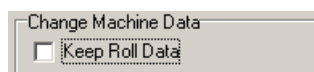


**Append Calibrating Stand:** Behind the current calibrating stand (marked in the [Machine Explorer](#)) a new calibrating stand is created..



**Remove stand.**

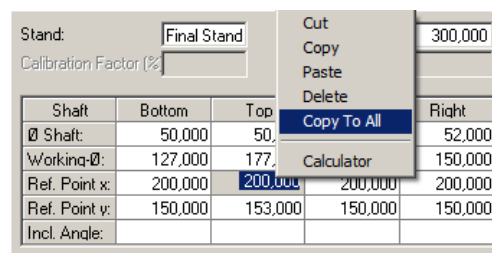
Pay attention to the sequence opposite the sheet running direction: first calibrating stands, afterwards forming stands. F01 always is the last forming stand in sheet running direction (the finishing stand), in case of tubes the welding station. C01 is the last calibrating stand (the finishing stand for shaped tubes).



In case the machine data are changed and rolls already exist in the project, the setting in [Options Rolls](#), **Change Machine Data** will be effective and decides how the rolls should be handled.

All values are shown in mm, if you have selected in [Options Profile List](#) the representation of values **Metric**. If you have selected **Imperial**, they are shown in inch.

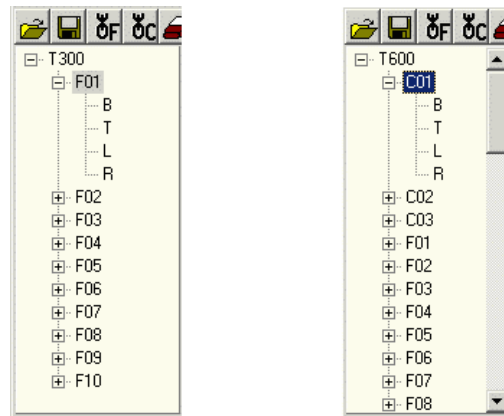
### Context Menu:



In certain input fields further functions are available from the context menu (right mouse button):

- **Cut, Copy, Paste, Delete:** Enables transfer of parameters via the clipboard.
- **Copy To All:** Copies the selected parameter to the corresponding input field in all other stands. If this function is called in the input field [Working Diameter](#) of the first stand, a window opens and asks **Increase working diameter per stand by?** The diameters of all previous stands are decreased by the given value, thus the diameter increase in sheet running direction. This prevents vertical buckling by lead (strip tension). Also the spring back of the profile legs is reduced between the stands.
- **Calculator:** Opens the [Calculator](#) and copies the selected parameter to it's display. After closing the calculator the result is transferred back to the input field. Press the **Enter** or **Tab** key afterwards.

### 3.3.8.1 Machine Explorer



Machine explorer of a typical roll forming machine (left) and tube forming machine (right)

The machine explorer displayed in the left part of the [Machine Window](#).

The entities in the table of stands are

- F = Forming Stand, for open profiles and welded tubes
- C = Calibrating stand, only for welded tubes

The machine explorer is just as the [Profile Explorer](#) organized against the sheet running direction. This means on the top is the last stand for the final profile.

A pure roll forming machine (for open sections, see left picture) has only forming stands (e.g. F01..F16). A tube forming machine (for [Welded Round Tubes](#) or [Shaped Tubes](#), see right picture) first (against the sheet running direction) has calibrating stands (e.g. C01, C02, C03). Behind C03 comes the welding station (F01), then the fin passes (e.g. F02, F03, F04), then the forming stands or break-down passes (e.g. F05..F10).

If you select a stand by mouse-click, the table on the right shows the belonging data:

[Stand Name](#)      [Distance to Next Stand](#)

(for calibrating stands only:)

[Calibrating Factor](#)   [Deformation Degree](#)

and the table with the data of the shafts/axles:

[Diameter Shaft](#)      [Working Diameter](#)  
[Reference Point](#)    [Inclination Angle](#)

If the stand has additional side axles, you can see the data after mouse-click on the additional axles symbol (e.G. **L2**).

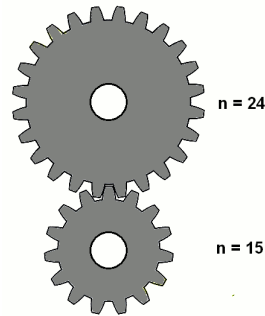
The stand name first is F01..Fnn for forming stands and C01..Cnn for calibrating stands. In the field [Stand Name](#) you can modify the name. The new name is shown in the machine explorer.

### 3.3.8.2 Machine

The machine name belongs to machine data and is displayed in the top part of the [Machine Window](#).

Enter a specification that characterises your machine or the location of your machine. The name will also be shown in the [Machine Explorer](#), in the header bar of the [Machine Window](#), and in the field [Machine](#) of the [Project Data Window](#)..

### 3.3.8.3 Transmission Ratio



Example of a gear with transmission ratio  
15 teeth : 24 teeth = 1 : 1.6

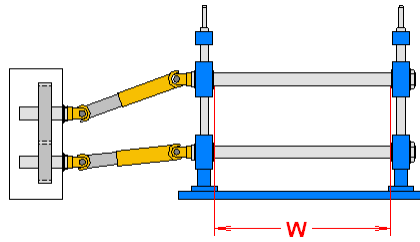
The transmission ratio (also called gear transmission ratio) belongs to machine data and is displayed in the top part of the [Machine Window](#).

Most roll forming machines have the gear transmission ratio 1 : 1.4. In order to have the same circumferential speed (peripheral velocity) of the top and bottom roll at the profile web, the working diameter (pitch diameter) of the top roll must be the bottom roll diameter multiplied by 1.4. This enables a non-slip traction of the sheet and better drawing the profile through the machine. Why have roll forming machines this transmission ratio? The reason is: Normally, profiles are guided through the machine with the open side on the top, this means the band edges are bent upwards by the rolls. This makes it easier to set-up the machine, because the operator can better look into the profile during jog mode (Below the profile, the machine base impedes the view, it would be necessary to work with a mirror) . If the top roll has a larger working diameter, profiles with higher legs can be processed without risk that the band edges touch the top shaft spacers. Below the profile there is no need to consider this. This is why material can be saved and a smaller roll diameter is sufficient. Unequal working diameter require unequal rotation speed of the shafts and with it a transmission ratio of e.g. 1 : 1.4. However, some roll forming machines are using the transmission ratio 1 : 1, mostly if small profiles with small legs should be processed.

Enter dividend and divisor of the transmission ratio between the bottom and the top roll. You need this information for defining the [Working Diameter](#) of the driven shafts, if you want to get equal velocities of the bottom and the top roll.

When the working diameter of a shaft is set, the working diameter of the counterpart shaft can be calculated dependent on the transmission ratio, see [Working Diameter](#).

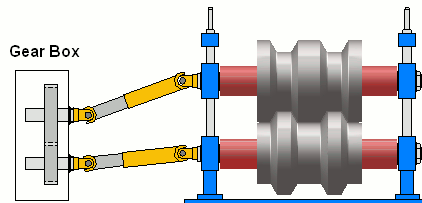
### 3.3.8.4 Working Width



The working width **W** belongs to the machine data and is displayed in the top part of the [Machine Window](#).

The working width of the machine is the maximal available roll space. If you do not utilize the maximal space left and right of the bottom and top forming rolls, you must use [Spacers](#) for fixing the position of the forming rolls. Between two alternatives can be selected: **Automatic Spacers** and **Spacer Rolls**, see [Tutorial, Roll Tools, Creating Spacer Rolls](#).

### 3.3.8.5 Spacers



**Spacer Diameter** and **Spacer Material** belong to the machine data and are displayed in the top part of the [Machine Window](#).

Spacers (red, see picture) fix the horizontal position of the rolls on the shaft. Between two alternatives can be selected: **Automatic Spacers** and **Spacer Rolls**, see [Tutorial, Roll Tools, Creating Spacer Rolls](#).

The [Parts List](#) is able to show spacers by final diameter, blank diameter and material (see [Parts List Columns](#)).

**Material** is the name of an [Additions File](#), which is used to calculate the blank size from the finished size of a roll. To assign a material to a roll, click with the mouse cursor into the input field. Select the desired additions file within the file select window.

If you want to create new additions files, press **Edit** in the [Expanded Roll Tool Window](#) and select "File, Save as..".

The material (file name) can be used as a column of the [Roll Parts List](#).

### 3.3.8.6 Stand Name

The stand name belongs to machine data and is displayed in the bottom part of the [Machine Window](#).

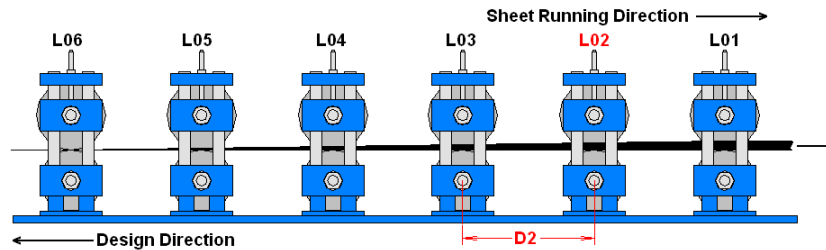
The stand name first is F01..Fnn for forming stands and C01..Cnn for calibrating stands. You can modify the name for your own needs. The modified name is displayed in the [Machine Explorer](#) afterwards.

The stand name can also be displayed in the [Profile Explorer](#), if in [Edit, Explorer Show stand](#)



from machine data is selected.

### 3.3.8.7 Distance to Next Stand



The distance between stands is the horizontal distance between the current and the previous stand in sheet running direction (center roll to center roll). Example: The distance between stands **D2** in profile list **L02** is the distance between **L02** and **L03**. For the first stand enter the length of the spheroidizing zone. Please estimate the value.

The distance to next stand belongs to the machine data and is displayed in the bottom part of the [Machine Window](#).

#### Principle of operation

If you create a new profile list, the distance between stands is taken from the [Machine Window](#) and copied into the [Profile List](#) data. The distance between stands is used by the calculation of the [Stress of Edge](#).

### 3.3.8.8 Calibrating Factor

The calibrating factor belongs to machine data and is displayed in the bottom part of the [Machine Window](#). The input field is active only in case of calibrating stands. The calibrating factor has effect on the developed length stands during the function [Shaped Tube Calibration](#).

The calibration factor says how much % the developed length of a tube decreases when the rolls of the calibrating stand mesh in. Read the documentation of your machine for the determination of the calibrating factor.

### 3.3.8.9 Deformation Degree

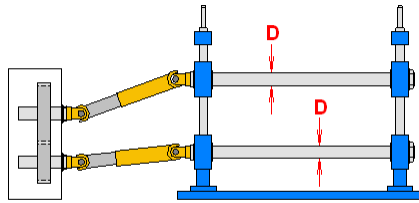
The calibrating factor belongs to machine data and is displayed in the bottom part of the [Machine Window](#). The input field is active only in case of calibrating stands. Use the deformation degree to have an influence on the shapes of the calibrating stands during the function [Shaped Tube Calibration](#).

Partition the 100% forming between the round tube in the welding station and the final product "Shaped Tube" to all calibrating stands in a suitable manner. If the sum of the deformation degrees of all calibrating stands is not 100%, PROFIL adapts them linearly. To get a final product with small allowances, select a smaller deformation degree for the last calibrating stand (C01) than for the others.

If the final product is a round tube, the deformation degree is unconsidered. Use the [Calibrating Factor](#) for decreasing the diameter of the tube.

If there are round tube calibrating stands behind the welding station (i.e. calibrating to a round tube with a smaller diameter), and behind them shaped tube calibrating stands, enter a **Calibrating Factor**, but no **Deformation Degree** in the round tube calibrating stands. As a result of it the tube keeps round during the function [Shaped Tube Calibration](#).

### 3.3.8.10 Diameter Shaft



The diameter of the shafts **D** belongs to machine data and is displayed in the bottom part of the [Machine Window](#) for each shaft.

Read the documentation of your machine for the determination of the diameter of the shafts.

#### Hint:

When a roll is created, the **Diameter Shaft** is taken from the machine data and copied to the roll data. It is shown in the input field [Diameter Shaft](#) in the [Expanded Roll Tool Window](#) and can be modified in case of e.g. the roll is a not driven idle roll on a [Bushing](#).

### 3.3.8.11 Working Diameter

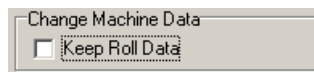
The working diameter (pitch diameter) belongs to machine data and is displayed in the bottom part of the [Machine Window](#) for each shaft.

The working diameter is the double of the distance between the [Reference Point](#) and the center line of the shaft. Pay attention that it can be a real diameter, i.e. a diameter that can be measured at the roll. It is also possible that it is a fictitious diameter, which cannot be measured. For a more detailed description see chapter [Reference Point](#).


If you define the working diameters, pay attention to the [Transmission Ratio](#) of the machine (equal velocity at the important points of the profile) and if the shafts are adjustable. Read the documentation of the machine for the optimum value.

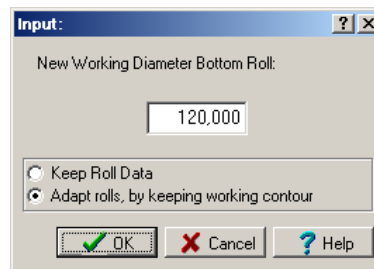
Experienced designers increase the working diameter of top and bottom roll by a small value of e.g. 1mm in each stand in sheet running direction. As a result the profile is drawn through the machine and the spring back and longitudinal strain behind each stand is decreased. Though the profile web climbs up in the machine (opposite of [Center Line Forming](#)) if the bottom shafts are not adjustable.

