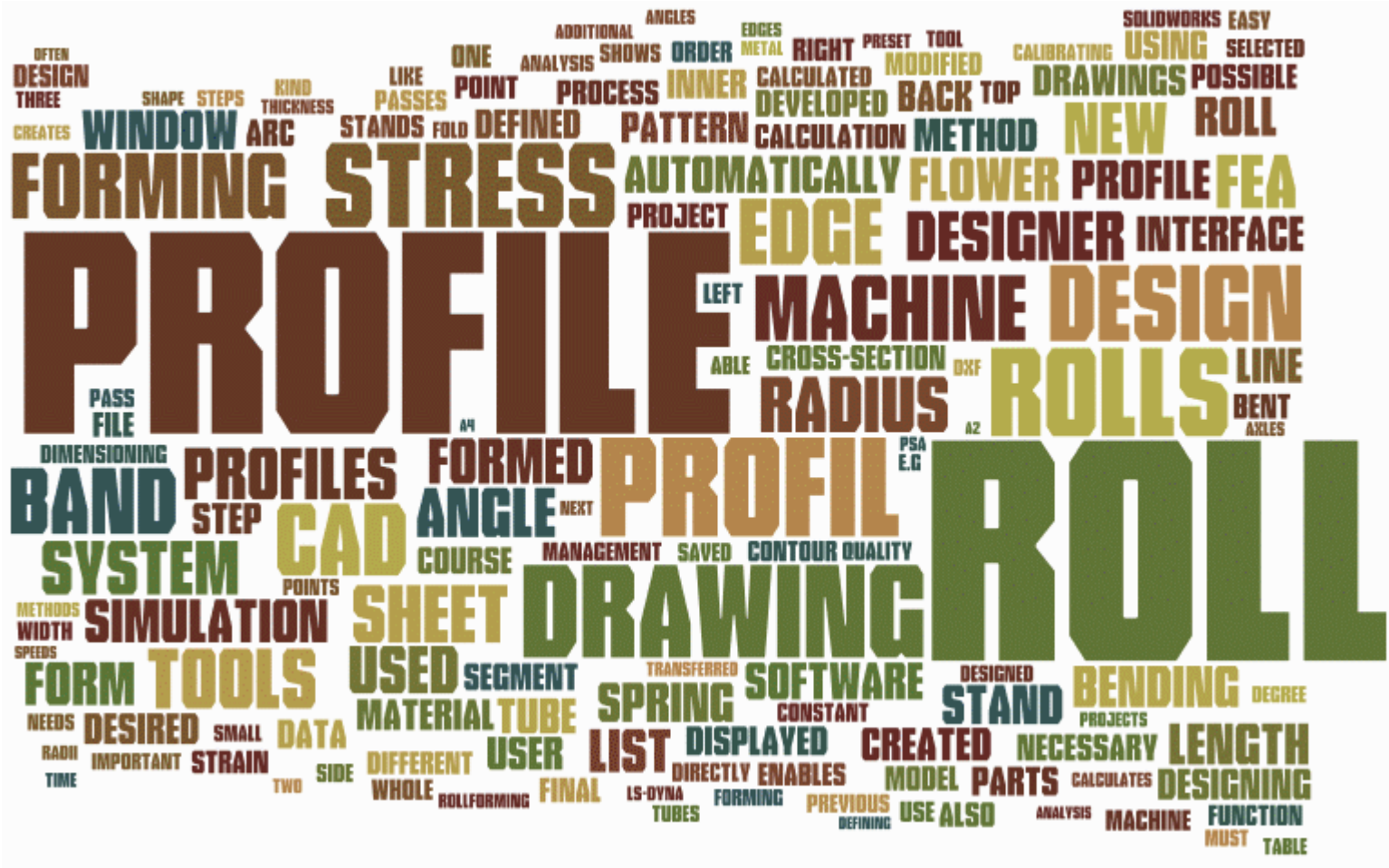
PROFIL Rollform Design Software



User Manual

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Notice

This document is the manual of the software package PROFIL release 6.3. Last revision: December 2023.

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

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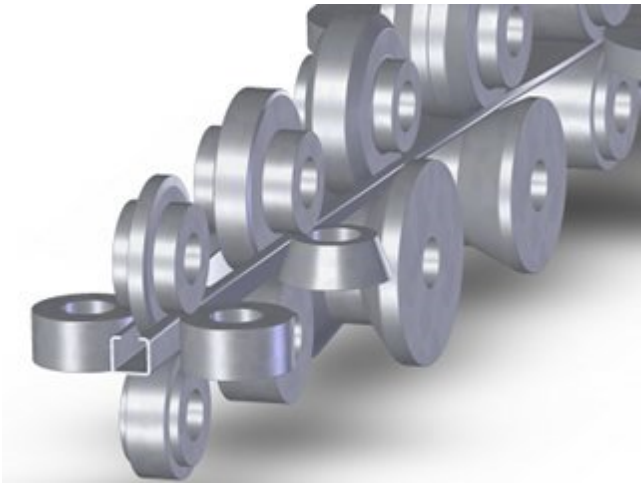
1 General

1.1 Roll Forming



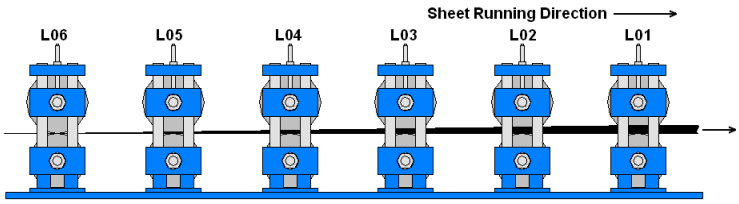
Roll forming

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

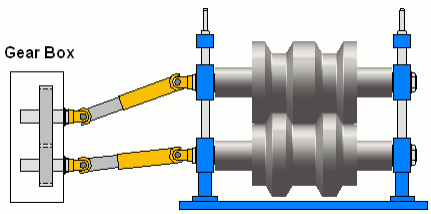
Any kind of shapes can be roll formed, simple and complicated ones. Some examples for roll formed profiles are: U and C channels, struts, door frames, sliding door rails, shutter profiles, trapezoidal profiles, corrugated sheet, screen doors, glass spacer bars, roller blinds profiles, glass shelves profiles, wall and roof cladding, roof bows and trusses, panels, gutters, purlins, fence posts, greenhouse profiles, grape stakes, logistic tracks, drawer slides, studs, beams, scaffolding profiles, 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



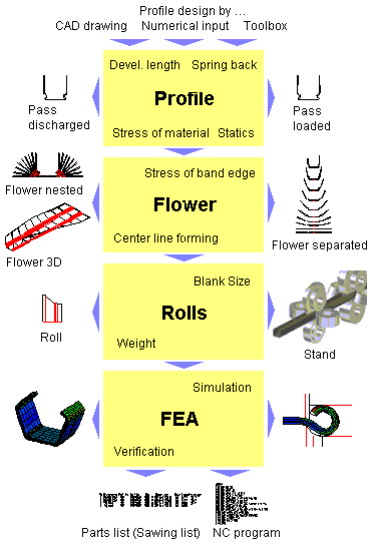
The **roll forming machine** consists of a machine base with a set of **roll forming stands**, arranged in series. Mostly between 6 and 32 stands are needed, dependent on the complexity of the profile.

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.

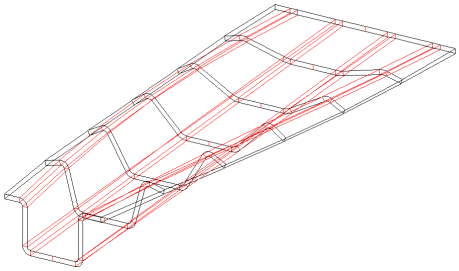
Roll Design



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

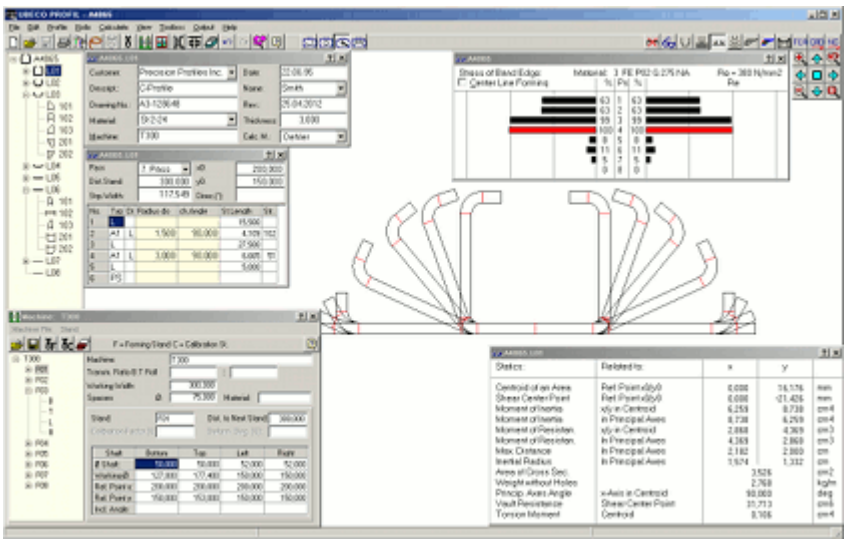
checked approximately by the **Stress of Band Edge Calculation** or by the **Profile Stress Calculation** (**PSA**).
After completion of the roll design, the **FEA** method can be used for a final check. It also helps to evaluate if the designed roll tools are able to form the desired profile with the 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 unsymmetrical, 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 (64bit version) 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, BricsCAD, and ZWCAD 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 600 systems have been sold. Customers in more than 50 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

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 6.x

PROFIL - Rel. 6.3 - 01-Dec-2023

New features:
General:
- FEA-Material Model is used also in the PROFIL basic version.
- New calculation methods for Stress of Band Edge, PSA, Spring Back, Stess on the outer side ofthe bending zone.
- New Material Database with stress-strain curves from tensile tests from **thyssenkrupp Steel®** and **SSAB®** (**DOCOL®**).

FEA Finite element analysis:
- Postprocessing with new evaluation of the strain of band edge..

Roll design:
- Shoulder roll are possible now by Moving rolls.

Calculate:
- Trapezoidal Profile Forming is possible now also with Bending Method A2..A4.

User interface:
- WINDOWS screen scaling and format is considered now.

PROFIL - Rel. 6.2 - 01-Dec-2022

New features:
General:
- Spanish version, for Spain and Latin-American countries.

FEA Finite element analysis:
- Automatic Refining: Since the profile is more deformed in the bend zones and at the band edges, it makes sense to double the count of elements in these critical zones.
- FEA output window more clearly by tree view explorer.
- Improved output progress protocol.
- In case of a long sheet the pass of the flower pattern is shown in all stands whose rolls have contact.
- Meshing preview with explorer and button bar; the sheet length for the preview can be set separately from the origin length.
- Bugfix: The simulation in imperial units is possible now.

Profile design:
- Import DXF and import ActiveX: Extension to closed polylines and extension to AcDb2dPolyline.
- Bugfix: Wrong strip width after changing PS to P. All entries behind P are unconsidered.

CAD interface:
- SolidEdge: Text height for roll and part number is exported, too.

User interface:
- Path and file selection dialogs are changed to Vista layout.
- Bugfix: In case of imperial units the mass texts are not selectable by mouse click.

PROFIL - Rel. 6.1 - 01-Dec-2021

New features:
General:
- Profile, Element, Fillet: Any two line or arc elements are selected that should be rounded out with each other. Elements between the selected ones are removed, especially correction elements that arose by non-tangential connections in a faulty CAD drawing.
- ActiveX interface to ZWCAD: A low-cost CAD system with perpetual license that is popular in Asia. It is fully DWG compatible and has functions and commands like AutoCAD. The comfortable interface enables creating profile and roll drawings in the just opened CAD document and imports user drawn profile and roll contours from CAD.
- View Roll Tools: Sometimes, if the roll and part numbers are long and the rolls are small, the numbers obscure each other. Now long numbers are rotated automatically in order to prevent obscuring. The Rotation Angle can be preset.

FEA Finite element analysis:
- After finishing the FEA simulation of the profile project a new Python Scipt shows important FEA results as a graphic automatically, e.g. the forces of the top shafts of each machine stand. The forces are important for configuring the shaft diameter, bearings, cardan shafts, gear box, and the drive power. The script easily can be adapted to individual user's needs.
- Stress-Strain Curve as analytic function: If the designer quickly wants to validate his project by FEA and the exact material data are not available, it makes sense to use an approximation. A new feature is to use an analytic function in the integrated Curve Generator.
- Stress-Strain Curve Generator: Input fields can be modified by PgUp/Dn keys.
- Desired final pass L01 is shown in the sheet center after calculating Spring Back.

Bugfixes profile design:
- Leaving the input field Strip Width is not possible.
- Missing input fields in all Toolboxes (Korean version only).
- Missing update of the Stress of Band Edge Window after toggling loaded-discharged (F9).
- TheStress of Band Edge Window has to be reopened after Undo/Redo. and Open Project.

Bugfixes roll design:
- Number Key Sequence can be preset for Variable $SA.
- Program loop if a superfluous corner point has a radius.
- Plausibility Check, inaccuracy in case of large radii.
- Overlap between arcs in case of a very small contour kink.
- Number Key is not considered if it differs from the previous.

Bugfixes machine:
- Missing user request if a Machine Stand is removed.

Bugfixes CAD interface:
- STEP Output and others: Checking improper characters and double layer names.
- Output via ActiveX to BricsCAD, wrong text rotation angle.

Bugfixes parts list:
- Run time error if no working sheet is opened in Excel.

Bugfixes user interface:
- Print Button, function changed to Print Preview.

PROFIL - Rel. 6.0 - 01-Dec-2020

New features:
General:
- 64bit version for WINDOWS 7,8,10 enables efficient handling especially for larger projects during output.
- Roll Tool Window, Print Preview, Create Parts List, Create NC: Real diameter instead of intersection diameter of the tangents.
- Profile List: Leading zeros are displayed after entering ## in the field Pass.
- Export and all other **Save** functions: User request if file already exists.

FEA Finite element analysis:
- New menu item Advanced for expert settings.
- Contacts are active only, if the sheet is near or between the rolls. This speeds up the simulation time.
- Start Position in front of the stand: Calculation dependent on the real roll radius instead of the radius at the intersection point of the tangents.
- Guideways are saved in the project file. If Restart is selected, the saved guideways are taken instead of calculating them newly dependent on the roll diameters. Thus restart is possible after modifying roll diameters.
- Improved Automatic Roll Meshing.
- New button FEA Output in the Button Key Bar.

Roll Stock Management / Profile Catalogue:
- New database format SQL 64bit.
- Database converter for the transfer of old roll and project data to the new SQL database.

Bugfixes general:
- Roll Numbering for more than 10 rolls works correctly now.
- Stress of Band Edge/PSA: No error anymore for empty profile lists.
- Print Preview: Correct form feed for NC output now.
- Roll Stock Management: No negative roll width anymore if rolls are defined from the right to the left.
- Double pass names check works correctly now.
- Short Keys are saved to the ini file correctly.
- 3D-Model -> CAD now also appears in the Short Keys selection box.
- ACAD ActiveX Import: No abort anymore if the CAD drawing contains more than one polyline.

Bugfixes FEA Output:
- No sporadic DEATH < ENDTIM anymore.
- The profile does not stick to the top roll anymore.
- In case of guiding the birth contact start is not too late anymore, now it always starts at the largest radius.
- Profile Meshing now works correctly in case of succeeding arcs with the same length,
- No double curve point anymore in the velocity curve for a small distance between stands.
- No run-time error anymore during FEA output, if a stand contains no rolls.
- No wrong z meshing anymore after opening old projects,
- In the Curve Generator min. allowed fracture strain is now 0.01.
- In a symmetric project, the rolls now are exported to FEA correctly if they are defined from the right to the left.

1.10 What's new - Releases 5.x

PROFIL - Rel. 5.7 - 01-Dec-2019

New features:
General:
- 8 Training videos for roll form design available.
- Machine training slides available.
- Button Repeat Last Operation.

FEA Finite element analysis:
- Checking project name and simulation path.
- Direct start of the FEA Solver and LS-Run.
- Change to MPP and mpiexec for parallel jobs.
- Enter key now works like TAB key. This avoids unintended closing of the window.
- New button for Saving (export) the Material Data.
- Pass name and roll number are shown in the Meshing Preview Window.
- Solid model: Null shells on each sheet side and at the lead and tail end.
- A traffic light remembers if the Plausibility Check is not yet proceeded.
- Roll files (.mod) now contain 1 stand only, thus quicker output.

Profile design:
- The Sheet Thickness limit is increased from 10mm to 20mm.

Tube design:
- Automatic round tube forming.

NC output:
- Optionally G02/G03 Commands with radius R instead of I/K parameters for the distance to the arc center point.
- NC Output in Fortran notation, e.g. 4. instead of 4

CAD interfacing:
- SolidEdge: 3D-ActiveX-Output.

Bugfixes:
- The Lankford parameters now are saved correctly.
- The Contour Tracking does not change the profile side anymore.
- STEP Output is always metric, also in an imperial project.
- No runtime error anymore during Append Profile List in case the file name contains a dot.
- No runtime error anymore during Copy to Clipboard.
- Error handling if Profiltr.dll is missing.
- Calculator results in the Profile Header Window now are copied to the input field correctly.
- Toolbox Tube Forming updates the explorer correctly.
- Output FEA LS-Dyna: Now correct part name for the planned pass cross-section.

PROFIL - Rel. 5.6 - 01-Dec-2018

New features:
FEA Finite element analysis:
- Output FEA LS-Dyna: The corresponding training slides to each parameter tab can be displayed and help understanding during input.
- Output, FEA, LS-Dyna, Rolls, **Enable Plan vs. actual analysis**: The designed passes are shown in LS-PrePost between the rolls for comparing.
- Output, FEA, LS-Dyna, Profile, **Preview**: While entering the FEA meshing parameter the meshed sheet is shown, either flat or in any pass.

- Output, FEA, LS-Dyna, Rolls, **Preview**: While entering the FEA meshing parameter the meshed rolls are shown.
- Output, FEA, LS-Dyna, Material: Material properties are saved into the **project file**. Henceforth the MAT-file is only needed for data transfer.
- Output, FEA, LS-Dyna, Rolls, **Visible rolls**: The roll file only contains rolls that are in contact with the sheet.
- Output FEA LS-Dyna: Button **Save Project**.
- Output, FEA, LS-Dyna, Start, **With simulation result (Restart)**: The stand number is asked to confirm the correct assignment.

Profile design:
- Calculate Stress of Edge: The user can toggle between **band edge only** and **within the whole profile cross-section**.
- Calculate, Plausibility Check: Extension: The first profile list in sheet running direction does not contain the flat sheet or contains rolls.
- Calculate, Plausibility Check: Extension: Mixed symmetrical and unsymmetrical profile lists are detected.

Tube design:
- Profile, Calibrate Strip Width, if the strip width is reduced in fin passes.
- Machine Window: **Lead** (strip tension) as menu function.
- Toolbox Tube Design: Modifying the values in the input fields step by step with the PgUp/Dn keys and simultaneous preview in the drawing area.
- Toolbox Tube Design: Fin pass and break down pass consider the calibration factor.

CAD interfacing:
- Settings ActiveX: Input entities from AutoCAD are extended to **BLOCKS** and **POLYLINES**.
- Settings Files: Input entities from DXF file are extended to **BLOCKS** and **POLYLINES**.

General:
- Settings Keyboard: **Delete** and **Insert** can be selected as shortcuts.

Bugfixes:
- Output FEA, LS-Dyna, Guiding: Determining the reference point for guiding now is more safe.
- Machine Window: Appending the first stand after installation and initial start is possible now.
- Output, FEA, LS-Dyna, Start: The input of the stand number after user query now is interpreted correctly.
- Output FEA LS-Dyna: If the profile is symmetrical, the CONTACTS to the rolls on the left side are not exported anymore.
- Settings Keyboard: User defined shortcuts are not mixed anymore.
- Output FEA LS-Dyna: Default values meshing, Young's modulus, and density now are calculated from Metric to Imperial correctly.
- Output FEA LS-Dyna: No division by 0 anymore in case of very small arcs.
- Output FEA LS-Dyna: The passes of the flower pattern for the plan vs. actual analysis now have the origin strip width for the simulation model Shell.

PROFIL - Rel. 5.5 - 01-Dec-2017

New features:

FEA Finite element analysis:
- Output, FEA, LS-Dyna, Start: Start with preformed profile.
- Output, FEA, LS-Dyna, Others: Avoiding waves at the tail end.
- Automatic profile and roll meshing.
- Output, FEA, LS-Dyna, Start: Overdrive speed factor for speed-up the sheet motion between the stands can be preset by the user.
- Output, FEA, LS-Dyna, Start: Spring back calculation by using the implicit solver is supported.

- Output, FEA, LS-Dyna, Others: Continuous treatment of thickness and plastic strain (IRCQ).
- Output, FEA, LS-Dyna: FEA file names are built from project name + explorer pass name.
- Output, FEA, LS-Dyna, Start: $ variables for the FEA project name are supported.
- Output, FEA, LS-Dyna, Rolls: Roll name can be assembled by new $ variables.
- Output, FEA, LS-Dyna, Material: Material files from the LS-PrePost MatLib (.k files) can be imported.
- Output, FEA, LS-Dyna, Material: Lankford coefficients of the plastic anisotropy of the rolled precursor sheet metal can be considered.
- Output, FEA, LS-Dyna, Contact: Welding of the sheet edges.

Roll stock management:
- Stock Management Search: Improved roll searching and replacing in the project by rolls from the stock management.
- Stock Management Save: Automatic check of existing similar rolls before saving a roll to the roll stock management

Roll design:
- Window Read CAD-Contour/ Scan Profile Drawing: Scan a profile contour and read contour from CAD: Context menu with snap points also for the start and end point.
- Roll, Split between Corners: Split roll at the quad point of an arc and at any arc point by angle input.

Machine:
- Machine window: Copy stand to the clipboard and replace stand by the clipboard content.
- Profile, Remove: When removing a pass, also the corresponding stand is removed after user query.
- Profile, Insert/Append: When appending/inserting a pass, a new stand will be created after user query. The data are taken from the previous stand, the next stand, or the stand in the clipboard.

General:
- View, Flower, separated: View flower separated: In addition to automatic vertical distance also constant distance, preset in Settings, Drawing.
- **PROFIL** Start with Open Project by mouse cursor drag and drop of a project file on the **PROFIL** desktop icon.

Bugfixes:
- Output, FEA, LS-Dyna: Memory error at LWPOLYLINE with Color=ByLayer.
- Output, FEA, LS-Dyna, Guiding: Rounding error at last time entry in .bnd.
- Output, FEA, LS-Dyna, Rolls: Roll flanks now are correct in case of large bore whole diameter.
- Output, FEA, LS-Dyna, Rolls: Partial rolls now are correct in case of odd ratio roll angle/segments count.
- Output, FEA, LS-Dyna: No program break anymore in case of missing roll flank.
- Output, FEA, LS-Dyna: No program break anymore during FEA output in case the profile list starts with a P.
- Output, FEA, LS-Dyna, Start: Start with preformed profile now works with solid models, too.
- Calculate Stress of Edge: The shown stress now is zero for the flat sheet.
- Output, FEA, LS-Dyna, Start: Guiding error solved in case of start with preformed profile.
- Output, FEA, LS-Dyna, Guiding: No chamfers anymore at the symmetry edge, improved meshing.
- Output, FEA, LS-Dyna, Profile: No small split elements anymore at automatic meshing.
- Output, FEA, LS-Dyna, Guiding: Guiding now for 180° arcs possible, too.
- Output, FEA, LS-Dyna: Considers contact conditions correctly at different distances between stands.
- Multiple Variables in a term now are handled correctly.
- Output, FEA, LS-Dyna, Start: The output files does not contain unnecessary contacts, parts, and bodies in case of start with preformed profile.
- View, Flower, separated: The left profile side now is also considered for the vertical distance.
- Profile Explorer Click now also refreshes the Machine Window, if the count of stands is larger than the count of passes.

PROFIL - Rel. 5.4 - 01-Dec-2016

New features:

- Separate calculation method of the Developed Length for each arc element.
- Dialog window for parameterization of User Defined Calculation Methods with function view as a graph.
- Extended Mirror function for rolls.
- Increased count of Profile Elements and Roll Corner Points, whereupon also high complicated profiles can be processed.
- Output, FEA, LS-Dyna: Threading the profile into the roller stand by chamfering the lead end in width and in thickness direction and by reducing the speed.
- Output, FEA, LS-Dyna, Profile: Automatic Meshing.
- Output, FEA, LS-Dyna: Automatic mesh refining for shells with aspect ratio > 4.
- Output, FEA, LS-Dyna, Rolls: Partial rolls instead of full rolls in order to optimize the simulation time.
- Extended Plausibility Check.

Bugfixes:

- File, Plot: The CAD button is visible again, also in case of large fonts are selected.
- View, Flower, 3D: Arcs now are displayed correctly again.
- Rolls: Side rolls can be turned now.
- Dimensioning: Split side rolls now can be dimensioned, too.
- Machine window: The current stand does not changes anymore, if more stands are parameterized than profile lists exist.
- Output, FEA, LS-Dyna: No solver stop anymore because of too small time steps in the guiding curves.
- Output, FEA, LS-Dyna: Guiding curves now start at the correct blank position.
- Output, FEA, LS-Dyna: No sporadic time back-step anymore in the guiding curves.
- Output, FEA, LS-Dyna, Material: Modifying the Young's modulus now takes effect on the stress-strain curve immediately.
- Output, FEA, LS-Dyna: Bore holes in the sheet, defined as circles, are possible again.

End of Support:

- The FEA System **ABAQUS** is not supported anymore by **PROFIL**..

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.
- Settings ActiveX: New ActiveX interface to SolidEdge and BricsCAD.
- Settings, 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.
- Calculate, Stress of Band Edge now calculates in the sheet center, not on the bottom side anymore.
- Output Drawing -> CAD via ActiveX to SolidWorks: Zero dimensions are suppressed now.
- Roll, Arched Extension with negative preset now is also possible in case the arc merges to the roll flank tangentially.
- The sign of the Angle Dimension and Diameter Dimension now is shown correctly under the Chinese and Korean Windows.

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:

- Importing a stress-strain-curve from a text file, e.g. from a tensile test, in Output, FEA.
- Settings, 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 Settings, Drawing and Settings, 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 Settings, Rolls or Settings, 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 Settings, 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 Settings, 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.

1.11 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 Mode 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:

- Settings, 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.
- 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 Settings, Calculate.
- Mirror Roll: Roll No. and Part No. are kept is this is preset in Settings, Rolls.
- Roll Stock Management: New import function.

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.

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.
- 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 Settings 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 Settings 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 Settings Keyboard.
- Parts list can be created not only in a fixed output path, but also in the path of the belonging project file, see Settings 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.

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 Settings 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-Contournow 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. Settings Explorer for clear selection of 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 Settings 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 Settings ActiveX.
- Track Holding Method A4 with constant intersection point on the inside or outside as selected (preset in Settings Profile List).
- Calculation Method DIN 6935 of the developed length by table or formula as selected (preset in Settings 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.

1.12 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 Settings Drawing).
- Automatic adaptation of the rolls when changing the working diameter or the reference point (Setting in Settings 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 Settings Drawing).
- Decimal places of the co-ordinates of the NC program can be preset (Setting in Settings NC).

Bugfixes:

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

1.13 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 “Settings, Rolls” previously.

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 "Settings, Parts List, Set-Up Columns, Designation").

Q: Although I checked "Spacers" in Settings, 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 "Settings, Files" is disabled. Why?

A: You have enabled ActiveX. Uncheck the boxes in "Settings, ActiveX". Then you can set-up the file interface.

Q: In "Settings, 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 "Settings, 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 loosing 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 (Settings, 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 "Settings, 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". Alternatively, you also can use the functions "File, Partial Project, Add on/Save as"

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 flattened 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.

Q: 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).

Q: 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 Settings, General.

Q: I got a DXF file from a customer and cannot import it. Why?

A: Autodesk modifies the file format occasionally, because DXF is not a standard format. For data exchange, it is a good idea to save the file from CAD in an older format, e.g. "Save As.." "AutoCAD 2004 DXF". PROFIL will import it without any problems.

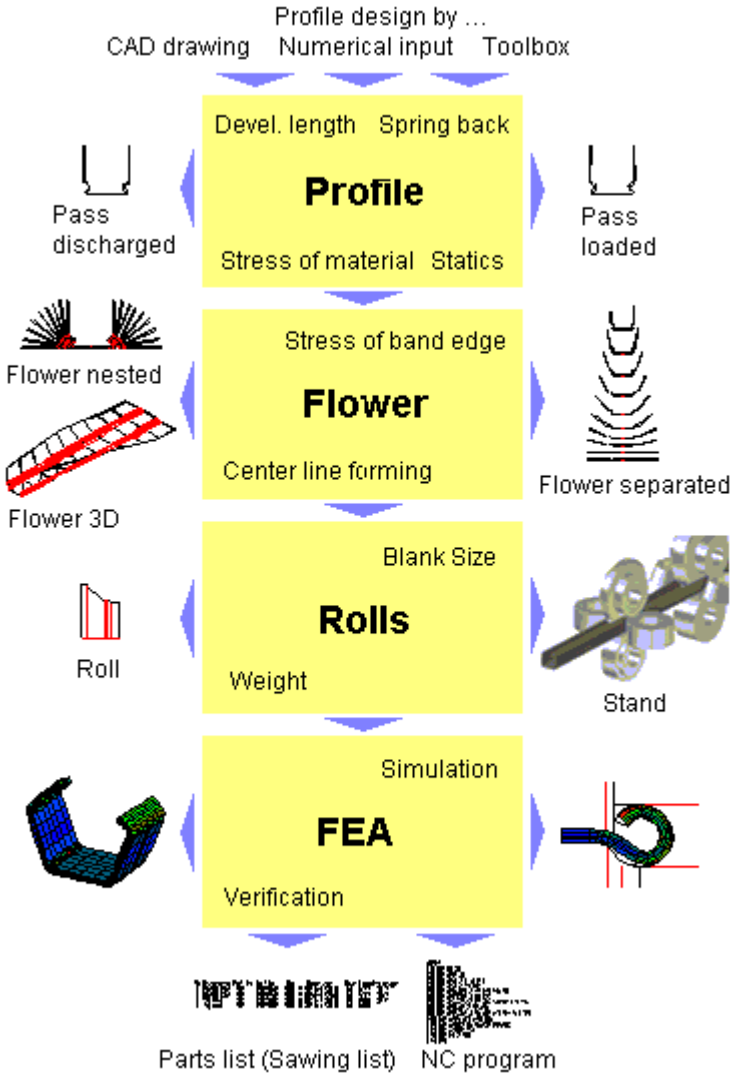
Q: Because the text height is too small on a high resolution screen, I modified the text size scaling in Windows. Now some PROFIL windows do not show the full content anymore. Why?

A: This is a well known WINDOWS problem. Microsoft recommends this workaround:

- In the Explorer, right-click on **Profil.exe** shortcut and select **Properties**.
- Select tab **Compatibility**.
- Dependent of the user rights:
- For local Admin rights: First call **Change settings for all users** and then select **Change high DPI settings**. If the user only has local rights click directly onto **Change high DPI settings**.
- In the next window select **Use this setting to fix scaling problems..** and **Override high DPI scaling behavior...**.
- Try both **System** and **Application.**
- After that, click **Ok** to save the changes and start PROFIL.

2 Tutorial

2.1 How to Work



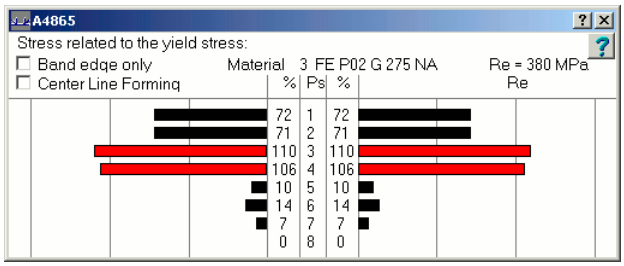
To design the roll tools for an open, cold rolled profile, proceed the following steps:

- 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:

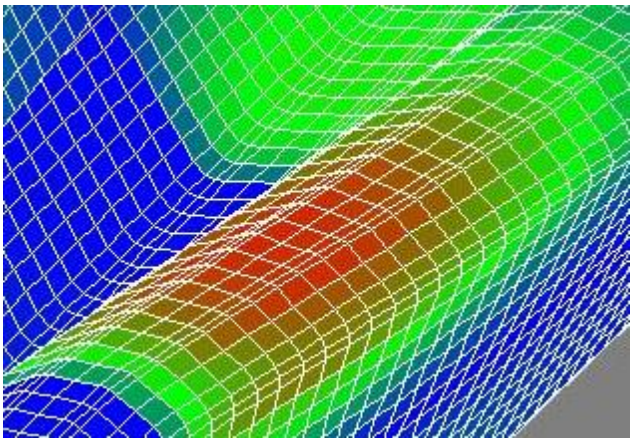


Stress of band edge as a bar diagram

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

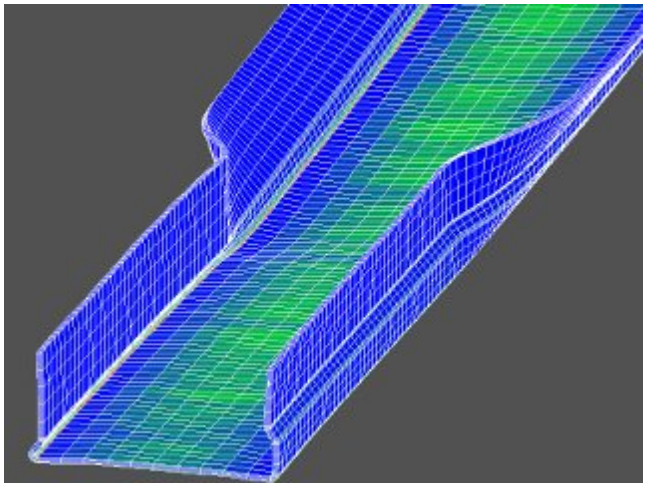


PSA - Profile-Stress-Analysis, shown in the PROFIL drawing area

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, shown in the PROFIL drawing area

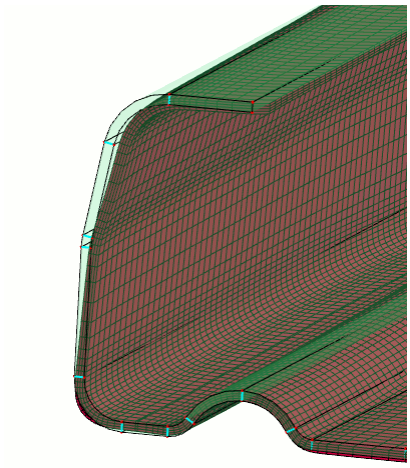
Step 3: FEA - Finite-Element-Analysis

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.**
For target/actual comparison the profile contour from the profile list is overlaid in light green

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.

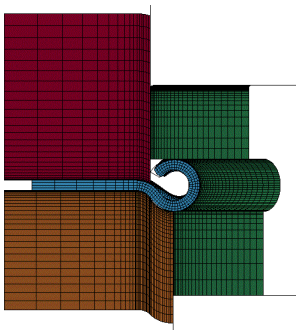
To proceed this very precise calculation of the stress and the longitudinal deformation within the whole profile, an interface has been provided to the leading FEA systems **LS-DYNA** from Livermore Software Technology Corp.

The necessary steps are:

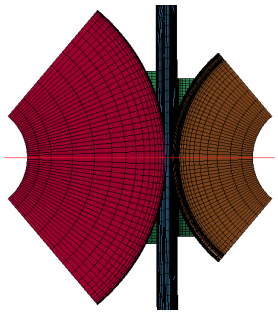
- Design the profile, the flower pattern and the roll tools, afterwards create the interface files by using the function Output FEA LS-Dyna.
- Proceed the FEA-simulation by using **LS-DYNA Solver**.
- Check the result of the simulation with **LS-PrePost** (see picture).

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.

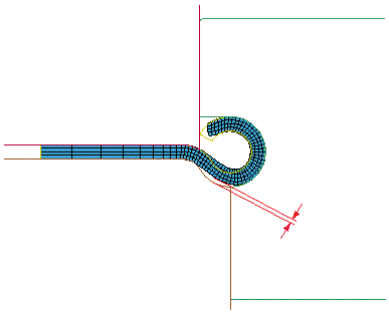
Enable plan vs. actual analysis
Prerequisite: You have enabled **Show designed profile pass from flower pattern between rolls** in Output, FEA, LS-Dyna, Rolls. The designed passes are shown in **LS-PrePost** during evaluating the simulation result between the rolls without having influence on the result (i.e. contacts are not defined). The part names are the names of the passes or stands that are shown in the Explorer in design or in rolling direction. This simply enables plan vs. actual analysis (planned contour = pass contour as designed, actual contour = FEA result).



Show one stand, the blank, and the profile pass



Click onto any roll point on the stand centerline

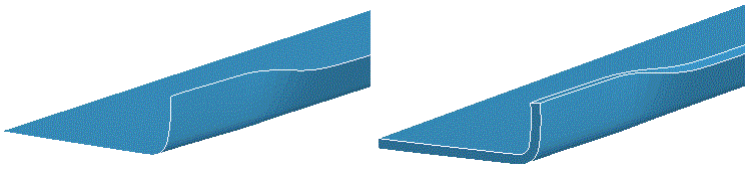


Check the deviation between designed pass and FEA result

After running the **LS-Dyna-Solver** proceed in that way:

- Start **LS-PrePost**
- **File, Open, LS-DYNA Binary Plot** --> Select .d3plot file
- Right button bar: **Model, SelPart** --> Show one stand, the blank, and the profile pass
- Move the sheet into the stand by using the arrow buttons
- Bottom button bar: **View, Front**
- Right button bar: **Model, Section, Model, BaseId, KeepSection** --> Click onto any roll point on the stand centerline --> **Cut**
- Bottom button bar: **View, Top** --> Check the deviation between designed pass and FEA result

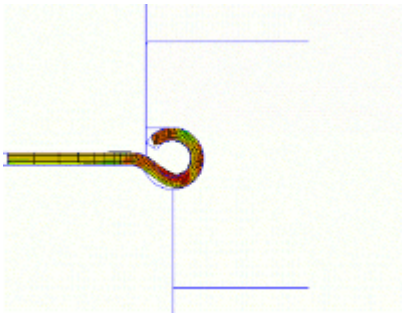
Sheet thickness for the shell model:



After opening a d3plot file of a shell model, you first see the thin shells in the sheet centerline (left). In order to see the sheet thickness (right) proceed in that way:

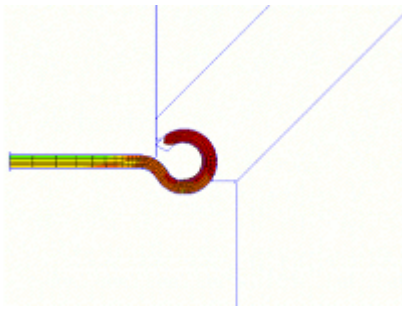
- Call **Model, Appear**
- Select **Thick**
- Hit **AllVis**

Optimizing the roll tools



Faulty profile

After cutting and rotating to the 2D view the planned contour (from the flower pattern) can be compared with the actual contour (from the FEA simulation). The example shows how a faulty roll tool set forms a profile that has not the desired circular cross section at the band edge. The designer can see clearly that the side roll with vertical axis raises the curved profil bottom. The simulation gives valuable information which rolls of which stand have to be redesigned.

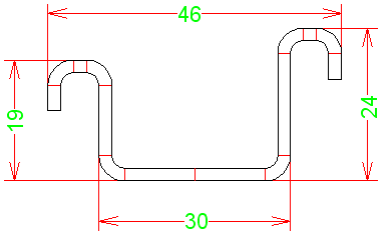


Error-free profile

After redesigning the rolls and restart the simulation (by using the time-saving Restart Function) the profile has the desired circular cross section at the band edge. Side rolls with increasing inclination angle prevent the raising effect of the curved profile bottom. Furthermore the modified bending sequence improves the profile form. Now the rolls can be manufactured without producing scrap.

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 Mode.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.

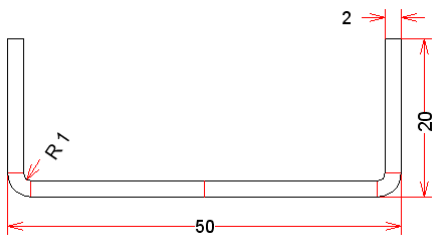
Hints:
- After designing the cross-section of the final profile, your profile project contains a profile list with the name **L01**. This name is shown in the Explorer.
- The next step is Designing the Flower Pattern. By doing this, the profile lists **L02**, **L03**, etc. are created. The last profile 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

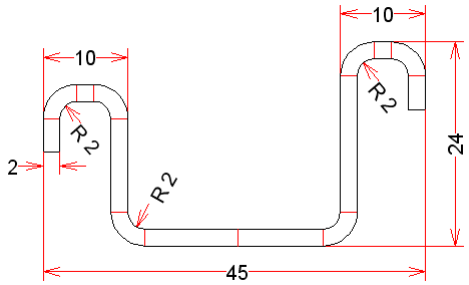


A profile U 50x20x2 (50 wide, 20 high, 2mm sheet thickness, 1mm inner radius) is described by the following input into the profile list:

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

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

Example 2

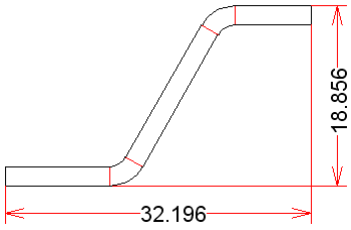


An unsymmetrical hat profile 45x24x2 (45 wide, 24 high, 2mm sheet thickness, 2mm inner radius) is described by the following input into the profile list:

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 Settings 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:

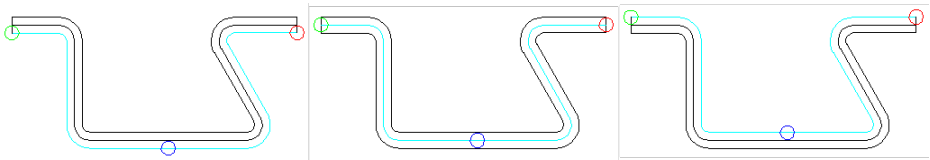
- If you have made a mistake in entering an arc, you must not modify the angle or the radius of the arc. By doing so you would bend up the arc with constant straight length. For correction toggle to the Draft Mode, which enables to modify angle and radius without bending the arc.
- By using the numerical 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 unsymmetrical 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.

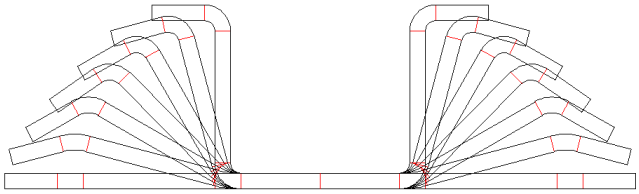
To search for a similar project, proceed as follows: Open the Profile Catalogue and create a Filter with the desired filter conditions, e.g. outer dimensions or classification key. If a suitable filter already exists, you only need to select it from the drop-down-list. See Filter for more info.

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 Mode 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 (in designing direction, 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.

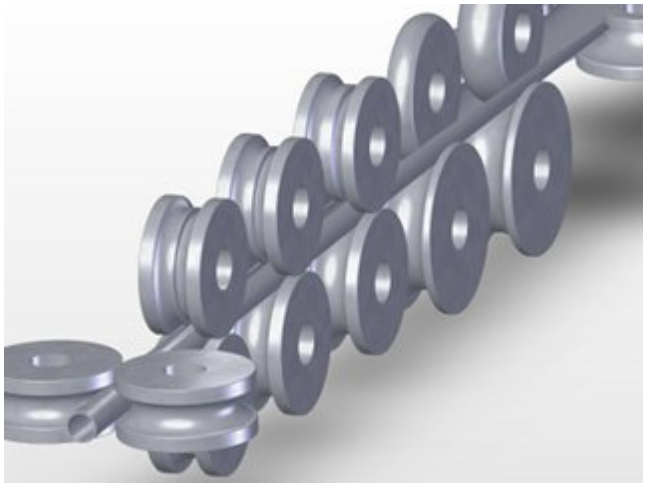
The function File Print prints out the drawing, the profile list, the table of statics and the bar diagram of the stress of band edge.

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 be 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.

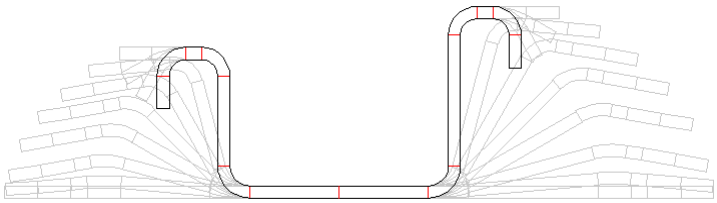
Before using the Toolbox Tube Design it is recommended to prepare the machine data in the Machine Window.

Hints:

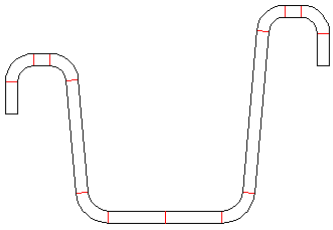
- In certain input fields the values can be modified gradually by pressing the **Pg Up/Dn** keys on the keyboard. The step interval can be preset in Options, Mouse.
- During input a preview is shown in the Drawing Area. The preview disappears in case of pressing the **Cancel** button. After pressing the **Ok** button, the preview remains valid in the project.

2.5.3 Automatic Flower Creation

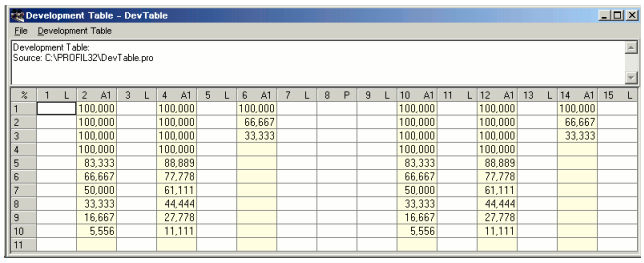
If the flower design of a similar profile already exists and a new profile should be developed by the same scheme, creating the flower pattern is quite simple.



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.



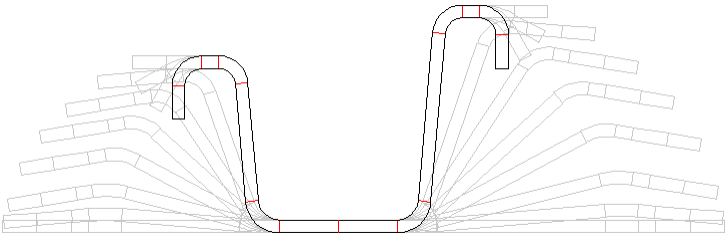
Development Table – DevTable

File Development Table

Development Table:
Source: C:\PROFIL32\DevTable.pro

%	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		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															

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.

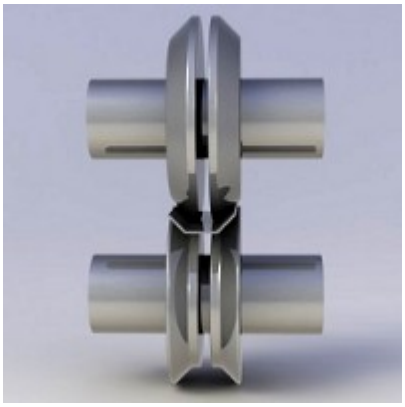
Similar means: the profile list has the same profile element sequence regarding line and arc types. If it is not, you can move certain columns (function **Development Table, Column**) or you can create and edit a new development table manually.

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



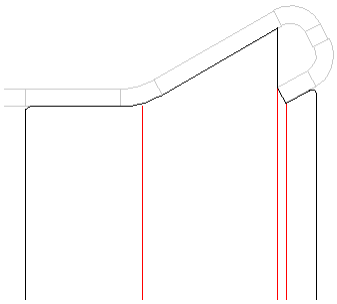
Select among 3 various methods, automatically, with the help of the CAD-system or by re-using of existing rolls from the roll stock management:

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.

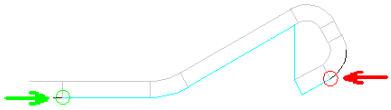


The profile drawing displayed in the Drawing Area is scanned. Only the contour of the profile and other yet existing rolls is considered that is visible from the shaft. A roll is created that touches the complete visible contour.

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.

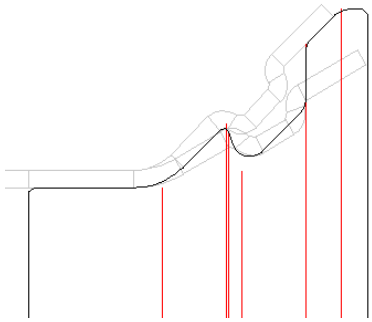
Select the drawing of the rolls by using function View Rolls. Now the center lines of the shafts will be displayed. Select one of the center lines for defining the type of roll. Afterwards call function Roll Scan Profile Drawing.



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

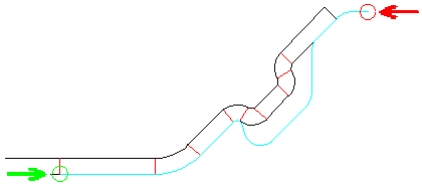
First draw the desired roll contour inside your CAD system. It is very useful, if you transfer the drawing of the profile pass to the CAD system (function Drawing -> CAD) if you use the contour of the pass for your drawing.

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 filter the rolls:

- 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. Call Roll, Roll Stock Management, Search and select in the next window which **Search criteria** should be used. Afterwards the Roll Table only shows rolls that are similar to the desired roll.
- 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. When 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. **Replace roll in project by roll from roll stock**
copies the selected roll from the database to the project, deletes the selected roll in the project and replaces it by the roll from the database.

2.6.5 Adapting the Roll Tools

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

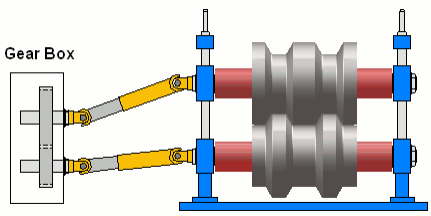
Conical Extension	Cylindrical Extension
Arched Extension	Split at Corner
Split between Corners	Join
Turn	Move
Mirror	
Cut	Copy
Paste	Delete

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.

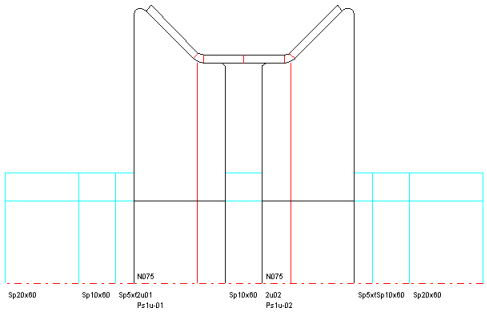
A Clearance Angle is used to reduce wear out of rolls, if the rolls have different circumference speeds because of the different heights of the legs of the profile. By using the function Gap the clearance angle can be continued in the successive profile segments.

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 that 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 Settings, 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 dependent on special number keys for spacer rolls only in Settings, 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 Settings, Spacer Rolls, the spacer rolls are split automatically dependent 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

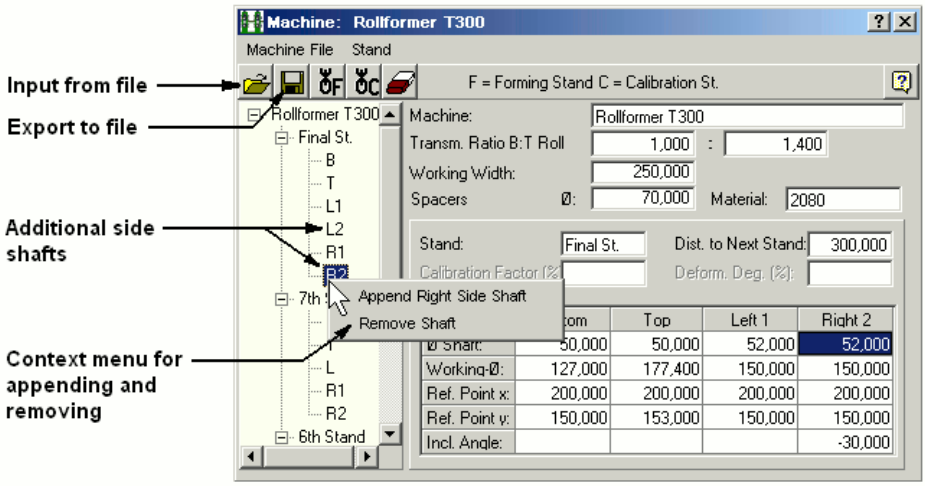
Create a new record in the Roll Table of the Roll Stock Management by using the button **Insert Roll** of the database navigator and enter the roll data into the record. Use the button **Post edit** to save the record. While the new roll is still active, for every corner or the roll create a new record 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.
.

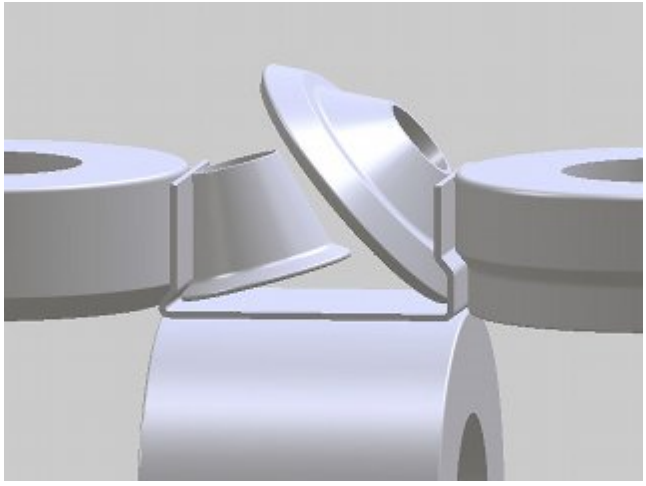
Working Instructions:

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 affect 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 affect also for the selected shaft. Because the machine data are handled within the project, also the undo and redo functions are available.

Interactive operation:

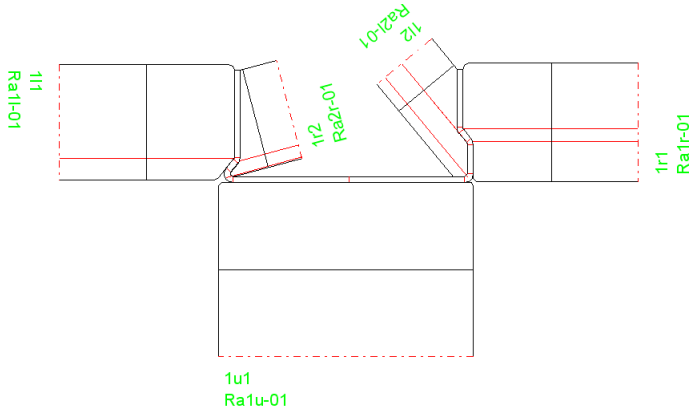
The machine window works interactively with the drawing area (similar to the profile list and roll tool window). This means the effect of data modifications are shown in the drawing immediately. Vice versa, if the user selects or modifies drawing entities, the content of the data windows is updated.

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° = 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 (Settings, 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.

2.8 Training Videos

Lesson 1 - Defining a simple U profile

You learn how to define a simple U profile, symmetric and unsymmetrical, by using the Toolbox Profile Design. Then you learn to understand the structure of the profile list, the reference point, and the meaning of the different profile elements.

Lesson 2 - Defining a complicated profile

You learn how to import a complicated profile, symmetric and unsymmetrical, by using the Read CAD-Contour function. You learn how to detect a faulty CAD drawings and what to do in case of import errors.

Lesson 3 - Creating the flower pattern

You learn how to create the flower pattern for a simple U profile by manual specification of the bending sequence. You see how to consider the Stress of Band Edge during design and what to do if the stress exceeds the yield stress of the material. Then you get to know the different Bending Methods: Constant developed length method, constant radius methods, track holding method and a method for compensating the spring back. At the end you see how to transfer the designer's know-how to a new project and how to create a flower pattern automatically by using the Development Table.

Lesson 4 - Defining machine properties

You learn how a roll forming machine and a tube forming machine is defined in the Machine Window. The difference between forming stands and calibration stands is shown. You learn to to set-up the shaft and axles properties.

Lesson 5 - Creating roll tools

You learn how to create rolls, first the very quick method by Scanning the Profile Contour. For special needing it is necessary to define a roll contour that differs from the profile contour. It is shown how to Create it in CAD and import it.

Lesson 6 - Verifying roll design by FEA

You learn how to verify the roll design by FEA - Finite Element Analysis. The FEA system LS-Dyna is a suitable instrument to check if the designed rolls are able to produce the desired profile with the requested allowances. The handling procedure is shown for the example bumper project.

Lesson 7 - Export CAD drawings and manufacturing data

You learn how to transfer drawings and 3D models to CAD. Export parts list and sawing list respectively for cutting the roll blanks from the bar is shown. You learn to know the three options to export milling data: G codes to the machine control, DXF files to a CAD/CAM system, and print output.

Lesson 8 - Designing shaped and round tubes

You learn how to use the automatic Shaped Tube Calibration for tubes that should be formed from from a round welded tube. Then you see the automatic Round Tube Forming, the development of a tube flower pattern, and how to modify the passes manually. Afterwards the creation of roll tools for breakdown passes, fin passes and calibration stands by using the Toolbox Tube Design is shown.

(Will be continued soon)

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 Settings, 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 Settings, 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..

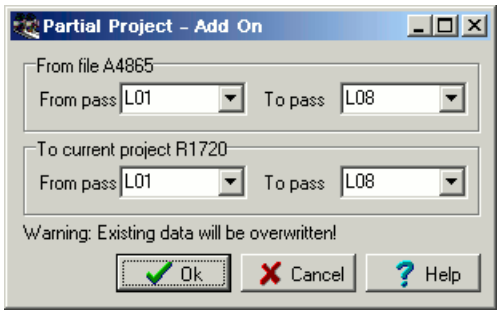
3.1.1.5 Partial Project Add On

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.

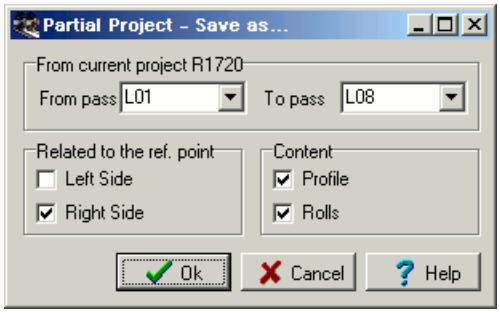
- If the added project contains rolls, they will be locked against automatic renumbering. By using the function Roll Renumber the rolls can be unlocked.

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 Settings, Files). 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 Settings, 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.

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 Settings, Files). 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

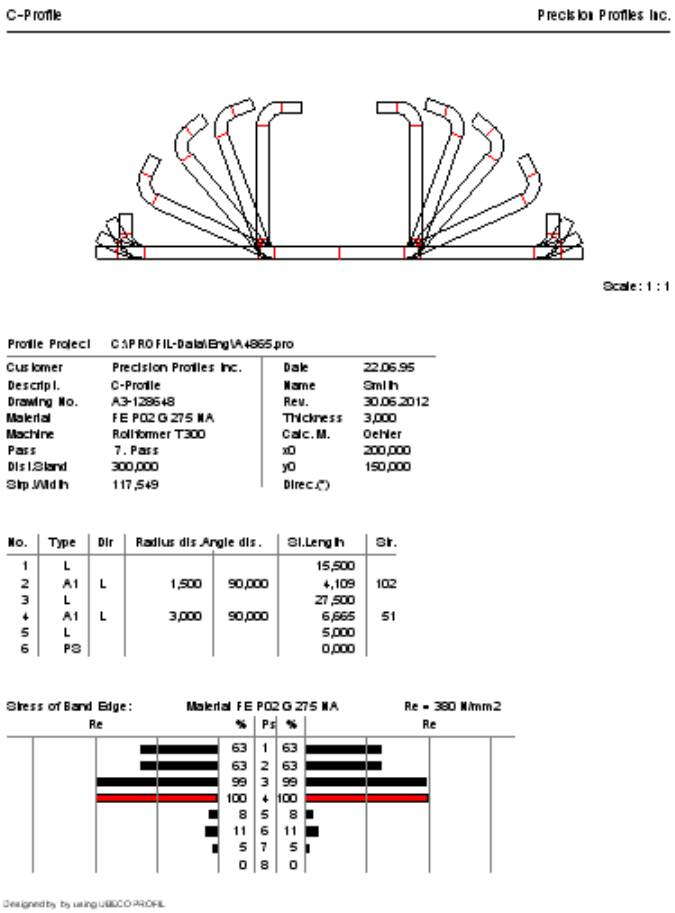
drawing object is saved in the file. The function Drawing -> CAD does the same; the name and the path of the temporal output file, however, are taken from Settings Files.

- **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 Settings, 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 Settings, 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 Settings, 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 Settings, General for the set-up of resolution and background color. See also Edit, Copy.
- **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 Settings 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

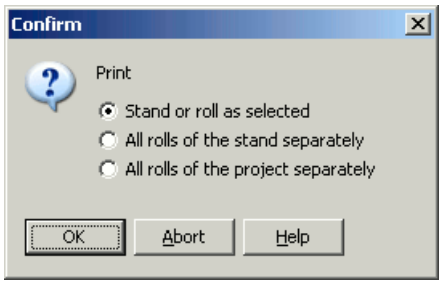
Use this function to print out the drawing that is displayed in the Drawing Area and the data of the selected Profile Project. In order to select the desired printer, print mode, and content of the print-out, call File, Print Preview previously.

Calling the function

Call this function optionally by:
- Main menu: **File, Print**.
- Button **Print** in the Button Key Bar.

Principle of operation

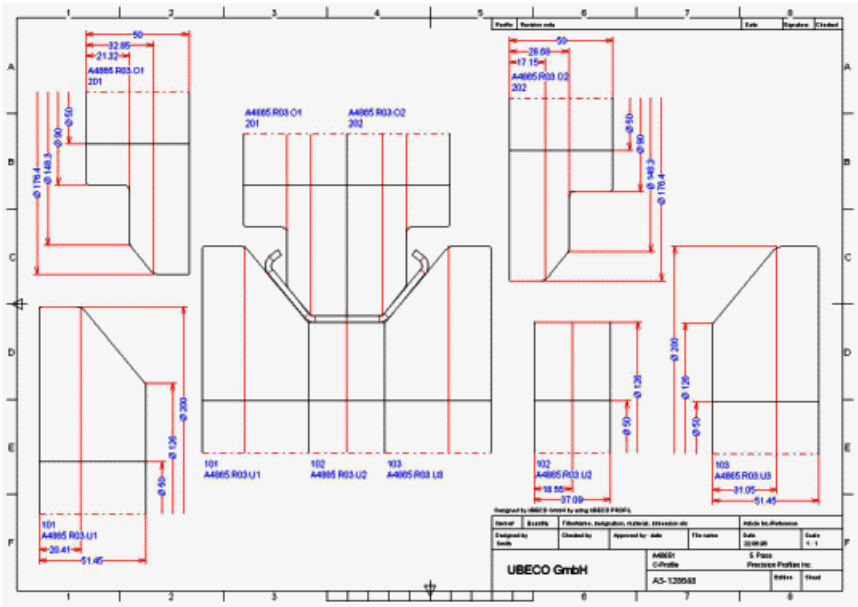
If View Pass, View Statics, View Flower Nested, View Flower Separated, or View Flower 3D is the current view, the content of the Drawing Area (or what is selected) is printed. If needed, successive pages are printed.



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

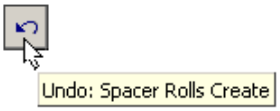
3.1.2.1 Undo

Use this function to reverse the most recent operations.

Calling the function

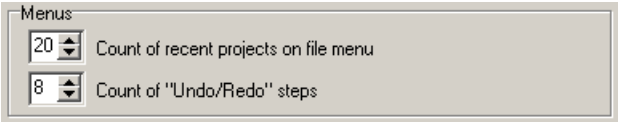
Call this function optionally by:
- Main menu: **Edit, Undo**.
- Button **Undo** in the Button Key Bar.

- Shortcut: **Ctrl Z** (Setup in Settings, Keyboard).



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

Setup



Select in Settings, 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

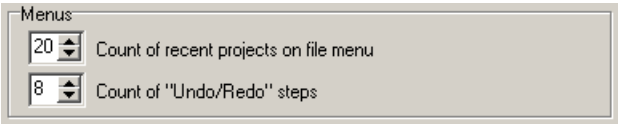
If you called Undo too often in error, you can reverse the effect of the last undo command by using this function.

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 Settings, General, **Menus** how many steps should be available for Re**do**.

Hints:

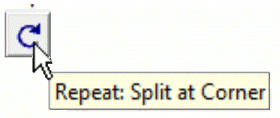
If you did this in error, you can Undo the commands.

3.1.2.3 Repeat

This function repeats the most recently called (last called) operation. It makes it easier to repeat an operation several times successively.

Calling the function

Call this function optionally by:
- Main menu: **Edit, Repeat**.
- Button **Repeat** in the Button Key Bar.
- Shortcut: **F4** (Setup in Settings, Keyboard).



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

3.1.2.4 Copy

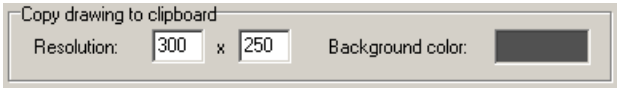
Use this function to copy the drawing from the Drawing Area as a bitmap to the Windows clipboard. This is useful to transfer the picture to any Windows application (e.g. Word, Paint and others) by using the function **Edit, Paste** in the other application.

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 Settings 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.

3.1.2.5 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.

3.1.2.6 Material

This function opens the Material Window, which shows the machine data from the current Project.

Calling the function

Call this function optionally by:
- Main menu: **Edit, Material**.
- Button **Material** in the Button Key Bar.
- Mouse click into the input field **Material** in the Project Data Window.

3.1.2.7 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.

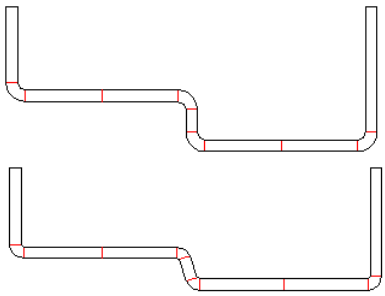
Calling the function

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 be 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 Settings Profile list, **Profile list window** how many windows you want to see.

3.1.2.8 Draft Mode

The **Draft Mode** 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 Mode**

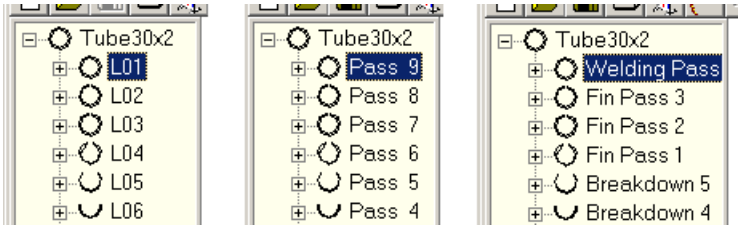
Principle of operation

Draft Mode 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 mode is displayed by a changed background color. The draft mode 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 Mode 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 Settings, Drawing, "Colors of Drawing Area, Background".

3.1.2.9 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.

3.1.2.10 Settings

Use this function to adapt PROFIL to your individual needs.

Calling the function

Call this function optionally by:
- Main menu: **Edit, Settings**.
- Button **Settings** in the Button Key Bar.

Select the following settings:
- Settings General
- Settings Drawing
- Settings Profile list
- Settings Calculate
- Settings Rolls
- Settings Spacer Rolls

- Settings Database
- Settings Parts List
- Settings NC
- Settings Files
- Settings ActiveX
- Settings PSA
- Settings Keyboard
- Settings Mouse

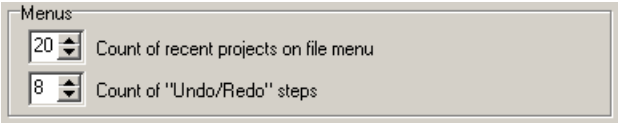
Principle of operation

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

3.1.2.10.1 General

Use this function for these general settings:

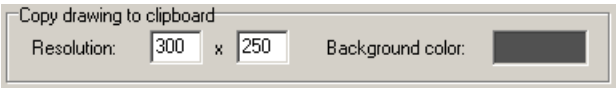
Menus:



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.

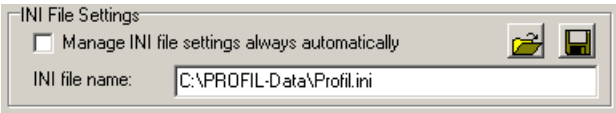
Copy drawing to clipboard:



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.

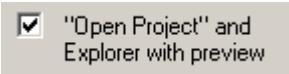
Load/Save INI file settings once: Buttons for loading and saving the setting from and to respectively the INI file.

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

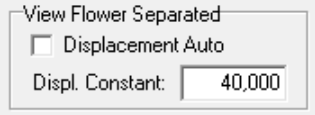


While working with larger projects the screen refresh takes more time, because refreshing the preview pictures is necessary after each modification. This may impede quick working. Simply switch off the preview.

3.1.2.10.2 Drawing

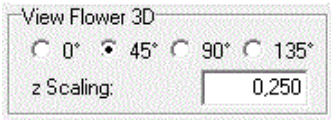
Use this function to set the parameters for views and drawings:

View Flower Separated



Select the vertical displacement of the passes for View Flower Separated.
Displacement Auto: The displacement of the passes is set automatically dependent on the pass height. Flat passes are closer together for a compact representation.
Displ. constant: Select a constant displacement of the passes.

View Flower 3D:



Select the angle for the View Flower 3D, View, PSA - Profile Stress Analysis. 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%.

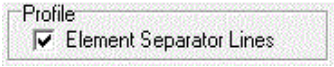
Grid:

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 Settings, Colors.
- Disable the grid lines with Settings, Drawing, Grid or **Button Grid on-off**.

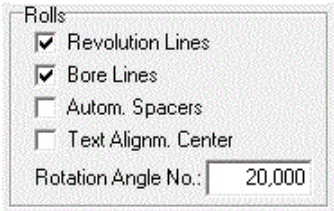
Profile:



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.
- **Rotation Angle Numbers:** Long roll and part numbers can obscure each other. In order to prevent it they are rotated automatically. The rotation angle can be preset.

Hint: The difference between **Automatic Spacers** and **Spacer Rolls** is described in Creating Spacers.

Rotation Angle Numbering:

Long Roll Numbers and Part Numbers are rotated automatically in order to prevent obscuring. Select the desired rotation angle.
The rotation is within an axle if
- more than one roll exists on an axle/shaft,
- the roll- and part numbers need more space than the roll width allows.

Drawing -> CAD also transfers the rotated roll and part numbers to CAD.

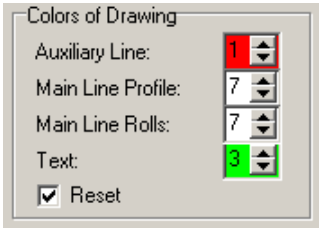
Text Height:

Select the text height for the table of statics, the roll numbering and the dimension texts on the Drawing Area.

3.1.2.10.3 Colors

Use this function to preset the colors in the Drawing Area:

Colors of drawing:

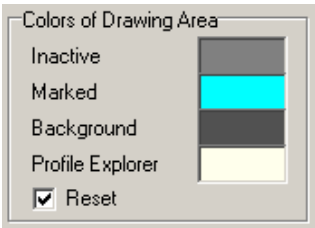


Auxiliary line, main line profile, main line roll, text: Select the colors for the drawing. The numbers of the colors are the same as in AutoCAD:

0 = by block
1 = red
2 = yellow
3 = green
4 = cyan
5 = blue
6 = magenta
7 = white

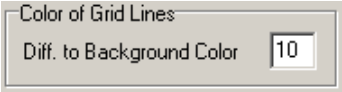
The selected colors will be transferred into the CAD and NC output file.
Default: All colors of the drawing are reset to the default colors.

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:

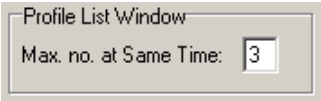


The **color of the grid lines** should differ only a little from the **background color of the drawing area**. Select how much it should differ. If the background color is dark, the grid lines are shown in a color that is a little bit lighter, otherwise darker. Recommended is a value of 6..12. A larger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

3.1.2.10.4 Profile List

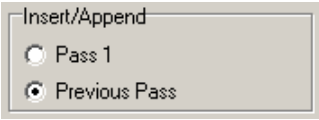
Use this function to set the parameters for the profile list:

Profile list window



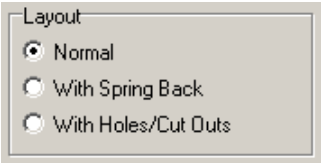
Select the **Max no. at Same Time** of Profile List Windows. For example, if you select 3, the 4th window closes the 1st again. So you do not need to close superfluous windows by yourself.

Insert/Append



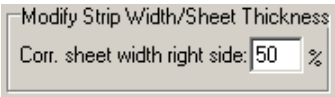
Select, if the Profile List of **Pass 1** or of the **Previous Pass** should be inserted/appended, when you call the function Profile, Insert or Profile, Append.

Layout



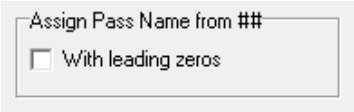
Select among 3 possible layouts of the Profile List Window: **Normal** is the smallest window without spring back and holes/cut-outs. **With Spring Back** displays the Spring Back, this means, besides the discharged state and the loaded state too. **With Holes/Cut-Outs** displays the Holes/Cut-Outs.

Modify Strip Width/Sheet Thickness



When the functions Profile, Modify Sheet Thickness and Profile, Modify Strip Width are applied on a unsymmetrical 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.

Assign Pass Name

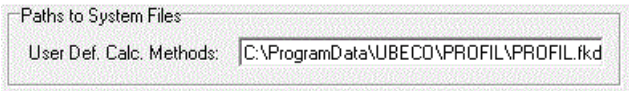


If you insert several # in the Profile List Window in the input field Pass and press <Enter>, these characters are replaced by the profile list number in sheet running direction. Select if leading zeros should be used in order to have constant count of places.This is also used for roll numbering if you enter the Variable **$PS** (Pass No) in Settings Rolls, **Number Keys**.

3.1.2.10.5 Calculate

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

Paths to System Files:

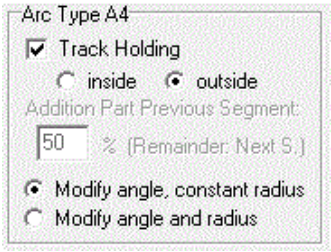


User defined calculation methods: The input field is used for selecting the path to the Factor File, e.g. if the file should be installed in a network path for common access. Since **PROFIL** rel. 5.4 the factor file does not need to be edited by a text editor. Select and parameterize individual Calculation Methods in the Developed Length Window.

Use the **Edit** button to call the text editor (set in Settings 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.

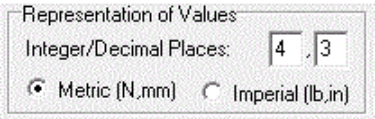
Arc Type A4



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.