#### Calling the function

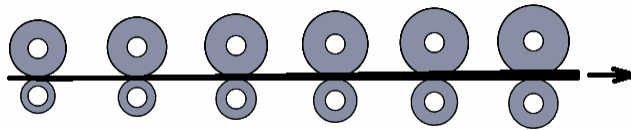


When you are going to modify the working diameter, select in [Options, Rolls, Changing Machine Data](#), **Keep Roll Data** if the rolls or if the roll contour only should keep unchanged during this operation. Modify the reference point optionally by:

-  [Machine Window](#): **Working-Ø**. Enter the new working diameter for the selected stand and shaft type.
- Context menu (right mouse button click on a roll or shaft dash dot centerline in the [Drawing Area](#)): **Modify working diameter**, if you want to enter a new working diameter.
- Context menu (right mouse button click on a roll or shaft dash dot centerline in the [Drawing Area](#)): **Calculate working diameter from transm. ratio**, if the new diameter should be calculated from the [Transmission Ratio](#) and the working diameter of the opposite roll.

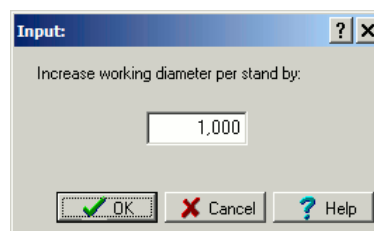


After calling the function the input window appears with the question **New Working Diameter?** In case of **Modify working diameter** the topic working diameter is shown and can be modified. In case of **Calculate working diameter from transm. ratio** the result of the calculation is shown and can be confirmed or modified. Furthermore the topic setting of the switch **Changing Machine Data, Keep Roll Data** in [Options, Rolls](#) is shown and can be modified.



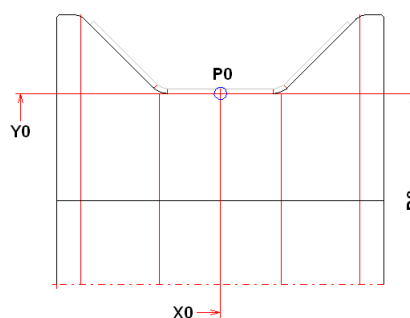
#### Lead (Strip tension):

It's a common practice to keep the strip under tension by gradually increasing the roll pitch diameters. Thus also the circumferential speed of the rolls is increased from stand to stand. This causes strip tension in sheet running direction and prevents vertical buckling. Also the spring back of the profile legs is reduced between the stands. In the [Machine Window](#), enter the desired working diameter and call the function **Copy To All** of the context menu (right mouse button).



A window opens and asks **Increase working diameter per stand by?** The diameters of all previous stands are decreased by the given value, thus the diameter increase in sheet running direction.

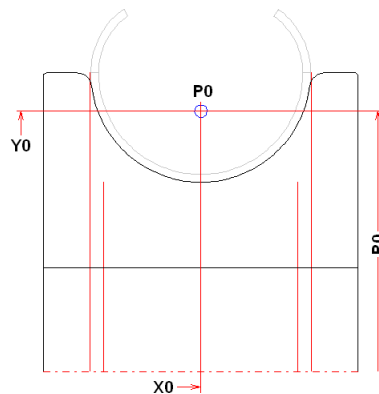
#### 3.3.8.12 Reference Point



Pic. 1: The roll reference point is identical to the profile reference

The roll reference point **P0** with the drawing coordinates **X0** and **Y0** is a part of the machine data and is displayed for each shaft in the bottom part of the [Machine Window](#). The [Working Diameter](#) (pitch diameter) **D0** (=2x working radius **R0**) is related to this reference point. The reference point of the machine is the connection to the profile and is measured in the drawing's coordinates. Mostly it is identical to the [Reference Point X0/Y0](#) of the profile list (see pic. 1). In this case the

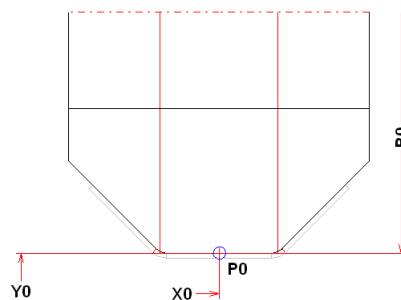
reference point is on the roll contour and the working diameter can be measured at the roll.



Pic. 2: The roll reference point is in the centroid of the profile

For special applications, the roll reference point **P0** can be set to any point, e.g. to the centroid of an area of the profile cross-section as shown in pic. 2 for a round tube. Also in this case the [Working Diameter D0](#) (=2x working radius **R0**) is related to this reference point. Pay attention that the working diameter cannot be measured at the roll.

This principle also is valid for top and side rolls. For each shaft type a separate column exist in the [Machine Window](#) with reference point coordinates and working diameter.



Pic. 3: The roll reference point is at the top side of the sheet

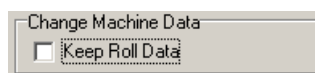
For top rolls it makes sense to set the roll reference point **P0** to the top side of the profile, i.e. to the point that is moved upwards from the [Reference Point X0/Y0](#) by the sheet thickness (see pic. 3). Thus the [Working Diameter D0](#) (=2x working radius **R0**) can be measured at the roll. But the disadvantage is that the machine data have to be adapted to varying sheet thickness. This can be avoided by proceeding in the following way:

Shaft	Bottom	Top	Left	Right
Ø Shaft:	50,000	50,000	52,000	52,000
Working-Ø:	126,000	176,400	220,000	220,000
Ref. Point x:	200,000		200,000	200,000
Ref. Point y:	150,000		150,000	150,000
Incl. Angle:				


Pic. 4: Empty input fields x/y cause moving the reference point by the sheet thickness

If in the [Machine Window](#) both input fields (x and y) for the top rolls are empty respectively 0, the reference point for the top rolls is moved upwards automatically by the sheet thickness from the reference point of the bottom rolls. This way can be used to create top rolls with a desired diameter independently from the sheet thickness.

### Calling the function



When you are going to modify the reference point, select in [Options, Rolls, Changing Machine Data, Keep Roll Data](#) if the rolls or if the roll contour only should keep unchanged during this operation. Modify the reference point optionally by:

-  [Machine Window](#): **Ref. Point x, Ref. Point y**. Enter the absolute drawing coordinates.
- Context menu (right mouse button click on a roll or shaft dash dot centerline in the [Drawing Area](#)): **Modify Reference Point**. Select **axial and radial**, **axial only**, or **radial only** and choose the new reference point by clicking onto a drawing element. The nearest end point is caught.

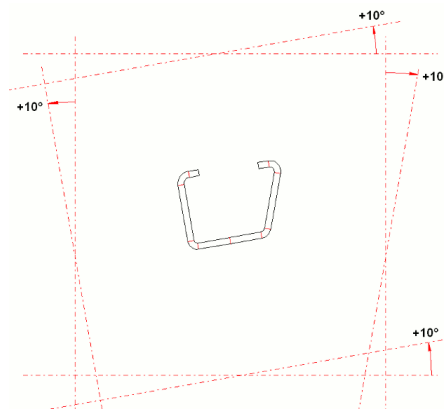
#### Principle of operation

- **Axial and radial**: The reference point is set to the nearest end of the selected drawing element.
- **Axial only**: The reference point is moved axially (i.e. in the direction of the roll centerline) without changing the radial component.
- **Radial only**: The reference point is moved radially (i.e. in the direction of the roll radius) without changing the axial component.

#### Hint:

- The roll reference point can be set independently from the [Profile Reference Point](#). Usually, both reference points are set to the same coordinate.

### 3.3.8.13 Inclination Angle

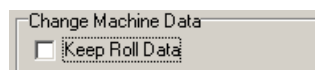


The inclination angle belongs to machine data and is displayed in the bottom part of the [Machine Window](#) for each shaft.


**Bottom/top rolls:** The normal shaft position is horizontal (inclination angle 0). In special cases, e.g. if subsequent processing of the profile behind the exit of the roll forming machine needs a certain angular positioning, it is not possible to turn the profile to the optimal angle required for roll forming purposes. Thus it is necessary to turn the machine shafts in the optimal angular position. A positive inclination angle rotates the shaft counterclockwise, a negative clockwise (see picture).

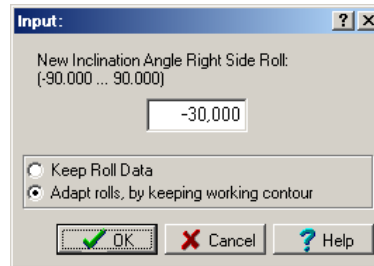
**Side rolls:** The normal axle position is vertical (inclination angle 0). A positive inclination angle rotates the shaft to the outside, an negative to the inside (see picture).

#### Calling the function



When you are going to modify the inclination angle, select in [Options, Rolls, Changing Machine Data, Keep Roll Data](#) if the rolls or if the roll contour only should keep unchanged during this operation. Select stand shaft/axle. Modify the inclination angle optionally by:

-  [Machine Window](#): **Incl. Angle**. Enter the new inclination angle either positive or negative (see picture).
- Context menu (right mouse button click on a roll or shaft dash dot centerline in the [Drawing Area](#)): **Modify inclination angle**.



After calling the function the input window appears with the question **New inclination angle?** and an admissible range of values for the input. Furthermore the topic setting of the switch **Changing Machine Data, Keep Roll Data** in [Options, Rolls](#) is shown and can be modified.

### 3.3.9 Read CAD-Contour/Scan Profile Drawing

The **Window Read CAD-Contour** appears, if you have called one of the functions [Profile, Read CAD-Contour](#) or [Roll, Read CAD-Contour](#) or [Roll, Read CAD Roll](#) or **Insert Roll from CAD** of the [Roll Table](#).

The **Window Scan Profile Drawing** has the same appearance and appears, if you have called the function [Roll Scan Profile Drawing](#) for creating a roll automatically from the profile drawing in the [Drawing Area](#).

In the first case (creating profile list or roll by reading the contour from CAD) the whole drawing is read from CAD and displayed in the window, this can take some seconds. The progress bar at the bottom shows the status.

In the second case (creating a roll by scanning the profile drawing shown on the PROFIL drawing area) a hidden lines algorithm is applied to the profile drawing and all non-visible contours are displayed in **inactive color** (by default gray). Afterwards the contour tracking reads the visible contour only (from the selected shaft).

If there are too many entities in the drawing and the contour tracking does not give the desired result, select one special layer and/or one special color in the input fields **Select Layer** and **Select Color**. If the contour tracking is not correct yet, delete the entities within CAD that you do not need and call again the window **Read CAD-Contour**.

The [Navigator](#) helps you to zoom, to move and to fit the drawing. Furthermore the Wheel Mouse is supported.

The read drawing contains the markers **Contour Start Point** (green) and **Contour End Point** (red) and - if you have called **Profile List Read CAD-Contour** - the additional marker **Reference Point x0/y0** (blue). Set the markers to the correct points of the contour by using the buttons in the button bar. The entities' color changes to the mark color (by default light blue) and shows which contour is taken by pressing the **Ok** Button. If the automatic contour tracking does not find the desired contour: see topic **Controlling the contour tracking manually**.



**Contour Start Point**

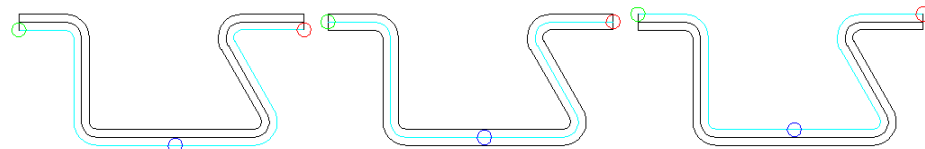


**Reference Point x0/y0**



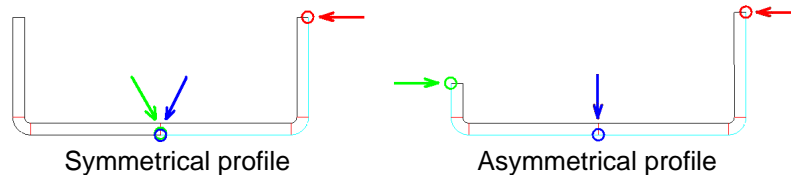
**Contour End Point**

**Designing a profile by using the CAD-system:**



Scanning the profile bottom side, the sheet center line, and the profile top side

Optionally, scanning the profile bottom side, the sheet center line, or the profile top side (bottom and top related to the reference point) is possible. In case profile legs touch each other and have common lines, it can be valuable to scan the opposite side.



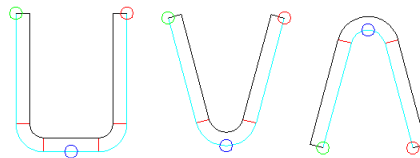
Symmetrical profile

Asymmetrical profile

(You have called this window with the function [Profile, Read CAD-Contour](#)) If you design a symmetrical profile (left picture), set both the marker **Reference Point x0/y0** (blue) and the marker **Contour Start Point** (green) to the [Reference Point X0/Y0](#) of the profile. If you design an asymmetrical profile (right picture), set the marker **Contour Start Point** (green) to the left corner of the profile and the marker **Reference Point x0/y0** (blue) to the [Reference Point X0/Y0](#) of the profile. For both set the marker **Contour End Point** (red) to the right corner of the profile. The color change to the mark color (light blue by default) must mark the lower contour line (in the reference point) of the profile. Otherwise: see topic **Controlling the contour tracking manually**. In case of a symmetrical profile you only need to append a [Profile Element PS](#) (for Point Symmetrical) to the end of the profile list. This mirrors the right side to the left.

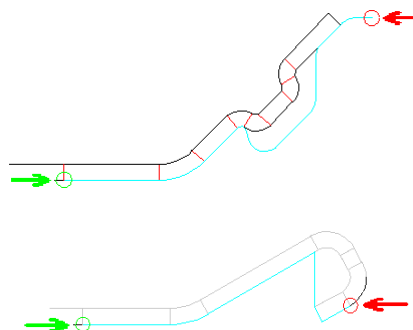
#### Snap points for setting the profile reference point:

By clicking on a drawing element (line or arc), the desired point (contour start point, reference point, or contour end point) will be set to the nearest end point of the drawing element.