Representation of values:



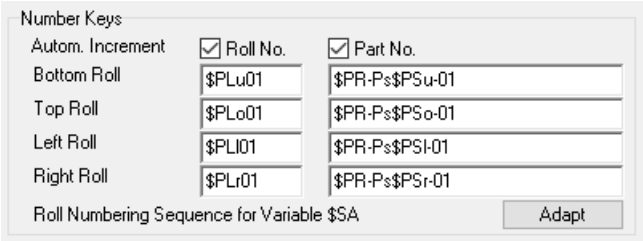
Integer/Decimal Places: Set the number of integer places (digits left of the decimal separator) and the decimal places (digits right of the decimal separator) for the representation of values in the data windows. For normal profile size in metric system (unit mm) use 4 integer places and 3 decimal places. 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.

3.1.2.10.6 Rolls

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

Number Keys:



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.

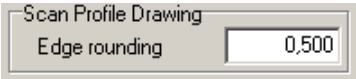
Autom. Increment: Select, if the roll number and the part number should be incremented automatically (number +1), when you split a roll with the function Roll Split at Corner and Roll Split between Corners. Precondition: the last digit must be numeric (0..9).

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.

Roll Numbering Sequence vor Variable $SA: Instead of **Autom. Increment** (that numbers rolls of each shaft separately) the Variable $SA can be used for numbering the rolls of a whole stand consecutively. Select the sequence by pressing **Adapt**.

Scan Profile Drawing:



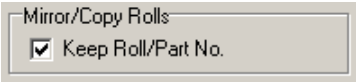
Edge rounding: Enter the radius (fillet) for the first and the last corner of a roll, when you create a roll automatically by using the function Roll Scan Profile Drawing.

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



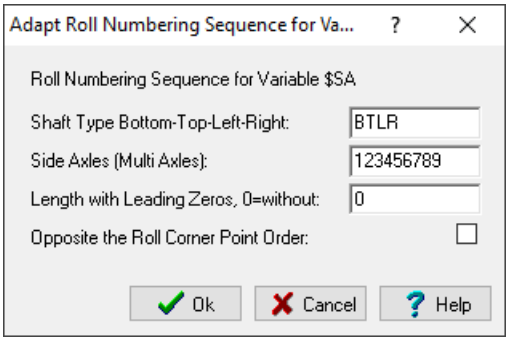
Keep Roll/Part No.: If checked, the roll/part number is not renumbered if a roll is mirrored (function Rolls Mirror) at a roll edge (bottom/top rolls) or from one side to the opposite side (side rolls). It also is not renumbered if a roll is copied to another position via the clipboard (function Roll 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-Contouror Roll Scan Profile Drawing).

3.1.2.10.6.1 Roll Numbering Sequence

Roll Numbering Sequence for Variable $SA



The Variable $SA in Number Keys enables consecutive roll numbering of a whole stand. Select the numbering sequence.

Shaft Type Bottom-Top-Left-Right: The entry e.g. **BTLR** causes numbering of the bottom shaft rolls first, then top shaft rolls, then left and finally right axle rolls.

Side Axles (Multi Axles): In case the machine has more than one let or right side axles, select here the axle sequence. The entry e.g. **123456789** causes numbering of the axle 1 first, then 2 etc. (correspondent to the numbering in the Machine Window).

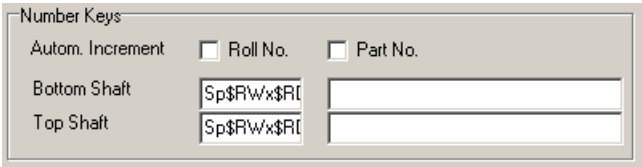
Length with Leading Zeros. 0=without): The entry e.g. 3 causes numbering 001, 002, 003 etc. The entry 0 suppresses leading zeros. Also valid for Variable $TA.

Opposite the Roll Corner Point Order: Normally, the rolls are numbered in the same order as the numbering order of the Roll Corner Points in the Roll Tool Window. By checking this check box you can select reverse order. Also valid for Variable $TA.

3.1.2.10.7 Spacer Rolls

Use this function to set the parameters for Creating Spacer Rolls.

Number Key

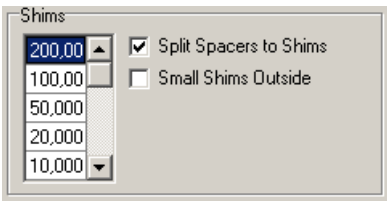


This setting enables the automatic numbering of the roll number and the part number for spacer rolls only, when spacer rolls are created newly. If you want to use the number keys for existing rolls, call the function Roll Renumber.

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. **SpRWxRD** 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

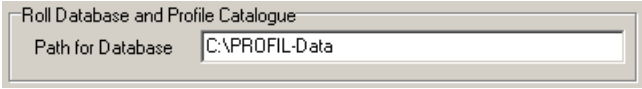
The Expanded Roll Tool Window opens. You can enter default values, which are copied into a spacer roll dataset during Creating Spacer Rolls.

Hint: The difference between automatic spacers and spacer rolls is discussed in Tutorial, Roll Tools, Creating Spacer Rolls.

3.1.2.10.8 Database

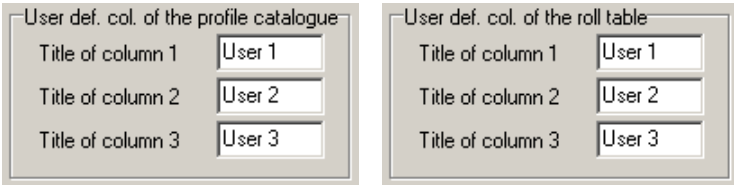
Only with option Database.

Profile Catalogue and Roll Database:



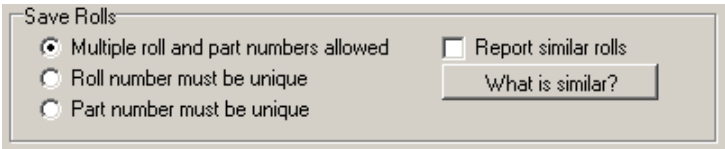
Path for Database: Enter the path to the database files. After clicking into the input field, the Explorer appears. The path must be valid, either on the local disk or on a network server. If the database files do not exist, they will be created, when you call the Profile Catalogue or the Roll Stock 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. In the 64bit version, the file of the profile catalogue is **Profiles64.sdb** and of the roll stock management **Rolls64.sdb**. For converting of 32bit files to 64bit files a converter is available.

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.

Save Rolls



Multiple roll and part numbers allowed: Roll number as well as part number can exist several times, i.e. several rolls can have the same roll or part number. If a roll with an already existing roll or part number is saved, a new roll appears in the database without user request.

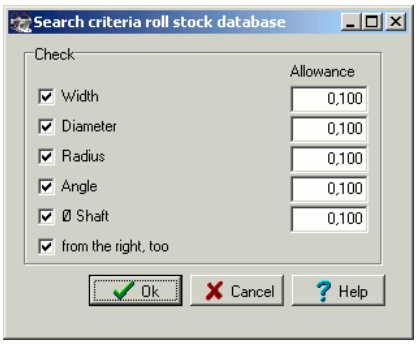
Roll number must be unique: Only one roll can have this roll number. If a roll is saved with an already existing roll number, the user is asked **Roll with roll number already exists. Overwrite? Yes** overwrites the existing roll with new roll data. **No** breaks saving the roll. If necessary, the roll number has to be modified in the Roll Tool Window before saving.

Part number must be unique: Only one roll can have this part number. If a roll is saved with an already existing part number, the user is asked **Roll with part number already exists.**

Overwrite? Yes overwrites the existing roll with new roll data. **No** breaks saving the roll. If necessary, the part number has to be modified in the Roll Tool Window before saving.

Report similar rolls: Checks if similar rolls already exist in the roll stock before saving a roll. If they do, the count of found rolls is reported and the similar rolls appear in the roll stock window. If the question **Save nevertheless?** is answered with **Yes**, the topic roll is saved as a new roll, otherwise the roll is not saved. The user can decide discretionary if he wants to insert one of the found rolls in his project, see function **Replace roll in project by roll from roll stock** in the Roll Table.

What is similar?
If **Report similar rolls** is switched on, you can enter here the criteria for for searching similar rolls:



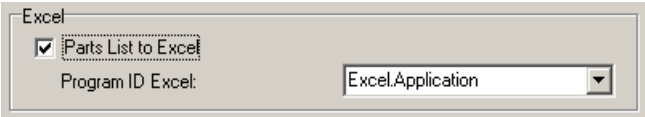
The window **Similarity criteria rolls** opens with these entries:
- **Width:** Valid for widths of all roll corner points.
- **Diameter:** Valid for diameters of all roll corner points.
- **Radius:** Valid for radii of all roll corner points.
- **Angle:** Valid for contour angles between all roll corner points to the next.
- **Ø Shaft**
- **from the right, too:** Means that also rolls are found that are saved mirrored.

Select by checking the boxes which search criteria should be used. For each criterion, you can enter an allowance. Avoid allowance 0, because unsafe results can occur.

3.1.2.10.9 Parts List

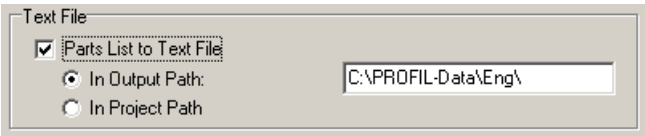
Use this function to set the parameters for the creating of the parts list of the rolls by the function Output Create Parts List.

Excel:



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:



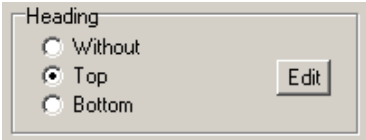
Parts List to Text File: Check here, if a text file should be created by the function Output Create Parts List. The name of the text file is always the same as the project name, but has the extension **.txt**.

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)



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 Settings 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.

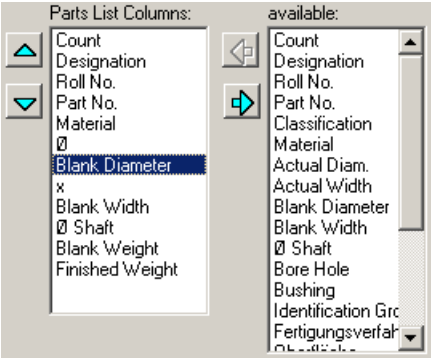
In case of Excel please define the heading within your initial work sheet.

Set-up Columns:

After pressing this button the Set-Up Parts List Columns Window opens.

3.1.2.10.10 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 Settings Parts List.

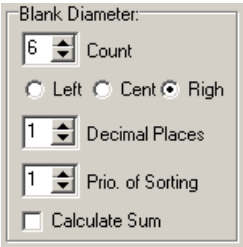


available: This list box shows which parts list columns can be selected:

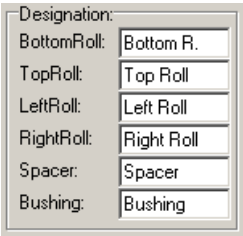
Count	count of rolls with same (visible) parts list properties.
Designation	fixed text from table **Designation**
Roll No.	from the Roll Tool Window.
Part No.	from the Roll Tool Window.
Classification	from the Roll Tool Window.

Material	name of the Addition File
Actual Diam., Width	from the Roll Tool Window.
Blank Diam., Width	actual dimensions with additions from the Additions File that is set in **Material**. Unit mm or inch.
Ø Shaft	from the Machine Data.
Bore Hole	from the Expanded Roll Tool Window.
Bushing	from the Expanded Roll Tool Window.
Identification Groove	from the Expanded Roll Tool Window.
Treating	from the Expanded Roll Tool Window.
Surface	from the Expanded Roll Tool Window.
Addition	from the Expanded Roll Tool Window.
Remark	from the Expanded Roll Tool Window.
Blank Weight	weight of the cylindrical tool blank with the blank size. Unit kg or lb. Multiplied with **Count**.
Finished Weight	weight of the finished roll; the outer contour and the bore hole are considered. Unit Kg or lb. Multiplied with **Count**.
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

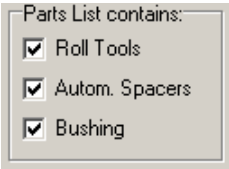
Parts List Columns: From the list **available** any entry can be copied into the **Parts List Columns** table by using the button with the left arrow. Afterwards the entry can be moved by using the buttons with the up and down arrow. To remove an entry from the list, use the button with the right arrow.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



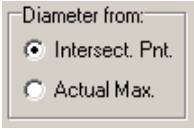
Column Properties: For each entry in the **Parts List Columns** table the properties can be set: **Count** (of characters), **Position** (left, center or right), **Decimal Places**, **Priority of Sorting** and if the **Sum** should be calculated. Decimal places, sorting and sum are only possible for 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: 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: 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.

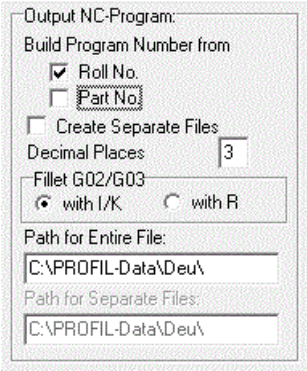


Diameter from Intersection Point/Actual Maximum: Select if the maximum roll diameter is calculated from the intersection point of the tangents (Roll Corner Point) or from the actual diameter on the fillet. The calculation of the Additions and with it the blank diameter is dependent on this selection.

3.1.2.10.11 NC

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

Output NC Program:



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 Mode **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.

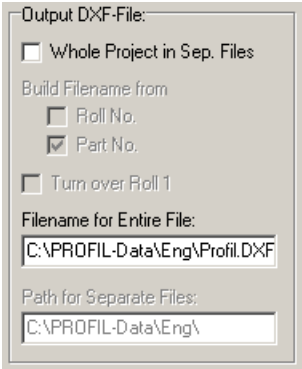
Fillet G02/G03

Fillets in the NC contour are exported as G02 (clockwise) or G03 (anticlockwise). Subject to the regulations in DIN 66025 the arc center is given relative to the arc start point as parameter I (x axis) and K (z axis, rotation axis). Many machine controls also accept parameter R (fillet radius), however this is not covered by DIN. Select if you want to use IK or R parameter.

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:

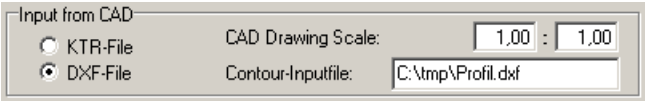
Enter path for the separate files. The filename is built automatically by using roll number or part number. After clicking into the input field, the Explorer appears.

Hint: Select by View, Show, Spacer Rolls whether the NC output should contain spacer rolls.

3.1.2.10.12 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:

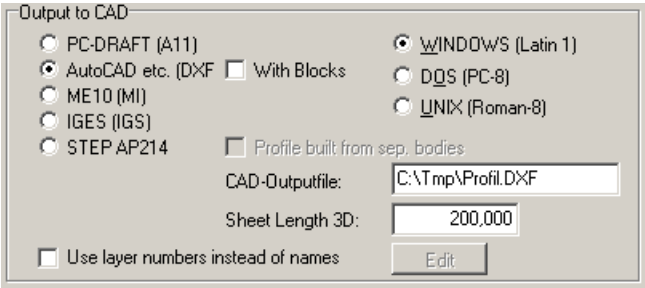
KTR-format: file format defined by UBECO, see Contour File (KTR Format). Use this format if a macro in your CAD system proceeds the contour tracking and saves the drawing elements in sorted order. Such macros currently are available for the CAD systems PC-DRAFT, AutoCAD until rel. 12, ME10, and CADDA.. 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, ELLIPSE, and BLOCK-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 Settings 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 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.

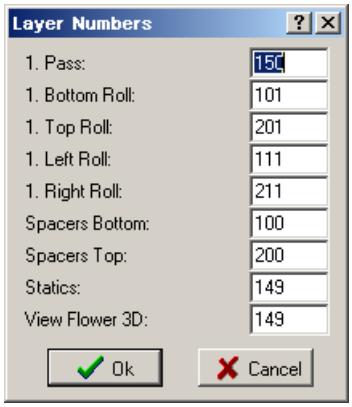
DOS (PC-8) converts to the PC-8 symbol set for DOS-based systems.

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:



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 Settings ActiveX the ActiveX Output to CAD is disabled.

Hint: Select by View, Show, Spacer Rolls whether the file output should contain spacer rolls.

3.1.2.10.13 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.

Element entities
LINE, ARC, POLYLINE, LWPOLYLINE, ELLIPSE, and BLOCK-entities can be imported.

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:

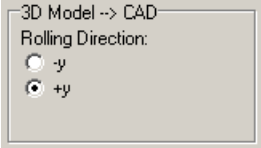
- dimension line color, extension line color, text color, text height, number of decimal places = as preset in PROFIL,
- dimension unit = decimal,
- suppression of zeroes = on.

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



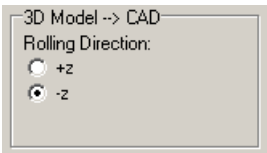
The function Output 3D Model -> CAD transfers the 3D model of all stands of the whole roll tool set should be transferred to CAD. Select, if the rolling direction should be shown in positive or negative y direction.

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 an 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.
- Before you transfer a 3D model by Output 3D Model -> CAD, select in SolidEdge a document Part. Otherwise an error messages appears.
- Colors an 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. Since BricsCAD Rel. 16 also 3D models can be transferred from PROFIL to BricsCAD.

Properties of the ZWCAD interface:

Since PROFIL Rel. 6.0.3 and ZWCAD Rel. 2021 this interface has been provided to transfer profile and roll tools drawings from PROFIL to ZWCAD and to transfer profile and roll contours from ZWCAD to PROFIL.

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:

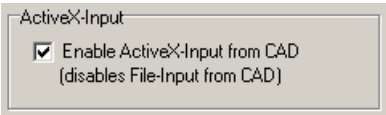
  AutoCAD 14: AutoCAD.Application.14
  AutoCAD 2000: AutoCAD.Application.15
  AutoCAD 2004: AutoCAD.Application.16
  AutoCAD 2005: AutoCAD.Application.16.1
  AutoCAD 2006: AutoCAD.Application.16.2
  AutoCAD 2007: AutoCAD.Application.17
  AutoCAD 2008: AutoCAD.Application.17.1
  AutoCAD 2009: AutoCAD.Application.17.2
  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
AutoCAD 2022: AutoCAD.Application.24.1 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
  SolidWorks 2016: SldWorks.Application.24 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 Bricscad 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
  BricsCAD Release 16: BricscadApp.AcadApplication.16.0
- **BricscadApp.AcadApplication**: Connects to the current BricscadApp release, this is the last installed one.
- **ZWCAD.Application.nn**: Connects to ZWCAD release nn. Use this Program ID, if you have installed more than one ZWCAD release and you want to select a special one:
  ZWCAD Release 2021: ZWCAD.Application.2021
- **ZWCAD.Application**: Connects to the current ZWCAD release, this is the last installed one.

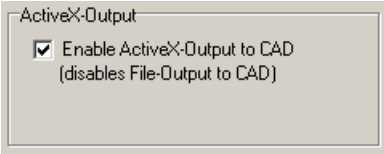
If the drop-down-list is empty, no CAD system is installed on your computer that is able to be controlled via ActiveX.

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 Settings Files) will be disabled.

Enable ActiveX Output to CAD:



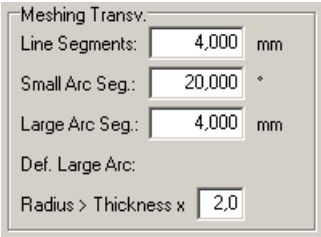
This function enables the ActiveX output and switches the function Output Drawing -> CAD and the matching button on the Button Key Bar to ActiveX. At the same time the file output (see Settings Files) will be disabled.

Hint: Select by View, Show, Spacer Rolls whether the ActiveX output should contain spacer rolls.

3.1.2.10.14 PSA

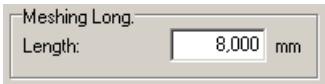
Use this window to set-up the PSA - Profile Stress Analysis.

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.

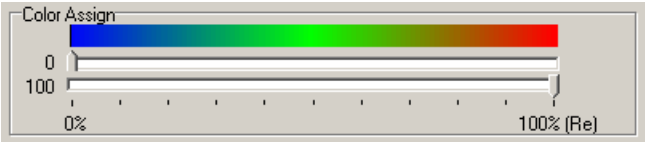


Length: select the desired length of the shell segments in longitudinal direction.

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

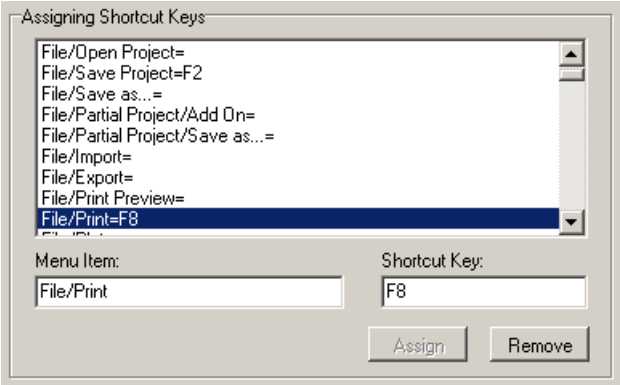


Select by moving the tabs which relative stresses should be assigned to the colors blue and red. The stresses are related to the yield point of the selected material. 0% means no stress and 100% means the yield point stress is reached.

3.1.2.10.15 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:

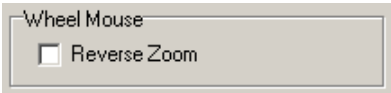
F2 .. F12 (F1 is reserved for the context sensitive help)
Shift + any character
Ctrl + any character
Alt + any character
Alt Gr + any character

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.

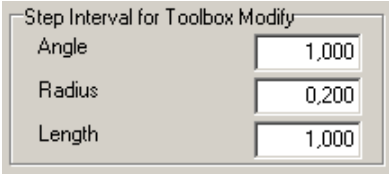
3.1.2.10.16 Mouse

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

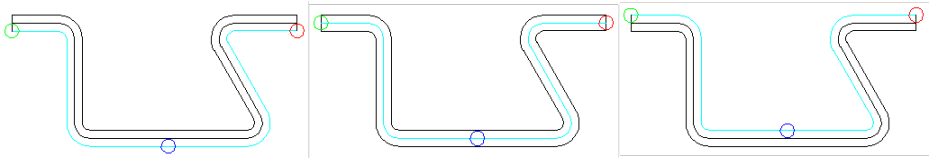
Use this function to import a contour, which you have prepared in your CAD system, into PROFIL in order to generate a Profile list. Whether a Contour File (KTR format) or a Contour File (DXF format) is being read, you select in Settings, Files, from CAD. If you want to read files with frequent varying 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 unsymmetrical profile, then the start point is the next element behind element **P**.
Call this function optionally by:

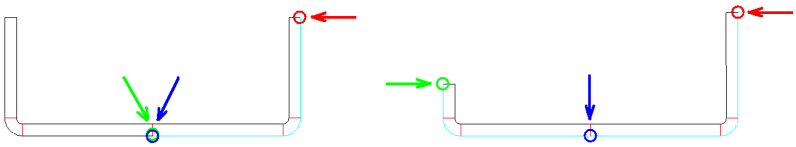
- Main menu: Profile**, Read CAD Contour**.
- Button **Read CAD Contour** in the Button Key Bar.

If you have enabled the ActiveX Input from CAD in Settings 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 Settings, 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.

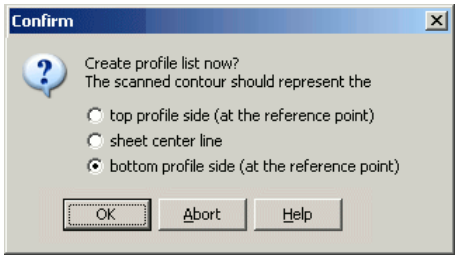


Symmetrical profile Asymmetrical profile

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 unsymmetrical 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 haft 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. Use the function Profile, Element, Fillet to combine such elements with the neighbors and to get the desired radius. Alternatively correct the CAD drawing by using the CAD's fillet function, 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 Settings 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

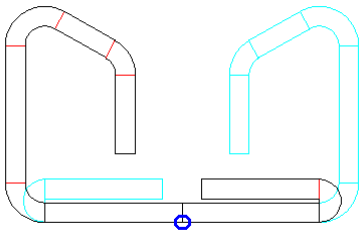
Before you call this function, select the pass whose profile list should be emptied.
Call this function optionally by:
- Main menu: **Profile, Empty**.
- Context menu (right mouse button click on any profile element of the desired pass in the Drawing Area): **Profile, Empty.**

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



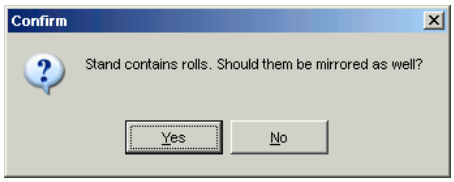
Use this function to mirror a unsymmetrical profile list at the Reference Point X0/Y0.

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.

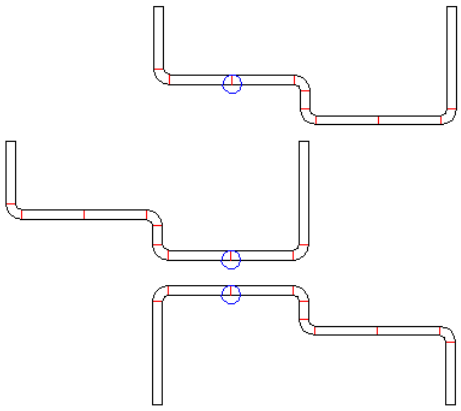


If the stand contains rolls, they can be mirrored as well. In this case the user is asked to agree. The width of top and bottom rolls are mirrored at the Roll Reference Point. Side rolls are exchanged and the width is mirrored at the roll reference point in addition.

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

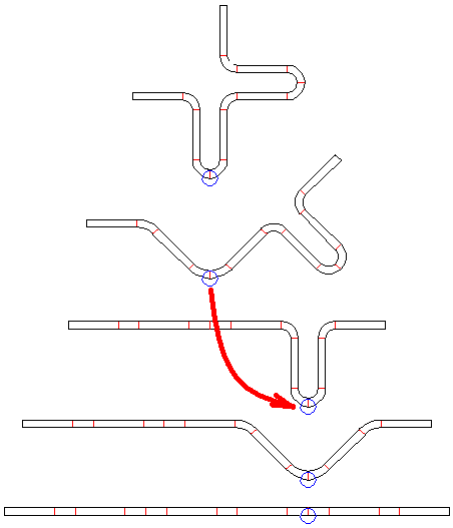
If you modify the start element of a symmetrical profile list (with **Symmetrical Point PS**), the list becomes unsymmetrical (with **Point P**). However, this cannot be redone with this function.

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.

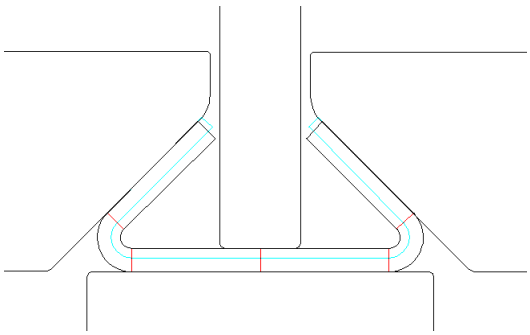
Principle of operation

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.

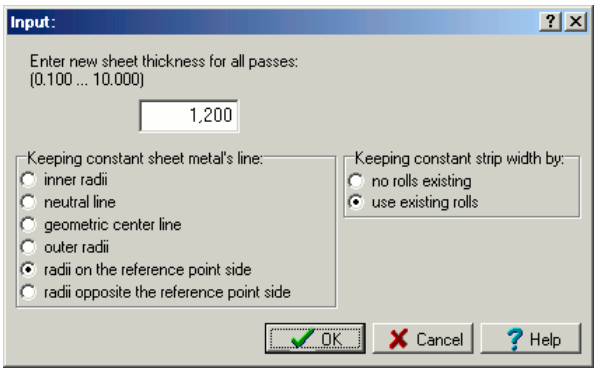
3.1.3.6 Modify Sheet Thickness



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 bad 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 unsymmetrical (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 Settings, Profile List, topic **Modify Strip Width/Sheet Thickness**.

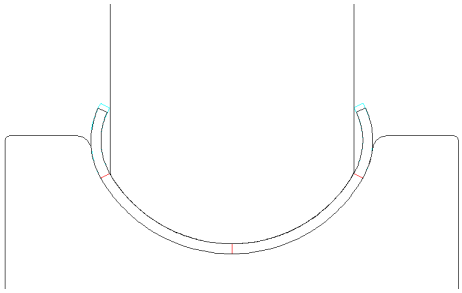
The method is useful in combination of the selection **Constant Inner Radii** or **Constant Outer Radii** for checking if existing rolls are suited for various sheet thickness. Another application is to learn how to design rolls for a sheet thickness interval. Pay attention by using this method: The 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

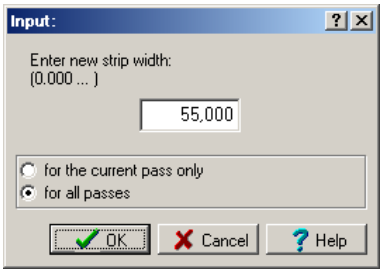


After defining the desired profile cross-section the strip width of the flat sheet is defined, too. Sometimes the question comes up, whether an already existing strip with a differing width can be used for forming this profile or whether a strip with a standard width can be used. Another important application 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.



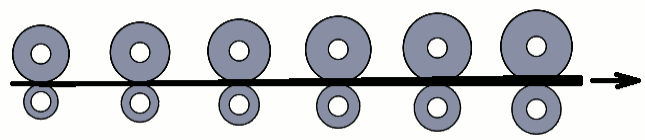
After calling this function, a dialog window is opened. Enter the desired new strip width and select, whether the new strip width should be effective on the current pass only or on all passes of the whole flower pattern.

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 bad 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 unsymmetrical (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 Settings, Profile List, topic **Modify Strip Width/Sheet Thickness**.

3.1.3.8 Calibrate Strip Width



If **Lead (Strip tension)** is selected in Machine, Working Diameter, i.e. tension by gradually increasing the roll pitch diameters, the sheet is stretched in longitudinal direction and bulged in transverse direction. The function **Profile, Calibrate Strip Width** can be used to consider the resulting bulge.

Calling the function

Call this function optionally by:

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

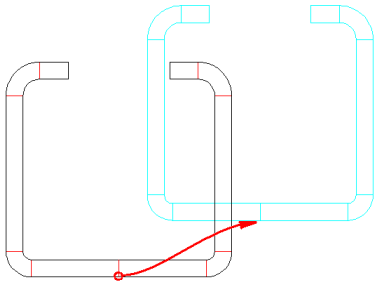
After calling this function, a message window is opened with the question **Calibrate sheet width now dependent on the calibration factor (%) in the machine window, starting from pass no. against the sheet running direction**. The pass number (from the Profile Explorer) for the start of the action is shown.

Principle of operation

The function **Profile, Calibrate Strip Width** calls the function Profile, Modify Strip Width for each pass and increases the strip width gradually against the sheet running direction. The amount is taken from the Calibration Factor from the Machine Window. Thus the strip width is decreased in sheet running direction.

Hint:
The diameter of round tubes is decreased by the same factor because of L = π*D.

3.1.3.9 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.

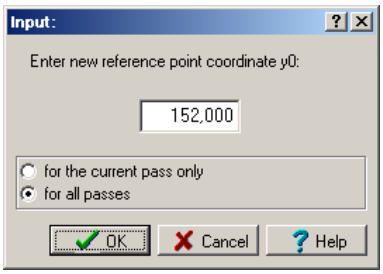
The function **Modify Reference Point** enables to modify the x and y coordinates of the reference point either for the current pass or for all passes.

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.

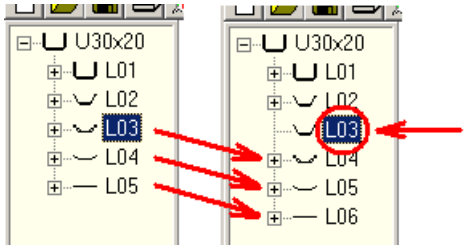
Principle of operation

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 is visible again by the button **Fit** of the Navigator.
- Select View, Flower Nested in order to check all passes for proper position.

3.1.3.10 Insert



Use this function to insert a new Profile List at the current position. This means to insert a stand.

Calling the function

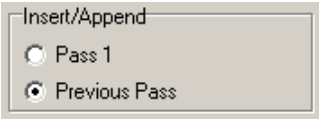
Before you call this function, select the pass at whose position a new profile list should be inserted. Call this function optionally by:

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

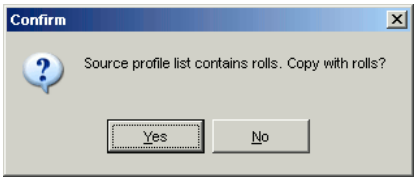
Principle of operation

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.

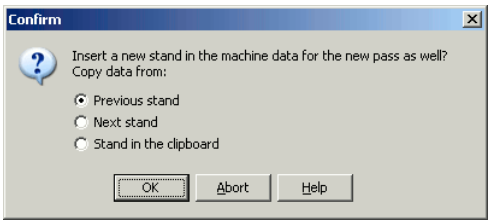
Setup



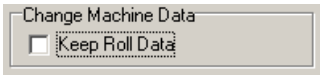
Select in Settings Profile List, **Insert/Append** if the contents of **Pass 1** or of the **Previous Pass** will be copied into the inserted list.



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 Settings Rolls) and that rolls are modified dependent on the data of the shafts of the new stand (in the Machine Window).



When a new profile list is inserted, a fitting stand can be inserted in the machine data at the same time. Select if the data should be copied from the previous stand, the next stand, or the stand in the clipboard.

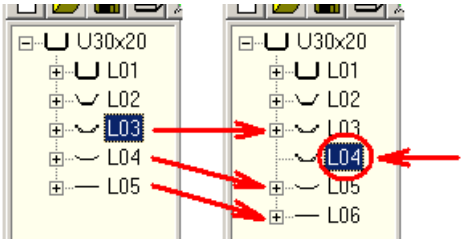


Furthermore the setting **Keep Roll Data** in Settings Rolls determines how the rolls are treated by changed machine data.

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.

3.1.3.11 Append



Use this function to append a Profile List behind the current one.

Calling the function

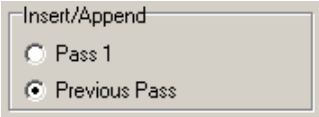
Before you call this function, select the pass behind whose position a new profile list should be appended.
Call this function optionally by:

- 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.

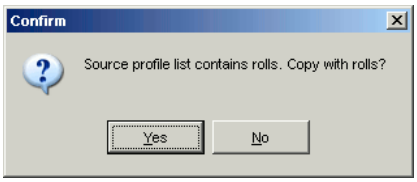
Principle of operation

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.

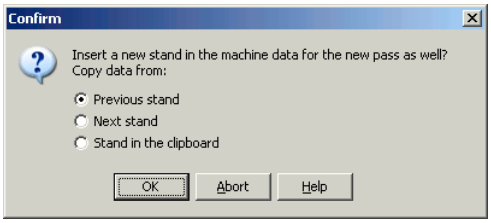
Setup



Select in Settings 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 Settings Rolls) and that rolls are modified dependent on the data of the shafts of the new stand (in the Machine Window).



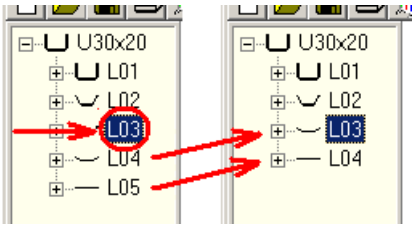
When a new profile list is appended, a fitting stand can be appended in the machine data at the same time. Select if the stand data should be copied from the previous stand, the next stand, or the stand in the clipboard.

Furthermore the setting **Keep Roll Data** in Settings 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.

3.1.3.12 Remove



Use this function to remove a Profile List from the set of lists.

Calling the function

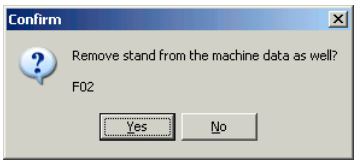
Before you call this function, select the pass that should be removed.
Call this function optionally by:

- 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.

Setup

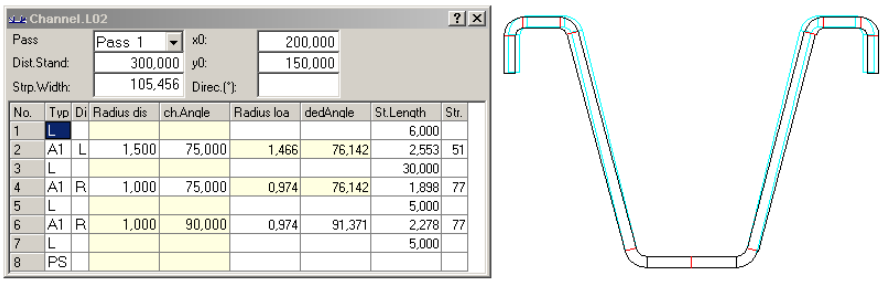


When a profile list is removed, the corresponding stand can be removed from the machine data at the same time.

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.

3.1.3.13 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

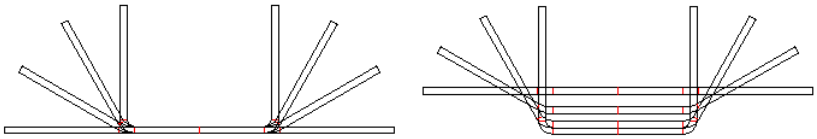
If a single arc of a pass is selected (by clicking of the arc in the Drawing Area or in the Profile, Window (thus, the arc is drawn in the mark color), only the loading of this arc is switched on and off. If no arc is selected (by button Inspect or by the right mouse button), the loading of all arcs 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 Settings 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.

3.1.3.14 Center Line Forming



Bottom line forming (left) and center line forming (right)
with lowering to constant centroid's height (100%)

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

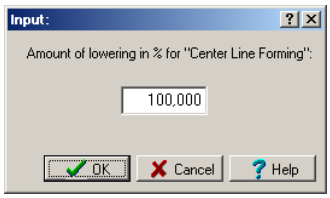
In order to decrease the way of the band edge and with it to decrease the stress of band edge it is recommended to lower the profile web from pass to pass. This means the flat strip enters into the machine in a higher position than the web of the final profile leaves the machine. This can be done by modifying the Y0 value of the profile Reference Point. In order to keep the circumference speed of both top an 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).

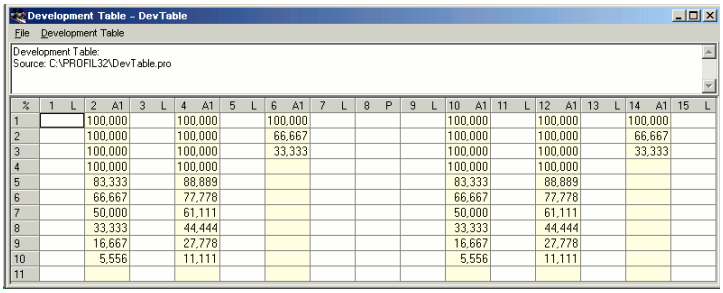
Principle of operation

The Y coordinates of the Reference Point of all passes except L01 are modified dependent 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.

3.1.3.15 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		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:

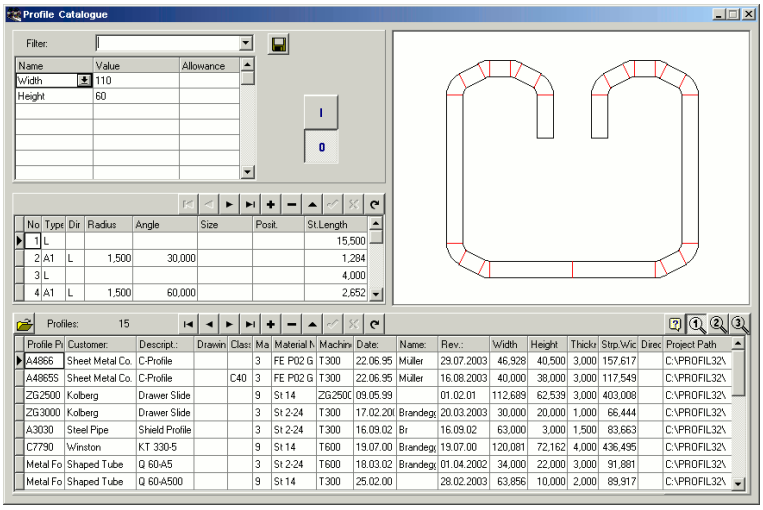
- The target profile list structure of the new profile must be the same as the source structure, this means the angles of the development table must be in the angle columns (A1..A4 in the table header). If it isn't, you can adapt the table by using the function **Development Table, Columns**.
- If an 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 Settings 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.

3.1.3.16 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.

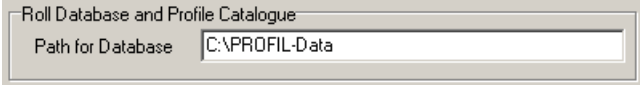
Profile Catalogue Save

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

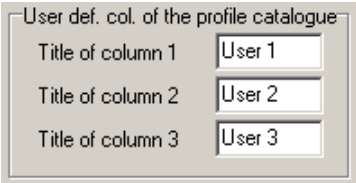
- **Current Project:** The currently opened project is stored.
- **All Projects of a Path:** The path selection window appears and all projects of the selected path are stored.

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

Setup



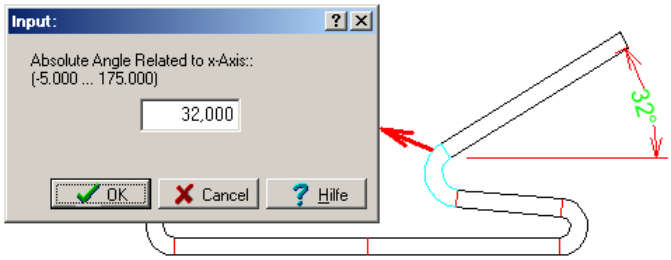
Set the path to the database in Settings Database.



Title of column 1..3: In Settings Database, enter the titles for the last 3 columns of the Profile Table. You can define these columns for your own needs.

3.1.3.17 Element

3.1.3.17.1 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:

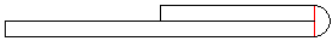
- Main menu: **Profile, Element, Abs. Angle**.
- Context menu (right mouse button click on the profile element in the Drawing Area): **Profile, Element, Abs. Angle.**

Principle of operation

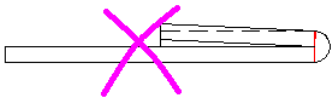
If you have selected a profile element **Line**, the absolute angle is displayed only without edit feature. If you have selected a profile element type **Arc**, you can modify the absolute angle between given limits. Valid limits are corresponding to a relative arc angle from 0..180°. The displayed absolute 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. The simplifies the function **Absolute Angle**.

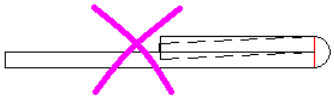
3.1.3.17.2 Open Fold



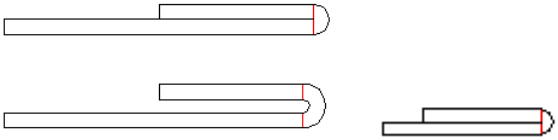
Problem: The band edge should be folded to a flattened hem by 180 degree with inner radius zero. The sheet surfaces should touch closely and should not spring up.



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.



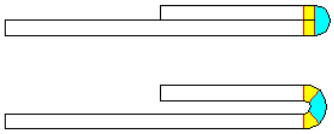
Not possible: Usually a bending zone is over bent in order to compensate the spring back. If this is tried with the 180 degree bend, the sheet would penetrate itself. This is not possible.



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?

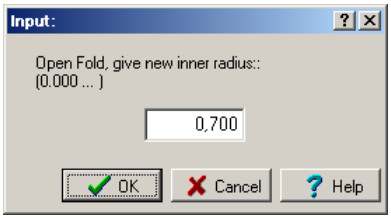


Explanation: Imagine, the new arc (with the larger inner radius) consists of three parts. The center part has exactly the developed length of the final 180 degree bend with inner radius zero. The two small parts have the developed length that is subtracted from the adjoining line segments. When the fold 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:

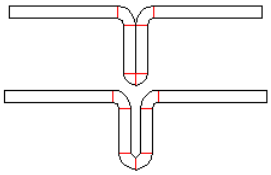
- Main menu: **Profile, Element, Open Fold**.
- Context menu (right mouse button click on the arc element for opening in the Drawing Area): **Profile, Element, Open Fold**.



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).

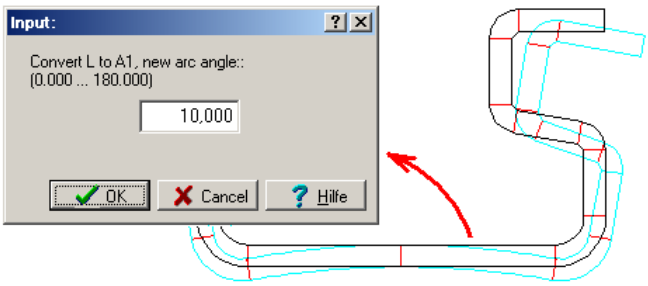
Principle of operation

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.

3.1.3.17.3 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.

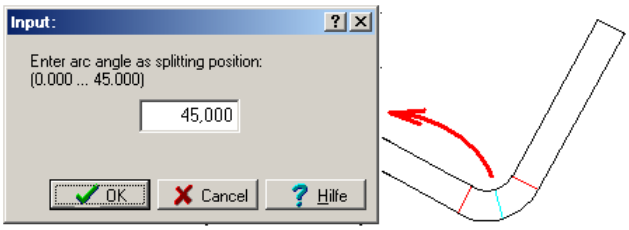
Principle of operation

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.

3.1.3.17.4 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.

3.1.3.17.5 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**.

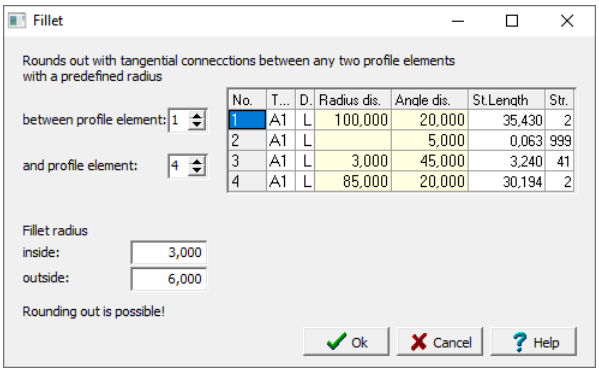
Principle of operation

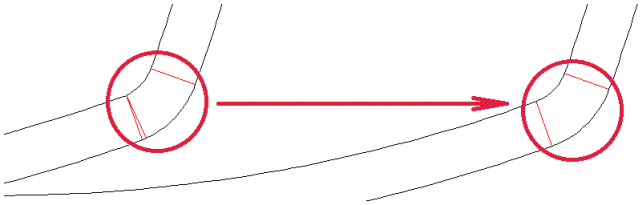
The selected profile element is joined with the next profile element. For arcs is effective: The Strip Width will keep constant if the Calculation Method DIN 3965 is selected. It will change marginally, if Oehler is selected, because the straight length calculated by the Oehler method is dependent on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

Hint:

Redo joining by the function Profile, Element, Split.

3.1.3.17.6 Fillet





This function rounds out two Profile Elements with a desired fillet radius and removes correction elements.

Application spectrum

- A faulty CAD drawing has been imported by Profile, Read CAD Contour. Faulty means: The drawing elements (lines and arcs) have not tangential connections. This happens if not the proper snap points are used during the arc creation. **PROFIL** recognizes it and inserts correction element as a small arc with radius 0, that restores the necessary tangential connection (see element no. 2 in the picture). It is strongly recommended to correct the defect, otherwise the rolls will create a groove on the sheet surface that will be seen by inspecting against the light.
- You find wrong radii after importing or radii that do not match the specification. By using this function you easily can modify the radii whereupon the rest of the profile keeps unchanged.

Calling the function

Before you call this function, select the profile element that should be removed or whose radius should be modified. Call this function optionally by:

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

Principle of operation

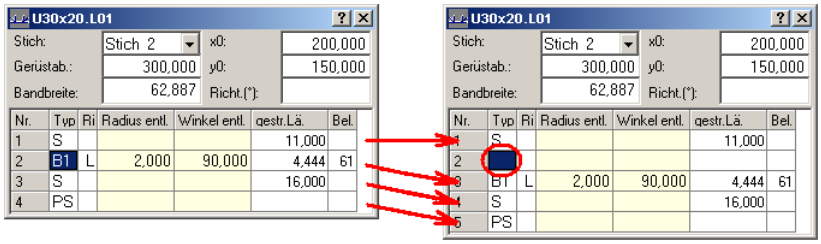
The window **Fillet** is opened and the table shows a part of the Profile List. Select in the box **between profile element** and in the box **and profile element** the elements that should be rounded out. The selected elements remain in place and change it's length dependent on the fillet radius. All profile elements between the selected are removed and replaced by a new arc with the selected **Filled radius inside** or **outside**.

Hints:

- Points (**P** or **PS**) are not allowed in the range between the selected profile elements.
- Any selection of lines or arcs ist possible, with any direction, also unequal.
- Any count of profile elements to remove between the selected ones is possible.

- Sometimes a mathematical solution does not exist, also the length of the selected elements can be too short. In theses cases a red message is shown.
- Alternatively you can use the CAD function **Fillet**. Empty the profile list (Profile, Empty) and import newly (Profile, Read CAD Contour).

3.1.3.17.7 Insert



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**.

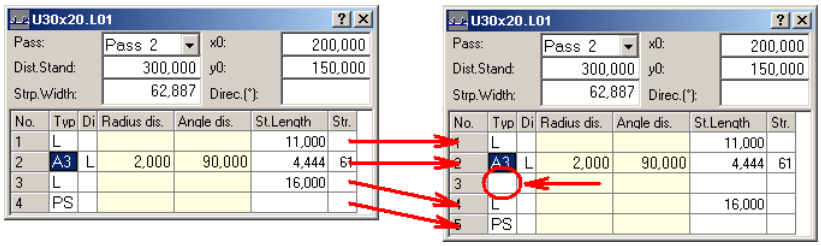
Principle of operation

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.

3.1.3.17.8 Append



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**.

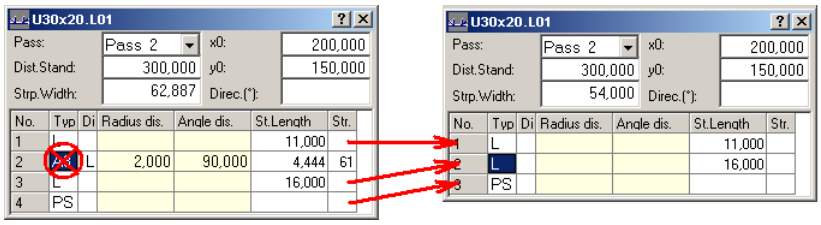
- Context menu (right mouse button click on the profile element in the Drawing Area): **Profile, Element, Append**.

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.

3.1.3.17.9 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.

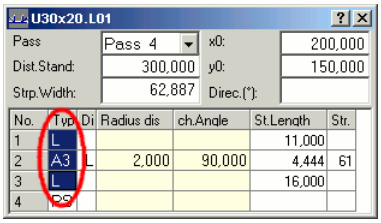
3.1.3.17.10 Copy

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.



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

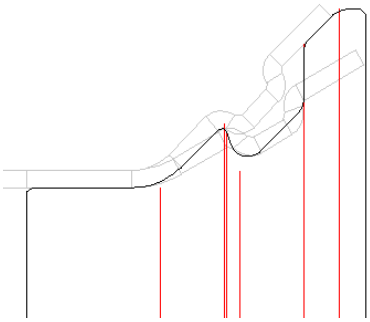
The marked profile elements are copied to the clipboard can be inserted or appended inside **PROFIL** at any other position. The content of the clipboard keeps available until **PROFIL** is terminated or you copy a roll to the clipboard (function Roll Copy)

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 Settings Files from CAD. If you have enabled the ActiveX Input from CAD in Settings 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.



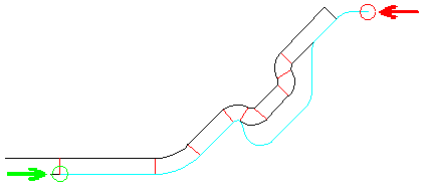
Use this function in the roll designing stage, if the function Scan Profile Drawing cannot produce the desired result and you prepared the roll contour in CAD, because the roll should get a contour that differs from the profile contour considerably. Only the working contour (without side flanks) should 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

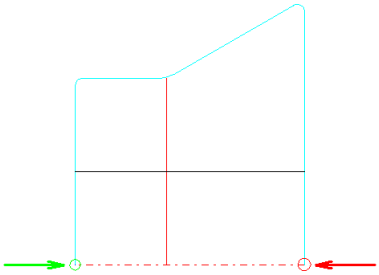
Use this function to import from the CAD System a roll that already exists but still is not available in a PROFIL Project File. This is useful in case you want to import rolls from third party systems or if you want to reuse old rolls. An important property of this function is that the roll keeps unchanged 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 Settings Files from CAD. If you have enabled the ActiveX Input from CAD in Settings 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.

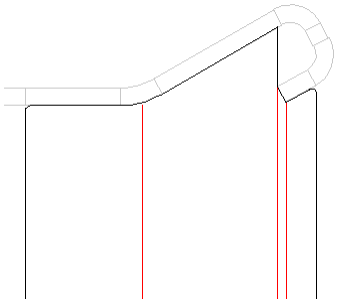
Hints:
Further function for creating rolls are:

- 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 Settings 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.

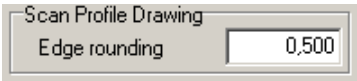
Principle of operation

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



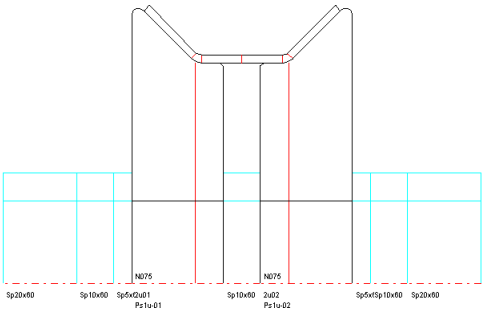
Select in Settings Rolls, **Scan Profile Drawing** the fillet radius of the automatic edge rounding at the left and right edge of each roll.

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

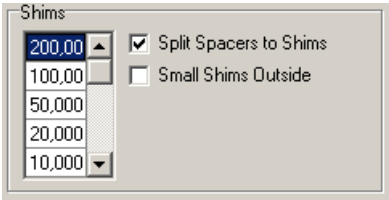
Call this function optionally by:

- Main menu: **Rolls, Spacer Rolls, Create**.
- Context menu (right mouse button click on a roll in the Drawing Area): **Spacer Rolls, Create.**

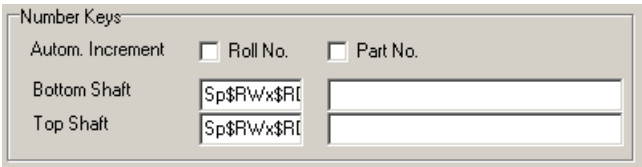
If the bottom or top shaft is selected by mouse click on an existing roll, the spacer rolls will be created for the selected shaft only. Otherwise, if nothing is selected (button **Inspect**), both bottom and top shaft will be furnished with spacer rolls. If spacer rolls already exist, a dialog box asks 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 Settings, Spacer Rolls, the spacer is split into shims dependent 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 dependent on **Number Keys** in Settings, Spacer Rolls. For spacer rolls, the variables $RW (roll width) and $RD (roll diameter) are of interest. Example: The number key SpRWxRD 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 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 Settings, 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.

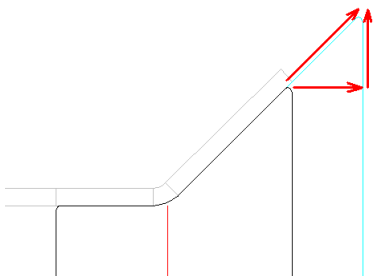
Calling the function

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, th 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.**

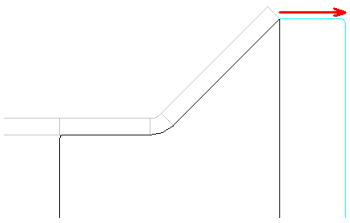
The input window appears and asks **Give width for the conical extension** or **Give diameter for the conical extension**. In case of **Width** you can select between relative **By Width** and absolute **To Width**. In case of **By Width** the input can be negative in order to get a smaller roll.

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.**

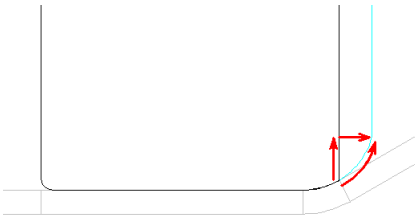
The input window appears and asks **Give width for the cylindrical extension**. Select between relative **By Width** and absolute **To Width**. In case of **By Width** the input can be negative in order to get a smaller roll.

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.

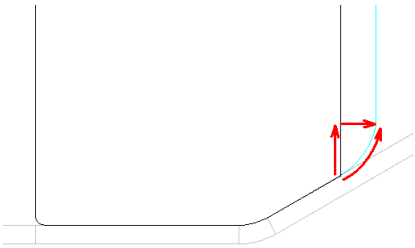
Hint:
Alternatively you can append a Conical Extension or an Arched Extension.

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 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)



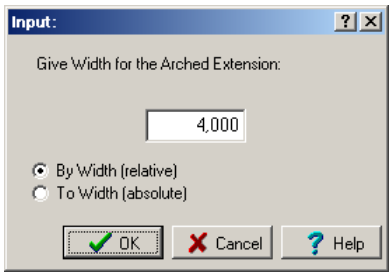
If the roll ends with a line, an arched extension can be added with a selectable radius. In this case the arched extension always is convex, this means it points to the centerline of the roll.

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.



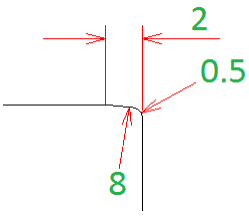
If the roll ends with a line, an additional input window appears and asks **Give radius for the arched extension**.

Principle of operation

The arched extension will be attaches 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. If the input value is too large, a message appears and the value is shortened so that the arc merges to the roll flank tangentially.

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.

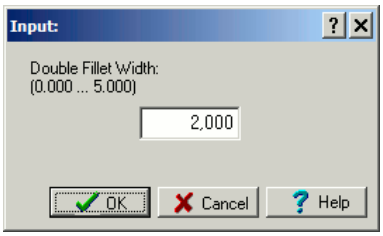
Application spectrum

- 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:

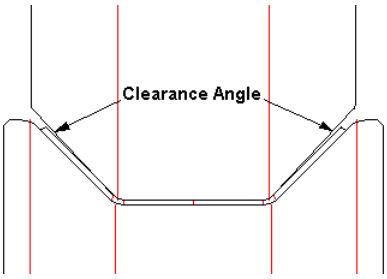
- Width >= 1.5mm (0,075in): Large radius = 8mm (0,4in), small radius = 0.5mm (0.025in).
- Width < 1.5mm (0,075in): Large radius = 5mm (0,25in), small radius = 0.3mm (0.015in).

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 a 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

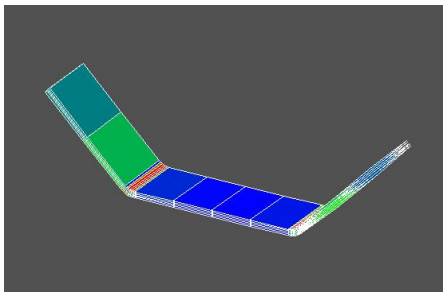
Each roll has a Working Diameter (pitch diameter), this is the diameter at the profile web (the horizontal part of the profile, see picture above). If the Gear Transmission Ratio of the machine is 1:1, top and bottom rolls should have the same working diameter in order to have equal circumferential speeds 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

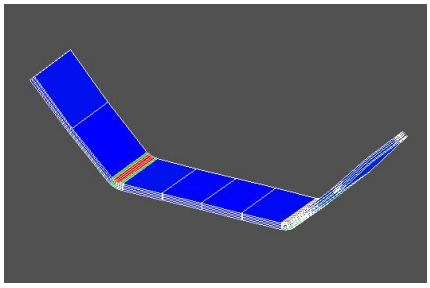
Imagine, the flat sheets runs into a stand like in the picture above. First it will get contact with the roll surfaces that have the largest diameters. The top roll has its largest diameter at the center cylindrical part, whereas the bottom roll has it at the side shoulders. By turning on the top roll 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 archived 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 consist of four angles is bent per side.

Effect of the clearance angle



In order to show the effect of the clearance angle, running of a flat sheet into a stand with 45 degree rolls was simulated by using the FEA (Finite Element Analysis) method. Without clearance angles considerable strain is shown in the legs (green: medium strain).

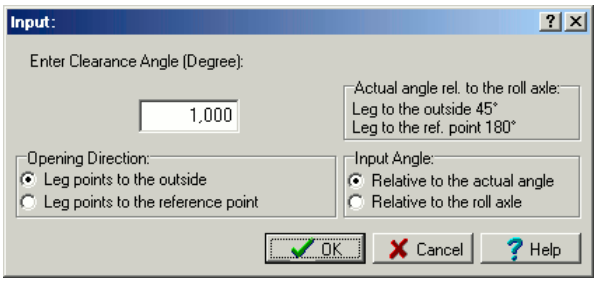


With clearance angles the legs are not strained (blue color). The red color shows high plastic strain in the bending zones, which is necessary that the profile does not spring back after leaving the machine.

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)**.

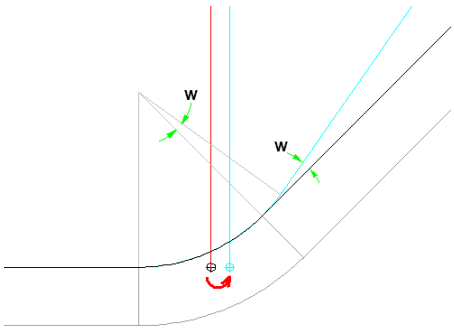
Opening Direction: In most cases the legs of the clearance angle point to the outside direction (view from the reference point). This direction **Legs point to the outside** is predefined in **PROFIL**. If

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 help defining the right value for the input field. Pay attention on the sign, it must be entered, too.

Principle of operation

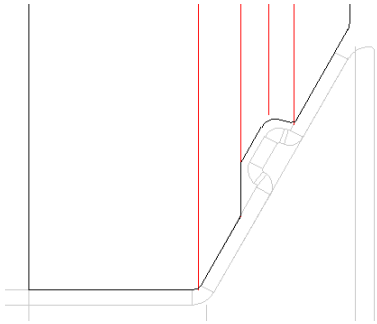


In order that the fillet of the roll clings to the profile, **PROFIL** keeps the center of the arc constant; the angle of the arc is extended or reduced by the clearance angle **w**. The Roll Corner Point (intersection point of the tangents) moves accordingly.
A positive clearance angle creates or extends 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 created between two Roll Corner Points or for the whole roll.

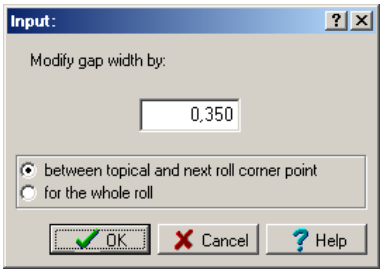
Application spectrum

- 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:

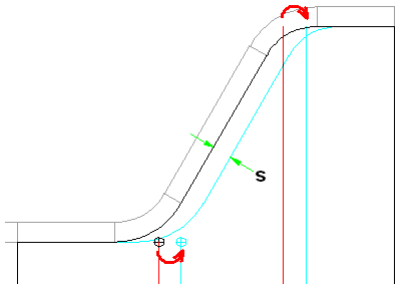
- Main menu: **Rolls, Gap**.
- Context menu (right mouse button click on the corner of the desired roll in the Drawing Area): **Gap**.



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.

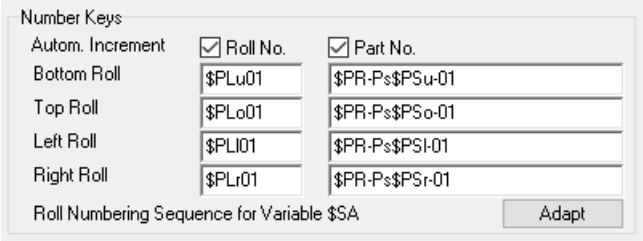
Calling the function

Call this function optionally by:

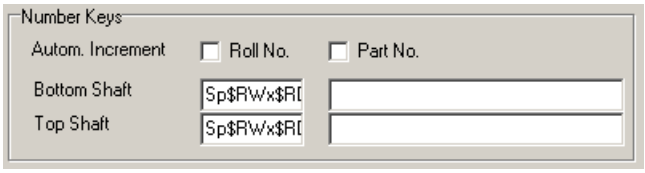
- Main menu: **Rolls, Renumber**.
- Context menu (right mouse button click on the left or right corner of the desired roll in the Drawing Area): **Renumber**.

Principle of operation

Renumbering works dependent on the **Number Keys**. For the forming rolls, the number keys in Settings Rolls are used:



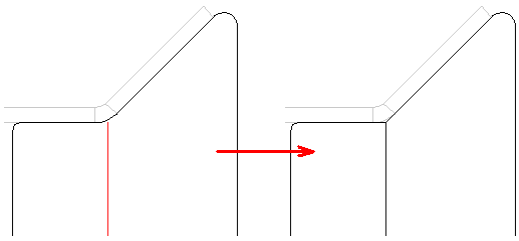
For the spacer rolls, the number keys in Settings Spacer Rolls are used:



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**.

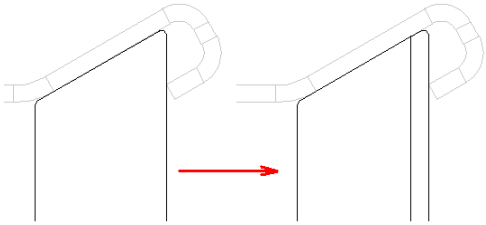
- Context menu (right mouse button click on the left or right corner of the desired roll in the Drawing Area): **Split at Corner**.

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.

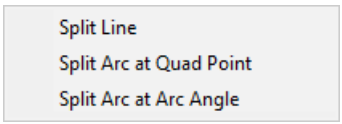
3.1.4.13 Split between Corners



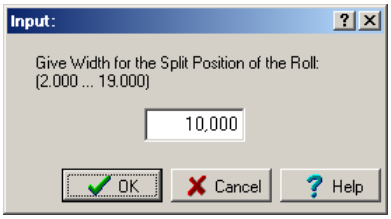
Use this function to split a roll between two Roll Corner Points.

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**.

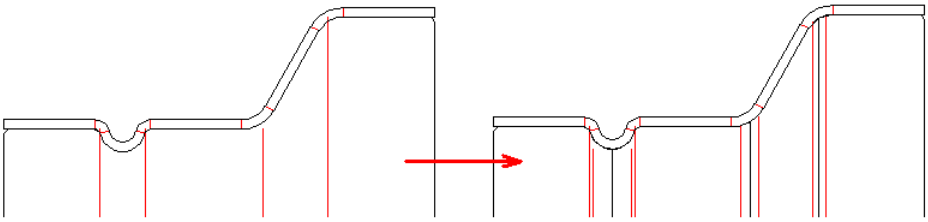


Split Line:



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.

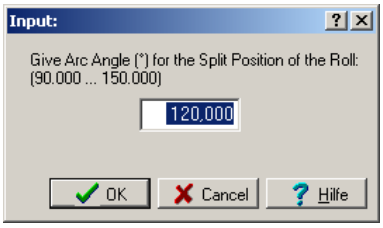
Split Arc at Quad Point:



Split at Quad Point (left) and at Arc Angle (right)

The split point is the quadrant point of the arc, i.e. the arc point of the maximum or minimum roll diameter.

Split Arc at Arc Angle:



The input window appears with the question **Give Arc Angle (°) for the Split Position of the Roll** and an admissible range of values for the input, which begins at the start angle and the end angle. In the input field appears the mean value between start and end angle as a proposal. You can modify this proposal or overwrite it.

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**.

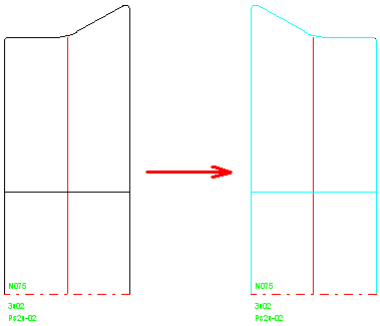
Principle of operation

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:

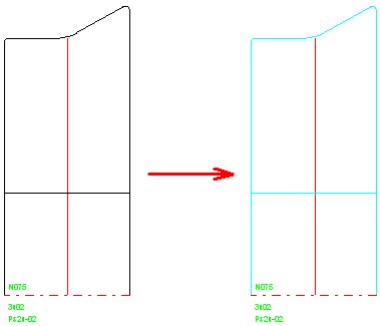
- Main menu: **Rolls, Turn**.
- Context menu (right mouse button click on the certain corner point of the desired roll in the Drawing Area): **Turn**.

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

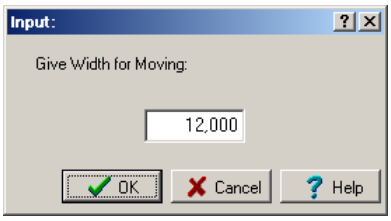


Use this function to move a certain roll or all rolls on the shaft.

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

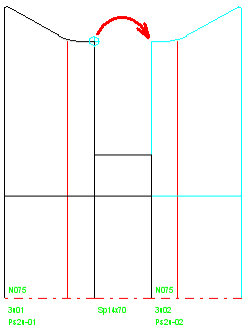
This function creates a mirrored roll. By selecting the roll flank, you can define if the mirrored roll should be created at the right or left outer edge of the roll set on the same shaft/axle. As desired, the roll can be mirrored at the Roll Reference Point or at any position. The roll also can be mirrored to the opposite shaft/axle.

Calling the function

Before you call this function, identify the roll by clicking on a Roll Corner Point of the roll that should be mirrored. If you want to mirror **At Roll Flank**, 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. Doing so, you can mirror any roll of a roll set to the right or left outer edge. 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**.

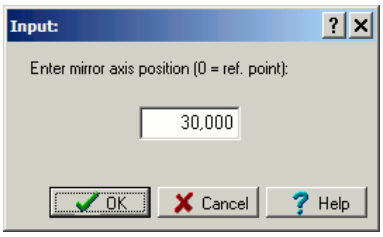
Submenu: Select from the submenu:



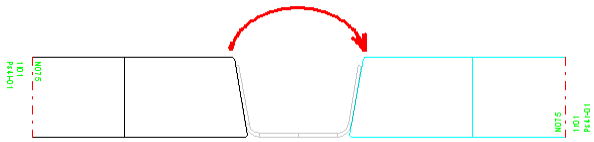
At Roll Flank:
Use this function to mirror a roll at the right or left outer edge of a roll set. Whether the roll is mirrored to the left or to the right, you have defined by clicking on the left or right Roll Corner Point previously.

At Ref. Point:

This function creates a new roll by mirroring at the Roll Reference Point. This is useful for easier designing rolls for symmetrical profiles: create the rolls for one half only and mirror them afterwards. Select the origin roll by clicking on any roll corner point.



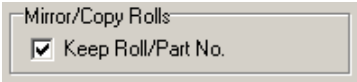
At any Position:
This function creates a new roll by mirroring at any position. An input window is opened and you can enter width coordinate. Input 0 means reference point. Select the origin roll by clicking on any roll corner point.



To the opposite Axle:
This function creates a mirrored roll on the opposite shaft/axle. This means, a left side roll will be mirrored to a right side roll and vice versa. Also a roll on the bottom shaft can be mirrored to the top shaft and vice versa This simplifies designing rolls for symmetrical profiles. Select the origin roll by clicking on any roll corner point.

Principle of operation

The new roll with same size is created dependent 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 Settings, 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 that way you also can duplicate a roll. Furthermore, in that 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 that way you also can duplicate a roll. Furthermore, in that 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 that 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.

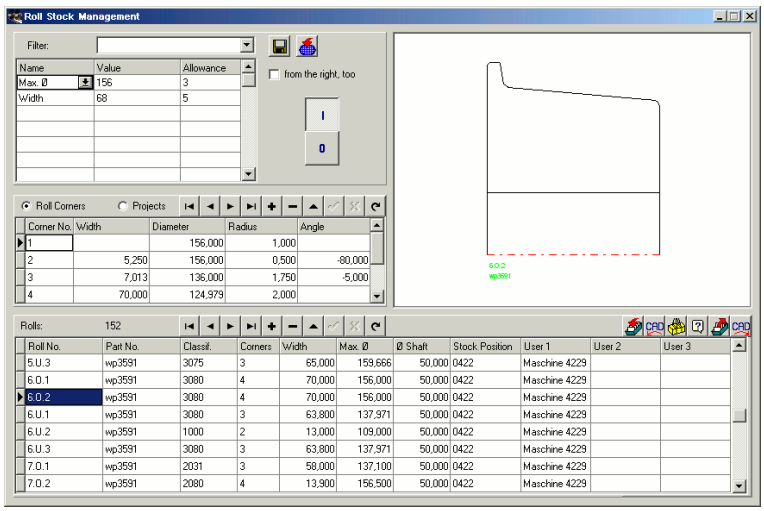
Calling the function

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.

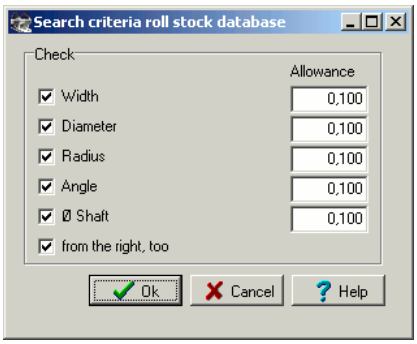
Calling the function

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 Search

Use this function to check if a designed roll (within your project) already exists in the roll stock. If a roll exists with same or similar properties, the designed roll can be replaced by the stock roll for re-use. For replacing, use the function **Replace roll in project by roll from roll stock** in the Roll Table.



Before calling this function, select the designed roll in the project that should be replaced by a stock roll. After calling this function the window **Search criteria roll stock database** opens with these entries:
Width: Valid for widths of all roll corner points.

- **Diameter:** Valid for diameters of all roll corner points.
- **Radius:** Valid for radii of all roll corner points.
- **Angle:** Valid for contour angles between all roll corner points to the next.
- **Ø Shaft**
- **from the right, too:** Means that also rolls are found that are saved mirrored.

Select by checking the boxes which search criteria should be used. For each criterion, you can enter an allowance. Avoid allowance 0, because unsafe results can occur. After pressing the **Ok** button, the Roll Stock Window opens and shows the rolls that fit to the given criteria. If no roll is shown, uncheck one or more check boxes or increase the allowances in order to make the search more tolerant. E.g. if you uncheck the criterion **Diameter**, rolls are shown that have the desired contour but have a different diameter. You can re-use this roll nevertheless by modifying the height adjustment of the shaft.