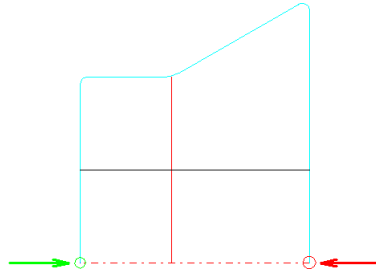
In case of **Reference Point** you can use further snap points by opening the context menu (right mouse button click), if the imported CAD contour has no element breakup at the desired position: **Line Center Point** (left), **Arc Quad Point 270°** (center), and **Arc Quad Point 90°** (right). The picture shows the typical applications. If the reference point should be set to another point that does not exist in the drawing, it is recommended to proceed like this: Split the element in CAD at the desired position. Then also the window "Read CAD Contour" contains split elements and the reference point can be set by direct mouse click.

#### Designing a roll set by using the CAD-system or by scanning the profile drawing:



(You have called this window with the function [Roll Read CAD-Contour](#) or [Roll Scan Profile Drawing](#)) Set the marker **Contour Start Point** (green) to the left corner of the roll and the marker **Contour End Point** (red) to the right corner of the roll. The color change to the mark color (light blue by default) must mark the roll contour without the edges (see picture). Rounding radii can be attached later in the roll list. Otherwise: see topic **Controlling the contour tracking manually**.

**Inserting a roll from CAD into the project or into the roll database:**



(You have called this window with the function [Roll, Read CAD Roll](#) or [Insert Roll from CAD](#) of the [Roll Table](#)) Set the marker **Contour Start Point** (green) to intersection point of the left edge with the centerline and the marker **Contour End Point** (red) to the intersection point of the right corner with the centerline of the roll. The color change to the mark color (light blue by default) must mark the outer contour of the roll. Otherwise: see topic **Controlling the contour tracking manually**.

#### Controlling the contour tracking manually:

If the automatic contour tracking does not find the desired contour, you can control it manually by selecting certain drawing elements with the mouse cursor.

If the blue point exists (reference point x0/y0 in case of defining the profile list by reading CAD contour), the start element from the blue point directing to the red point (contour end point) can be defined by mouse click. The direction to the green point (contour start point) always is opposite (turned by 180 degree).

If the blue point does not exist (in case of defining a roll) , the start element from the green point directing to the red point can be defined by mouse click.

If the automatic contour tracking does not find the desired contour at any intersection, you can define the correct connection by mouse click. If a wrong connection was given in error, you can use the **Reset** button to start newly.

After pressing the **Ok** Button the tracked contour is taken as a profile contour or a roll contour.

## 3.4 Files

### 3.4.1 Profile Project

The profile project contains the project data, all [Profile Lists](#) and the [Roll Tool Data](#) for all stands.

The profile project is saved in the project file, which has a user defined name with the extension **.pro**. In this way the project file contains all data belonging to a profile design and can be archived and passed on to other users of PROFIL.

The project data are:

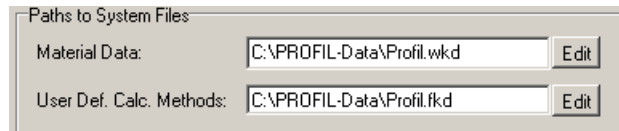
<a href="#">Customer</a>	<a href="#">Date</a>
<a href="#">Description</a>	<a href="#">Name</a>
<a href="#">Drawing No.</a>	<a href="#">Revision</a>
<a href="#">Material</a>	<a href="#">Thickness</a>
<a href="#">Machine</a>	<a href="#">Calculation Methods</a>



If needed, the profile project name can be used as variable **\$PR** for the automatic roll numbering, see [Options Rolls](#).

### 3.4.2 Material File

The material file contains the data of different sheet materials, which are necessary for the calculation of [Spring Back](#) and [Stress of Edge](#). The material file has the name **PROFIL.WKD**, if you have selected in [Options Profile List](#) the representation of values **Metric**. If you have selected **Imperial**, the material file name is PROFIL.WK1.



Select the desired material file by using the function [Options Calculate](#), **Material Data** and open it with the **Edit** button. The columns of the material file are:

**No:** Identification number of the material.

**Material:** Name of the material.

**K1, K10:** Factors of spring back for inner radius = thickness of sheet respectively 10\* thickness of sheet.

**c0.5, c1, c2, c3, c6:** Critical minimum bending factors for the thickness of sheet 0.5 ... 6 mm, which are the base for calculating the minimum bending radius =  $cn \cdot \text{thickness of sheet}$ . In the Imperial System the critical minimum bending factors are called c0.02, c0.04, c0.08, c0.12, c0.24 and are valid for the thickness 0.02 .. 0.24 in.

**Re:** Stress at the yield point in N/mm<sup>2</sup>. This size is used for calculating the relative stress of the band edge corresponding to the yield point from the strain of the band edge. In the Imperial System the yield point has the unit psi (pounds per square inch, lb/in<sup>2</sup>).

**E:** Young's modulus in 1000 N/mm<sup>2</sup>. It is used for calculating the stress of the band edge from the strain of the band edge. In the Imperial System the Young's modulus has the unit kips (kilo pounds per square inch, 1000 lb/in<sup>2</sup>).

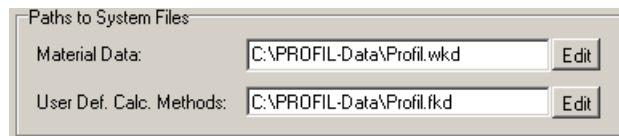
**rho:** Density of the material in kg/dm<sup>3</sup>. In the Imperial System the density has the unit pci (pounds per cubic inch, lb/in<sup>3</sup>).

View the material file with the function [Options Calculate](#), **Edit** and select the desired material. The same function can be used to enhance the file with new materials. Be sure to use ascending numbering.

If the material file does not contain your desired material, you can enhance the material file by new materials. You need to find out the material data by experiment and by material catalogue. Or use a similar material from the file.

### 3.4.3 Factor File

In the factor file PROFIL.FKD you can define your own calculation methods for the developed length. Decide first, if you want to use a [Factor Method](#) or a [Addition Method](#).



Select the desired factor file by using the function [Options Calculate](#), **User Def. Calc. Methods** and open it with the **Edit** button.

The file has one or more lines with a leading **N**, followed by the name of the method. The letter **#** indicates comment lines.

The line with leading **N** contains the name of the method with max. 9 letters, this name is displayed inside the [Project Data Window](#).

For the **Factor method** there are now some lines with a leading **F** and the contents:

- |           |  |
|-----------|--|
| Column 1: | <b>F</b> for the Factor Method;  |
| Column 2: | Limit of the relation inner radius/sheet thickness, for the factor of the position of the neutral line, the correction addend and the correction factor; |
| Column 3: | Factor for the position of the neutral line;   |
| Column 4: | Correction factor in % (pos. or negative);   |
| Column 5: | Correction addend in mm (pos. or negative).  |

The lines have to be sorted in ascending order and the last F-line should have a large value in column 2 (for example 999) to ensure proper calculations even for unusual proportions.

For the **Addition method** follow some lines with a leading **Z**:

- |           |  |
|-----------|--|
| Column 1: | <b>Z</b> for the Addition Method                               |
| Column 2: | Limit of the sheet thickness for correction factor and addend; |
| Column 3: | Limit of the bending angle for correction factor and addend;   |
| Column 4: | Limit of inner radius for correction factor and addend;        |
| Column 4: | Correction factor in % (pos. or negative);                     |
| Column 5: | Correction addend in mm (pos. or negative).                    |

The lines have to be sorted in ascending order first by column 2, then by column 3 and then by column 4. The last F-line should have a large value in column 2 (for example 999) to ensure proper calculations even for unusual proportions.

The **E**-line marks the end of the file.

To define own methods modify the file, which contains only examples. You can modify the examples or append new methods.

### 3.4.4 Machine File

The machine file is useful for exchanging machine data (which are shown in the [Machine Window](#)) between projects. After finishing a project a set of machine data have accumulated (which are saved with the [Project](#) in the project file). Now you can export the machine data into a machine file in order to re-use them and import them in another new project. For this, use the functions [File](#), [Export](#) and [File, Import](#) or the export and import functions in the [Machine Window](#).

The machine file is a file with the extension **.M01**. Select a short name of machine for file name to

allow easy identification.

### 3.4.5 Contour File (KTR Format)

Use the contour file to transfer the geometry of a profile or a roll tool drawn in the [CAD System](#) to PROFIL.

The KTR-format is a file format defined by UBECO. For these CAD systems macros are available, which generate a contour file: PC-DRAFT, AutoCAD, CADD.A.. Please contact your dealer.

To generate a contour file, proceed the following steps:

- Draw the contour in the CAD system. Select a [Reference Point](#) (this is mostly the center point of the lower line of the web of the profile) and draw the lower line of the profile, beginning in the reference point.
- In the CAD system, call the macro which generates the contour file.
- In PROFIL, select the [Profile Element](#) in the [Profile Lists Window](#) for the start of the contour. The start point normally is element 1, except you read the 2nd half of an asymmetrical profile, then the start point is the next element behind element **P**. If you have selected the first element, the reference point x0/y0 and the direction are set to the start point of the contour. If you have selected another element, the new profile elements will be appended to the existing ones without changing the reference point. So you can append the second half of an asymmetrical profile behind the point **P**.
- Selection discharged or loaded by using the menu switch [Profile, Loaded](#), just as you have drawn the contour.
- Call the function [Profile, Read CAD-Contour](#) and you get the profile list for the contour. Check the profile list by using the function [View Pass](#).

If you are using another CAD system, you alternatively can use the [Contour File \(DXF Format\)](#)

### 3.4.6 Contour File (DXF Format)

Use the contour file to transfer the geometry of a profile or a roll tool drawn in the [CAD System](#) to PROFIL.

The DXF-format is a file format defined by Autodesk, used from nearly all CAD systems. The file can contain unsorted LINE, ARC, POLYLINE, LWPOLYLINE, and ELLIPSE entities, the contour tracking is done within PROFIL.

To generate a contour file, proceed the following steps:

- Draw the contour in the CAD system. Select a [Reference Point](#) (this is mostly the center point of the lower line of the web of the profile) and draw the lower line of the profile, beginning in the reference point.
- Save the contour into a DXF-file. In AutoCAD, use the function DXFOUT. Create a small CAD macro to simplify this operation.
- In PROFIL, select the [Profile Element](#) in the [Profile Lists Window](#) for the start of the contour. The start point normally is element 1, except you read the 2nd half of an asymmetrical profile, then the start point is the next element behind element **P**. If you have selected the first element, the reference point x0/y0 and the direction are set to the start point of the contour. If you have selected another element, the new profile elements will be appended to the existing ones without changing the reference point. So you can append the second half of an asymmetrical profile behind the point **P**.
- Selection discharged or loaded by using the menu switch [Profile List Loaded](#), just as you have drawn the contour.
- Call the function [Profile, Read CAD-Contour](#). The window [Window Read CAD-Contour](#)

appears and you define the desired profile contour in this window. Afterwards you get the profile list for the contour. Check the profile list by using the function [View Pass](#).

For generating a roll tool, proceed the same steps, but:

- Draw the contour by using and modifying the drawing of the profile, which you have transferred by using the function [Drawing -> CAD](#).
- Read the contour file by using the function [Roll Read CAD-Contour](#). The window [Window Read CAD-Contour](#) appears and you define the desired roll contour in this window. The result is the roll tool with the desired roll contour.

### 3.4.7 Additions File

The additions file contains the additions for diameter and width, which are necessary for the calculation of the blank sizes out of the finished sizes. The file is utilized, when you call the function [Output Create Parts List](#).

**Section Roll Diameter:** The first column contains the blank diameters and the second the related finished diameters. The meaning is: in the second column the value is searched that is yet larger or equal to the finished diameter of the roll (in the case of fillets the intersection point of the tangents). The related value in the first column is the matching blank diameter (the bar diameter).

**Section Roll Width:** The first column contains the finished diameters and the second the related additions for the width. The meaning is: in the first column the value is searched that is yet larger or equal to the finished diameter of the roll. The related value in the second column is the matching addition for the width, which is added to the finished width and is rounded to full mm (respectively 0.1 inch).

The name of the additions file is the name of the roll material with the file extension **.add**, e.g. **2080.add**, if you have selected in [Options Profile List](#) the representation of values **Metric**. All values are in mm. If you have selected **Imperial**, the file name has the extension **.adi** and all values are in inch. The name of the roll material is copied into the column **Mat.** of the parts list.

Use the function **Edit** in the row **Material** in the [Expanded Roll Tool Window](#) to edit the additions file and to adapt it to your own needs.

### 3.4.8 Drawing Template

The drawing template is a DXF file that contains a drawing frame, a title block, and [Variables](#). It is used by the function [File Plot](#) in order to send a standardized assembly drawing to the output device.

You can use any DXF file as drawing template. Use the button **Open Template** in the [Plot Assembly Plan](#) window to open one of the existing templates or define a template by yourself by using any CAD system. You also can modify an existing template for your needs by e.g. separating the rolls in the [Assembly Plan](#) window and by saving the template afterwards.

PROFIL is able to fill out the title block automatically, if the drawing template contains [Variables](#). Position and text height are considered. Variables that represent drawing objects are replaced by the ones from the current drawing.

## 3.5 Profile List

The profile list describes the geometry of the profile for one pass. The name of a profile list ends always with **.Lnn**, while nn is the number of the pass and L01 is the last pass (finishing pass). The numbering is opposite the sheet running direction.

The profile list data are:

[Pass](#)  
[Dist. stand](#)  
[Strip width](#)  
[Reference Point X0/Y0](#)  
[Direction](#)

and a table of maximum 199 (Full Version) or 15 (LT Version) [Profile Elements](#). In conjunction with the profile list data these data describe numerically the geometry of the profile.

The table of profile elements has the following columns:

[Number](#)  
[Type](#)  
[Direction](#)  
[Radius](#)  
[Angle](#)  
[Size](#)  
[Position](#)  
[Straight Lenth](#)  
[Stress](#)

Comment for users of updates from earlier versions of **PROFIL**: In earlier versions the profile lists were saved in separate files for each profile list. These files had the extension **.Lnn**. To import such files use the function [File Import Profile List](#).

### 3.5.1 Pass

The pass belongs to the profile list data. Other names for the pass are station, stand.

Profile lists are always numbered opposite the sheet running direction (L01 = finishing stand). In the input field **Pass** you can automatically number the passes in sheet running direction. Enter into the input field **Pass** of any profile list, e.g. the scheme **Pass ##** or **##. Stand**. Press ENTER and in all profile lists your input appears; at the same time **##** is replaced by the pass number in sheet running direction. If the first pass is flat, i.e. has no arcs, it becomes number **00** (for the flat sheet). Use this function when all passes are complete.

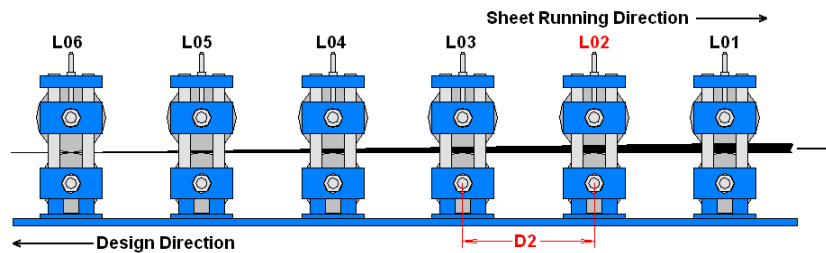
If additional profile lists are inserted in a profile project, the numbering in the field **Pass** is not renewed automatically. So it is necessary to renumber the pass number like this: Point to the arrow down key, select from the drop-down-list the desired numbering scheme and press ENTER. In order to delete an item from the list, press the **Backspace** key.

If needed, the pass number can be used as variable \$PS for the automatic roll numbering, see [Options Rolls](#).

The pass number can be displayed in the Profile Explorer to name the profile lists in sheet running direction. Toggle by using the function [Edit, Explorer](#).

See also: [Profile List](#).

### 3.5.2 Dist. Stand



The distance between stands is the horizontal distance between the current and the previous stand in sheet running direction (center roll to center roll). Example: The distance between stands **D2** in profile list **L02** is the distance between **L02** and **L03**. For the first stand enter the length of the spheroidizing zone. Please estimate the value.

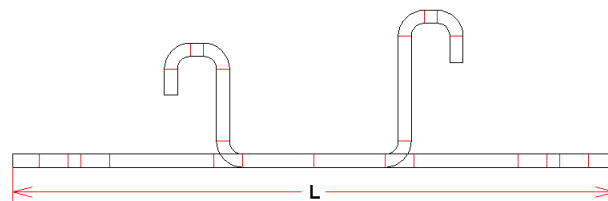
The distance between stands belongs to the [Profile List](#) data and is displayed in header of the the [Profile List Window](#).

#### Principle of operation

If you create a new profile list, the distance between stands is taken from the [Machine Window](#) and copied into the profile list data. The distance between stands is used by the calculation of the [Stress of Edge](#).

See also: [Profile List](#).

### 3.5.3 Strip Width



The **Strip Width L** is the width of the flat sheet that is needed to form the profile with the desired cross-section.

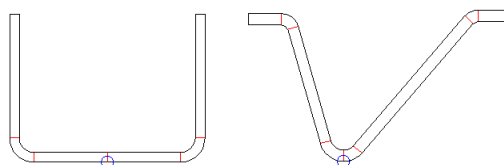
**PROFIL** contains different [Calculation Methods](#) to calculate the strip width.

The strip width belongs to the [Profile List](#) data and is displayed in header of the the [Profile List Window](#).

#### Hint:

- By using the function [Profile, Modify Strip Width](#) the profile elements at the band edges can be modified in order to have a profile list with a given strip width.

### 3.5.4 Reference Point X0/Y0



The **Reference Point** defines the start point of the profile in the xy plane.

The reference point normally should be at or near the center of the profile's web on the sheet metal's bottom side (see left pic.). If the profile has not a horizontal web, the reference point can be set on the bottom point of an arc segment instead (see right pic.). Because both legs turn around the reference point during forming, it is recommended to put it in such a position in order nearly to have equal leg heights on both sides, also in case of a unsymmetrical profile. Thus the [Stress of Band Edge](#) is distributed equally to both sides. This also can be optimized by selecting a suitable [Direction](#). Some results of the function [Calculate Statics](#) are related to the reference point.

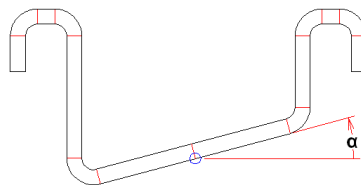
You can set the start point to any point. Practical values are (0,0) or (200,150) dependent on the used CAD system (200,150 is the center of a DIN-A3-page). In order to move the reference point to another position, use the function [Modify Reference Point](#).

The reference point belongs to the [Profile List](#) data and is displayed in header of the the [Profile List Window](#).

#### Hints:

- Modify the first profile element in the [Profile List](#) (the element that starts at the reference point) by using the function [Modify Start Element](#).
- Some [Statics](#) are related on the reference point.
- The [Roll Reference Point](#) can be set independently from the profile reference point. Usually, both reference points are set to the same coordinate.

### 3.5.5 Direction



The **Direction** determines the start angle of the profile in the [Reference Point X0/Y0](#). It is defined like in CAD systems:

- 0° = to the right
- 90° = to top
- 90° = to bottom

Most profiles have a horizontal web. The reference point is in the middle of it and the direction is 0°. Sometimes, if it has not a horizontal web, the reference point has to be set to a point on an angular part of the profile (as shown in the example drawing). Then the direction is positive for an ascending and negative for a declining part.

The direction belongs to the [Profile List](#) data and is displayed in header of the the [Profile List Window](#).

#### Hints:

- By using the function [Read CAD-Contour](#) the direction is automatically set to the direction of the contour.
- During flower pattern creation, do not forget to turn the profile in the machine in order to get the direction 0° for the flat sheet. This is important why the decoiled strip always is horizontal. This can take place step by step within several passes.

### 3.5.6 Profile Elements

The profile elements are the components of the [Profile List](#) that describe the geometry of the profile as a spread sheet table. Each profile element describes either a straight, not bent or a bent part of the profile or a reference or a symmetrical point.

The column **Type** marks whether the profile element describes a straight or a bent part or a point.

The straight part is represented by a **L** (Line) in the column **Type** and the length in column **St. Length**.

The bent part is represented by an **A** (Arc) in the column **Type** and a following number, which defines the [Bending Method](#). In column **Di.** (Direction) the bending direction is displayed, **L** for left and **R** for right. Further data are the **Radius**, the **Angle** and the **St. Length** that is calculated by different [Calculation Methods](#).

**P** is the reference point of an asymmetrical profile and marks the start of the second half of the profile.

**PS** is the reference and symmetrical point of a symmetrical profile. You have to enter data for one side of the profile only.

**P** and **PS** will be used as a [Reference Point](#) for the function [Calculate Statics](#).

#### 3.5.6.1 Number

The number is a column of the table of [Profile Elements](#) of the [Profile List](#) and numbers the profile elements continuously.

If you modify the number of the profile elements, e.g. by using one of the functions [Element Insert](#), [Element Append](#) or [Element Remove](#), the profile elements will be automatically renumbered.

See also: [Profile List](#).

#### 3.5.6.2 Type

The type is a column of the table of the [Profile Element](#) of the [Profile List](#) and marks, whether the profile element is a straight or a bent part of the profile.

The following types are possible:

##### **L = Line (or S = Strecke):**

The profile element describes a straight part of the profile. Further data are only the length of the straight part in the column [Straight Lenth](#). If the part is punched, furthermore the columns [Size](#) and [Position](#) can be filled out.

##### **A = Arc (or B = Bogen):**

The profile element describes a bent part of the profile. The number behind the letter defines the [Bending Method](#). Further data are: [Direction](#), [Radius](#), [Angle](#), [Straight Lenth](#) (will automatically be calculated). Not allowed are data in the columns: [Size](#) and [Position](#).

##### **P = Point:**

The profile element describes the reference point of a non symmetrical profile. Afterwards the description of the 2nd half of the profile follows. Only one element **P** is allowed. The point is also be used as [Reference point](#) by using the function [Calculate Statics](#).

##### **PS = Symmetrical point:**



The profile element describes the symmetrical point of a profile. Only one element **P** is allowed and must be the last element in the profile list. The point is also be used as [Reference Point](#) by using the function [Calculate Statics](#).

See also: [Profile List](#).

### 3.5.6.3 Direction

The direction is a column of the table of the [Profile Elements](#) of the [Profile List](#) and marks, whether a bent part turns left (**L**) or right (**R**), based on the reference point.

See also: [Profile List](#).

### 3.5.6.4 Radius/Angle Discharged

Radius and angle discharged are columns of the table of [Profile Elements](#) of the [Profile List](#) and describe the inner radius and bending angle of a bent part in discharged state, this means after the profile has left the roll tools and has sprung back. So the profile list **L01** discharged describes the geometry of the desired profile.

See also: [Radius/Angle Loaded](#)

See also: [Profile List](#).

### 3.5.6.5 Radius/Angle Loaded

Radius and angle loaded are columns of the table of [Profile Elements](#) of the [Profile List](#) and describe the inner radius and bending angle of a bent part in loaded state, this means while the profile is in contact with the roll tools. For compensating the [Spring Back](#) this angle must be larger and the radius must be smaller than in discharged state.

The loaded state is automatically calculated out of the discharged state, if you have selected a [Material](#) in the [Project Data Window](#) and the [Material File](#) contains the factors **k1** and **k10** for spring back for this material. Otherwise also the calculation of the discharged state out of the loaded state is possible.

Radius and angle are shown in the [Profile List Window](#), if you have selected the layout **With Spring Back** in [Options Profile List](#).

See also: [Radius/Angle Discharged](#)

See also: [Profile List](#).

### 3.5.6.6 Size

The size is a column of the table of the [Profile Element](#) of the [Profile List](#) and defines the diameter of a hole or the width of a cut-out. [Holes/Cut-Outs](#) are allowed only in straight [Profile Elements](#) of type **L**.

For displaying the size in the [Profile List Window](#), select the layout **With Holes/Cut-Outs** in [Options Profile List](#).

See also: [Profile List](#).

### 3.5.6.7 Position

The position is a column of the table of the [Profile Elements](#) of the [Profile List](#) and defines the center of a hole or a cut-out. [Holes/Cut-Outs](#) are only allowed in straight [Profile Elements](#) of type L. The position is related to the start point of the profile element.

For displaying the size in the [Profile List Window](#), select the layout **With Holes/Cut-Outs** in [Options Profile List](#).

See also: [Profile List](#).

### 3.5.6.8 Straight Length

The straight length is a column of the table of the [Profile Elements](#) of the [Profile List](#) and displays the length of a straight element or the calculated straight length of a bent element, dependent on the selected [Calculation Methods](#).

See also: [Profile List](#).

### 3.5.6.9 Stress

The stress is a column of the table of the [Profile Elements](#) of the [Profile List](#) and displays the relative stress in transverse direction on the outer side of an arc in % related to the fracture stress of the material. If values are greater than 100%, you must expect cracks. Furthermore the [Statics](#) is not valid.

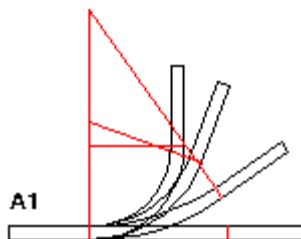
See also: [Profile List](#).

### 3.5.6.10 PE

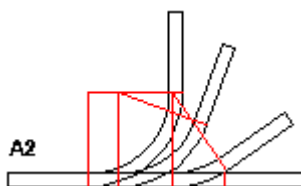
For future use only.

## 3.5.7 Arc Types

PROFIL knows 4 various arc types (bending methods), which are treated differently while bending:

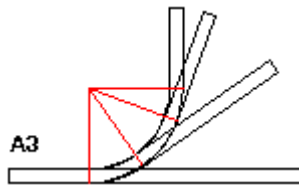


**Arc type A1** has a constant straight length. By changing the bending angle the bending radius is calculated and modified. Also possible: by changing the radius the angle is calculated and modified. This method often is used in case of sharp bending (this means small inner radii).

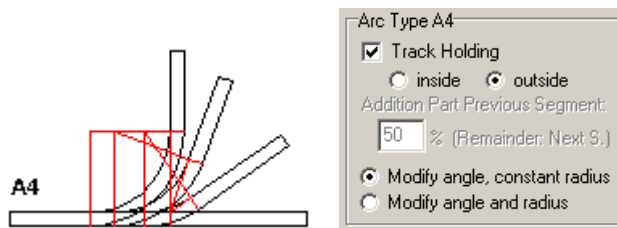


**Arc type A2** has a constant straight length together with its previous segment and has a constant

bending radius. The result of changing the bending angle is a modified straight length of the arc. In order to keep constant the sum of all lengths the length of the previous segment is adapted. PROFIL calculates the new lengths if you define a new bending angle. The method often is used in case of large radii. Because bending mostly takes place in the horizontal part of the profile, the use of side rolls can be avoided.



**Arc type A3** is similar to arc type **A2**, but instead of the previous segment the next one is used for the length compensation. PROFIL calculates the new lengths. Also this method often is used for large radii.



**Arc type A4** is a combination of **A2** and **A3**. Select in [Options Calculate](#), "Arc Type A4, Addition Part Previous Segment" how much of the residual length should be added to the previous segment. The rest is added to the next segment.