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.

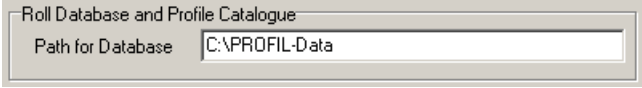
Stock Management Save

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:

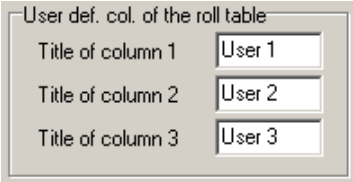
- **Roll:** The marked roll is stored.
- **Shaft:** All rolls of a shaft are stored. Define the shaft by marking any roll of the shaft.
- **Stand:** All rolls of all shafts of one stand are stored.
- **Project:** All rolls of all stands of the whole project are stored.

If the window Roll Stock Management is not yet open, it will be opened after calling one of these functions.

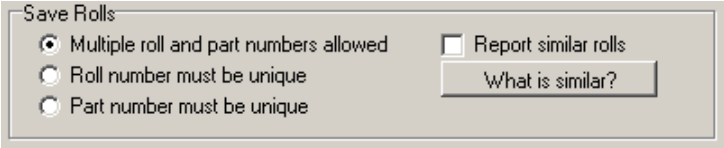
Setup



Set the path to the roll database in Settings Stock Management.



Title of column 1..3: Enter in Settings Stock Management the titles for the last 3 columns of the Roll Table. You can define these columns for your own needs.



Save Rolls: In Settings Stock Management, you can define wether multiple roll and part numbers are allowed or if the must be unique.Furthermore you can check if similar rolls already exist in the roll stock before saving a roll. In **What is similar?** you can enter the criteria for for searching similar rolls.

3.1.4.23 Corner

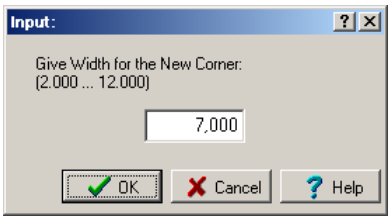
3.1.4.23.1 Append

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**.



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

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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:

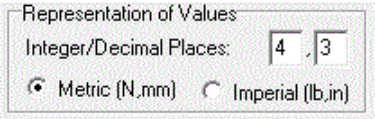
- Main menu: **Calculate, Statics**.
- Button **Statics** in the Button Key Bar.

Principle of operation

The **Statics Window** is opened and shows these statics parameters:

Reference Point	Centroid
In Principal Axes	Shear Center Point
Moments of Inertia	Moments of Resistance
Max. Distance	Inertial Radii
Area of Cross	Weight
Principal Axes Angle	Vault Resistance
Torsion Moment	

Setup



Select in Settings 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 unsymmetrical 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:

$$\partial = \sqrt{\frac{i_{min}}{A}}$$

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/dm3).

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 unsymmetrical 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):

$$\tau_t = \frac{M_t}{W_t} = \frac{M_t \cdot e_{max}}{J_t}$$

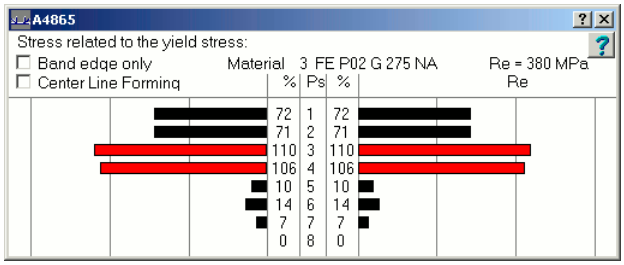
M_t = applied torque,
J_t = torsion moment,
e_{max} = maximum distance

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



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.

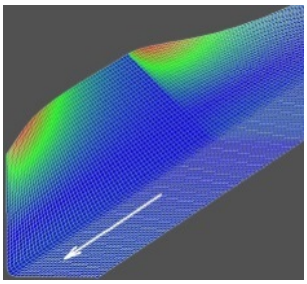
Principle of operation

The window **Stress of Band Edge** is opened. The strains are calculated by using an approach model that depends on an empirical analysis. The results are converted to relative stress related to the yield point and are displayed in a bar diagram. The header shows the Material entered in the Project Data Window and the stress of the yield point for the selected material (Re). The calculation is based on the mesh settings of **PSA (Profile Stress Analysis)**. Please check, if in View, PSA a proper meshing is shown. If not, please adapt the shell size in Settings, PSA.

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, also called downstream).

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 she 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.



After calling the function, the profile cross section is meshed considering the PSA meshing parameters, preset in Settings PSA. In order to check if the preset meshing parameters are suitable for the profile, call View PSA - Profile Stress Analysis. Afterwards a special 3D curve is calculated for each longitudinal fiber that approximates the real course of the band edge. From the length of this

curve, PROFIL calculates the strain and the relative stress by considering Young's modulus and yield stress.

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 or the first and the last bending step),
- in the first and the last passes the stress should be smaller than in the other passes.

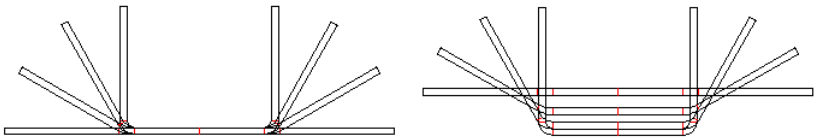
You define by using the switch Profile, Loaded, whether the stress of the band edge is calculated for the discharged or the loaded state of the profile. The status of the switch is displayed by a coloured background of the table fields in the profile list window.

Band edge only: If selected, the stress is calculated at the left and right band edge only. Edge waviness and bow down (symmetrical profiles) and twist (unsymmetrical profiles) mostly is caused by exceeding the yield stress at the band edge. If not selected, the stress is calculated in the whole profile cross section.. This is important to avoid center waviness and ripples already in the design phase.

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.
- If the calculation is too slow, possibly a too high resolution is preset in Settings PSA.

3.1.5.2.1 Center Line Forming



Bottom line forming (left) and center line forming (right)
with lowering to constant centroid's height (100%)

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

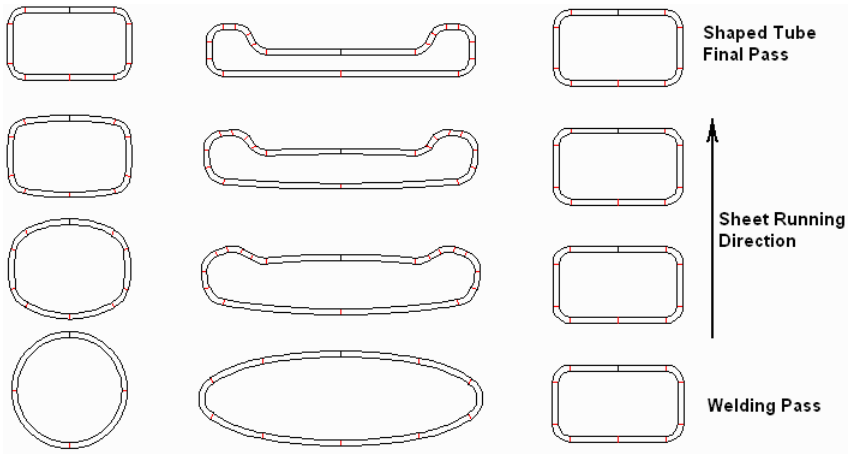
Center Line Forming is a checkbox in the window Calculate, Stress of Edge. For a preview, switch the calculation method for the stress of the band edge from bottom line forming, i.e. constant level of the web of the profile, to center line forming and back again. So you quickly can check, if center line 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.

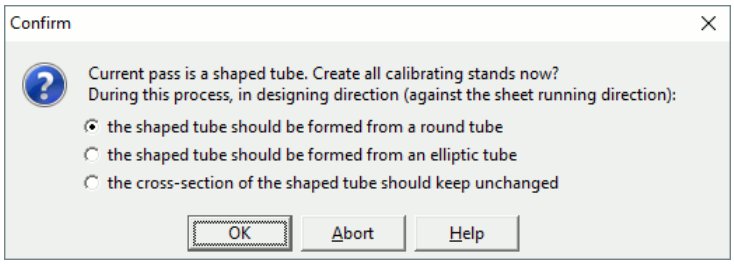
Calling the function

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 from 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 from an elliptic tube.** This method should 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 a 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 or 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.

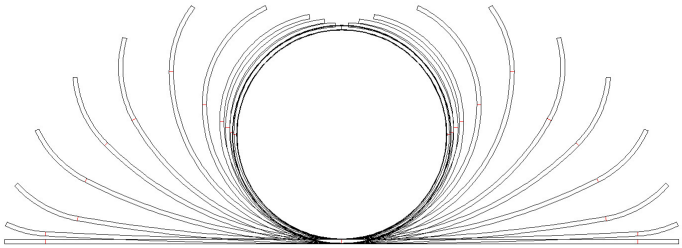
The Reference Points (x0/y0) both for the shaped tube and for all calibration passes are modified in that way that the centroids of the area are in the center of the round tube of the welding station.

Hints:
- Any cross section can be used. Thus this function is also suitable for calibrating (sizing) of round tubes. In this case the round cross section keeps unchanged in all calibration stands. The tube changes its diameter dependent on the calibrating factor.
- If you selected **round tube**, create the tube flower pattern by using the automatic Round Tube Forming or use the toolbox 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 Round Tube Forming



This function creates the flower pattern for a given round tube automatically. The flower pattern consists of

- the round tube that is designed by the user,
- the passes for a given count of fin passes,
- the passes for the break down stands.

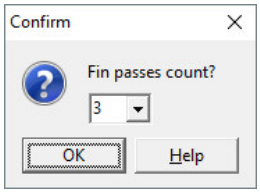
Calling the function

Before you call this function, define the final round tube. Use either - or:

- The function Welding Pass from the Toolbox Tube Design, if the final product should be a welded round tube.
- The welded tube that has been created by the function Calculate, Shaped Tube Calibration from a shaped tube.

Select the welding pass.
Call this function by:

- Main menu: **Calculate, Round Tube Forming**.



After checking if the selected cross-section is a closed round tube, a dialog window is opened with the question: **Fin passes count?** Enter the desired count.

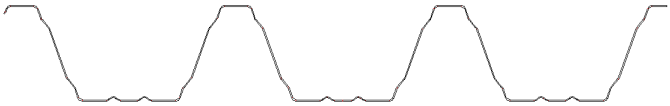
Principle of operation

Starting with the current profile the passes for flower pattern for all stands in the defined machine data are created automatically until the flat sheet is reached. If profile lists with a number higher than the current yet exist, you are asked if they can be deleted.

Hints:

- After creation of the tube flower pattern use the function Center Line Forming to lower the profile web in order to get a constant centroid of all passes (center of the final tube).
- By using these functions of the Toolbox Tube Design you can optimize the tube flower pattern:
  - Fin Pass
  - Break Down Pass
  - Break Down Pass, W-Forming
- Afterwards create the rolls for the tube forming passes by using these functions of the Toolbox Tube Design:
  - 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

3.1.5.5 Trapezoidal Profile Forming



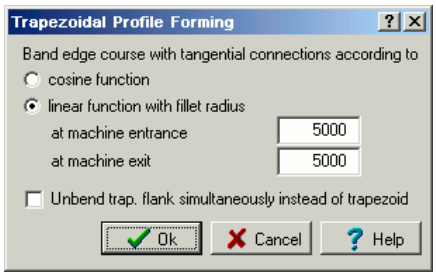
When wide sheet metal strips are roll formed to a trapezoidal profile, the course of the band edge has a crucial influence on the quality of the product. On the one hand the course must be as short as possible in order to form the band edge elastically only. The shortest connection between two points 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).

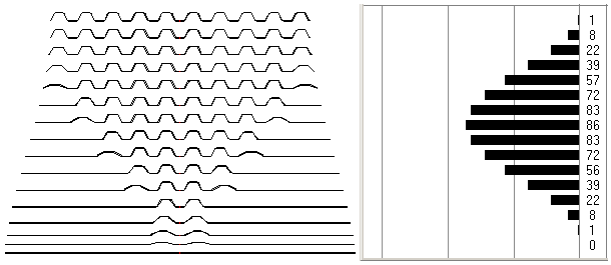
Calling the function

Before you call this function, define the final trapezoidal profile by the function Trapeziodal 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:

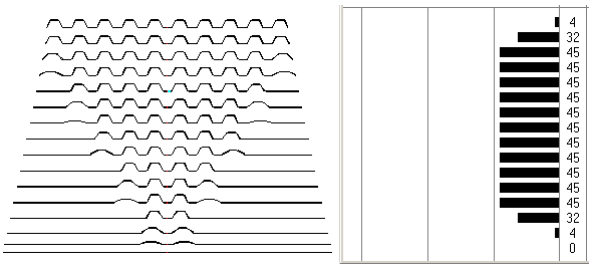
- Main menu: **Calculate, Trapezoidal Profile Forming**.



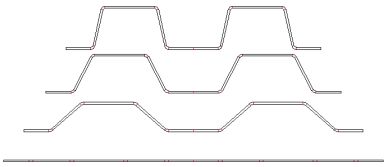
A dialog window is opened with the question: **Band edge course with tangential connections according to:**



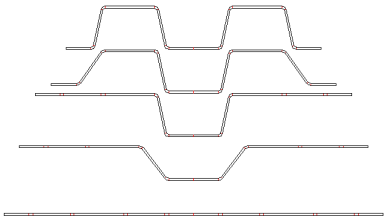
- **Cosine function:** The **cosine band edge course** (left) creates a **sine course of the stress of band edge** (right). The advantage of this method is the very smooth transition both from the entering flat sheet and to the ready product leaving the machine.



- **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.



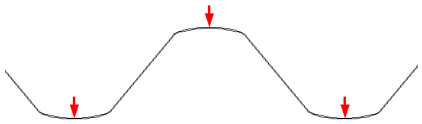
The **trapezoid's flank is formed simultaneously**: more stands are necessary, the material can better "flow" to the inside. Problem: the band edge moves up and down, this causes higher stress. The stress can be reduced by Center Line Forming.

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 trpezoid. 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.

3.1.5.6 Required Count of Stands

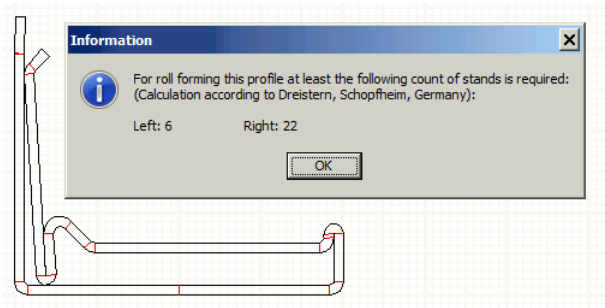
During processing a quotation and a cost assessment the cost of a roll tool set or a roll forming machine must be estimated roughly. The expected count of passes or stands for a given profile cross section is needed for this operation.

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



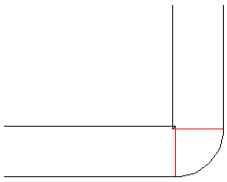
A window opens and shows the estimated count of stands for the left and right half of the profile. Left side is the side left of the Reference Point X0/Y0, provided that the Start Direction points to the right. The calculation is according the method of the company **Dreistern, Germany**.

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.

3.1.5.7 Plausibility Check

This function is useful to check the whole project on:



Negative inner radius

- Errors in the logical structure of the profile list, e.g.: empty profile elements, negative radii, existence of P or PS, both P and PS existing, missing distance between stands.

Arc overlap

- Overlap of roll contour arcs.
- Missing shaft diameter
- Double and unnecessary roll corner points
- The first profile list (in sheet running direction) does not contain the flat sheet or contains rolls

Calling the function

Call this function by:

- Main menu: **Calculate, Plausibility Check**.
-  Button **Plausibility Check** in the Button Key Bar.

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.
The Button Key Bar shows:

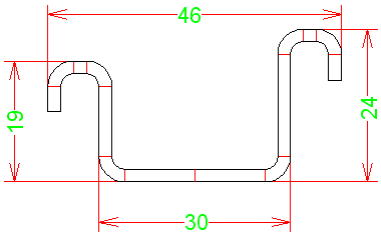
-  Error found in the project.
-  Plausibility Check is necessary.
-  Plausibility Check finished, Project is error-free.

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

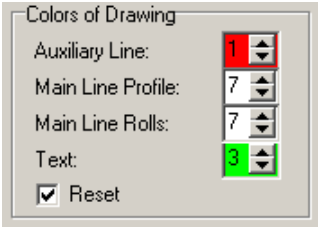
Select the pass that should be displayed. Call this function optionally by:

- 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



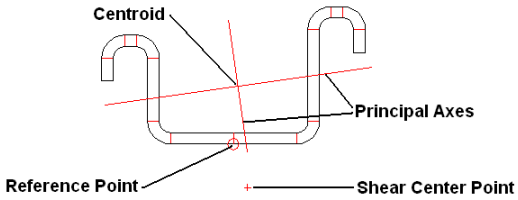
Select the **Auxiliary Line Color** and the **Main Line Color Profile** in Settings Drawing.

In Settings 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.

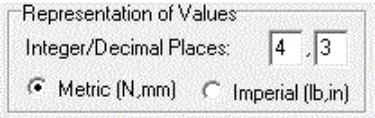
Principle of operation

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:

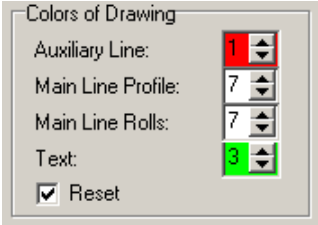
- Centroid = Origin of the **Principal Axes**
- In Principal Axes = Large axes system
- Max. Distance = Lengths of the **Principal Axes**
- Principal Axes Angle = Angle of the larger axis of the **Principal Axes** against the horizontal line
- Shear Center Point = small cross
- Reference Point = small 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 Settings 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.

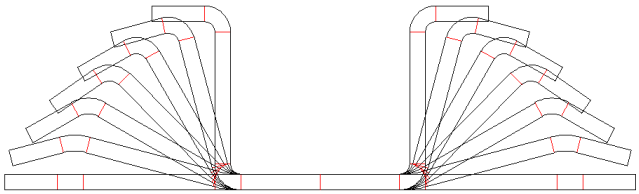


Select the **Auxiliary Line Color**, **Main Line Color Profile**, and **Text Color** in Settings Drawing.

Preset the text height for the table of statics in Settings 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

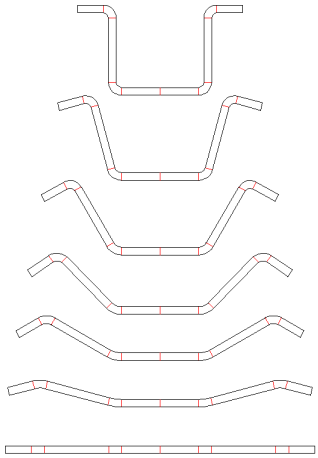
Call this function optionally by:
- Main menu: **View, Flower, Nested**.
- Button **View Flower Nested** in the Button Key Bar.

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



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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

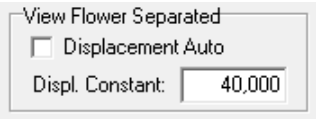
Call this function optionally by:

- 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.

Setup

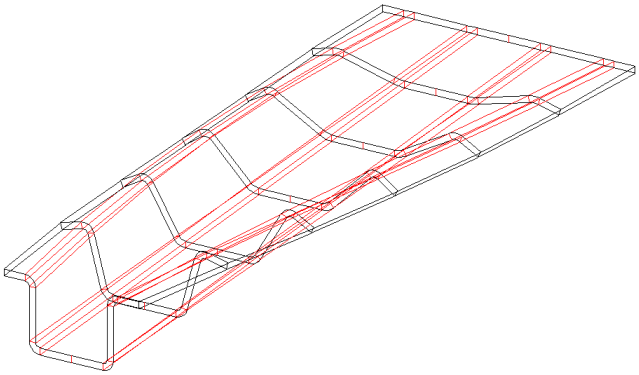


Select in Settings Drawing whether the vertical displacement of the passes should be set automatically for a compact representation or if you want to set the displacement to a constant value.

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

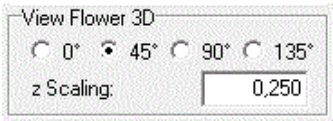
Call this function optionally by:

- 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

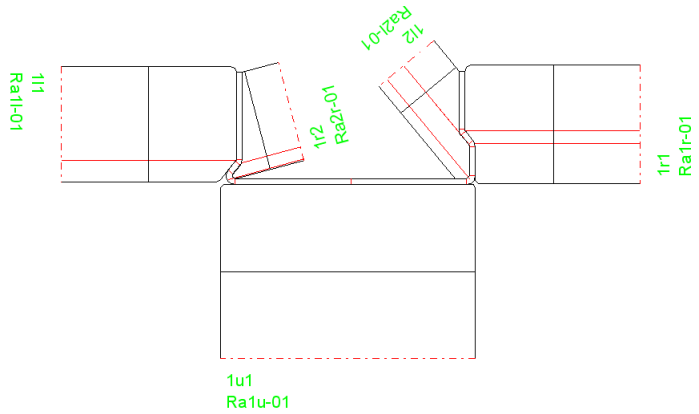


Select the view port angle, and the scaling factor of the z axis in Settings Drawing. The settings also can be modified in the context menu (right mouse button).

Hints:

- The Navigator helps you to zoom, to move and to fit the drawing.
- By using the Navigator 3Dyou 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

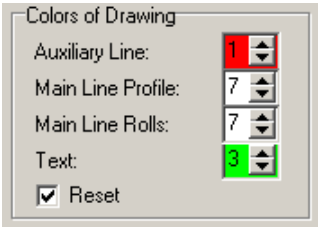
Select the roll stand that should be displayed. Call this function optionally by:

- Main menu: **View, Roll Tools**.
- Button **View Roll Tools** in the Button Key Bar.

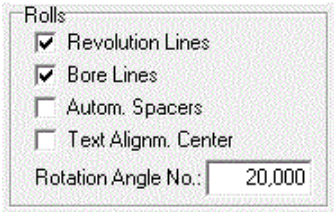
Principle of operation

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 Settings Drawing.



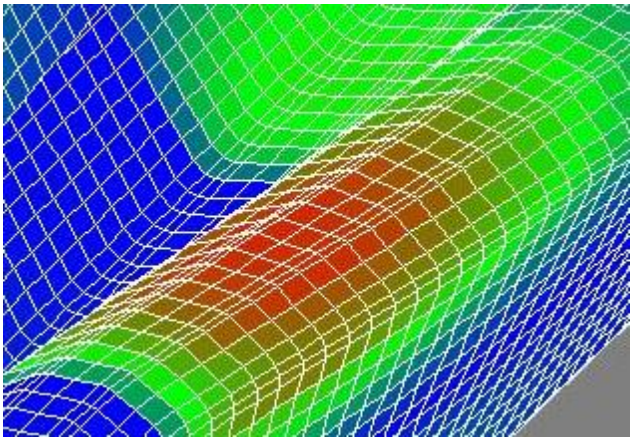
In the same dialog, you can define if the roll drawing should contain **Revolution Lines**, **Bore Lines**, and **Automatic Spacers**. **Text Alignment Center** positions roll number and part number in the center of a roll instead on the left corner. Rotation Angle Numbers: Long roll and part numbers can obscure each other. In order to prevent it they are rotated automatically. The **Rotation Angle** can be preset here.

Preset the text height for the roll number and part number in Settings 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

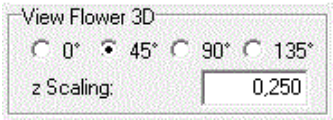
A 3D drawing is created in the Drawing Area that shows the strip while running through the roll forming machine.

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 Settings 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



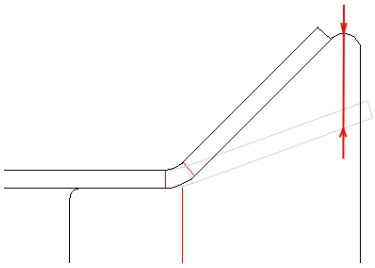
Select the initial view port angle in Settings Drawing or modify the angle by using the context menu (right mouse button).

Hints:
- The Navigator helps you to zoom, to move and to fit the drawing.
- By using the Navigator 3Dyou 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.

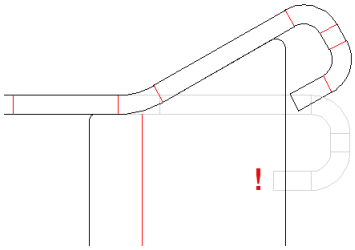
3.1.6.8 Show

3.1.6.8.1 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 easy to find out which point of the running-in profile touches first which roll of the current stand. I 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.

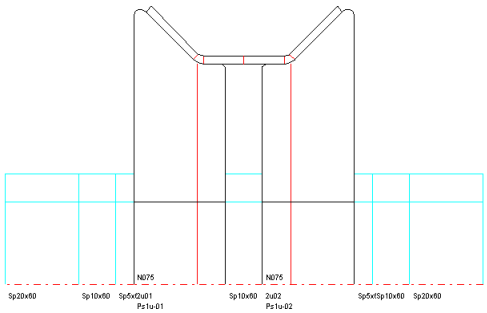
Calling the function

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 Settings, 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 Settings, Drawing, Colors of drawing area). Printing and transfer to CAD is not possible.

3.1.6.8.2 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**.

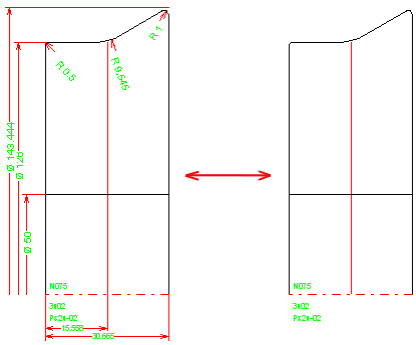
Principle of operation

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.

3.1.6.8.3 Dimensioning



In case the dimensioning should be not visible, it can be switched off.

Calling the function

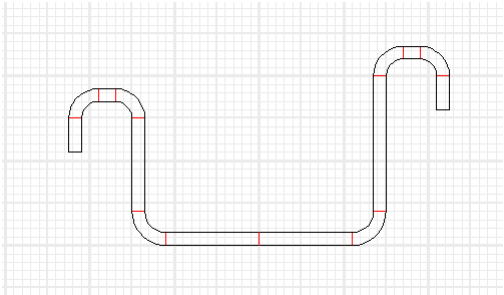
Call this function by:

- 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).

3.1.6.8.4 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 Settings, Drawing the desired grid lines distance.
- Preset in Settings, 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

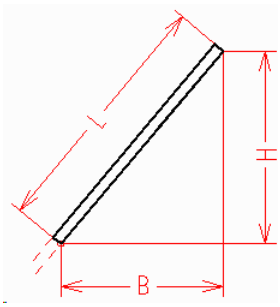
Hat-Profile

Z-Profile

Hints:

- In certain input fields the values can be modified gradually by pressing the **Pg Up/Dn** keys on the keyboard. The step interval can be preset in Options, Mouse.
- During a preview is shown in the Drawing Area. The preview disappears in case of pressing the **Cancel** button. After pressing the **Ok** button, the preview remains valid in the project.

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.

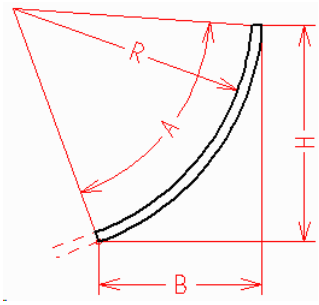
The window **Line** is opened. If you enter one of the three variables **Length**, **Width** and **Height**, the other two will be calculated. The length is always positive.

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.

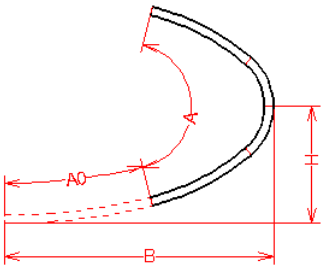
The window **Arc** is opened. If you enter **Angle** and **Radius**, width and height will be calculated. If the angle is negative, an arc to the right will be generated. If you enter **Width** and **Height**, angle and radius (and direction) will be calculated. The length is always positive.

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.

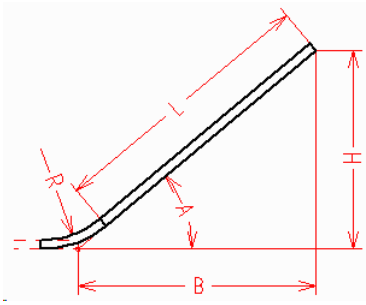
Enter the half **Major Axis Length B** and the half **Minor Axis Length H**. The major axis length always must be larger than the minor axis length. In case of
H > 0: A counter clockwise partial ellipse is created.
H < 0: A clockwise partial ellipse is created.

Enter the **Start Angle A0** and the **Apex Angle A** of the partial ellipse. In case of
A0 = 0° or 180°: The partial ellipse starts at the minor axis.
A0 = 90° or 270°: 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°.

Calling the function

Call this function by:

- Button **Arc <90° - Line** from the Toolbox Profile Design.

The window **Arc <90° - Line** is opened.

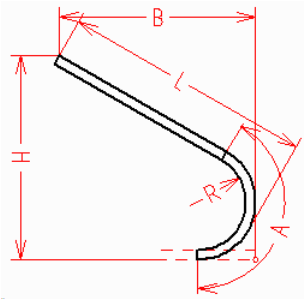
If you enter **Angle** and **Length**, width and height will be calculated. If you enter **Angle** and **Width**, length and height will be calculated. If you enter **Angle** and **Height**, length and width will be calculated. If you enter two of the three variables length, width and height, all other 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.

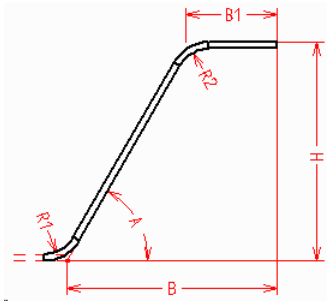
If you enter **Angle** and **Length**, width and height will be calculated. If you enter **Angle** and **Width**, length and height will be calculated. If you enter **Angle** and **Height**, length and width will be calculated. If you enter two of the three variables length, width and height, all other 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.

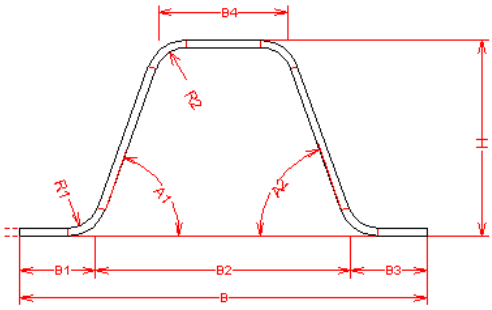
If you select **Angle** and **lower Width**, the height will be calculated. If you select **Angle** and **Height**, the lower width will be calculated. If you select **Width** and **Height**, the angle will be calculated. Angle and height can be negative too; in this case first an arc to the right and 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 Trapeziodal 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:

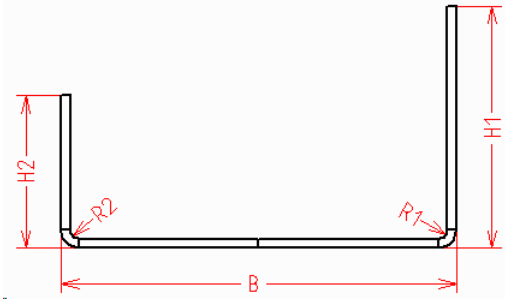
- Button **Trapeziodal Profile** from the Toolbox Profile Design.

The window **Trapeziodal 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 unsymmetrical.

Calling the function

Call this function by:

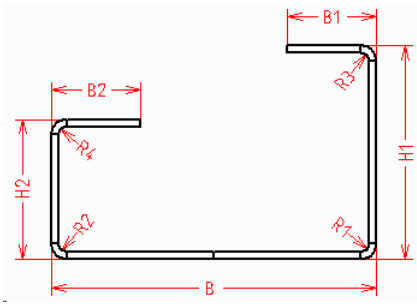
- Button **U-Profile** from the Toolbox Profile Design.

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 unsymmetrical.

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 unsymmetrical.

Calling the function

Call this function by:

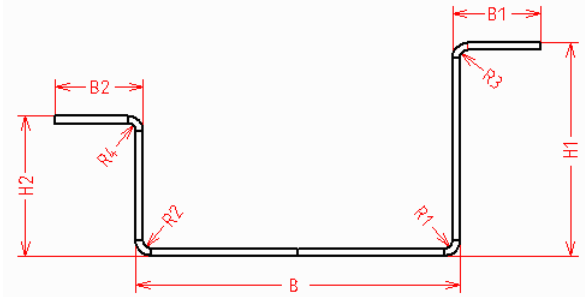
- Button **C-Profile** from the Toolbox Profile Design.

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 unsymmetrical.

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 unsymmetrical.

Calling the function

Call this function by:

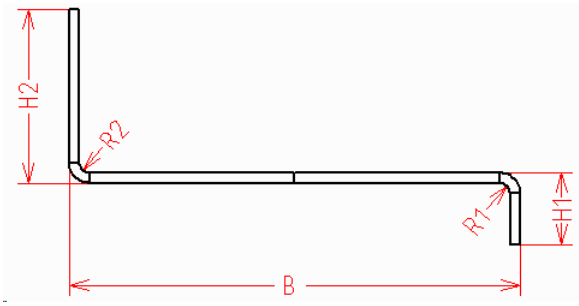
- Button **Hat-Profile** from the Toolbox Profile Design.

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 unsymmetrical.

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.

Calling the function

Call this function by:

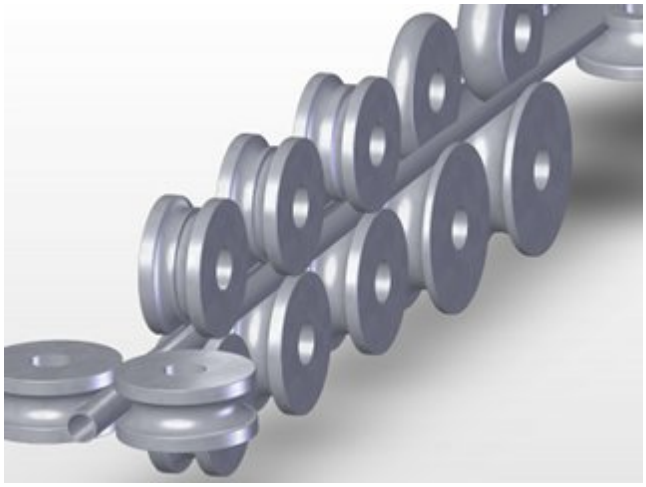
- Button **Z-Profile** from the Toolbox Profile Design.

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

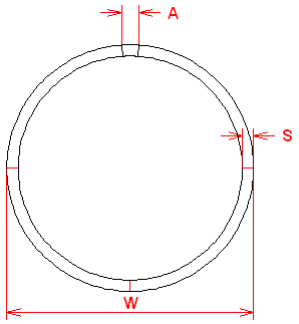
With other kinds of profile lists the Toolbox Tube Design will not work. Modifications by hand or be the Toolbox Modify are allowed, however.

- 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.

Hints:
- In certain input fields the values can be modified gradually by pressing the **Pg Up/Dn** keys on the keyboard. The step interval can be preset in Options, Mouse.
- During input a preview is shown in the Drawing Area. The preview disappears in case of pressing the **Cancel** button. After pressing the **Ok** button, the preview remains valid in the project.

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.

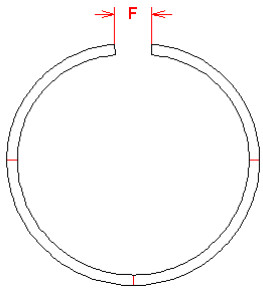
The window **Welding Pass** is opened. Enter these parameters into the input fields:
- **tube diameter D**,
- **sheet thickness s**,
- **addition for welding A**.

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** (Flange Gap) 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.

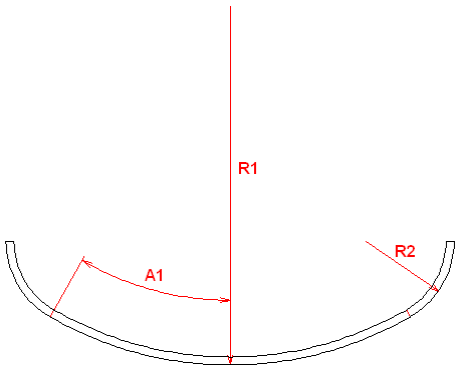
If a Calibrating Factor is preset in the Machine Data of the appropriate stand, a window opens with the question **For this fin pass a calibration factor (%) is provided in the machine data. Consider it for the developed length?** If you agree, the Strip Width is increased dependent on this factor. So in sheet running direction (also called downstream), the strip width will be decreased.

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 th **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.

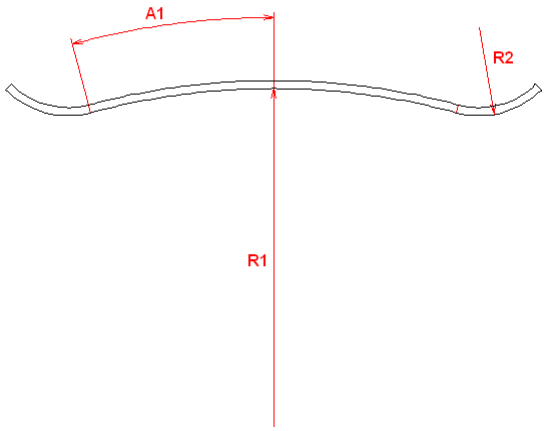
If a Calibrating Factor is preset in the Machine Data of the appropriate stand, a window opens with the question **For this break down pass a calibration factor (%) is provided in the machine data. Consider it for the developed length?** If you agree, the Strip Width is increased dependent on this factor. So in sheet running direction (also called downstream), the strip width will be decreased.