**Track Holding:** Select arc type **A4** and check in [Options Calculate](#), "Arc Type A4, Track Holding". The residual length is divided automatically to the previous and next segment that the strip is guided straightaway (this means with constant intersection point of the tangents). Select if the intersection point of the tangents **inside** or **outside** should be kept constant while bending.

**Modify angle, constant radius:** By using this setting A4 works like A2 and A3 by keeping the radius constant while modifying the angle.

**Modify angle and radius (radius/angle method):** After entering a new angle an input window opens and asks for the desired new radius. Firstly, this method seems to be absurd, because (in sheet running direction) a part of the arc after bending is pressed to flat again. However, the method can be chosen expediently in these cases:

- Existing rolls (with certain angles and radii) should be re-used.
- Spring back should be compensated. This works in the same way as the function [Open Fold](#), see chapter **Why doesn't this fold spring up?** First, in sheet running direction, more material as necessary is bent to an arc. In the last stand, a part at the beginning and at the end of the arc is bent to flat again, whereas the middle part is bent to the final angle. During this operation, all three part spring back. The middle part springs back to a larger radius, the other flat parts however want to reconvert to an arc again. This means, the springback acts in different directions. By a smart selection of the radius in the input window the springback can be compensated completely. Recommended values for the radius are: Until 1mm sheet thickness: Final radius plus 1.0 .. 1.5mm addition, until 2mm sheet thickness: Final radius plus 1.5 .. 1.7mm addition, until 3mm sheet thickness: Final radius plus 1.7 .. 2.5mm addition. The input window for entering the radius opens when an arc should be unbent by modifying the arc angle, see [Designing the Flower Pattern](#).

#### Hints:

- Enter the bending method into the column **Type** of the [Profile List](#).
- Normally bending with modification of the neighbor segment (arc type **A2**, **A3**, and **A4**) only makes sense if the neighbor is a line type segment (type **L**). However, if it is an arc type segment, the material is moved without bending. This behavior can be intended by the user, if he wants to modify the profile list in this way. In order to bend up the profile, this trick may be helpful: Before bending, select the position in the profile list where the material should be moved

to. Then insert a profile segment of type **L** and with length zero at this position by using the function [Element Insert](#). During bending of the arc, the new line segment will be lengthened and the profile will be bent up.

### 3.5.8 Holes/Cut Outs

If you have selected in [Options Profile List](#) the layout **with holes/cut-outs**, in the [Profile List Window](#) the input fields **Size** and **Position** are shown. Use these fields to add holes/cut-outs to any [Profile Element](#) of type **L**.

Enter the diameter of the hole or the maximum width of cut-out into the field **Size**.

Enter the distance from the center of the hole/cut-out to the start of the profile element into the field **Position**.

#### Principle of operation

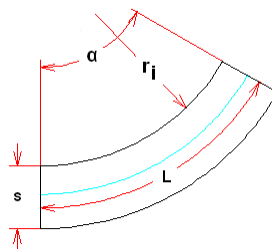
Holes/cut-outs will weaken the profile - this will be considered by function [Calculate Statics](#). Statics is only related to the cross sectional area, so longitudinal size has no influence. But it must be considered in strength calculations based on the statics. **PROFIL** does not deal with strength calculations.

#### Hints:

- [View Pass](#) or any other view function will display your defined holes/cut-outs. They will be transferred to the CAD drawing by function [Drawing -> CAD](#).
- You may also print the profile list including holes/cut-outs Use the function [File Print](#).  
Precondition is that you have selected the layout **with holes/cut-outs** in [Options Profile List](#).
- If the [Profile Element](#) should get more than one holes/cut-outs, you have to split it by using the function [Profile, Element, Split](#).

### 3.5.9 Calculation Methods

The [Strip Width](#) of a profile is the sum of the [Straight Lengths](#) of all [Profile Elements](#). In case of a profile elements of [Type L](#) (Line) the straight length is identical to the line length. Otherwise, in case of a profile elements of [Type A](#) (Arc) the straight length has to be calculated by assuming a neutral line that keeps it's length constant during bending. Thus the straight length is identical to the neutral line.



$$L = 2\pi \left( r_i + f \cdot \frac{s}{2} \right) \cdot \frac{\alpha}{360^\circ}$$

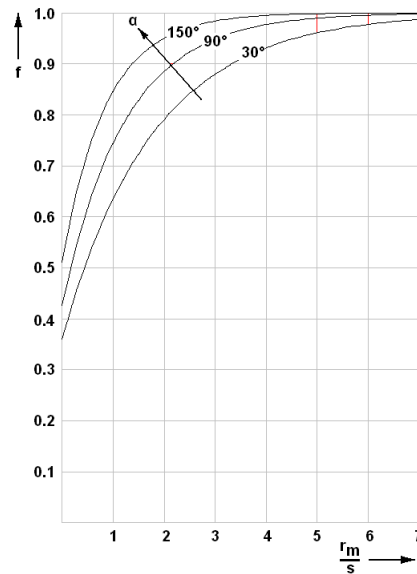
with:

- L** = Straight length or length of the neutral line (blue line, see picture)
- ri** = Inner radius
- f** = Factor for the position of the neutral line (0 = inner sheet side, 1 = center line)
- s** = Sheet thickness
- a** = Bending angle

In case of large radii the neutral line is in the sheet center accurately, the factor *f* for the position of the neutral line is 1. The smaller the radius is, the more the neutral line is moved to the inner side of the sheet, the factor is <1. For sharp bending, i.e. with inner radius near 0, the factor is about 0.4 .. 0.5.

In order to get the **factor f** for the position of the neutral line, several methods exist. Two of them have delivered an optimal performance in practice, the Oehler and the DIN method.

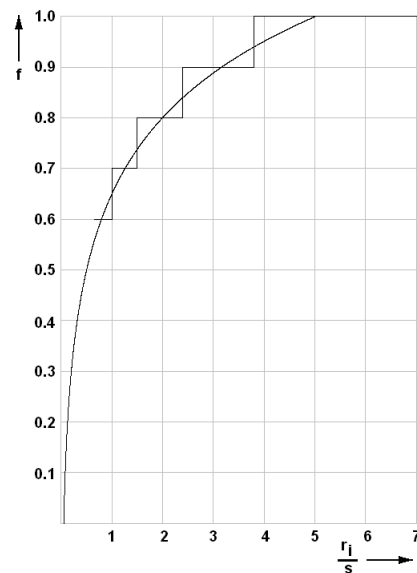
### Oehler Method



$$f = f(r_m, s, \alpha)$$

The Oehler method determines the **factor f** as a function of the **medium radius rm**, the **sheet thickness s**, and of the **bending angle  $\alpha$**  (in the diagram shown as a set of curves). This is why the Oehler method is the most precise method in case of pure bending.

### DIN 6935 Method



Calculation by formula:

$$f = 0.65 + \frac{1}{2} \cdot \lg \frac{r_i}{s}$$

Determining from table:

$r_i / s$	>0.65 .. 1.0	>1.0 .. 1.5	>1.5 .. 2.4	>2.4 .. 3.8	>3.8
f	0.6	0.7	0.8	0.9	1.0

with:

**f** = Factor for the position of the neutral line  
**ri** = Inner radius  
**s** = Sheet thickness

DIN 6935 is a practical calculation method as a function of inner **radius ri** and **sheet thickness s**, for pure bending too. Select in [Options Calculate](#), if it should be calculated **by table** or **by formula**. In the diagram, both alternatives are shown.. Because the DIN method has only two parameters, it is less precise than the Oehler method. However, because it is quite simple to use (especially by handling the table), it is the preferred method for manual determining the straight length.

#### User Defined Methods

Use your own [User Defined Calculation Methods](#) based to your experience, especially for not pure bending.

#### Hints:

- Select the desired method in the [Project Data](#).
- Do not modify the calculation method if your project already has more than one stands or the flower pattern is already completed. Thus you would get different strip widths in the passes.

### 3.5.10 User Defined Calculation Methods

In addition to default methods of Oehler and DIN 6935 you can enhance **PROFIL** with your own calculation methods to get the straight length.

Call [Options Calculate](#) and **Register User Defined Calculation Methods**. Edit the [Factor File](#) to define your own calculation procedure of type [Factor Method](#) or [Addition Method](#).

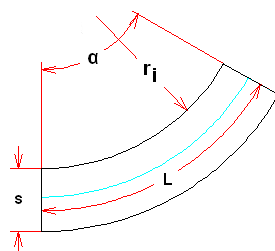
### 3.5.11 Factor Method

The length of the neutral line of the sheet is calculated by using the factor method. The position of the neutral line is defined by a factor **f**, which is between 0 and 1:

**f = 0** means, the neutral line is at the inner side of the sheet;  
**f = 1** means, the neutral line is in the center of the sheet.

In addition you can enter a correction factor (**k1** in %) and a correction addend (**k2**).

The straight length of a bent part of the sheet is calculated by:



$$L = 2\pi(r_i + f \cdot \frac{s}{2}) \cdot \frac{\alpha}{360^\circ} \cdot (1 + \frac{k_1}{100}) + k_2$$

with:

**L** = Straight length or length of the neutral line (blue line, see picture)  
**ri** = Inner radius  
**f** = Factor for the position of the neutral line (0 = inner sheet side, 1 = center line)  
**s** = Sheet thickness  
**a** = Bending angle  
**k1** = Correction factor in % (pos. or neg.)  
**k2** = Correction addend in mm (pos. or neg.)

The [Factor File](#) contains **factor f**, **correction factor k1** and **correction addend k2** dependent on the relation inner radius/sheet thickness.

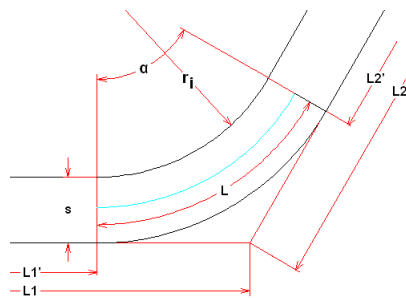
See also: [Addition Method](#)

### 3.5.12 Addition Method

The addition method works with the theory that the outer lines of the straight parts before and behind a bent part of the profile are lengthened until they intersect another. This point often is used as dimensioning point in technical drawings. The lengths of these new lines are added and lead to a straight length that is too large, first.

To this result a (negative) correction addend **k2** from a table is added. Furthermore a correction factor **k1** (in %) can be used.

The straight length of a bent part of the sheet is calculated by:



$$L = (L_1' + L_2' - L_1 - L_2) \cdot \left(1 + \frac{k_1}{100}\right) + k_2$$

with:

- L** = Straight length or length of the neutral line (blue line, see picture)
- L1, L2** = extended length of the straight parts of the sheet  
(to the intersection points of the tangents)
- L1', L2'** = length of the straight parts of the sheet
- k1** = Correction factor in % (neg.)
- k2** = Correction addend in mm (neg.)

The [Factor File](#) contains **correction factor k1** and **correction addend k2** dependent on sheet thickness, bending angle and inner radius.

See also: [Factor Method](#)

## 3.6 Roll Tools

### 3.6.1 Roll Number

The roll number belongs to the roll data and is displayed in the header of the [Roll Tool Window](#).

For the roll number, all numbers and characters, included special character, are allowed.

Define the roll number, so that it marks the install location in the machine, e.g.  
03102 = Stand 03, Shaft 1 (bottom shaft), Roll No. 02.

The roll number can be created automatically, if you enter a number key into the dialog [Options Rolls](#) for the roll number, before you generate a roll by using the function [Roll Read CAD-Contour](#) or [Roll Scan Profile Drawing](#).

The roll number can be incremented automatically, when you split a roll by using the function [Roll Split at Corner](#) or [Roll Split between Corners](#). Select the **Automatic Increment** in [Options Rolls](#) for

the roll number.

The roll number should not be engraved into the roll, because later the roll could be installed in another stand, if you want to re-use the roll. For engraving use the [Part Number](#).

The roll number appears in the drawing in the [Drawing Area](#) and in the CAD-output file, which you generate with function [Drawing CAD](#) and determines the layer name. Furthermore the roll number is inserted in the [Parts List](#) and determines the program number of the [NC-Program](#).

### 3.6.2 Part Number

The part number belongs to the roll data and is displayed in the header of the [Roll Tool Window](#).

For the part number, all numbers and characters, included special character, are allowed.

If the rolls should be re-used and the part number is engraved into the roll, it is recommended, not to code the profile project and the install location into the part number. Pay attention that the part number is unambiguous, this means, allocate each part number only once.

The part number can be created automatically, if you enter a number key into the dialog [Options Rolls](#) for the part number, before you generate a roll by using the function [Roll Read CAD-Contour](#) or [Roll Scan Profile Drawing](#).

The part number can be incremented automatically, when you split a roll by using the function [Roll Split at Corner](#) or [Roll Split between Corners](#). Select the **Automatic Increment** in [Options Rolls](#) for the part number.

Use the [Roll number](#) for marking the install location in the machine and use the [Classification](#) for marking the type of the roll.

### 3.6.3 Classification

The classification code belongs to the roll data and is displayed in the header of the [Roll Tool Window](#).

For the classification, all numbers and characters, included special character, are allowed.

Define the classification key so that it marks the type of a roll, e.g.

CY100/50 = cylindrical roll 100 diam./50 width

CO120/30/50 = conical roll 120 diam./30 degree/50 width

With the help of the classification code it is easier to find a roll in the roll database for re-use.

### 3.6.4 Width

The width of the roll belongs to the roll data and is displayed in the header of the [Roll Tool Window](#).

The width is only an output field, this means, you cannot enter data into this field. The width is calculated out of the difference between the width of the first and last corner point of a roll.



### 3.6.5 Max. Diameter

The maximum diameter of the roll belongs to the roll data and is displayed in the header of the [Roll Tool Window](#).

The maximum diameter is only an output field, this means, you cannot enter data into this field. The diameter is identical to the intersection point of the corner point with the largest [Diameter](#).