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 a Calibrating Factor is preset in the Machine Data of the appropriate stand, a window opens with the question **For this break down pass a calibration factor (%) is provided in the machine data. Consider it for the developed length?** If you agree, the Strip Width is increased dependent

on this factor. So in sheet running direction (also called downstream), the strip width will be decreased.

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

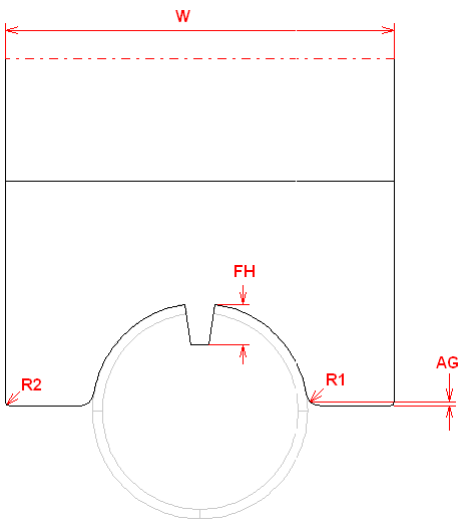
After pressing the **Ok**-button the tube profile for the break down pass is entered into the profile list. The first arc segment (at the bottom) has the desired angle and radius. The second one (at the outside) has the desired radius and its angle is calculated on this way that the developed length 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.

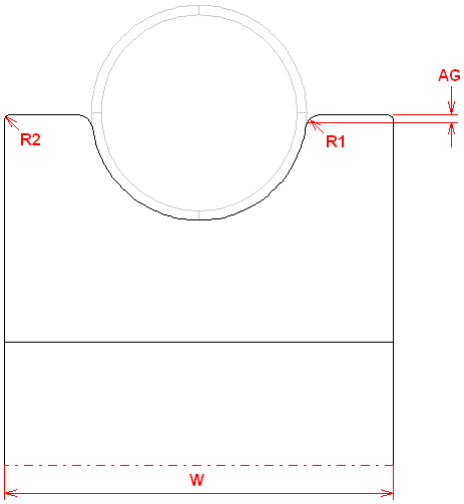
- **Fillet Radii R1, R2:** Select a fillet radius at the inside (at the tube) and a fillet radius at the outside (at the roll edges).
- **Fin Height FH:** Enter the fin height (Remark: The fin width is set in Fin Pass). If the opening in the tube is less than 1 mm or 0.1 in, no fin is created.
- **Half Air Gap AG:** Enter the desired vertical air gap (also called flange 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:

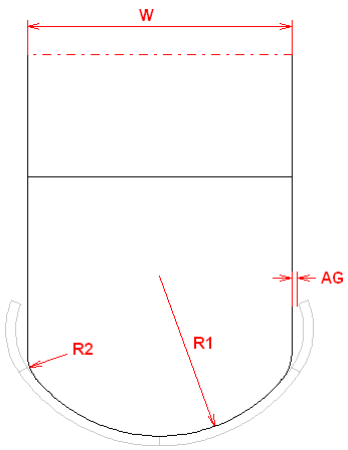
- **Total Width W:** If your entered value is too small, the width is set automatically to a minimum width that has enough space for all fillet radii. The result is rounded to 10 mm or 1 in.
- **Fillet Radii R1, R2:** Select a fillet radius at the inside (at the tube) and a fillet radius at the outside (at the roll edges).
- **Half Air Gap AG:** Enter the desired vertical air gap (also called flange 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:

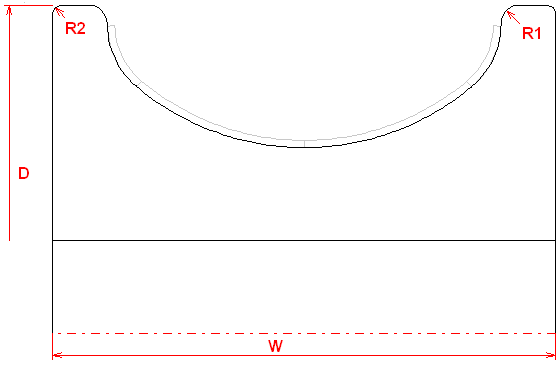
- **Total Width W:** This input is only necessary, if the roll should become smaller than the opening of the tube. If your entered value too big, your input is ignored and the width is set automatically to a maximum width by considering the desired air gaps at both sides (if the edges of the tube are 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:

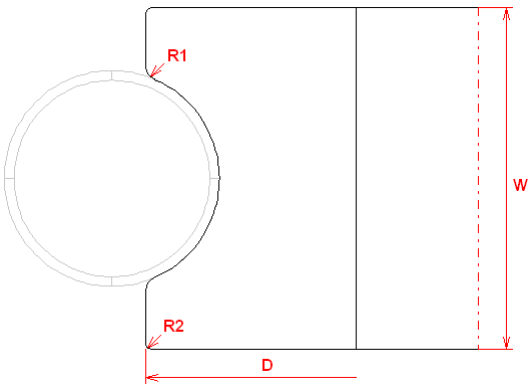
- **Roll Diameter D:** If your entered diameter is too small, that the roll cannot touch the tube, an error message appears.
- **Total Width W:** If your entered value is too small, the width is set automatically to a minimum width that has enough space for all fillet radii. The result is rounded to 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:

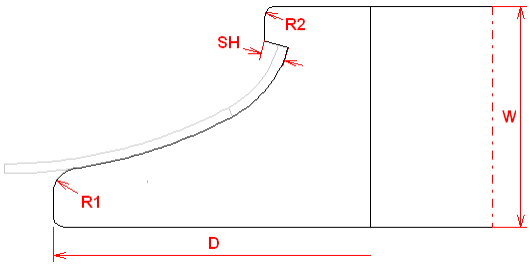
- **Roll Diameter D:** If your entered diameter is too small, that the roll cannot touch the tube, an error message appears. If it is too big, the diameter is set automatically that both side rolls touch together.
- **Total Width W:** If your entered value is too small, the width is set automatically 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.

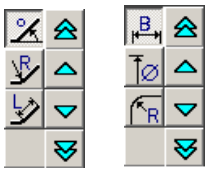
3.1.7.3 Modify

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, dependent 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.

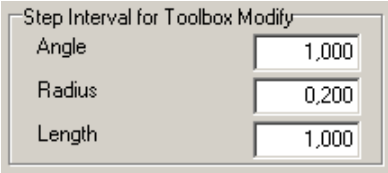
Width/Diameter/Radius

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 Settings 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 Settings 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 snap 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:

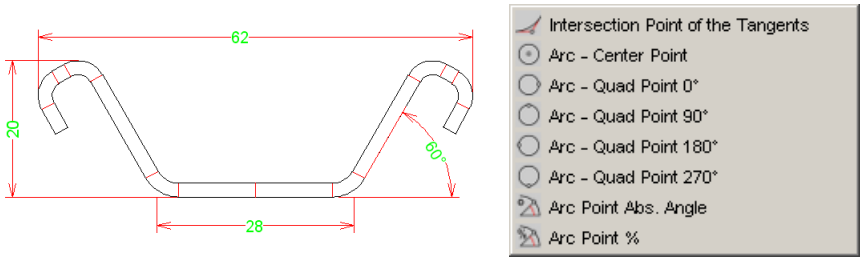
After you created a dimensioning, the dimensioning function starts again. This is useful for creating series of dimensioning of the same kind. For breaking off the function use the Esc key or select another dimensioning function.

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 boarders is possible (e.g. distance between shafts). The created dimensioning is visible only, when no roll is selected.

Snap points in the context menu:

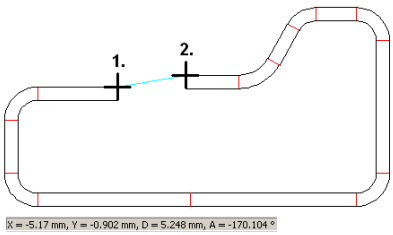


Examples for dimensioning by using the context menu

Open 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 snap 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:

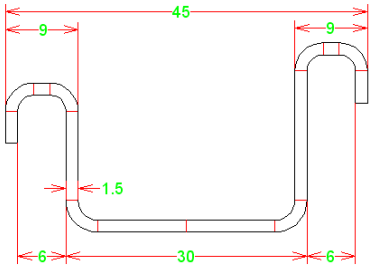
x = ... y = ... d = ... w = ...°

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.

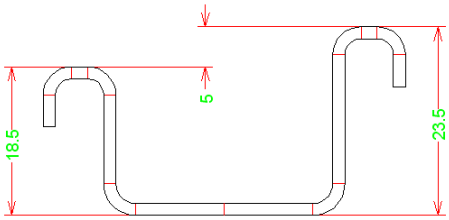
After calling this function you are asked to enter:

- **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.

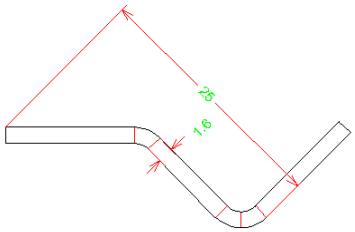
After calling this function you are asked to enter:

- **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.

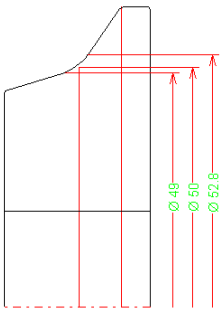
After calling this function you are asked to enter:

- **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 creates 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 don'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:

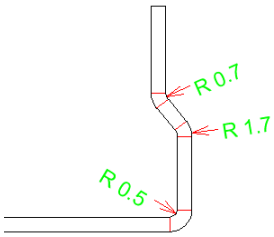
- Button **Diameter Dimension** from the Toolbox Dimensioning.

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.

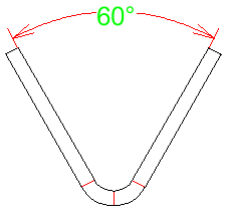
Calling the function

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

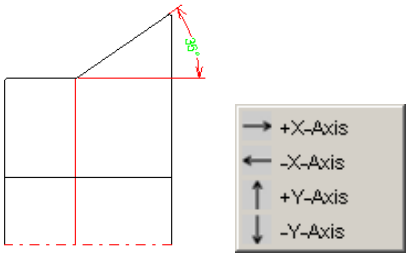


This function is a part of the Toolbox Dimensioning and can be used for angle dimensioning between two drawing elements.

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:

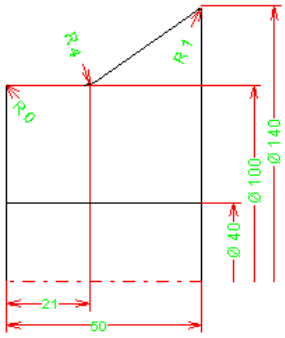
- +x - Axis
- -x - Axis
- +y - Axis
- -y - Axis

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

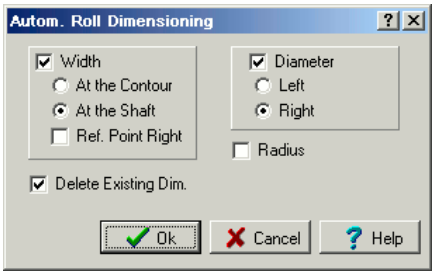


This function is a part of the Toolbox Dimensioning and can be used for automatic roll dimensioning (for roll tools only).

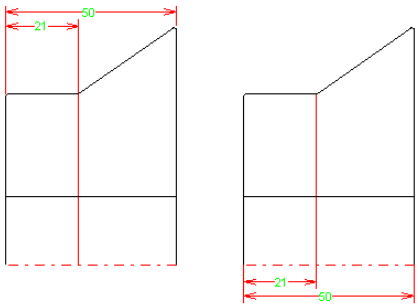
Calling the function

Call this function by:

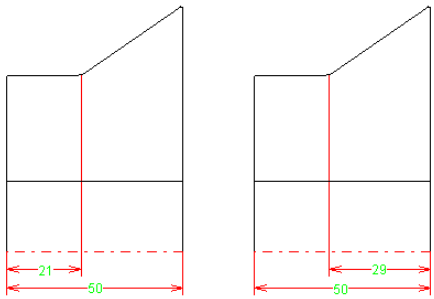
- Button **Automatic Roll Dimension** from the Toolbox Dimensioning.



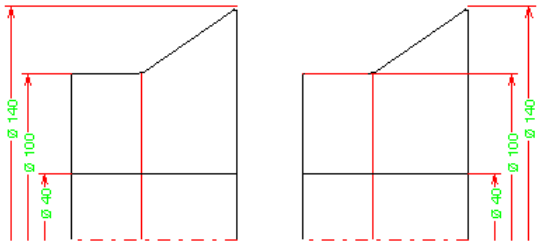
After calling this function you are asked to enter:



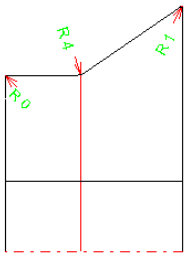
- **Width at Contour/at Shaft**: Select if the roll width should be dimensioned and whether the dimensioning should appear at the roll contour or at the shaft.



- **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:

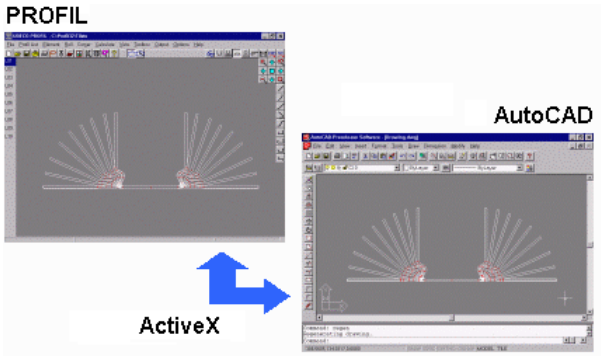
- **Dimension?** - identify a dimension text and the identified dimensioning will be deleted.

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

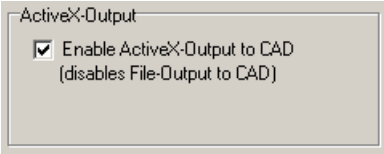
Select from the menu **View**, what should be transferred. Select by mouse-click, if a certain pass or a certain roll or if all (button Inspect) should be transferred.
Call this function optionally by:

- Main menu: **Output, Drawing -> CAD**.
- Button **Drawing -> CAD** in the Button Key Bar.

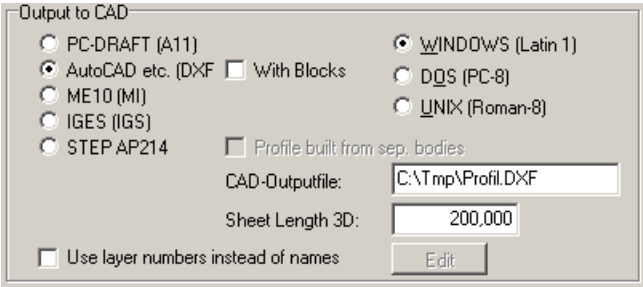
Principle of operation

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 Settings ActiveX, the drawing will directly be transferred to CAD (AutoCAD or SolidWorks or SolidEdge or BricsCAD or ZWCAD) 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 Settings, 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 Settings NC previously.

Before you create NC data you should use the function Calculate, Plausibility Check to check the whole profile project on errors.

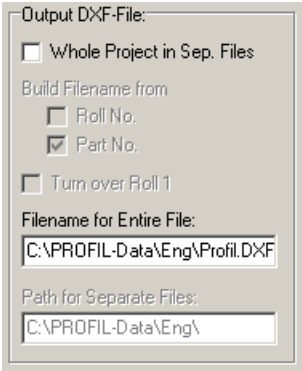
Call this function optionally by:

- Main menu: **Output, Drawing -> NC**.
- Button **Drawing -> NC** in the Button Key Bar.

Principle of operation

If the box **Separate Files** in Settings 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

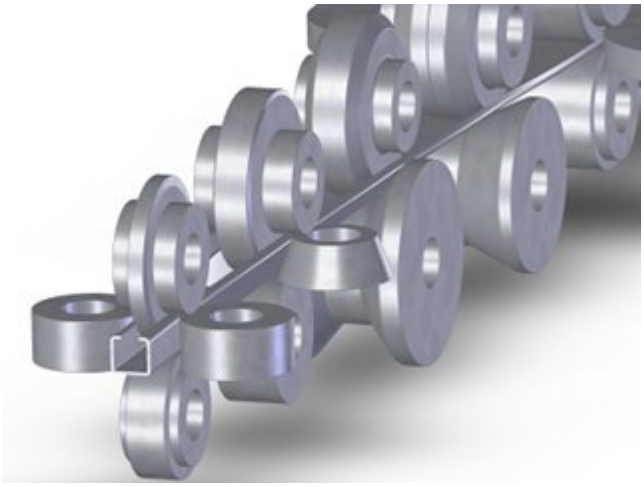


Which filename and which path are to be used, select in Settings NC.

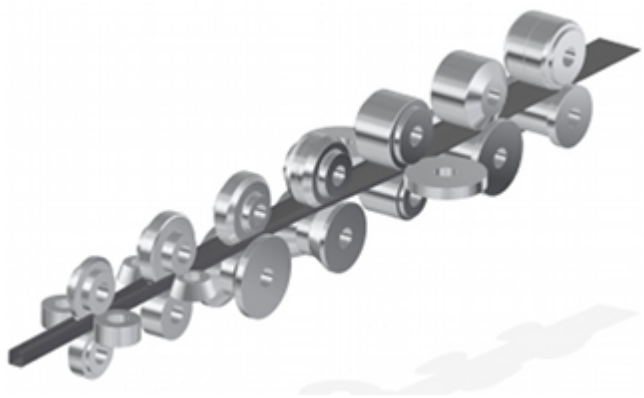
Hints:

- 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 Settings, 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 Settings, 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.

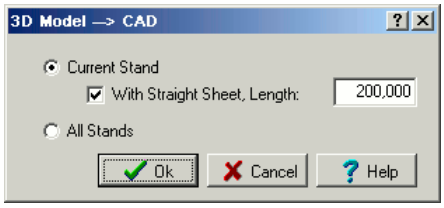
This function is useful e.g. for designing the side roll support. If the machine drawing already exists in CAD, the rolls simply can be inserted. Another application is to create photo-realistic images for

- **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

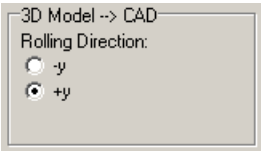
Select View Pass or View Rolls if you want to transfer a straight piece of profile or the roll tools of a stand.
Call this function by:

- 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 Settings, 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: The transfer of 3D models is possible in each perspective view.

ZW**CAD**: The transfer of 3D models is possible in each perspective view.

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**.

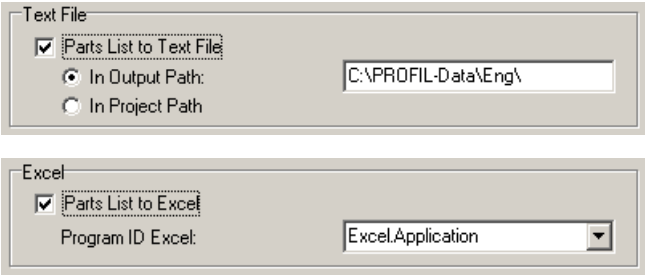
Principle of operation

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 Settings 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.

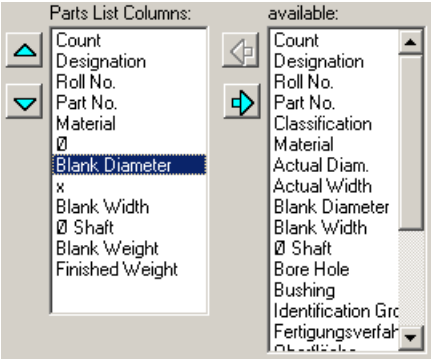
The file name of the parts list file is the same as the name of the project file, the file extension is **.txt**, however.

Setup



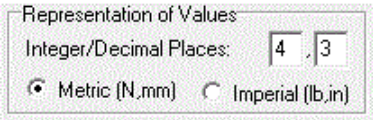
Preset it in Settings Parts List previously, whether a **Text File** is created or the part list is transferred to **MS Excel**.

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.

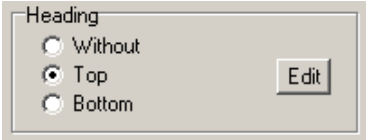


The composition of the parts list is fully user defined in the window Parts List Columns, call it by the button **Set-Up Columns** in Settings Parts List.

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 Settings 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 Settings 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 Settings 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:

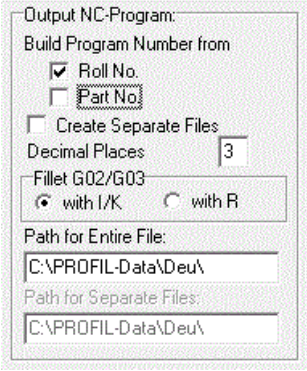
- Main menu: **Output, Create NC**.

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.

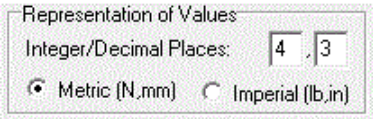
Afterwards, you can view it and append further machine commands to it, if you confirm **Yes** to the question **NC program has been created. View now?**. The text editor (set in Settings General) will be opened with the NC file. You can do the same with the function Edit NC.

Setup



Define in Settings 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 Settings NC.



Select in Settings 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 Settings 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

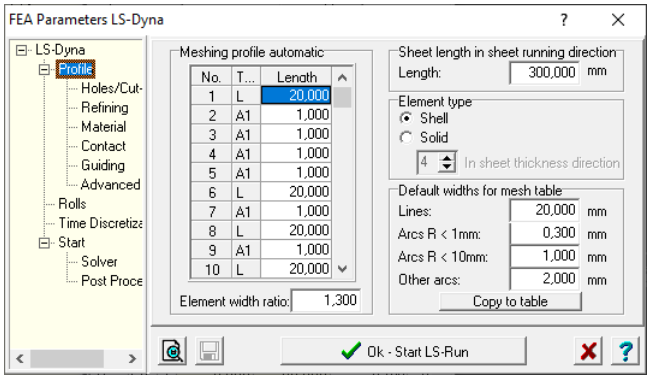
3.1.8.8.1 LS-Dyna

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**.

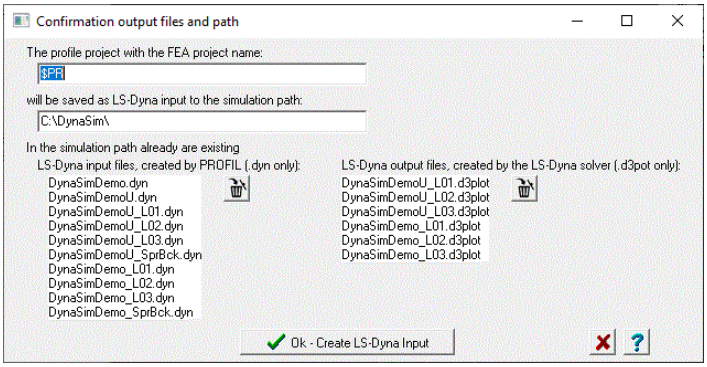


In the dialog window **FEA-Parameters LS-Dyna** is opened. These settings are necessary for the FEA simulation:

- Profile: Meshing in profile cross section and in in sheet running direction, element type
- Holes/Cut-Outs: Meshing the roll tools and holes/cut-outs
- Refining: Refining in the bend zones
- Material: Material properties, stress-strain-curve, import, moved to Material Window
- Contact: Roll tool contact, scale factor for sliding interface penalties
- Guiding: Threading the strip lead end into the stands
- Advanced: Modify after consulting the LS-Dyna support only
- Rolls: Meshing rolls axial and radial
- Time Discretization: Speed ratio for the time step size for mass scaling
- Start: FEA input/output path, start and restart, start position
- Solver: Action for the Ok button
- Post Processing: Automatic evaluation features

Principle of operation

After pressing the **Create LS-Dyna Input** button in the **FEA-Parameters LS-Dyna** window the **Confirmation output files and path** window opens first of all:



This window is good for checking the proper FEA project name and simulation path.This prevents overwriting earlier simulation results by mistake. If needed, both settings can be modified, For safety,

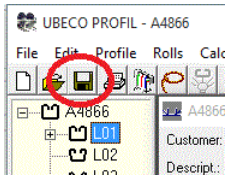
the bottom part of the window shows the existing files in the simulation path, separated by the creating system **PROFIL** and **LS-Dyna Solver**. For reason of clarity only .dyn files (PROFIL) and .d3plot files (LS-Dyna Solver) respectively are shown.

By using the two buttons **Delete** the path can be emptied before a new simulation is started in this path. The files can be deleted separately dependent on the creating system **PROFIL** or **LS-Dyna Solver**. Not only the shown files are deleted but all.

Create LS-Dyna Input After pressing the **Create LS-Dyna Input** button in the **Confirmation output files and path** window, these files 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>_CutOut.dyn File for holes and cut-outs.
<project name>_<pass name1>.dyn File for the first stand (in sheet running direction)
<project name>_<pass name1>.bnd File for guiding the first row of nodes (if parameterized)
<project name>_<pass name1>.mod Geometry of the rolls
<project name>_<pass name1>.pfl Pass of the flower pattern for plan vs. actual analysis
<project name>_<pass name2>.dyn File for the second stand (in sheet running direction)
<project name>_<pass name2>.bnd File for guiding the first row of nodes (if parameterized)
<project name>_<pass name2>.mod Geometry of the rolls
<project name>_<pass name2>.pfl Pass of the flower pattern for plan vs. actual analysis
... etc.
<project name>_SprBck.dyn File for spring back simulation
<project name>.blk Geometry of the blank sheet
<project name>.mod Geometry of the rolls (only for Rolls visible: All)
<project name>.mat Material properties of the blank sheet
<project name>.dxf contains the blank sheet for defining the holes/cut-outs
<project name>.txt for documentation, contains the project data, the FEA parametrization and messages
0_CLEAN_RESULTS_v2.bat is a batch file that removes all simulation results in the current path

<project name> is the FEA Project Name defined in FEA, LS-Dyna, Start.
<pass name> is the name of the pass or stand that is shown in the Explorer in design or in rolling direction.



Afterwards you can save all settings of the FEA parameters window and the material data 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.

Preview: See Profile and Rolls.

Save Project: While the window **FEA-Parameters LS-Dyna** is open, all FEA parameters and the material data can be saved into the project file.

Help: Depending on the opened tab sheet, the appropriate slides of the FEA training are shown and can be browsed by using the arrow buttons. opens the **PROFIL User Manual**.

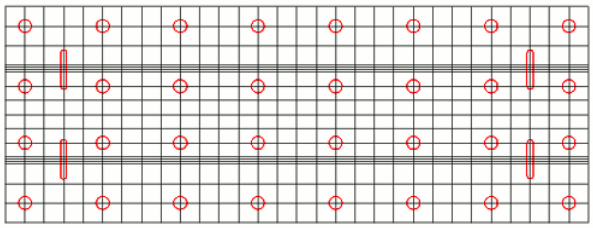
Starting the simulation

In Solver, select the action for the **Ok** button:

- Create the files for the FEA analysis only, e.g. if you want to define prepunched holes/cut-outs.
- Start the FEA solver immediately, if the solver is installed on the local computer.
- Call **LS-RUN** and use this program to start the analysis on a local or on a remote computer.

Defining prepunched holes/cut-outs

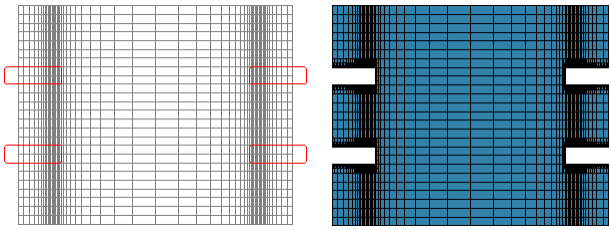
Element Type Shell: The function is fully available.
Element Type Solid: The function is not fully available, see **Current Restrictions for Element Type Solid**.
If prepunched sheet should be roll formed, the holes/cut-outs simply can be defined in a 2D CAD drawing with any shape and position. The simulation 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:



DXF file for a prepunched sheet

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 Settings, Profile List to layout **With Holes/Cut-Outs**). The center lines in the developed sheet in the DXF file are useful for positioning.



DXF-file for a slotted sheet (left) and FEA result (right)

- If you want to define **Holes/Cut-Outs** at the band edge, e.g. in case of a slotted sheet, draw the polyline like a punching tool in excess of the sheet edge. Do not terminate the polyline with the sheet edge.
- Pay attention that **Holes/Cut-Outs** does not span nodes that are a part of a Guided Node Row (see Others). This is why the solver has to delete the nodes within a **POLYLINE**.
- Save the drawing in **AutoCAD 2000 ASCII DXF** or in **AutoCAD 2004 ASCII DXF** format. Close the file in order to enable **PROFIL** to access to the file.

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 ration of 1 preferably, at the utmost 5.
Start the solver and the profile is simulated with holes/cut-outs.
Current Restrictions for Element Type Solid:

- The neighboring zones around holes and cut-outs are not refined automatically (as with element type shell). Fine pre-meshing of the sheet is necessary if holes and cut-outs exist.

- Slotted sheet is not possible, i.e. holes and cut-outs must not exceed the band edge (possible with element type shell only).
- If the main file <project name>.dyn is started with the solver, there is no automatic continuation with the first stand after proceeding the <project name>_CutOut.dyn file. So start <project name>_<pass name1>.dyn manually.

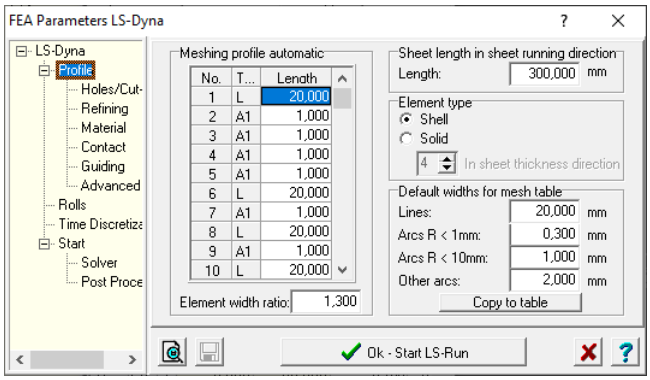
The restrictions will be purged in a later solver release.

Hints:

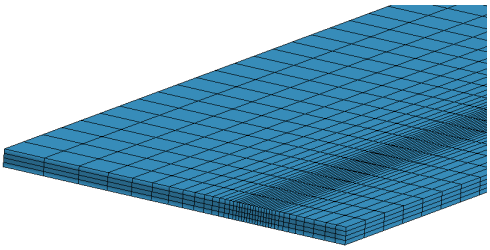
- Before you create the FEA simulation model you should use the function Calculate, Plausibility Check to check the whole profile project on errors. If you forgot it, PROFIL reminds you.
- In order to get a good simulation result, the maximum aspect ratio of the shell or solid elements of the flat sheet should preferably not exceed 8. 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 parametrization 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 ration occurred.

3.1.8.8.1.1 Profile

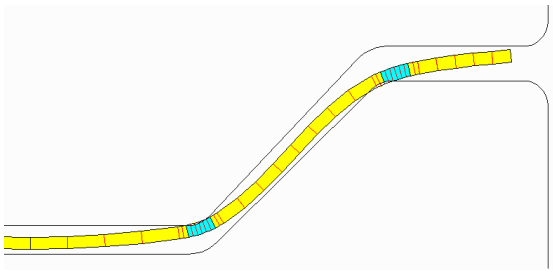
This dialog window is called by Output, FEA, LS-Dyna. It is used for parameterizing the meshing in profile cross section and in in sheet running direction.



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.



Meshing profile automatic: The sheet meshing must be small at the position of later narrow bend and can be wider in the straight parts or zones of less bend. However, the finite element simulation works most robust if the length deviation of neighboring elements is not too large. For arc segments (A1..A4) a fixed arc length for the FEA elements is preset. For line segments (L) a continuously increasing element width is created in that way, that starting from the neighboring arc the length is multiplied by a factor each. This is optimal for the FEA solver, because abrupt changeover is avoided. In order to avoid oversized width, the maximum width can be preset.



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, the elements neighboring to an arc segment must also be small. This is ensured by the automatic meshing of the line segments by using the **Element width ratio** for increasing neighboring elements.

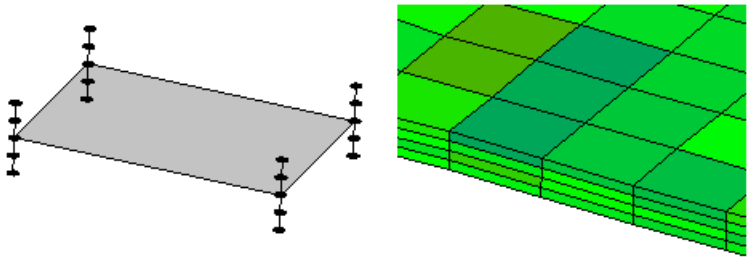
The table structure is composed like the Profile List. The rows correspondent to the Profile Elements of the final pass. For clear correlation the first two columns show the number and the type of the profile element. The third column shows the actual element width in case of an arc segment and the maximum element width in case of a line segment. Enter the desired element width for arc segments (A1..A4) and the desired maximum width for line segments (L) or use the default widths (see later in this chapter).

Element width ratio (recommended 1.3): All line segments (L) are split in such a way that the neighboring elements approximately are getting increasingly larger by this ratio. Example: If the arc width is 0.4mm, the first element of the neighboring line becomes 1.3 x 0.4mm, the next 1.3 x 1.3 x 0.4mm and so on. Special values are:
1: This creates constant element width by using the width of the neighboring arc segment.
0: This also creates constant element width, however the maximum width is used from the line rows of the table.

Meshing in Sheet Running Direction

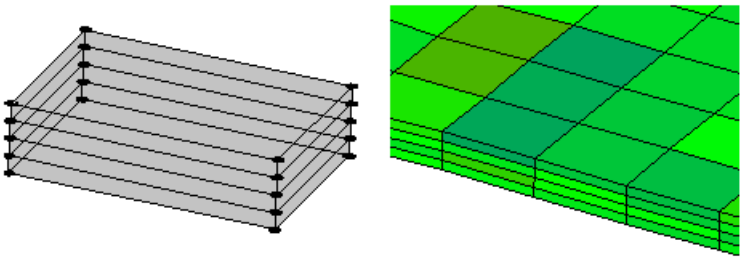
- **Length:** Select the desired sheet length for simulation in longitudinal direction. Possibly **PROFIL** will modify this length a little. This is why because of discretization sheet length, element length, and distance between stands must fit each other. See the modified sheet length after opening the dialog window again or by opening the documentation file <project name>.txt with a text editor.
- **Element length in longitudinal direction:** The length cannot be preset by the user, but it is calculated by **PROFIL** automatically dependent on the minimum element edge length in x direction. This is why the aspect ratio z/x should be between 5 and 6 optimally because of stable simulation. See also the protocol file <project name>.txt.

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.

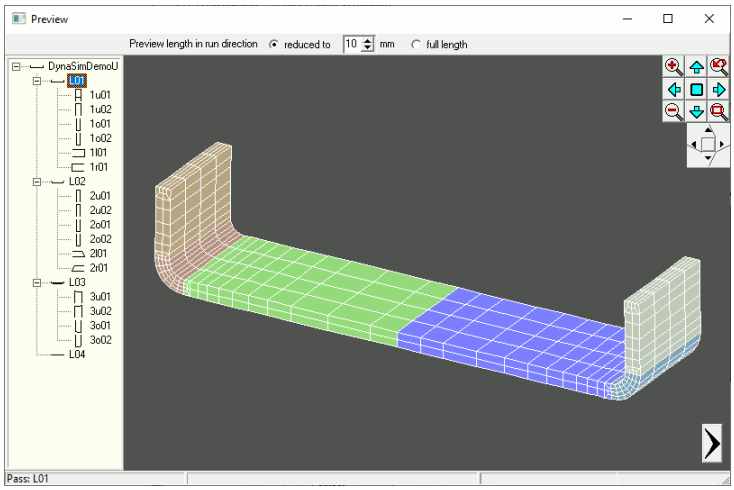


Solid element type with 4 solids in thickness direction

- **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. The basic setting is effective for line and arc segments. If needed, you can double this setting in Output, FEA, LS-Dyna, Others for arc segments only.

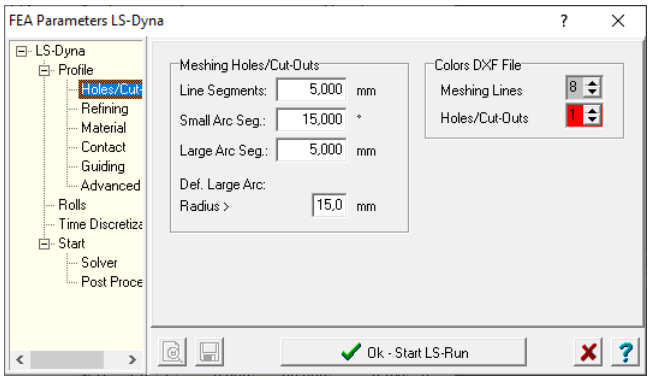
Default widths for mesh table: If the dialogue window **Output, FEA, LS-Dyna** is opened for the first time in a new project, the table **Meshing profile automatic** is filled out with the default widths. You can parametrize the default widths here dependent on the inner radii. The default widths are saved in the WINDOWS registry.
Copy to table: After modifying the default widths the table **Meshing profile automatic** is configured newly by key press. Also after modifying the profile list structure it might be necessary to reconfigure the table.



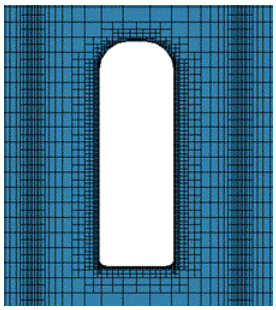
Preview: While entering the FEA parameter the preview of the meshed flat sheet is shown in the preview window. Switch to further passes by using the explorer on the left or the arrow buttons. The Navigator helps you to zoom, to move and to fit the drawing. By using the Navigator 3D you can rotate the drawing and toggle to a 2D view. Furthermore the **Space Mouse** is supported for rotating and moving the view. Since the **full length in run direction** will be too large to have a clear view of the meshing, select a **reduced length** in the top bar of the window.

3.1.8.8.1.2 Holes/Cut-Outs

This dialog window is called by Output, FEA, LS-Dyna. It is used for parameterizing the meshing of the roll tools and holes/cut-outs.



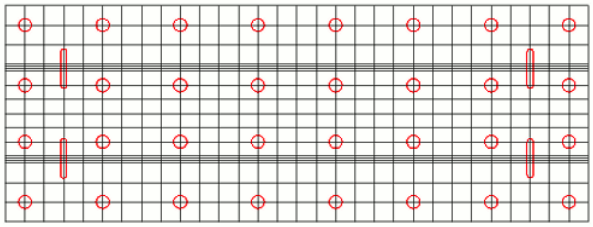
Meshing Holes/Cut-outs: Line and arc segments are meshed as follows: How to define holes/cut-outs in the flat sheet, read in the chapter **Defining prepunched holes/cut-outs** (later in this topic).



- **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.

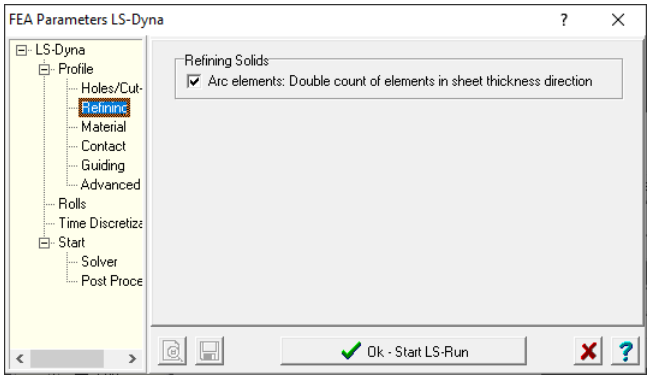
Colors DXF File: After pressing the **Ok** button for the first time, **PROFIL** creates a DXF file with the meshed flat sheet. After pressing the **Ok** button again, the holes/cut-outs (inserted from the user) are extracted from the file and prepared for the simulation. Get more details from the the chapter **Defining prepunched holes/cut-outs** (later in this topic).



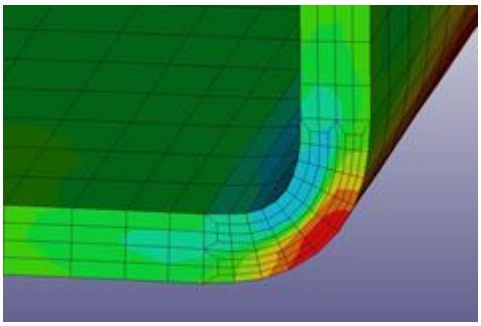
- **Meshing Lines:** Modify the color for the meshing lines. This is only necessary in case the CAD system is not able to display the default color grey.
- **Holes/Cut-Outs:** In order to enable **PROFIL** to extract the holes/cut-outs entities from the file, they must be drawn in the preset color (default color red).

3.1.8.8.1.3 Refining

This dialog window is called by Output, FEA, LS-Dyna. It is used for set-up refining in the bend zones and at the edges.



Since the profile is more deformed in the bend zones, it makes sense to double the count of elements in these critical zones. Thus a smaller count can be preset as basic setting. This lowers the simulation time, especially if large line segments exist.

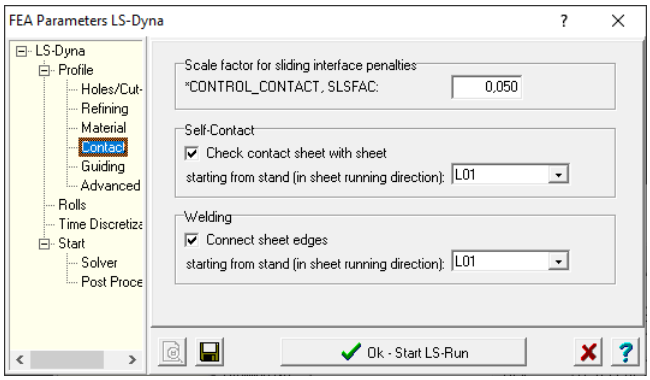


Refining Solids: Set the **count of elements in sheet thickness direction** in Output, FEA, LS-Dyna, Profile. The basic setting is effective for line and arc segments.

Arc elements: Double count of elements in sheet thickness direction: Because normally arc segments are more deformed than line segments, it makes sense to double the count vs. the basic setting. Check this box for this purpose. The setting is also effective for the band edges.

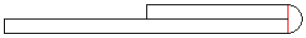
3.1.8.8.1.4 Contact

This dialog window is called by Output, FEA, LS-Dyna. It 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.

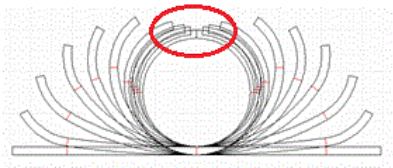
Self-Contact



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.

Welding



Connect sheet edges: Check this box, if a closed profile or tube should be welded, i.e. left and right band edge should be connected together non-detachably. The closed profile or tube can be formed onwards in succeeding calibrating stands (turks heads). Pre-condition: The Profile List structure must be unsymmetric in the final pass L01 at least, see hint.

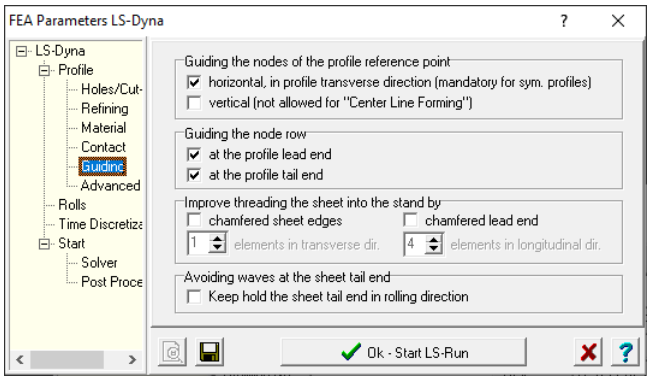
Starting from stand (in sheet running direction): Select the stand from the drop-down-list to define the position of the welding machine. LS-Dyna connects the sheet edges starting at this stand and prevents spring open.

Hint:

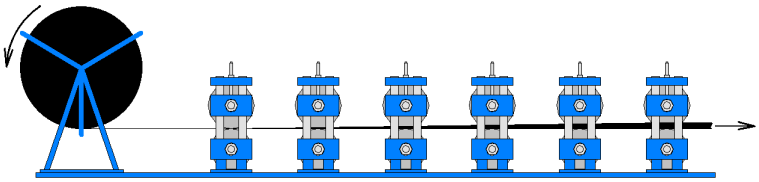
- In order to convert a symmetric profile list to a unsymmetric one it is recommended to use the function Profile, Modify Start Element twice. Set the start element to any different position, thereby the profile list becomes unsymmetric. Afterwards set it back to the origin point; the profile list keeps unsymmetric. For the FEA simulation it is sufficient to modify the final stand (L01) only.

3.1.8.8.1.5 Guiding

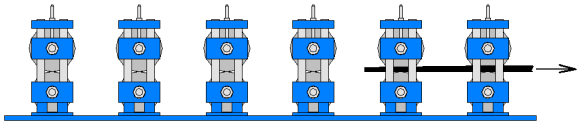
This dialog window is called by Output, FEA, LS-Dyna. It is used for set-up the guiding.



Why the profile must be guided?

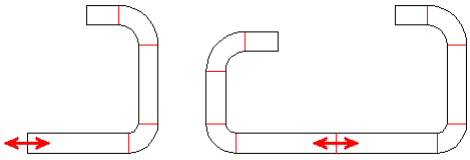


In reality, the sheet metal strip has a quasi-infinite length, because it is decoiled from a coil. However, a sheet with an infinite length cannot be simulated, because it would take infinite computing power or infinite time.



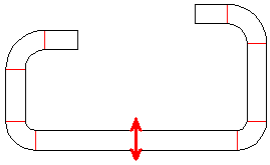
So for the simulation, a section with a finite length is used instead (setting in Profile, **Meshing in Sheet Running Direction**). The picture shows a length of about 2x distance between stands. However, the behaviour of this section if different to a sheet with infinite length. Notably the front may 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: Precut blanks 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. Dependent 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



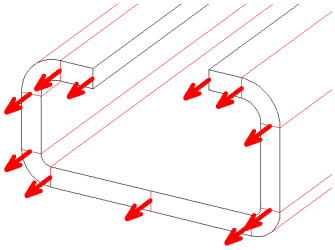
Guiding the profile reference point edge horizontally
for symmetrical (left) and unsymmetrical profiles (right)

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 unsymmetrical 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.



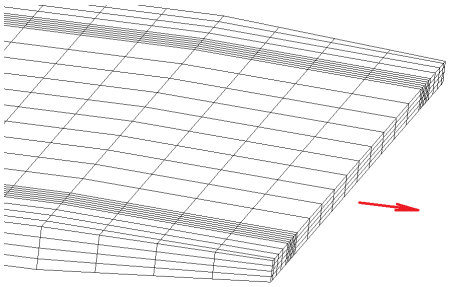
Guiding the first row of nodes at the profile lead end

Guiding the node row 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 row 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 certain count of rows can be cut off at the lead and tail end.

Improve threading the sheet into the stand by:

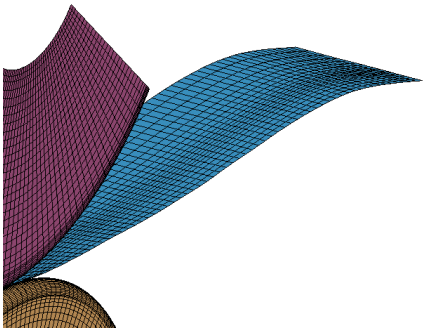


Chamfered edges and chamfered lead end

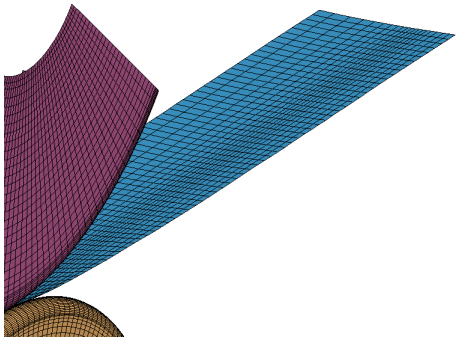
Chamfered sheet edges
Another method to improve threading the sheet into the roller stands is to create a chamfer at the sheet edges. This is a common used method in practice. Select the dimension of the chamfer by entering the element count in transverse and longitudinal direction.

Chamfered lead end
Additionally, the sheet thickness can be decreased by a chamfer. This also improves threading the sheet into the roll stands. Select the dimension of the chamfer by entering the element count in longitudinal direction. The thickness reduction is fixed to 50%. The chamfered lead end is possible only, if in Profile the element type **Solid** is selected.

Avoiding waves at the sheet tail end



Sheet tail end not kept hold



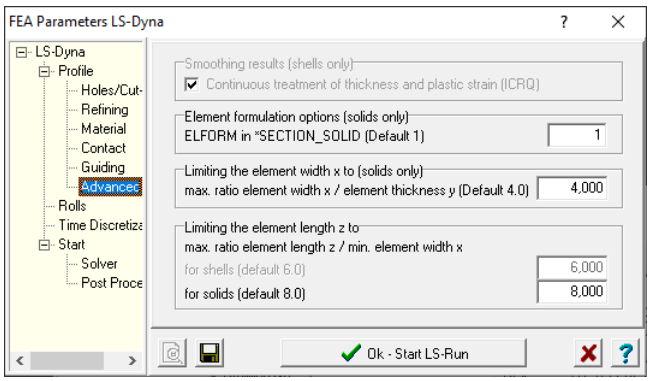
Sheet tail end kept hold

In case guiding is not chosen, the tail end of thin sheet may oscillate up and down. This phenomenon fits to the reality, when the decoiled sheet end is reached. However, it disturbs the analysis of the simulation result of the short section that is used for simulation.
Keep hold the sheet tail end in rolling direction: By checking this box the sheet tail end is kept hold in rolling direction by using the CONSTRAINED_GLOBAL command. This avoids unwanted oscillation and strain at the sheet tail end. **Hint:** The sheet front end always is kept hold independent of the checkbox in order to avoid horizontal movement by the rolls.

Hints:
- The method **Chamfered edges** and **Chamfered lead end** cannot be used with the method **Guiding the node at the profile lead end** simultaneously.
- If precut blanks should be roll formed, guiding and chamfers can be switched off in order to simulate also this process correctly.

3.1.8.8.1.6 Advanced

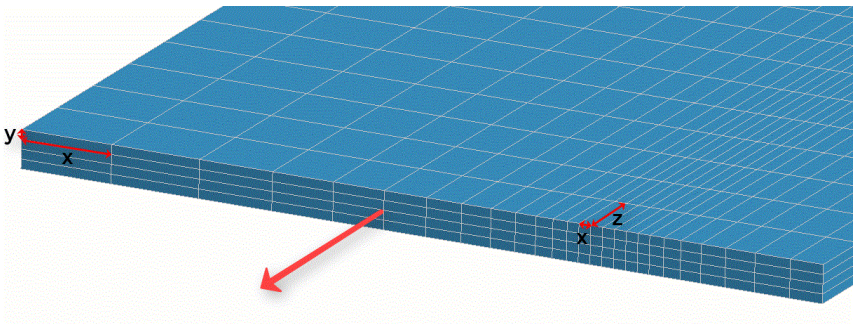
This dialog window is called by Output, FEA, LS-Dyna . It is used for expert settings only. Please do not modify the settings except you are requested by the LS-Dyna support.



Smoothing results (shells only):
A new option is available from **LS-Dyna R9.0** (Shells only). A nodal averaging technique is used to achieve continuity for some quantities across element edges. Applying this approach to the thickness field and plastic strains **ICRQ=1** can reduce alternating weak localizations sometimes observed in metal forming applications when shell elements get stretch-bended over small radii.

Continuous treatment of thickness and plastic strain: Checking this box sets **IRCQ=1** (needs more calculation time). Unchecking sets **IRCQ=0**. The option is available for the **SHELL** model only. In case of **SOLID** models it takes no effect.

Element formulation options (solids only)', ELFORM in *SECTION_SOLID (Default 1)
Get more information from the **LS-Dyna Keyword Manual Vol. 1** under the keyword ***SECTION_SOLID**. The parameter **ELFORM** defines the properties of the solid elements. The option is available for the **SOLID** model only.

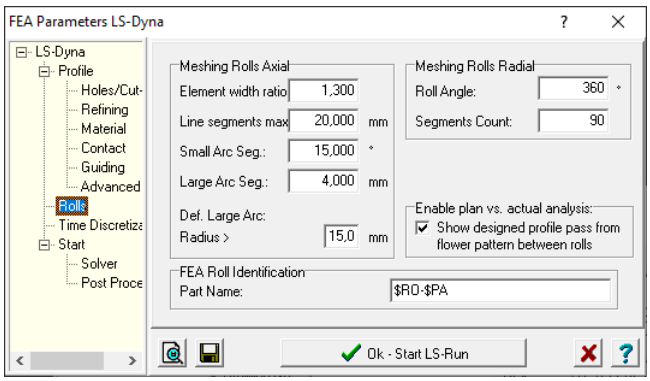


Limiting the element width x to max. ratio element width x / element thickness y (Default 4.0) (see left side of the picture)
In the tab window Profile the **Element width ratio** can be preset. It says how much neighboring line elements are increased based on the arc element width. In order not to get too large widths (and thus inappropriate corner ratio) the **max. ratio element length x / element thickness y** can be set. The option is relevant for the **SOLID** model and for ELFORM = -1 only.

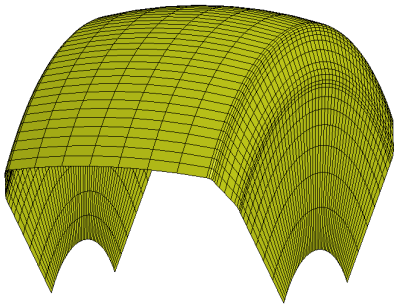
Limiting the element length z to max. ratio element length z / min. element width x (see right side of the picture)
The element length z in sheet running direction is calculated from the smallest element width x (usually occurring in sharp bends) multiplied by a factor that should not exceed 6 (shells) or 8 (solids) respectively. A larger corner ratio would be inappropriate for the simulation. For special applications the factor can be modified here.

3.1.8.8.1.7 Rolls

This dialog window is called by Output, FEA, LS-Dyna. It is used for parameterizing the meshing of the rolls in axial and radial direction.

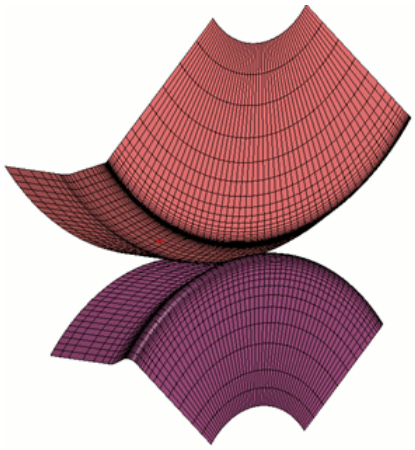


Meshing Rolls Axial: Line and arc segments are meshed in that way: For **arcs**, the arc segment angle or the arc segment length can be preset as alternatively. The first is sensible for arcs with small radius, the second for arcs with large radius. **Lines** are meshed automatically by using the preset **Element width ratio**. Starting with the neighboring arc segment, the width of line elements are multiplied by this factor. This is optimal for the FEA solver, because abrupt changeover is avoided. In order to avoid oversized width, the maximum width can be preset.



- **Element width ratio:** Select the ratio that is used for increasing the width of line elements starting with the neighboring arc segment. The recommended ratio is 1.3.
- **Line Segments max.:** Select the desired maximum length of the shell elements for line segments when the width increases more and more. This avoids oversized element widths. 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

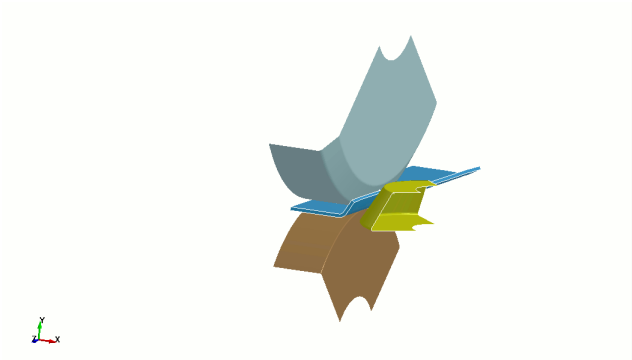


Example: Partial rolls with 100° roll angle and 50 segments

Roll Angle: Enter 360° if a full roll should be shown in the FEA result. This is useful in case the result should be displayed in a presentation, brochure etc. In order to optimize the simulation time, it makes sense to create a partial roll only. It is sufficient to create the part only that is touched by the profile., e.g. 80°.. 120°. Check after running the solver if the partial roll has been large enough.
Segments Count: Enter the number of segments for discretization or the roll or the partial roll. The number must not be larger than the roll angle, because the minimum allowable resolution is 1°.
Example: If a full roll is selected (**Roll Angle** 360°) and the **Segments Count** is set to 90, the rolls are split radially in 360° / 90 = **4°-Segments**.

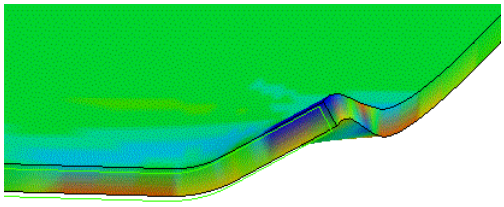
Visible Rolls
Select, how the rolls should be displayed during evaluation of the FEA result in **LS-PrePost**:

All: All rolls of the whole machine are shown. If the profile is symmetric, also the rolls of the left side are shown, even though they have no contact with the sheet. Combined with **Roll Angle = 360°** the rolls of the complete machine are shown in a proper style, useful for pictures for brochures etc.
Disadvantage: The simulation takes more time.



With sheet contact only: Only that rolls are shown that have contact with the sheet of the certain stand. This is dependent on the sheet length and the distance of the stands. Advantage: The simulation is faster.
Hint: If you have a very large project with a very large count of rolls, better use **With sheet contact only** in order to avoid a too large .mod file.

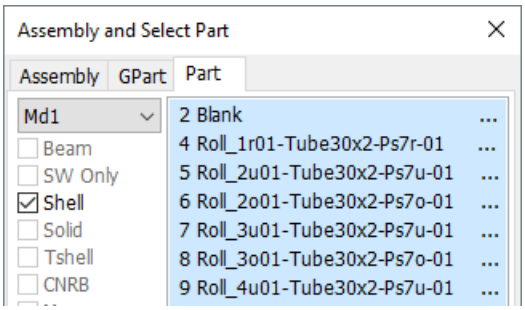
Enable plan vs. actual analysis



Plan vs. actual analysis, light green = planned contour

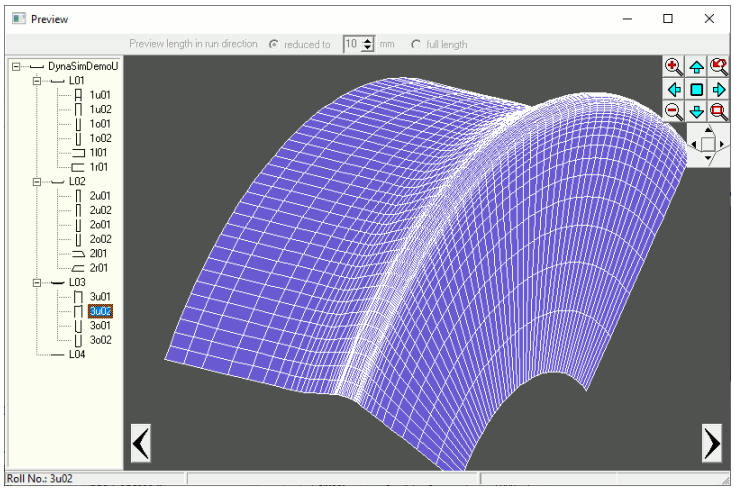
Show designed profile pass from flower pattern between rolls: The designed passes are shown in **LS-PrePost** during evaluating the simulation result between the rolls without having influence on the result (i.e. contacts are not defined). The part names are the names of the passes or stands that are shown in the Explorer in design or in rolling direction. This simply enables plan vs. actual analysis (planned contour = pass contour as designed, actual contour = FEA result).

FEA Roll Identification



LS-Dyna handles the roll tools under a part name. In **LS-PrePost**, call **Model, SelPart** and in the window **Assembly and Select Part** you will see the parts table of the existing rolls.
Part Name: Enter the scheme how the part names of the rolls should be assembled. Use fixed text or Variables. Useful variables are

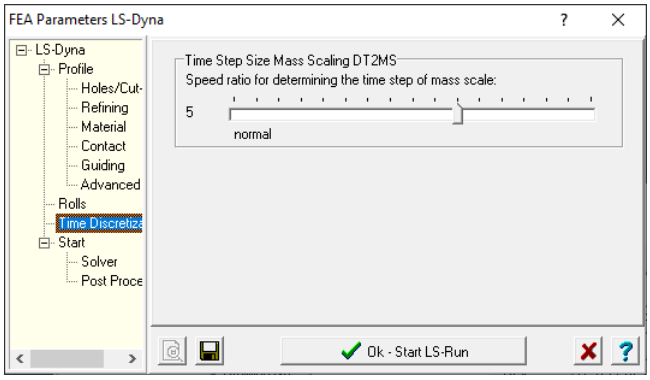
- $RO Roll-No.
- $PA Part-Nr.



Preview: While entering the FEA parameter the preview of a meshed roll is shown in the preview window. Switch to further rolls by using the explorer on the left or the arrow buttons. 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. Furthermore the **Space Mouse** is supported for rotating and moving the view.

3.1.8.8.1.8 Time Discretization

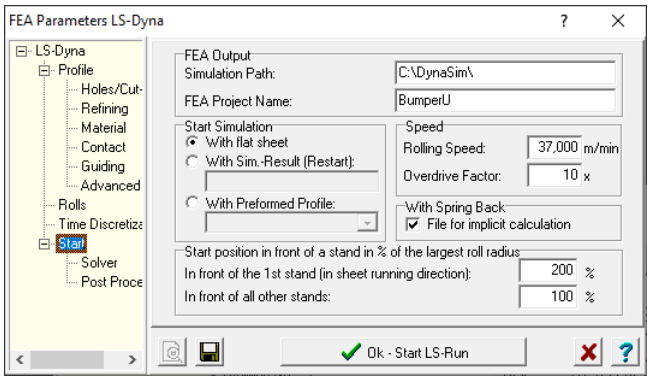
This dialog window is called by Output, FEA, LS-Dyna. It is used for set-up the speed ratio that determines the time step size for mass scaling.



Time Step Size Mass Scaling DT2MS: The best time step size is dependent on the used material and the element size and has great influence on robustness and precision of the simulation. Determining needs experience. In order to relieve the user from unnecessary decision, the best time step size is calculated by **PROFIL**. By using the slider the recommended value can be fine-tuned from very accurate until very fast.
Speed ratio for determining the time step of mass scale: Select with the slider, if the solver should work precise or fast. This corresponds to a speed factor of 0.1 .. 2.0. **PROFIL** calculates the best time step size from the speed factor, the smallest element edge length, and the material properties. Find the calculated time step size after creating the output files in the protocol file <project name>.txt.

3.1.8.8.1.9 Start

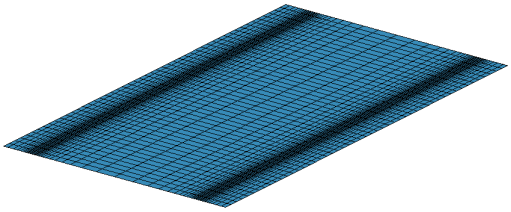
This dialog window is called by Output, FEA, LS-Dyna. It is used for set-up the FEA input/output path, start and restart, and the start position.



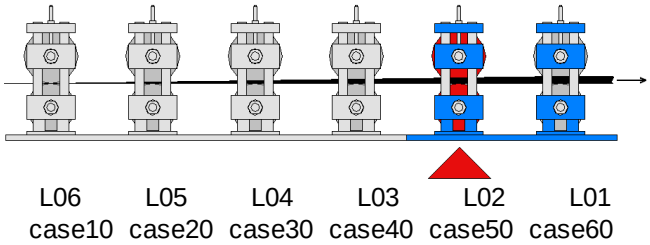
FEA Output, Simulation Path: Enter the network path for saving the output files of the FEA simulation model. Hint: Since the FEA solver needs 100% of computing power, the FEA system **LS-Dyna** should run on a remote computer, not on the PROFIL/CAD workstation.

FEA Output, FEA Project Name: The files of the simulation model are saved under this name. Use fixed text or Variables. Useful variables are:
$PR: Project Name
$CU Customer
$PD Description
$DR Drawing Number
The name must not contain umlaute, blanks, and other special characters; the will be replaced by substitute characters.

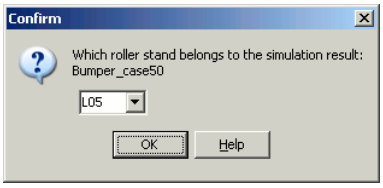
Start Simulation:



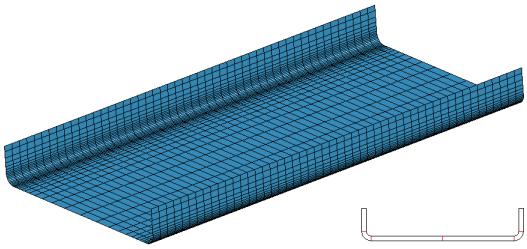
- **With flat sheet:** Select this option, if you want to analyse a project for the first time, i.e. a result from earlier simulation does not exist. The guideways are are calculated (see: **Start position in front of a stand in % of the largest roll radius**) and put in the project data. Do not forget to save the project file after creating the FEA simulation files. This is important in order to eventually restart with the same guideways after modifying roll diameters.



- **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** consist of 6 stands **L01 .. L06** (and **L07**=flat sheet additionally. The first simulation, starting with the flat sheet, produces the files**Example_L06.dynain** (correspondents to **L06**) until **Example_L01.dynain** (correspondents to **L01**). **Hint:** By toggling the Explorer to **Show pass from profile list** the files can be named in sheet running direction. Now you detect an error in stand **L02**, i.e. the simulation result **Example_L02.dynain** and all successive have to be repeated after modifying the roll of **L02**. Select in the file selection field **Example_L03.dynain**, i.e. the simulation result, that still was correct. The following simulation will base on this result. **Restart behind stand** does not calculate the guideways newly but uses the saved guideways from the last simulation **With flat sheet**.



PROFIL determines the belonging roller stand to the selected simulation result file automatically. Sometimes the proper stand is not found, e.g. if you changed the explorer display or if stands are removed or inserted before a new simulation. If so, you can select the proper stand from the drop-down list.

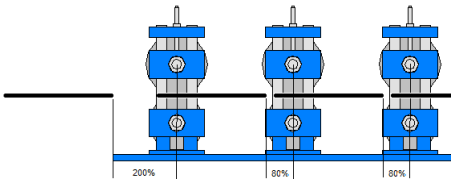


- **With Preformed Profile:** Instead of a flat sheet, you can start the simulation with a preformed profile. Select the profile list from your flower pattern, that should be used for forming the initial profile. The simulation starts with the next roll stand (in sheet running direction). Also here the guideways are are calculated (see: **Start position in front of a stand in % of the largest roll radius**) and put in the project data.

Speed:
Rolling Speed: Select the desired strip speed when the lead end of the strip enters each stand. The speed is crucial for proper threading the lead end into the stand. It is recommended to select a low speed. First, try the real machine speed. If the lead end is deformed to much, decrease the speed.
Overdrive Factor: Apart from that, when the lead end of the strip can move freely, the speed can be increased. Enter the factor for overdrive speed. Recommended is the factor 10, this means the tenfold speed is taken because of optimizing the simulation time.

With Spring Back: While running through the roll forming machine, the **explicit** working solver is used for simulation. This method works quickly, however the spring back cannot be calculated precisely. If you check **With Spring Back**, an additional file is created for the spring back calculation. After proceeding the roll forming simulation by using the explicit solver, start the spring back calculation by using the **implicit** solver (double precision) and by opening the additional file. Previously the result of the final stand should be checked and the guided nodes should be removed by using **LS-PrePost**. This is why they should not have influence on the spring back calculation. The additional file has the name **<project name>_SprBck.dyn**. It requires the file **<project name>_<pass name>.dynain** (from the final pass). This means, the simulation of the whole project has to be finished previously. Afterwards start the spring back calculation by using the implicit solver (double precision) with the file **<project name>_SprBck.dyn**.

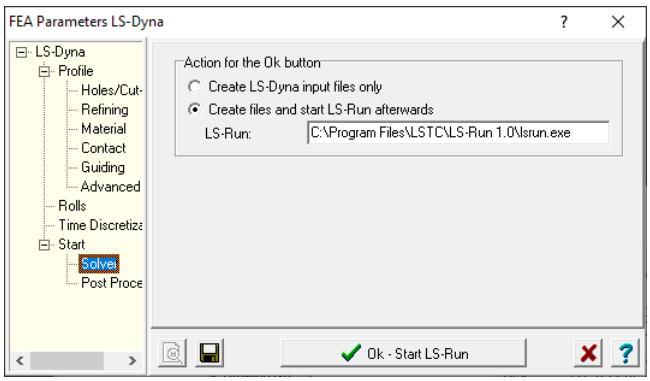
Start position in front of a stand in % of the largest roll radius:
The simulation of a stand is finished if the front end of the sheet is just before the next stand. This is the start position for the simulation of the next stand as well. Because the contact definitions for the next stand still are 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.



- **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 is **Guiding the node at the profile lead end** 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%.

3.1.8.8.1.10 Solver

This dialog window is called by Output, FEA, LS-Dyna. It is used for set-up the action for the Ok button at the bottom of the window.



Action for the Ok button

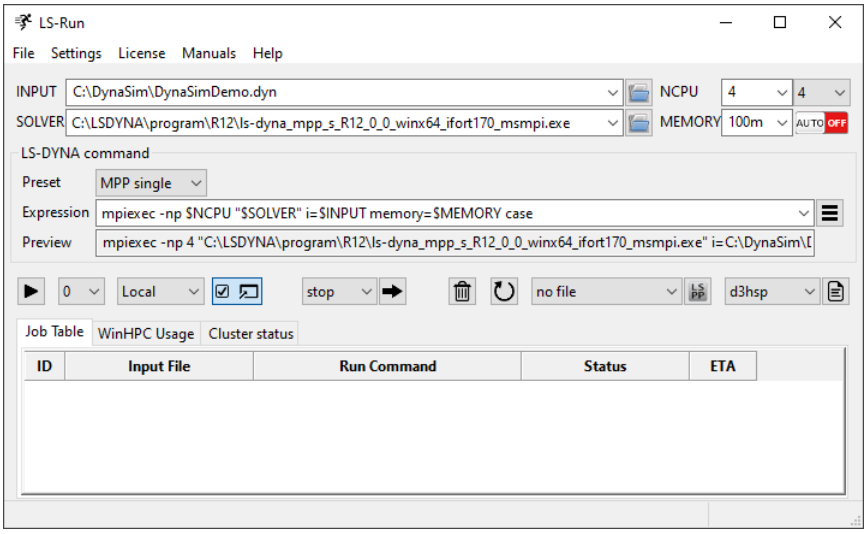
After parameterizing the FEA simulation can start. For this purpose the files with the FEA model must be created and the LS-Dyna solver must be started. The are two options:

Create LS-Dyna input files only: Select this option in case the solver is installed on a remote computer. This makes sense therewith the local computer can be used for other tasks furthermore while the simulation is running. The remote device can be a special powerful computer with higher CPU count and larger memory space. Select the network path for the simulation in Start, **FEA Output, Simulation Path**. Start the solver after creating the files by using **LS-Run** or the **LS-Dyna Program Manager** (both are shipped with the LS-Dyna package).

Create files and start LS-Run afterwards: This option starts **LS-Run** after exporting the files. **LS-Run** is shipped with **LS-Dyna** and is good for organizing the simulations.

- **LS-Run**: After clicking on the input field the file selection windows opens. Select path and name of the exe file of **LS-Run**.

Ok - Start LS-Run:



LS-Run is a part of the LS-Dyna package.

INPUT: Is been filled out by PROFIL with the project's main file <project name>.dyn.
SOLVER: Select the Solver MPP, single precision, rel. R12.
NCPU: Select the count of CPUs in your computer.
MEMORY: Select the amount of memory in word that should be used for simulation.
Preset: Select MPP, single precision.

Expression: Start command for the solver, here with variables.

mpiexec: A Microsoft tool, used for CPU managing and distributing the simulation on the CPUs.

Download from

https://www.microsoft.com/en-us/download/details.aspx?id=57467

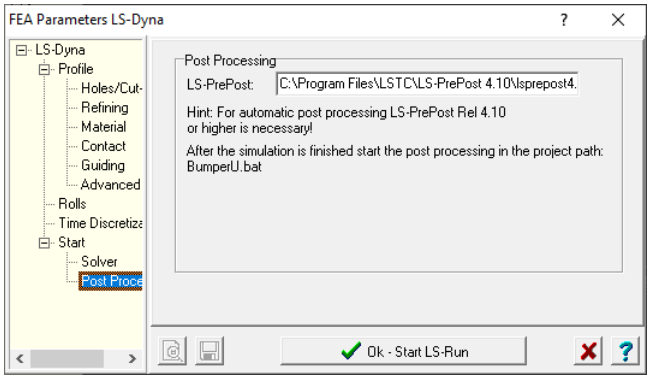
case: Causes that all roll forming stands are proceeded successively.

Preview: Resolved start command.

Starts the simulation.

3.1.8.8.1.11 Post Processing

This dialog window is called by Output, FEA, LS-Dyna. It is used for set-up the automatic evaluation features after finishing the simulation.



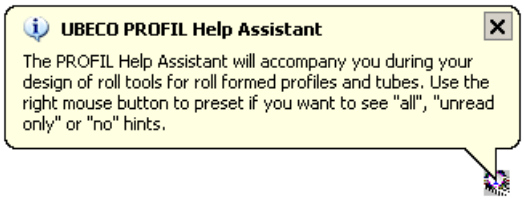
LS-Prepost:
Select the exe file of the post processor LS-PrePost. Release 4.10 or higher is necessay!

Proceeding:
A Python script is saved that creates the automatic evaluation files by using LS-PrePost in the subdirectory 0_postprocessing_<project name> under the procect path.
After finishing the simulation start the batch file <project name>.bat in the project path!

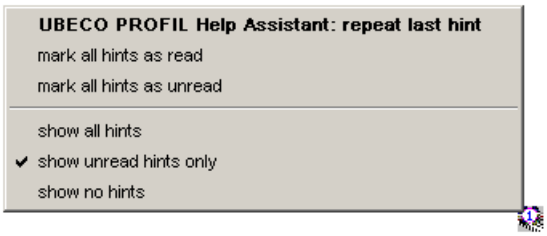
Required Settings:
See the document 2021_07_23_Rollforming_post-processing_draft.pdf (ask the author).