### 3.6.6 Spacer Roll

Spacer roll is a switch that belongs to the roll data and is displayed in the header of the [Roll Tool Window](#). It is set automatically by [Rolls, Spacer Rolls, Create](#). Dependant on the status of this switch the roll is treated differently:

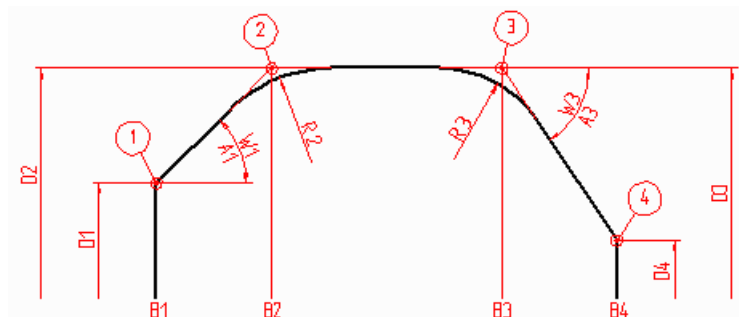
**Treating as [Forming Roll](#)** (box **Spacer Roll** unchecked):

- During [Renumbering](#) the **Number Keys** in [Options, Rolls](#) are used.
- In the [Part List](#), the appropriate designation for **Bottom Roll**, **Top Roll**, **Left Roll**, and **Right Roll** is taken from [Set-Up Parts List Columns](#).

**Treating as [Spacer Roll](#)** (box **Spacer Roll** checked):

- During [Renumbering](#) the **Number Keys** in [Options, Spacers](#) are used.
- In the [Part List](#), the appropriate designation for **Spacer Roll** is taken from [Set-Up Parts List Columns](#).
- [Rolls, Spacer Rolls, Remove](#) removes only this kind of roll.
- [View, Show, Spacer Rolls](#) toggles on-off the display and output of this kind of roll.

### 3.6.7 Corner Point



The contour of a roll is described by a set of roll corner points. Each one has the following data:

<a href="#">Width</a>	<a href="#">Diameter</a>
<a href="#">Radius</a>	<a href="#">Angle</a>

Imagine a rubber band, stretched across the corner points (1, 2, 3, 4, see picture), which are defined by width (B1, B2, B3, B4) and diameter (D1, D2, D3, D4). Each corner point may have a fillet radius (R2, R3), so that the connections to the neighbouring line elements are tangential. If the radius is larger than 0, the roll corner point is not a real point, but a virtual point of the roll, it is the intersection point of the tangents. The width is related to the width of the [Roll Reference Point](#). This means, the width can be positive or negative. Furthermore each corner point has an angle (W1, W3) to the next corner point. If the next point has a bigger diameter, the angle is positive, in the other case negative.

With the help of the rubber band model you can easily illustrate by yourself, what happens in the case of modifying and moving the corner points. The rubber band keeps stretched across the corner points; the radii cause a fillet in the corner points.

For each roll corner point a revolution line (with the auxiliary line color) can be drawn. Select it in

[Options Drawing.](#)

#### 3.6.7.1 Width

The width of the [Corner Point](#) belongs to the roll data and is displayed in the table of the corner points of the [Roll Tool Window](#).

You can modify the width by

- data input into the input field
- pressing **PgUp/Dn** on the keyboard
- using the [Toolbox Modify](#).

The width is related to the y-coordinate of the [Roll Corner Point](#); positive and negative values are possible.

#### 3.6.7.2 Diameter

The diameter of the [Corner Point](#) belongs to the roll data and is displayed in the table of the corner points of the [Roll Tool Window](#).

You can modify the diameter by

- data input into the input field
- pressing **PgUp/Dn** on the keyboard
- using the [Toolbox Modify](#).

#### 3.6.7.3 Radius

The radius of the [Corner Point](#) belongs to the roll data and is displayed in the table of the corner points of the [Roll Tool Window](#).

You can modify the diameter by

- data input into the input field
- pressing **PgUp/Dn** on the keyboard
- using the [Toolbox Modify](#).

The radius causes a fillet at the corner point with tangential connections to the drawing elements of the neighbouring corner points. In case of pure arc transitions width and diameter both of the selected and of the neighbouring corner points are modified, if necessary.

If the fillet is not possible, e.g. if the radius is too large and the arc would exceed the limit to the neighbouring corner point, this is shown in the drawing area and a warning is displayed in the bottom status bar.

#### 3.6.7.4 Angle

The angle of the [Corner Point](#) belongs to the roll data and is displayed in the table of the corner points of the [Roll Tool Window](#).

The angle is positive, if the next corner point has a larger radius, in the other case it is negative. The roll axle is the reference line for the angle. Thus a cylindric part of a roll has the angle 0°.

You can modify the angle by

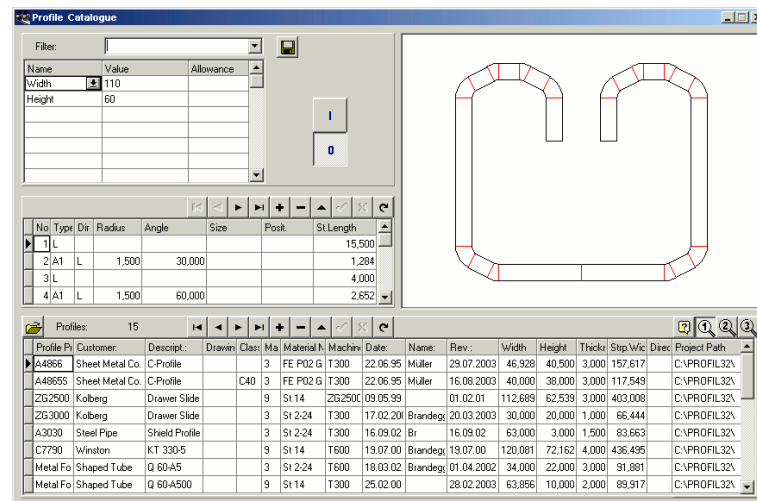
- data input into the input field
- pressing **PgUp/Dn** on the keyboard

If the angle is modified, the diameter of the next roll corner point is adapted. Width and radius of the topical and the next corner point keeps unchanged.

#### Hint:

Alternatively, the angle can be modified by using the function [Clearance Angle](#). In doing so, the width of the corner points are modified.

## 3.7 Profile Catalogue




Only with option Database.

While dealing with an inquiry the designer needs calculation data from similar profiles that are produced in the past. Designing new roll form projects should consider experience from earlier projects. The profile catalogue gives a quick overview of all produced roll formed parts. Any filters can be defined, named, saved and reloaded by the user for selecting the desired profiles.

The profile catalogue contains the [Profile List](#) of the final pass L01 each, and the project data of the [Profile Project](#). The belonging drawing of the final pass is quickly generated and displayed while browsing through the database. A special button in the profile window enables quick access to the project file. 3 different user defined views with desired columns are displayed by key-stroke.

### Calling the function

Before you call this function, select one of the views [View Pass](#), [View Statics](#), [View Flower Nested](#), [View Flower Separated](#), [View Flower 3D](#). This prepares opening the profile catalogue for the button in the top button bar. Call this function optionally by:

- Main menu: **Profile, Profile Catalogue**.
-  Button **Profile Catalogue** in the [Button Key Bar](#).

### Profile Catalogue Open

After calling this function, the window **Profile Catalogue** appears, which contains 4 areas:

- [Profile Table](#) (lower area): This is the real profile catalogue, each row displays the project data of one profile.
- [Profile Element Table](#) (in the middle, left). This area displays the profile list L01 (final pass) data of the profile selected in the profile table.
- [Drawing Area](#) (top, on the right): This area displays the drawing of the profile selected in the profile table.
- [Filter](#) (top, on the left): Any filters can be defined for reducing the amount of displayed profiles.

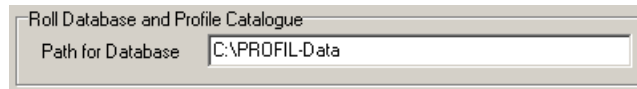
### Profile Catalogue Save

Use this function to store the final pass of one or more projects into the profile catalogue.

- **Current Project:** The currently opened project is stored.
- **All Projects of a Path:** The path selection window appears and all projects of the selected path are stored.

If the window Profile Catalogue is not yet open, it will be opened after calling one of these functions.

### Setup



Set the path to the database in [Options Database](#).

## 3.7.1 Profile Table

The lower area of the [Profile Catalogue](#) displays the table of the project data of the profiles. This table contains all profiles (if the filter is switched off) or a subset of the profiles (if the filter is switched on). Each row of the table displays the project data of the [Profile Project](#):

<a href="#">Customer</a>	<a href="#">Date</a>	
<a href="#">Description</a>	<a href="#">Name</a>	
<a href="#">Drawing No.</a>	<a href="#">Revision</a>	
<a href="#">Material</a>		<a href="#">Thickness</a>
<a href="#">Machine</a>		

Further columns contain the profile data of the [Profile List](#):

[Strip width](#)  
[Direction](#)

From the profile geometry are calculated:

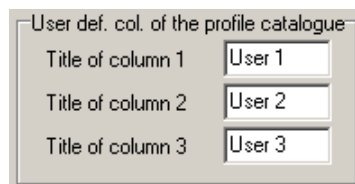
- **Width:** Total width of the profile
- **Height:** Total height of the profile

Furthermore are displayed:

- **Project Path:** This entry is used for quick access to the project by button **Open Profile Project**. The maximum field length is 40. Please consider that quick access only works if the path length does not exceed the maximum length.

Columns for manual input are:

- **Classification:** Organize the profiles in user defined groups dependent on the geometry. Assign a classification key to each group. This is useful for searching for special kind of profiles.



- **User 1-3:** Define the meaning of these columns by yourself. Enter suitable titles for these columns in [Options Database](#). Filtering for these columns is possible, too.

The belonging profile elements of the selected profile are displayed in the [Profile Element Table](#). Use a [Filter](#) to reduce the amount of displayed profiles. The drawing of the selected profile is not stored in the database; while browsing through the database it is quickly generated from the [Profile Element Table](#) and displayed in the [Drawing Area](#) instead.

The top bar of the profile table contains these indicators and buttons:



#### Open Profile Project

This button enables quick access to the belonging profile project of the selected profile. The belonging project will be opened. For the access the contents of the fields **Project Path** (maximum field length 40) and **Profile Project** (maximum field length 20) are used.

#### Number of profiles

displays the number of all profiles in the database (if the filter is switched off) or the number of filtered profiles (if the filter is switched on).



#### Database Navigator

for switching to the first/previous/next/last profile, for inserting, deleting and editing a profile, for posting and cancelling the edited data and for refreshing the data, e.g. if in a network several designers use the same database.



#### View 1/2/3

The count of displayed columns can be reduced for more clarity. Simply hide unwanted columns by moving the columns separator in the header row. The buttons **View 1/2/3** enable changing the view quickly.

### 3.7.2 Profile Element Table

The left area in the middle of the [Profile Catalogue Window](#) displays the [Profile Elements](#) of the profile, which is selected in the [Profile Table Window](#) at the bottom. Each line of the table shows the data of one profile element of the [Profile List](#) L01 (final pass):

[Number](#)  
[Type](#)  
[Direction](#)  
[Radius](#)  
[Angle](#)  
[Size](#)  
[Position](#)  
[Straight Lenth](#)

These data are automatically transferred, when a profile is inserted into the database via [Profile Catalogue, Save](#).

The belonging project data of the profile elements are displayed in the [Profile Table](#). Use a [Filter](#) to reduce the amount of displayed profiles. The drawing of the selected profile is not stored in the database; while browsing through the database it is quickly generated from the profile element table and displayed in the [Drawing Area](#) instead.

The top bar of the profile element table contains these buttons:



#### Database Navigator

for switching to the first/previous/next/last profile element, for inserting, deleting and editing a profile element, for posting and cancelling the edited data and for refreshing the data, e.g. if in a network several designers use the same database.

### 3.7.3 Drawing Board

The top area on the right of the [Profile Catalogue Window](#) contains the drawing area, which displays the drawing of the profile, which is selected in the [Profile Table](#) in the bottom area of the window. The drawing is always refreshed from the data of the [Profile Element Table](#), so that you have a clear idea how the selected profile looks like.

The colors of the drawing are the same as the colors of [Drawing Area](#) of the main window and can be set in [Options Drawing](#), too.

### 3.7.4 Filter

Define filters in the top area at the left of the [Profile Catalogue Window](#) for reducing the amount of displayed profiles in the [Profile Table](#) in the bottom area of the window. Each filter consists of a set of filter conditions (logical AND conjunction) and can be stored and recalled again. Filtering can be switched on and off.

#### Filter

Click on the arrow on the right of the input field and a drop-down-list is opened, which contains all existing filters. Select one of them and the according filter conditions are displayed in the table.

#### Table of Filter Conditions

##### Name

Select from the drop-down-list the name of the filter condition. The names are at first all columns of the [Profile Table](#): Profile project, Customer, Descript. and so on. Furthermore all columns of the [Profile Element Table](#) can be selected, but limited to the profile element numbers 1..8: Type 1..8, Dir. 1..8, Radius 1..8, Angle 1..8, Size 1..8, Posit. 1..8, St. Length 1..8.

##### Value

Enter the value of the filter condition. If the filter name in the same row determines a numerical field (e.g. width), the value has to be numerical, too (e.g. 156.3). In the other case an alpha-numerical value can be entered (e.g. Classif. = U100/50).

##### Allowance

Enter the allowance for the filter value. Example: If the filter value is 100 and the allowance is 0.1, all entities between 99.0 and 100.1 stay visible. The column **Allowance** is only evaluated, if the filter name in the same row determines a numerical field.

#### Buttons



##### Save Filter

If a filter name is displayed in the input field (i.e. if you have loaded an existing filter), you can use this button to save changed filter conditions. If not (i.e. if you have entered new filter conditions), you are asked to enter a filter name after pressing this button. Afterwards the new filter will be saved.



##### Filter On

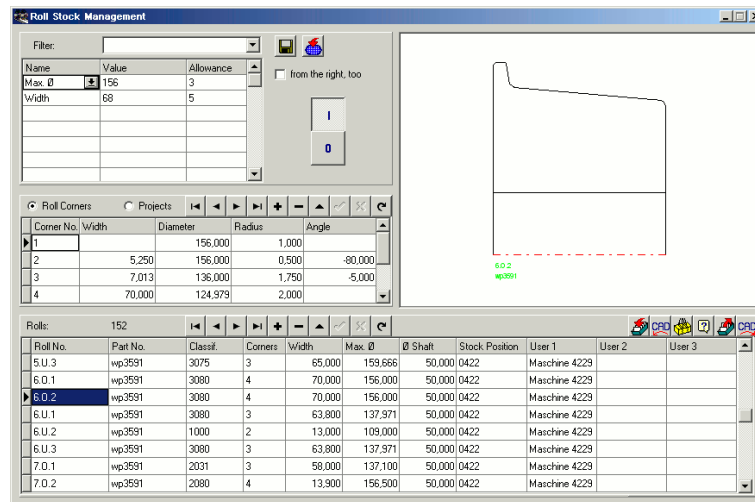
If the table contains filter conditions, the filter will be switched on. Afterwards the [Profile Table](#) displays those profiles only, which match to the filter conditions. In the top bar of the profile table the number of filtered profiles is displayed. If the profile table is empty, there is no matching profile in the database.



##### Filter Off

Use this button to switch off the filter. All profiles are displayed in the [Profile Table](#) again. In the top bar of the profile table the number of all profiles is displayed again.

## 3.8 Roll Stock Management



Only with option Database.


Afterwards the manufacturing of a profile has been finished, the roll tools are removed from the roll former and are taken to the roll stock, so the roll former can be set up for the next project. This is the right time to transfer the rolls from the project to the roll database.

The roll database gives information about the rolls in the roll stock. This helps while designing a new profile project, if you want to re-use existing rolls for reducing the costs. The roll database performs quick filter and search functions.

To transfer the rolls from and to the database, the clipboard can be used. All rolls of a shaft, a stand or of the complete project can be stored directly. Also a roll drawn in CAD can be stored. Rolls from the database can be transferred directly to CAD. If only a paper drawing of old rolls is available, the data can be entered directly into the database.

### Calling the function

Before you call this function, select the [View Roll Tools](#). This prepares opening the roll stock management for the button in the top button bar. Call this function optionally by:

- Main menu: **Rolls, Stock Management**.
-  Button **Roll Stock Management** in the [Button Key Bar](#).

### Stock Management Open

After calling this function, the window **Roll Stock Management** appears, which contains 4 areas:

- [Roll Table](#) (lower area): This is the real roll database, each row displays the data of one roll.
- [Roll Corner Table](#) (in the middle, left), can be switched to the [Project Table](#): This area displays the data of the roll selected in the roll table.
- [Drawing Area](#) (top, on the right): This area displays the drawing of the roll selected in the roll table.
- [Filter](#) (top, on the left): Any filters can be defined for reducing the amount of displayed rolls.

### Stock Management Save

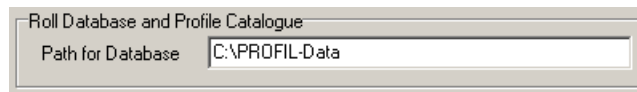
Use this function to store selected rolls into the roll database. Which rolls are stored, you define by selecting one of the following sub functions and by marking a roll in the [Drawing Area](#) respectively:

- **Roll**: The marked roll is stored.
- **Shaft**: All rolls of a shaft are stored. Define the shaft by marking any roll of the shaft.
- **Stand**: All rolls of all shafts of one stand are stored.
- **Project**: All rolls of all stands of the whole project are stored.

If the window Roll Stock Management is not yet open, it will be opened after calling one of these

functions.

### Setup



Set the path to the roll database in [Options Stock Management](#).

## 3.8.1 Roll Table

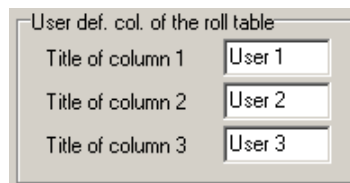
The lower area of the [Roll Stock Management Window](#) displays the table of the roll data. This table contains all rolls (if the filter is switched off) or a subset of the rolls (if the filter is switched on). Each row of the table displays the data of one roll. At first, there are the data from the top of the [Roll Tool Window](#):

<a href="#">Roll Number</a>	<a href="#">Width</a>
<a href="#">Part Number</a>	<a href="#">Max. Diameter</a>
<a href="#">Classification</a>	<a href="#">Diameter Shaft</a>
<a href="#">Number of Roll Corner Points</a>	

These data are automatically transferred, when a roll is inserted into the database or when a roll is taken from the database.

Furthermore, the roll table contains some data, which are to be entered by hand in case of need:

- **Stock Position:** Enter the location of the roll in the stock.



- **User 1-3:** Define the meaning of these columns by yourself. Enter suitable titles for these columns in [Options Stock Management](#).

The roll corner points of the selected roll are displayed in the [Roll Corner Table](#). The belonging projects of the selected roll are displayed in the [Project Table](#). Use a [Filter](#) to reduce the amount of displayed rolls. The drawing of the selected roll is displayed in the [Drawing Area](#).

The top bar of the roll table contains these indicators and buttons:

### Number of rolls

displays the number of all rolls in the database (if the filter is switched off) or the number of filtered rolls (if the filter is switched on).



### Database Navigator

for switching to the first/previous/next/last roll, for inserting, deleting and editing a roll, for posting and cancelling the edited data and for refreshing the data, e.g. if in a network several designers use the same database.



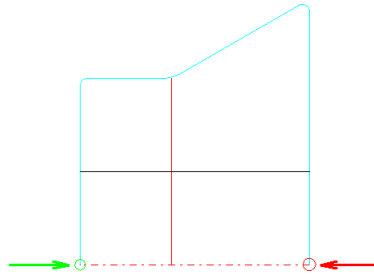
### Insert Roll from clipboard

transfers a roll from the clipboard to the database. The roll can be copied to the clipboard from the current project by using the function [Roll Cut](#) or [Roll Copy](#) previously.



### Insert Roll from CAD





reads a roll from a CAD drawing and inserts it into the database. This function is useful if older rolls should be inserted that are existing as a CAD drawing only (not in a profile project). Pay attention that the contour in the file has to start and to end on the centerline of the roll (in contrast to the function [Roll Read CAD-Contour](#)).

- **AutoCAD R14 or higher or SolidWorks 2003 or higher or SolidEdge ST7 or higher or BricsCAD rel. 15 or higher** (Precondition: In [Options ActiveX](#) Enable ActiveX to CAD is selected): The [Window Read CAD-Contour](#) appears (see picture), you can select the layer of the desired roll and define the contour of the roll by setting the start (green) and end point (red) of the contour (the intersection point of the edges and the center line in each case).
- **ME10 or OneSpace Designer Drafting or PTC Creo Elements/Direct Drafting** (Precondition: In [Options Files](#) **Output to CAD, ME10 (MI)** is selected): The file **Rolle.mi** is read from the same path that is set for the output file. This file must contain the drawing of one roll exactly. To get this file, select in ME10 the part that contains the roll (EDIT\_PART) and store the active part into a MI-file (STORE MI ' ' DEL\_OLD 'Rolle.mi'). The roll has to be drawn with the same colors and line types that are used by PROFIL, when a drawing is created in CAD.
- **Others** (Precondition: In [Options Files](#) **Output to CAD, A11, DXF or IGES** is selected): The KTR or DXF file is read that is set in [Options Files from CAD](#). This file must contain the drawing of one roll exactly.



#### Insert Roll from File

This function is equivalent to **Insert Roll from CAD**, but the roll is not taken from a fixed source. Instead of it, a file selection window opens and any DXF or KTR file can be selected as import file.



#### Copy Roll to Clipboard

copies the selected roll from the database to the clipboard. From the clipboard, the roll can be inserted into the current project by using the function [Roll Insert](#).



#### Drawing -> CAD

transfers the drawing of the selected roll directly to the CAD-system. This function uses the same settings as the function [Drawing -> CAD](#) in the main window.

### 3.8.2 Roll Corner Table

The left area in the middle of the [Roll Stock Management Window](#) displays the [Roll Corner Points](#) of the roll, which is selected in the [Roll Table Window](#) at the bottom. Each line of the table displays the data of one roll corner point. These are the corner point number and the data from the table of the [Roll Tool Window](#):

<a href="#">Width</a>	<a href="#">Diameter</a>
<a href="#">Radius</a>	<a href="#">Angle</a>

These data are automatically transferred, when a roll is inserted into the database or when a roll is taken from the database.

The belonging projects of the selected roll are displayed in the [Project Table](#). Use a [Filter](#) to

reduce the amount of displayed rolls. The drawing of the selected roll is displayed in the [Drawing Area](#).

The top bar of the roll corner table contains these buttons:

#### Roll Corners/Projects

These buttons change the whole area of this part of the window, so that the belonging [Projects](#) of a roll can be displayed.



#### Database Navigator

for switching to the first/previous/next/last corner point, for inserting, deleting and editing a corner point, for posting and cancelling the edited data and for refreshing the data, e.g. if in a network several designers use the same database.

### 3.8.3 Project Table

The left area in the middle of the [Roll Stock Management Window](#) displays the belonging projects of the selected roll in the [Roll Table](#) at the bottom. Each row of the table displays the data of one project. These are the project number, the project name (file name) and a subset of the data that are displayed in the [Project Window](#) (and belong to the [Profile Project](#)):

[Customer](#)  
[Description](#)  
[Drawing No.](#)

These data are automatically transferred, when a roll is inserted into the database or when a roll is taken from the database.

The roll corner points of the selected roll are displayed in the [Roll Corner Table](#). Use a [Filter](#) to reduce the amount of displayed rolls, also for given projects.

The top bar of the project table contains these buttons:

#### Roll Corners/Projects

These buttons change the whole area of this part of the window, so that the [Roll Corner Table](#) can be displayed.



#### Database Navigator

for switching to the first/previous/next/last project, for inserting, deleting and editing a project, for posting and cancelling the edited data and for refreshing the data, e.g. if in a network several designers use the same database.

### 3.8.4 Drawing Board

The top area on the right of the [Roll Stock Management Window](#) contains the drawing area, which displays the drawing of the roll, which is selected in the [Roll Table](#) in the bottom area of the window. The drawing is always refreshed from the data of the [Roll Corner Table](#), so that you have a clear idea how the roll looks like. If you enter the data manually (e.g. from a paper drawing) and the data input is not yet finished completely, non-rational drawings can be displayed.

The colors of the drawing are the same as the colors of [Drawing Area](#) of the main window and can be set in [Options Drawing](#), too.

### 3.8.5 Filter

Define filters in the top area at the left of the [Roll Stock Management Window](#) for reducing the amount of displayed rolls in the [Roll Table](#) in the bottom area of the window. Each filter consists of a set of filter conditions (logical AND conjunction) and can be stored and recalled again. The filter conditions can be entered manually into the table or can be generated automatically from the data of a roll from the clipboard. Filtering can be switched on and off.

#### Filter

Click on the arrow on the right of the input field and a drop-down-list is opened, which contains all existing filters. Select one of them and the according filter conditions are displayed in the table.

#### Table of Filter Conditions

##### Name

Select from the drop-down-list the name of the filter condition. The names are at first all columns of the [Roll Table](#): Roll No., Part No., Classification, No. of Corners, Width, Max. Ø, Ø Shaft, Stock Position, User 1-3. Furthermore all columns of the [Roll Corner Table](#) can be selected, but limited to the corner numbers 1..8: Width 1..8, Diameter 1..8, Radius 1..8, Angle 1..8. Furthermore all columns of the [Project Table](#) can be selected: Project, Customer, Description and Drawing No.

##### Value

Enter the value of the filter condition. If the filter name in the same row determines a numerical field (e.g. width), the value has to be numerical, too (e.g. 156.3). In the other case an alpha-numerical value can be entered (e.g. Part No. = BP3517).

##### Allowance

Enter the allowance for the filter value. Example: If the filter value is 100 and the allowance is 0.1, all rolls between 99.0 and 100.1 stay visible. The column **Allowance** is only evaluated, if the filter name in the same row determines a numerical field.

#### Buttons



##### Save Filter

If a filter name is displayed in the input field (i.e. if you have loaded an existing filter), you can use this button to save changed filter conditions. If not (i.e. if you have entered new filter conditions), you are asked to enter a filter name after pressing this button. Afterwards the new filter will be saved.



##### Create Filter from Clipboard

If the clipboard contains a roll, a new filter is created, whose filter conditions match to the roll exactly. Previously you are asked to enter an allowance, which will be inserted into the allowance column of all numerical fields. In this way, you can search for a suited roll, which you have created, e.g. by using the function [Roll Scan Profile Drawing](#) and afterwards copied to the clipboard by using the function [Roll Copy](#).

#### From the right, too

Since rolls can be installed turned, the search results can be expanded by rolls that match the filter conditions in inverse direction. If you check this box, these rolls will be found, too.



##### Filter On

If the table contains filter conditions, the filter will be switched on. Afterwards the [Roll Table](#) displays those rolls only, which match to the filter conditions. In the top bar of the roll table the number of filtered rolls is displayed. If the roll table is empty, no roll is matching.



##### Filter Off

Use this button to switch off the filter. All rolls are displayed in the [Roll Table](#) again. In the top bar of the roll table the number of all rolls is displayed again.

## 3.9 Others

### 3.9.1 Variables

PROFIL contains internal variables, which can be used in some functions for substituting e.g. a value or a name.

**Structure:** A variable consist of a \$ sign and two following capital letters, e.g. **\$PL**, or one capital letter and a numeral.

**Use:** Variables can be used for defining the roll number key (see [Options Rolls](#)) or for creating the [Drawing Template](#). When the rolls or the assembly drawings are created, the variables are replaced by the assigning value or name or by the assigning drawing object.

Variables from the [Project Data Window](#):

**\$PR** [Profile Project](#)  
**\$CU** [Customer](#)  
**\$PD** [Description](#)  
**\$DR** [Drawing No.](#)  
**\$MA** [Material](#)  
**\$MC** [Machine](#)  
**\$DA** [Date](#)  
**\$NV** [Revision](#)  
**\$NA** [Name](#)  
**\$ST** [Thickness](#)  
**\$CM** [Calc.-Method](#)

Variables from the [Profile List Window](#):

**\$PL** [Profile List No.](#) – only the number, against the sheet running direction, e.g. 7 in profile list L07.  
**\$PS** [Pass No.](#) - only the number, in sheet running direction  
**\$PN** [Pass Name](#) – full name, incl. number  
**\$DS** [Distance between Stands](#)  
**\$SW** [Strip Width](#)

Variables for the rolls:

**\$SA** consecutive roll number of the rolls of a stand  
**\$TA** consecutive roll number of the rolls of a shaft type (B, T, L, or R) of a stand

Variable from the drawing:

**\$SC** Drawing Scale (see [Plot](#))

Variables that represent drawing objects:

**\$AS** Roll stand assembly  
**\$Bn** Bottom roll n (if separated)  
**\$Tn** Top roll n (if separated)  
**\$Ln** Left side roll n (if separated)  
**\$Rn** Right side roll n (if separated)  
**\$MX** Roll mirrored at the x-axis (supplement to \$Bn, \$Tn, \$Ln, \$Rn)  
**\$MY** Roll mirrored at the y-axis (supplement to \$Bn, \$Tn, \$Ln, \$Rn)

These variables are used in the assembly plan only.

Further variables:

**\$CO** Company – Your own company name from the initial window  
**\$IM** Imprint - „Designed by \$CO by using UBEKO PROFIL“

### 3.9.2 CAD Systems

**PROFIL** is able to exchange data to and from different CAD systems. You only need to select the data exchange format in [Options Files](#):



CAD system **AutoCAD R14 and higher** or **SolidWorks 2003 and higher** or **SolidEdge ST7 or higher** or **BricsCAD release 15 and higher**: Use the built-in ActiveX interface (Setting in [Options ActiveX](#)).



CAD system **ME10 or OneSpace Designer Drafting or Creo Elements/Direct Drafting respectively**: Macros are available for generating a [Contour File \(KTR Format\)](#) and for reading a MI file. Please contact your dealer.

CAD system **PC-DRAFT**: Macros are available for generating a [Contour file \(KTR Format\)](#) and for reading a A11 file. Please contact your dealer.

Others: Please create macros by your own for quick data exchange via **DXF**. In order to transfer 3D models use the **STEP** format.

If you do not know the drive, path and file format for data exchange file files, please ask your system administrator.

If the used CAD system is running under the operating system UNIX, it is possible to exchange data via a network server. It does not matter if the generated files have the UNIX file format (only LF at the end of a line), PROFIL is able to read them. In the other direction, **PROFIL** generates files with CR/LF at the end of a line. UNIX-systems should be able to read them.



## 4 Installation

### 4.1 Contents of the Disks

Dependent on the different operating system and CAD systems the following files are necessary. If a necessary file is absent please order it or download it from the web-site <http://www.ubeco.com>

- **profil32.exe** Setup procedure for the demo version 32 bit for WINDOWS. (Contains the ActiveX interface to **AutoCAD**, **SolidWorks**, **SolidEdge**, and **BricsCAD** and any the file interface to any CAD system)
- **ProfilMe.zip** Interface to **CoCreate ME10** or **OneSpace Designer Drafting** or **Creo Elements/Direct Drafting** respectively
- **ubeco.psf** Personality File
- **\Marx\** Driver for the **Marx USB Hardlock**: CBUSetup.exe, readme.txt, version.txt

The following installation procedure requires these files on a disk in drive a:

Please start the installation with [Installing PROFIL](#).

### 4.2 Installing PROFIL



In order to install **PROFIL** for WINDOWS proceed as described:

- Installation: Start the setup program:  
a: **PROFIL32.exe**  
and follow the instructions of the setup procedure.
- Copy the Personality File **ubeco.psf** into the path of PROFIL. It will turn the Version on, this means the demo version becomes a LT or Full Version, e.g.  
copy a:ubeco.psf c:\program files\UBECO\PROFIL
- Set the Screen Resolution to 1024 x 768 (or higher) and select **Small Fonts**.

**PROFIL LT:** The software now is fully installed for WINDOWS.

**PROFIL Full Version:** With it PROFIL Full Version is installed and - in case you use **AutoCAD** or **SolidWorks** or **SolidEdge** or **BricsCAD** - the ActiveX-interface is ready.

Now install the hardlock driver (PROFIL Full Version only):

- [Driver Installation Marx USB Hardlock](#).

Further Information about adapting PROFIL to CAD systems:

- [ActiveX interface to AutoCAD and SolidWorks and SolidEdge and BricsCAD](#)
- [Interface to ME10\(CoCreate OneSpace Drafting, PTC Creo Elements/Direct Drafting\)](#)
- [Interface to other CAD systems](#)

### 4.3 Driver Installation for Marx USB Hardlock

These steps are necessary if you got a Marx USB hardlock shipped with your PROFIL Full Version. The USB hardlock is running under WINDOWS 98, WINDOWS ME (Aug. 2000 Release), WINDOWS NT 4.0 SP6, WINDOWS 2000, WINDOWS XP, WINDOWS XP/64bit, WINDOWS 7, and WINDOWS 8. Proceed the following steps in the described order:

- Login at WINDOWS as **Administrator**.

- Call (before connecting the Marx USB hardlock!):  
a:\Marx\CBUSetup.exe  
and follow the instructions of the setup procedure.
- Now connect the **Marx USB hardlock** to an **USB port** of the computer.
- Login at WINDOWS under the correct USERNAME, before you start **PROFIL.exe** for the first time (this is why **PROFIL** creates the working path under this USERNAME).

## 4.4 ActiveX Interface to AutoCAD, SolidWorks, SolidEdge, BricsCAD



[Object] The interface to AutoCAD R14, AutoCAD 2000 and higher, to SolidWorks 2003 or higher is integrated into the profil.exe file. If the CAD system is already installed and you install PROFIL afterwards, you do not need to do anything more, PROFIL notices that AutoCAD or SolidWorks is present and enables the interface automatically, if you answer **YES** to the question **AutoCAD/SolidWorks/SolidEdge/BricsCAD found on your computer. Do you want to enable the ActiveX-Interface?**, when you start profil.exe for the first time.

If you want to enable the interface manually, open [Options ActiveX](#):

### Program ID CAD System:

Determine the Program ID of the ActiveX target system. For explanation: The CAD system registers itself to WINDOWS with its Program ID. This information is stored in the system registry of WINDOWS. By this Program ID the connection to PROFIL is made. PROFIL notices which CAD releases are registered and displays them in a drop-down-list, when you click on the arrow symbol.

Open the drop-down-list and select one of the Program ID's:

- **AutoCAD.Application.nn**: Connect to AutoCAD Release nn. Use this Program ID, if you have installed more than one AutoCAD Release and you want to select a special one:  
AutoCAD 14: AutoCAD.Application.14  
AutoCAD 2000: AutoCAD.Application.15  
AutoCAD 2004: AutoCAD.Application.16  
AutoCAD 2005: AutoCAD.Application.16.1  
AutoCAD 2006: AutoCAD.Application.16.2  
AutoCAD 2007: AutoCAD.Application.17  
AutoCAD 2008: AutoCAD.Application.17.1  
AutoCAD 2009: AutoCAD.Application.17.2  
AutoCAD 2010: AutoCAD.Application.18  
AutoCAD 2011: AutoCAD.Application.18.1  
AutoCAD 2012: AutoCAD.Application.18.2  
AutoCAD 2013: AutoCAD.Application.19  
AutoCAD 2014: AutoCAD.Application.19.1 etc.
- **AutoCAD.Application**: Connects to the topical AutoCAD Release, this is the last installed one.
- **SldWorks.Application.nn**: Connect to SolidWorks release nn. Use this Program ID, if you have installed more than one SolidWorks release and you want to select a special one:  
SolidWorks 2003: SldWorks.Application.11  
SolidWorks 2004: SldWorks.Application.12  
SolidWorks 2005: SldWorks.Application.13  
SolidWorks 2006: SldWorks.Application.14  
SolidWorks 2007: SldWorks.Application.15  
SolidWorks 2008: SldWorks.Application.16  
SolidWorks 2009: SldWorks.Application.17  
SolidWorks 2010: SldWorks.Application.18  
SolidWorks 2011: SldWorks.Application.19  
SolidWorks 2012: SldWorks.Application.20



SolidWorks 2013: SldWorks.Application.21  
 SolidWorks 2014: SldWorks.Application.22  
 SolidWorks 2015: SldWorks.Application.23 etc.

- **SldWorks.Application:** Connects to the current SolidWorks release, this is the last installed one.
- **SolidEdge.Application:** Connects to the current SolidEdge release, this is the last installed one.
- **BricscadApp.AcadApplication.nn:** Connects to BricscadApp release nn. Use this Program ID, if you have installed more than one BricscadApp release and you want to select a special one:  
     BricsCAD Release 15: BricscadApp.AcadApplication 15.0
- **BricscadApp.AcadApplication:** Connects to the current BricscadApp release, this is the last installed one.

If the drop-down-list is empty, no AutoCAD or SolidWorks or SolidEdge or BricsCAD release is installed on your computer.

#### Enable ActiveX Input from CAD:

This function enables the ActiveX input and switches the function [Profile, Read CAD-Contour](#) and [Roll Read CAD-Contour](#) and the matching button on the Button Key Bar to ActiveX. At the same time the file input (see Options Files) will be disabled.

#### Enable ActiveX Output to CAD:

This function enables the ActiveX output and switches the function [Output Drawing -> CAD](#) and the matching button on the Button Key Bar to ActiveX. Furthermore the function [3D-Stand -> AutoCAD](#) will be enabled. At the same time the file output (see Options Files) will be disabled.

#### Peculiarity of the SolidWorks interface:

Since PROFIL Rel. 4.0 and SolidWorks 2003 this interface has been provided to transfer profile and roll tools drawings from PROFIL to a SolidWorks drawing and to transfer profile and roll contours from a SolidWorks drawing to PROFIL.

#### Peculiarity of the SolidEdge interface:

Since PROFIL Rel. 4.3 and SolidEdge Rel. ST7 this interface has been provided to transfer profile and roll tools drawings from PROFIL to SolidEdge and to transfer profile and roll contours from SolidEdge to PROFIL, preliminary only for 2D drawings. The extension to 3D models is planned for a later release.

#### Peculiarity of the BricsCAD interface:

Since PROFIL Rel. 4.3 and BricsCAD Rel. 15 this interface has been provided to transfer profile and roll tools drawings from PROFIL to BricsCAD and to transfer profile and roll contours from BricsCAD to PROFIL, preliminary only for 2D drawings. The extension to 3D models is planned for a later release.

## 4.5 Interface to ME10



(previously Hewlett-Packard ME10 and CoCreate OneSpace Drafting, now PTC Creo Elements/Direct Drafting)

After you have performed the [Installation PROFIL](#) and the ME10 installation, proceed these steps to install the interface to ME10:

- Extract file **ProfilMe.zip** in a user defined file exchange path, e.g.  
     c:\profil
- Modify the file **Startup.mac** in the ME10-directory. Open it with a text editor and append a new line at the end of the file. Enter an input call of the file **Customiz** in the profil directory, e.g.  
     input 'c:\profil\customiz'

- Edit the file **Customiz** in the profil directory and check the path name of the file exchange directory.
- Edit the file **Profil.mac** in the profil directory and check the section **Settings**. Enter the correct pathname of the file exchange directory.
- In PROFIL, call [Options Files](#) and select **ME10** and the operating system under which ME10 is running. The CAD Output file must be **Profil.mi** and the Contour Input file must be **Profil.ktr** in the user defined file exchange path.
- Check the data exchange between PROFIL and ME10. Operating instructions for the transfer are in the WINDOWS Help file **Profilme.hlp**.

Remarks: You can also connect PROFIL to the HP-UX version of ME10. Use the TCP/IP protocol for connecting the WINDOWS-PC to the UNIX-Workstation. Define two drive letters, one mapped to the UNIX path of the data exchange (profil.mi and profil.ktr) and the second of the project files (\*.pro). Enter in [Options Files](#) the drive letter and the file name for the CAD output file and the Contour Input file. Take care that in UNIX file names with uppercase and lowercase letters determine different files. While reading, PROFIL converts the UNIX file format (LF) automatically to DOS format (CR/LF).

## 4.6 Interface to other CAD-Systems

You need 2 CAD-macros. They should be created by someone who is experienced in CAD or by your CAD supplier.

**Macro 1** is for transferring the drawings generated by PROFIL to the CAD system. It has the task to load the temporary DXF or IGES file, which is set in [Options Files to CAD](#). Optionally the macro should examine which layers are in the file. If these layers yet exist in CAD, they should be deleted, because modified passes or rolls should be updated in CAD.

**Macro 2** is for transferring pass and roll contours drawn in CAD to PROFIL. If the CAD system is able to save DXF files (normally available in all CAD systems), the macro only needs to save the current drawing into the temporary [Contour File \(DXF Format\)](#) that is set in [Options Files from CAD](#).

If the CAD system is not able to save DXF files, the KTR file format can be used for the transfer. A description of the [Contour File \(KTR Format\)](#) is available from UBEKO.

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