3.1.9 Help

3.1.9.1 Assistant

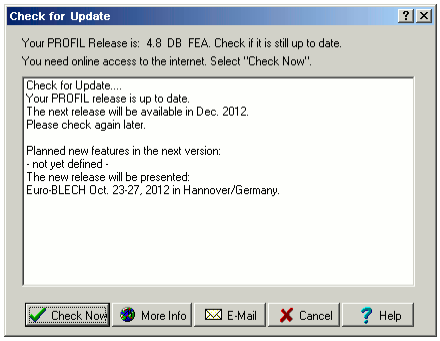


The help assistant accompanies you during your design work by showing helpful hints in a speech balloon. In case the display time was too short, click on the **PROFIL** symbol in the right corner and the hint will appear again.



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

Material

Window Visible/Hidden

Statics

Stress of Edge

Profile Catalogue or Roll Stock Management

Undo

Redo

Repeat

Settings

Help

Toolbox Profile Design

Toolbox Tube Design

Toolbox Modify

Toolbox Dimensioning

Plausibility Check

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

FEM-Ausgabe

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.

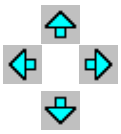
Principle of operation

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**

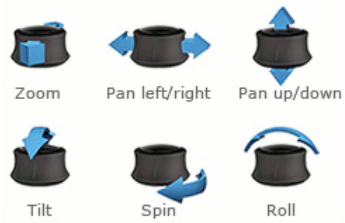
Use this function to zoom by clicking two opposite corners of a rectangle for the new view port. After clicking the first point a drag box is displayed in the predefined color for marked elements.

 **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. Use the arrow buttons to rotate the 3D view port. The center button resets to the initial view port (Setting in Settings Drawing, View Port Angle). By clicking on the background area you can switch to the 2D view.

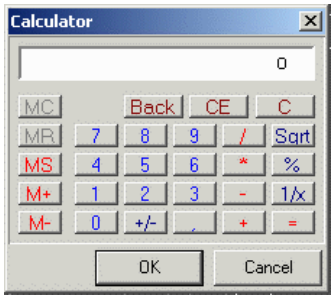


Space-Mouse: If you use a space mouse from 3DConnexion, the motion controller allows pan, zoom, and rotate the model simultaneously. The function keys are allocated by

- 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** mode.

Principle of operation

The current value of the input field is copied to the calculator's display.

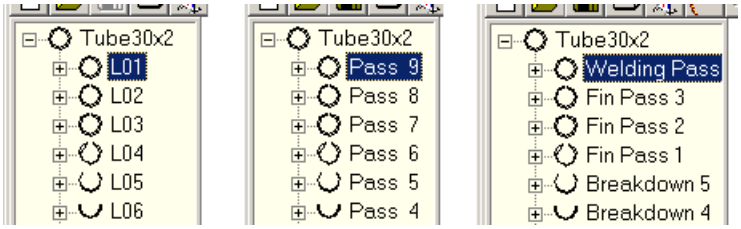
These basic arithmetical operations are possible: addition, subtraction, multiplication, and division. Furthermore: inversion, percentage calculation, square root, and memory functions.

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



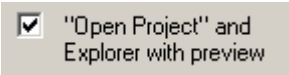
The **Profile Explorer** on the left side of the screen clearly shows the passes, stands, and rolls of a profile project. It can be used for quick navigating between all objects of the project. The tree diagram contains the project name, the pass numbers (L01 .. etc.), the roll number, and a small icon that shows the preview of the object.

Principle of operation

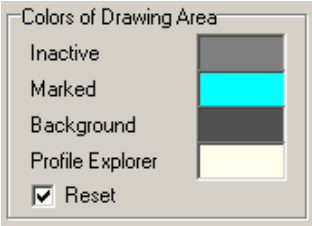
Clicking the **+** sign opens the tree node and the objects of the node become visible. The **–** sign closes the node. By clicking on an object the belonging drawing is shown in the drawing area. If necessary, the view changes automatically between flower, pass, stand, and roll. If another object is 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 Settings, General.

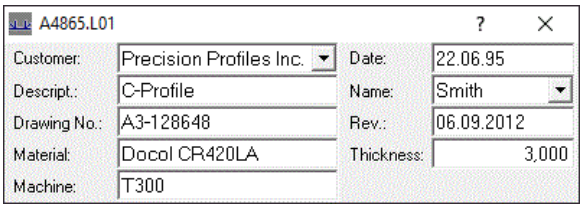


The background color can be preset in Settings Drawing. By moving the vertical split line between explorer and drawing area the width of the explorer can be modified. If more explorer entries exist than the window is able to display, a vertical scroll bar appears.

Hint:

- If needed, the profile list number can be used as variable **$PL** for the automatic roll numbering, see Settings 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:

Customer	Date
Description	Name
Drawing No.	Revision
Material	Thickness
Machine	Calculation Methods (removed)

Since **PROFIL rel. 5.4** the global input field **Calculation Methods** is removed. It is replaced by the individual Calculation Methods Developed Length.

Since **PROFIL rel. 6.2.1** a mouse click on the input field Material opens the Material Window. The earlier used Material File (.wkd, .wki) is not supported anymore.

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 Settings 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 Settings Rolls.

See also: Profile Project

3.3.2.4 Material

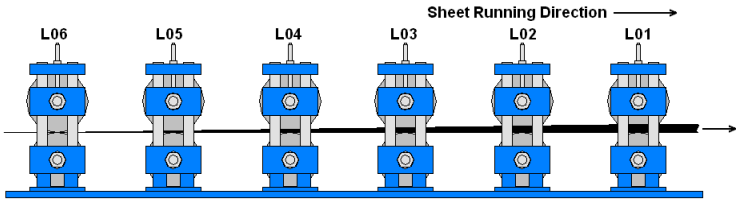
The name of the material belongs to the project data.

A mouse click on the input field opens the Material Window.

Dependent on material are the calculations of the Spring Back, the Stress of Edge, and PSA - Profile Stress Analysis. and the Profile Weight.

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

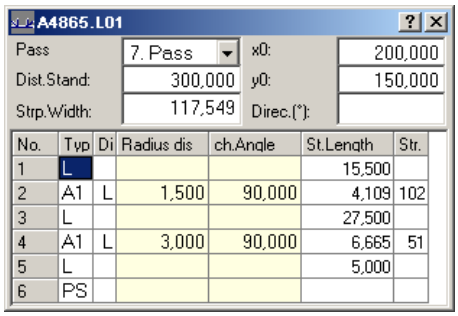
3.3.2.9 Thickness

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.

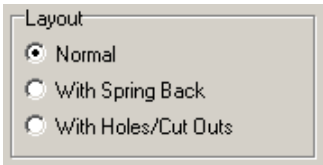
See also: Profile Project

3.3.3 Profile List

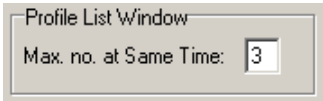


In this window, the Profile List is shown and you can edit it.

Select in Settings Calculate the number of decimal places and the unit mm or inch.



Furthermore select in Settings 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 Settings 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 Settings 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.

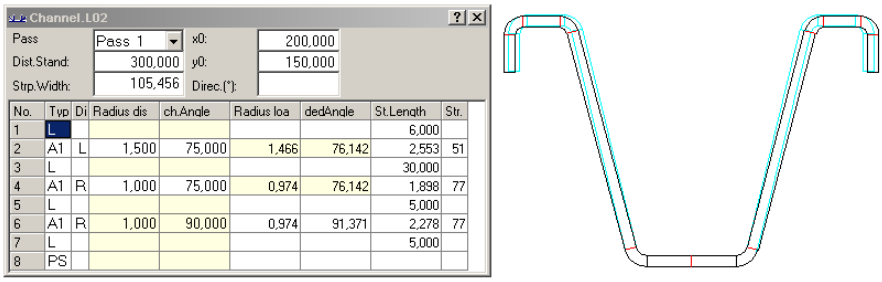
All values are shown in mm, if you have selected in Settings Profile List the representation of values **Metric**. If you have selected **Imperial**, they are shown in inch.

Hints:
In some input fields the values can be modified gradually by pressing the **Pg Up/Dn** keys on the keyboard. The step interval can be preset in Options, Mouse.
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



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 Spring Back is calculated from the Young's Modulus and the Yield Stress by using the **Kalpakjian** method.

The spring back is shown, if you select **With Spring Back** in Settings 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.

Hint: An alternative method to avoid spring back is the radius/angle method, see ArcTypes **B4**.

3.3.5 Developed Length

The developed length of each arc segment and with it the strip width normally is calculated dependent on sheet thickness, bend radius and angle, see Fundamentals. In reality, however, there are more dependencies. Also the material, the bending method and the bend angle sequence have influence. Anyway, this was the result of most recent research at the PtU of the Technical University of Darmstadt (Germany) and this insight meets the experience of the roll form designers.

This influence cannot be handled by a universally valid calculation method, but has to be defined individually and empirically. In **PROFIL** a separate calculation method can be applied to each arc segment. The user can select between a standard method, such as Oehler, DIN, centerline, or define user methods that can be equipped with empirical corrections. The latter are important in case larger angle increase is necessary because of limited count of stands and subsequently occurring deep drawing effects. Another important application is stiffening corrugations, whereupon standard calculation methods fail.

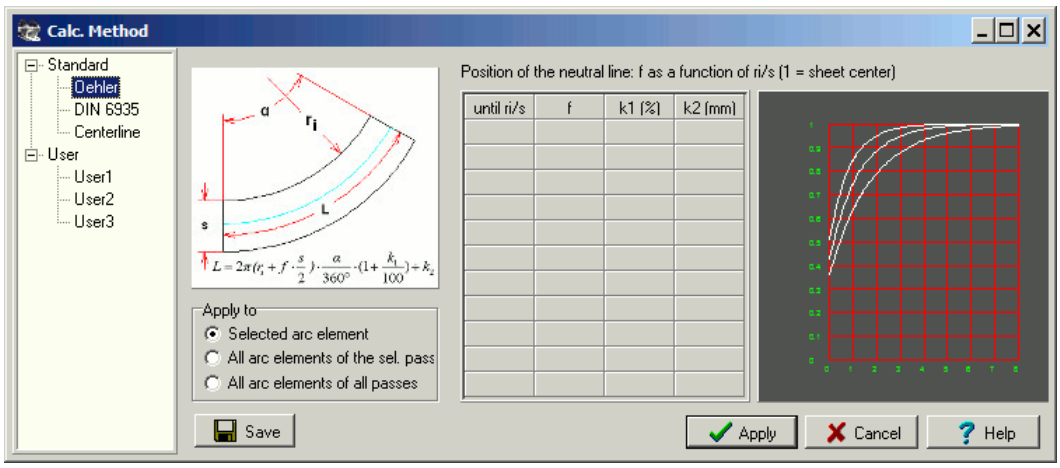
Calling the function

Before you call this function, select the desired profile element of type **Arc (A1..A4)** that you want to modify. Or select and profile element of type arc if you want to modify more than one profile element Call this function optionally by:

- Main menu: **Profile, Calc. Method**.

- After selecting a Profile Element of type **Arc (A1..A4)** in the Profile List Window, a white or colored button appears in the top center of the window. White means, the calculation method **Oehler** (default) is preset for this arc. Another color means, another method is preset. Click on the button.

The window **Calc. Methods** opens:



Principle of operation

Use this window to:

- Select a Calculation Method and apply to the selected arc, to all arc of a pass, or to all arc of all passes.
- Parameterize user defined calculation methods by using the Factor Method by filling out the factor table.
- Add empirically determined length corrections to the user defined calculation methods.
- Save the modified user defined methods.

Explorer (left side)

The explorer has two sections: Standard, internally defined methods that cannot be modified, and user methods as examples only that can be modified and extended by the user. The context menu, opened by a right mouse click on a user method, contains function for modifying the user methods:

- Insert
- Append
- Remove
- Rename

Apply to

- Selected arc element
- All arc elements of the selected pass
- All arc elements of all passes

Select, which arc elements should be modified after clicking on the **Apply** button.

Factor Table (center)

This table can be used for parametrization user defined calculation methods by using the Factor Method. Take care that the table ends with a large ri/s value (e.g. 999) that to all occurrent ri/s a proper factor for the position of the neutral line can be assigned.

Graph (right side)

The graph shows the factor f for the position of the neutral line dependent on the ratio ri/s (inner radius divided by sheet thickness). f = 1 means: The neutral line is the centerline of the sheet.

Save

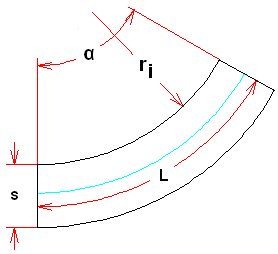
Saves the calculation into the Factor File, that is preset on Settings, Calculate. This file is imported, when **PROFIL** is started.

Apply
Applies the selected calculation method to the selected arc element.

See also: Fundamentals.

3.3.5.1 Fundamentals

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.



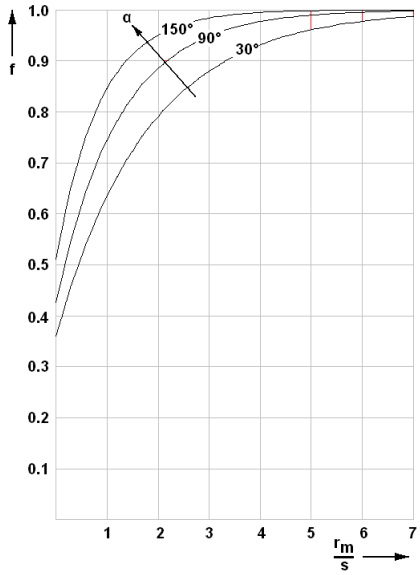
$$L = 2\pi(r_i + f \cdot \frac{s}{2}) \cdot \frac{\alpha}{360°}$$

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 toward 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.

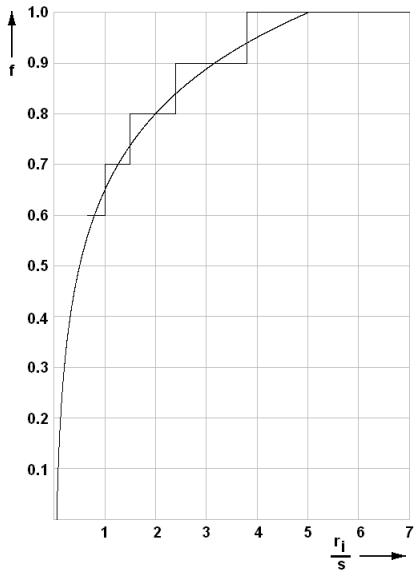
Oehler Method



$$f = f(r_m, s, \alpha)$$

The Oehler method determines the **factor f** as a function of the **medium radius rm**, the **sheet thickness s**, and of the **bending angle** α (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:

ri / 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 the Calculation Methods in the Developed Length Window, 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.

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.

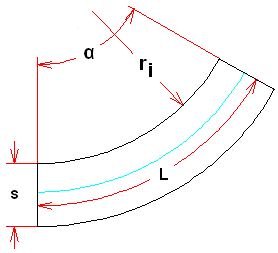
3.3.5.2 Factor Method

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:

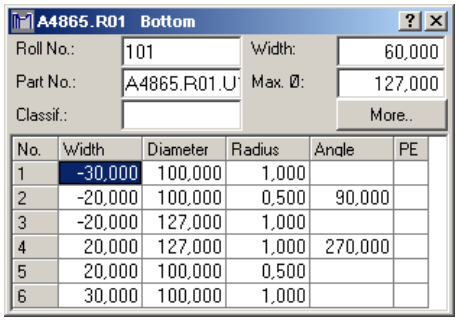


$$L = 2\pi(r_i + f \cdot \frac{s}{2}) \cdot \frac{\alpha}{360°} \cdot (1 + \frac{k_1}{100}) + k_2$$

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.

3.3.6 Roll Tool



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

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	

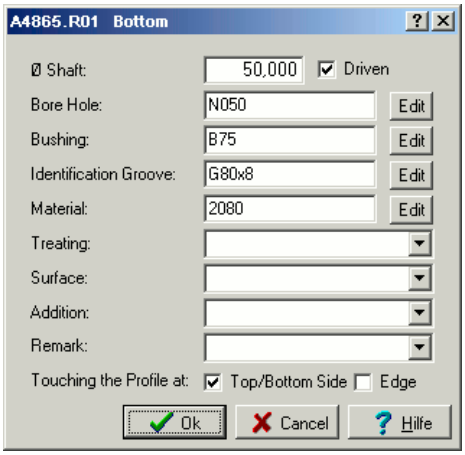
Click into a row of the table, and the corresponding roll corner point in the Drawing Area will be highlighted by a little cross-hair in the predefined mark color. Vice versa, if you identify a roll corner point in the drawing, the corresponding row of the table in the roll tool window will be activated.

All values are shown in mm, if you have selected in Settings Profile List the representation of values **Metric**. If you have selected **Imperial**, they are shown in inch.

Hints

- In some input fields the values can be modified gradually by pressing the **Pg Up/Dn** keys on the keyboard. The step interval can be preset in Options, Mouse.

3.3.7 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

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 Settings 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 Settings Rolls or in Settings, 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 Settings, Spacer Rolls, the field **Material** in 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.

3.3.7.1 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 Settings Drawing.

3.3.7.2 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.

3.3.7.3 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.

3.3.7.4 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 is the Roll Parts List, if this is set in Settings Parts List. Furthermore belonging bushing can be displayed in a separate column of the rolls.

3.3.7.5 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.

3.3.7.6 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 Settings, 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.

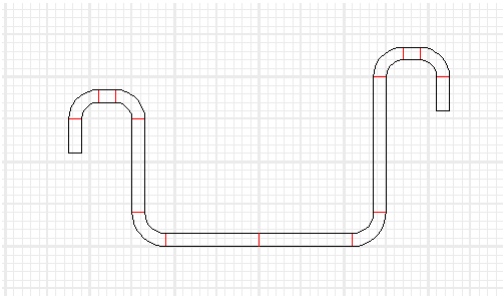
3.3.7.7 Treating, Surface, Addition, Remark

Treating, Surface, Addition, Remark are additional data, the field names of which can be defined by the user in Settings 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.

3.3.8 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 PSA - Profile Stress Analysis
View Statics
View Flower Separated
View Roll Tools

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.

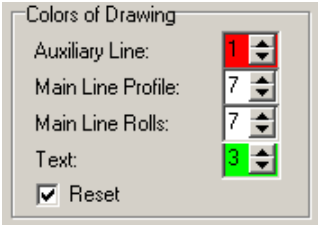
Inside the drawing of the roll tools, you can identify rolls and roll corner points. The data of the selected roll corner point are displayed in the bottom status bar. Simultaneously the corresponding Roll Corner Point in the Roll Tool Window will be activated, if the window is open.

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.

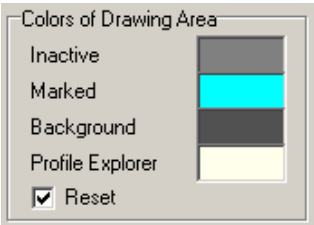
In Settings Rolls, **Keep Roll Data** select if the roll properties should be retained or if they should be adapted by retaining the roll contour only.

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.

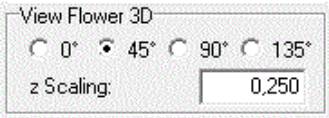
If you want to expand the drawing area, use the function Window Visible to switch off the windows.



Select the drawing entity colors in Settings Drawing, **Colors of Drawing**.



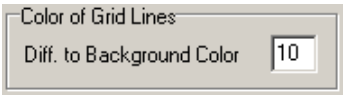
Select the drawing area colors in Settings Drawing, **Colors of Drawing Area**.



In case of a 3D drawing, select the 3D view port angle and the z scaling in Settings Drawing, **View Flower 3D**.

Select the text height in Settings Drawing, **Text**.

Select in Settings Drawing, **Grid** the grid lines distance and if the grid lines should be visible on the screen or not.

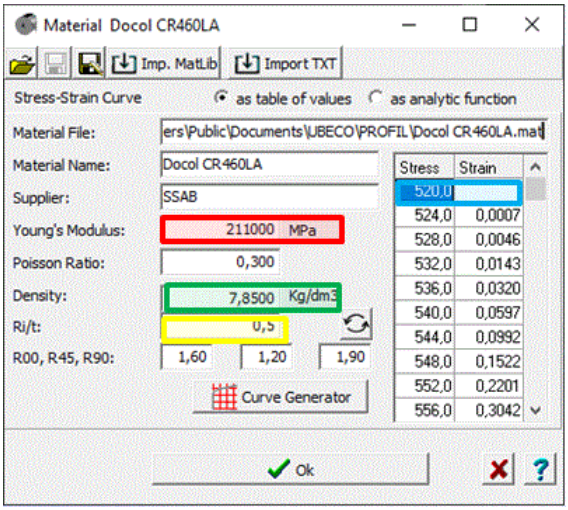


Select in Settings Drawing, how much the **color of the grid lines** should differ from the **background color.**

Use the function Drawing CAD to transfer the drawing, which is displayed in the Drawing Area, to your CAD System.

3.3.9 Material

This dialog window is called by Output, FEA, LS-Dyna. It is used for parameterizing the material properties, the stress-strain-curve, and for importing curves.



Colored input fields are used for calculations in the PROFIL basic version
All others are used for FEA output only

The material window shows the material date of the current Project. It is opened by the menu function Edit Material or with the button in the top Button Key Bar or by a mouse click into the input field **Material** in the Project Data Window. In the top line select:

- **Stress-Strain-Curve as table of values** for the **PROFIL** basic module (colored input fields) and the FEA extension (all input fields).
- **Stress-Strain-Curve as analytic function** for the FEA extension only.

Stress-Strain-Curve as table of values: This selection is useful if a table of values already exists (e.g. form a tensile test) of you want to approximate a given curve by using the curve generator.

Material File: This field shows the complete path and file name of the selected material file. A mouse click into this field opens the file explorer and you can select a different material file. The file name extension is .mat.

Material Name, Supplier: For describing the material and the source.

Young's Modulus: This is the slope of the Hook's line, i.e. the ratio stress to strain in the elastic area of the stress-strain curve. For steel material it is 211 MPa and for aluminum alloy about 70 MPa. In the **PROFIL** basic version the Young's modulus is used for calculating the Spring Back.

Poisson Ratio: This is the ratio of the relative change of sheet thickness to the relative change of sheet length during load.

Density: The PROFIL basic module calculates from the density the Weight of the profile ans shows it in the Statics Window.

Ri/t: Min. inner radius / sheet thickness, that should not be underrun in order to avoid cracks on the outside of a bending zone. Furthermore the Statics is not valid because of the notch effect. If Ri/t is given in supplier's data sheets, it can be entered here. If not it can be calculated approximately. Ri/t is used in the PROFIL basic version to calculate the row Stress in the Profile List Window.

If the min.inner radius / sheet thickness is unknown, it can be calculated approximately by pressing this button. Strip masterial until 500 MPa mostly has Ri/t = 0, i.e it can be bent sharp-edged. In the range of 600..800 MPa Ri/t is about 1, at 1000 MPa Ri/t rises to 2 and high-strength material of 1000..1500 MPa has Ri/t = 3.5 .. 4.

R00, R45, R90: The Lankford coefficients are a measure of the plastic anisotropy (i.e. different hardening in different directions) of the rolled precursor sheet metal. The number is the deviation in degree from the rolling direction. The coefficients are determined by tensile testing. Typical values for low-carbon, galvanized steel with sheet thickness 0.6mm are R00 = 1.60, R45 = 1.20 and R90 = 1.90.

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**.

1st Stress-Strain-Curve Point: This is the yield stress, i.e. the point in the stress-strain-curve, at which the elastic area ends and the plastic area starts. The PROFIL basic module uses the yield stress to calculate the Stress of Band Edge, PSA, and the Spring Back.

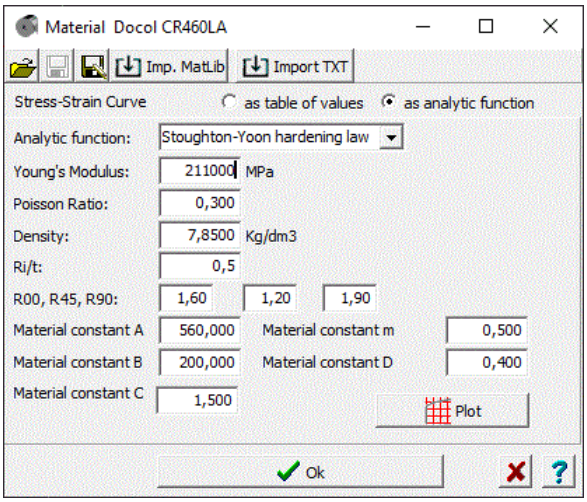
Material File open, save, save as ..: Use this function to select the material for a new project and to transfer material data from a project to a new project.

Ok **Ok Button:** Transfers the material data to the project. Afterwards the material data can be saved to the project file by using File Save in the PROFIL main window.

Import MatLib: A collection of material data is available in the LS-PrePost program path called MatLib. The .k files contain material properties an stress-strain-curves. They can be imported and saved as a **PROFIL** .mat file. If the message **No valid material data found** is shown, the material is not useful for roll forming applications. Because the MatLib files are created in metric units, toggle **PROFIL** in Settings Calculate to metric units before importing.

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. Toggle **PROFIL** in Settings Calculate to metric or imperial units before importing dependent on the file unit system. 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.



Stress-Strain-Curve as analytic function: Alternatively you can select an analytic function from a list. The function parameters can be varied until the desired yield point, the tensile stress, and the curve form is reached. Analytic functions make sense in the FEA output only.

EQ.1: Swift power law:

$$\sigma = K(\epsilon_0 + \epsilon_p)^n ,$$

where σ is true effective stress, ϵ_0 is the residual plastic strain at the initial yield point, K is a strength coefficient, ϵ_p is true effective plastic strain, and n is the work hardening coefficient.

EQ.2: Voce law with form:

$$\sigma = \sigma_0 + R_{sat}\left(1.0 - e^{-\zeta \epsilon_p}\right) ,$$

where σ_0 is the initial yield stress, R_{sat} is the stress differential between σ_0 and the saturated stress, and ζ is a strain coefficient.

EQ.4: Hockett-Sherby law:

$$\sigma = A - Be^{-C\epsilon_p^H} ,$$

where A, B, C, and H are material constants.

EQ.5: Stoughton-Yoon hardening law:

$$\sigma = A - Be^{-C\epsilon_p^m} + D\epsilon_p ,$$

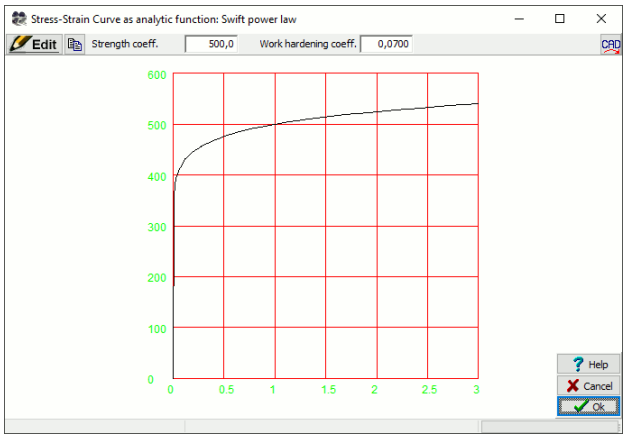
where A, B, C, m and D are material constants, such that $0 < m < 1.0$ and $D \geq 0.0$. According to Stoughton-Yoon, "with the exception of metals exhibiting Yield Point Elongation (YPE) effects, this function can represent the stress-strain response for both mild and AHSS steel and aluminum, from the initial yield point, throughout the small strain range, up to the highest strains realized in bulge tests". Note that if $D = 0.0$, this law reduces to the Hockett-Sherby law (ITYPE = 4). Also note that if $m = 1.0$ and $D = 0.0$, this law reduces to the ITYPE = 3 Voce law.

Extract from: LIVERMORE SOFTWARE TECHNOLOGY (LST), AN ANSYS COMPANY
LS-DYNA® KEYWORD USER'S MANUAL, VOLUME I, Keyword *DEFINE_CURVE_STRESS

Analytic function: Select between Swift power law, Voce law with form, Hockett-Sherby law and Stoughton-Yoon hardening law.

Material constant A .. D: By modifying the function parameters you can adapt the curve to your needs. Open

Plot: The selected curve is shown graphically.



Edit: The function parameters appear in the header and can be modified by checking the graphic curve simultaneously.

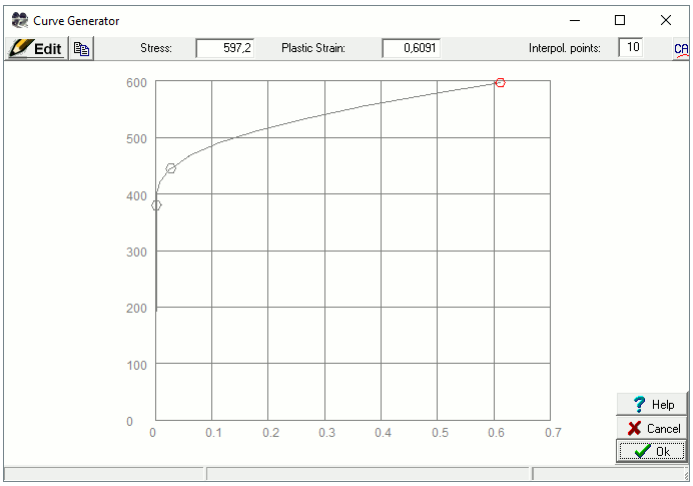
By using the **PgUp/Dn** keys on the keyboard the parameters can be modified step by step.

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 2 at least, better strain 3.
- 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.

3.3.9.1 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**. 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 that 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 dependent 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).

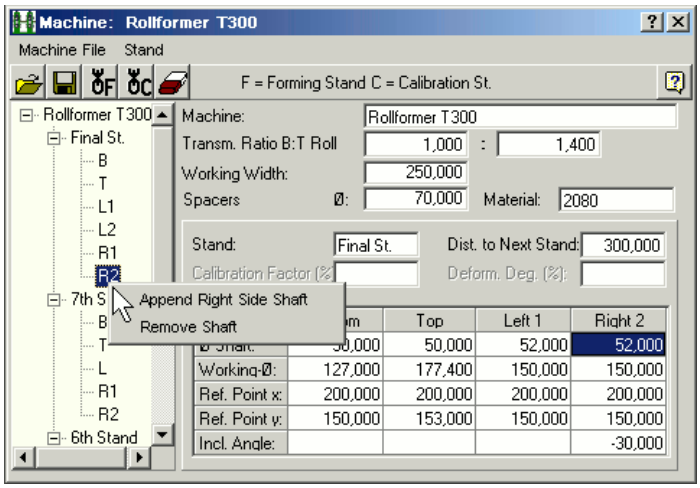
- **Drawing -> CAD:** Use this button to transfer the stress-strain-curve as vector drawing to the CAD system. The settings of the function Drawing -> CAD in the PROFIL main window are used as well.

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 Stain Curve Table** in **Output, FEA**.

3.3.10 Machine



The machine window shows the machine date 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
Calibrating Factor
(for calibrating stands only):
Deformation Degree

and the table with the data of the shafts/axles:

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 an be attached and removed again.

By calling the function **Append Forming Stand** or **Append Calibrating Stand** the shaft data are copied from the previous stand, the next stand, or the stand from the clipboard.

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.

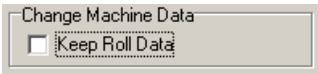
Export a Machine File. The machine data from the current project are saved into a file *.m01

Append Forming Stand: Behind the current stand (marked in the Machine Explorer) a new forming stand is created. The shaft data are copied from the previous stand without user request. So it is recommended to fill out the shaft data previously. If you want to copy the data from the next stand or from the stand in the clipboard, use the function **Append Forming Stand** in the main menu or in the context menu.

Append Calibrating Stand: Behind the current calibrating stand (marked in the Machine Explorer) a new calibrating stand is created. As noted above, the data are copied from the previous stand, too.

Remove stand.

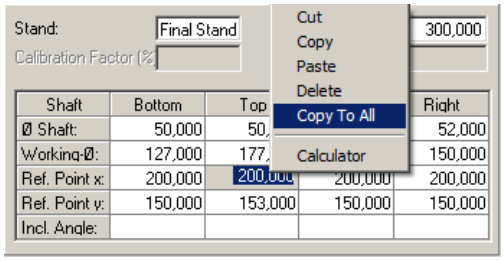
Pay attention to the sequence opposite the sheet running direction: first calibrating stands, afterwards forming stands. F01 always is the last forming stand in sheet running direction (the finishing stand), in case of tubes the welding station. C01 is the last calibrating stand (the finishing stand for shaped tubes).



In case the machine data are changed and rolls already exist in the project, the setting in Settings 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 Settings Profile List the representation of values **Metric**. If you have selected **Imperial**, they are shown in inch.

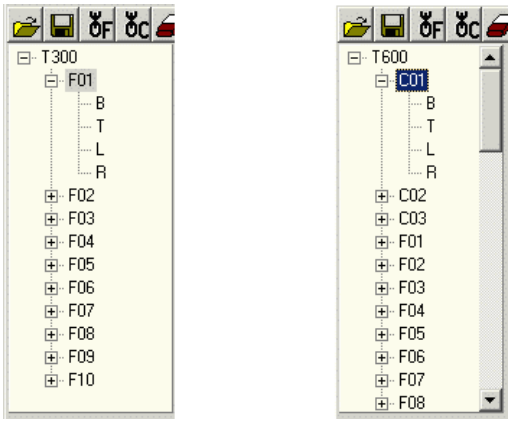
Context Menu:



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.

3.3.10.1 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
Calibrating Factor

(for calibrating stands only:)

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**).

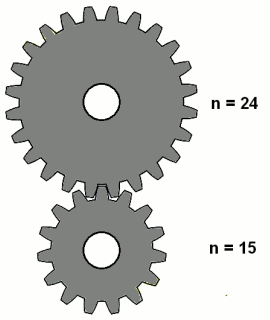
The stand name first is F01..Fnn for forming stands and C01..Cnn for calibrating stands. In the field Stand Name you can modify the name. The new name is shown in the machine explorer.

3.3.10.2 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 or 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..

3.3.10.3 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.

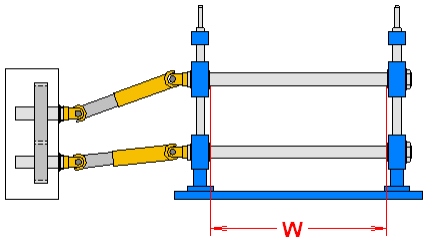
Most roll forming machines have the gear transmission ratio 1 : 1.4. In order to have the same circumferential speed (peripheral velocity) of the top and bottom roll at the profile web, the working diameter (pitch diameter) of the top roll must be the bottom roll diameter multiplied by 1.4. This enables 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.

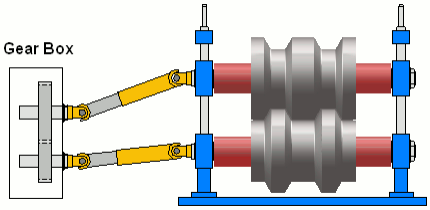
3.3.10.4 Working Width



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.

3.3.10.5 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.

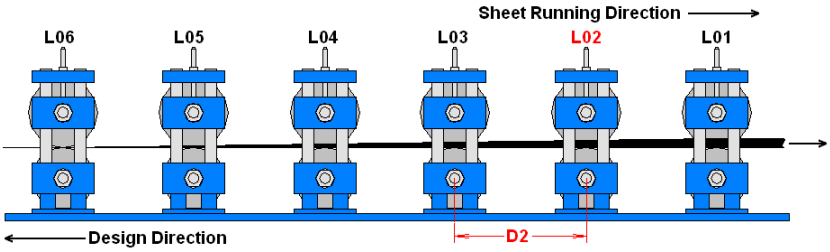
3.3.10.6 Stand Name

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.

3.3.10.7 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 stand **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.

3.3.10.8 Calibrating Factor

The calibrating factor belongs to machine data and is displayed in the bottom part of the Machine Window. The calibrating factor is used, if e.g. a tube is formed with high pressure and the strip width is reduced by this operation. As a result, the length of the tube is increased and the sheet width is increased.

The calibration factor says how much % the Strip Width of a tube decreases when the rolls of the certain stand mesh in. Read the documentation of your machine for the determination of the calibrating factor.

The calibrating factor is used by these functions for calibrating the strip width:

- Shaped Tube Calibration

- Profile, Calibrate Strip Width
- Profile, Calibrate Strip Width
- Fin Pass
- Break Down Pass
- Break Down Pass, W-Forming

3.3.10.9 Deformation Degree

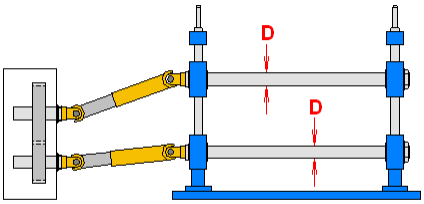
The deformation degree belongs to machine data and is displayed in the bottom part of the Machine Window. The input field is active only in case of calibrating stands. Use the deformation degree to have an influence on the shapes of the calibrating stands during the function Shaped Tube Calibration.

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.

3.3.10.10 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.

3.3.10.11 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 a fictitious diameter, which cannot be measured. For a more detailed description see chapter Reference Point.

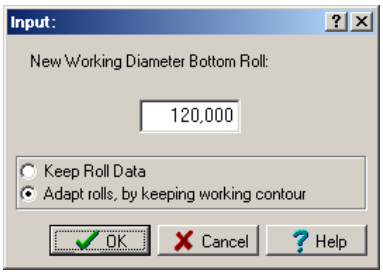
If you define the working diameters, pay attention to the Transmission Ratio of the machine (equal velocity at the important points of the profile) and if the shafts are adjustable. Read the documentation of the machine for the optimum value.

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.

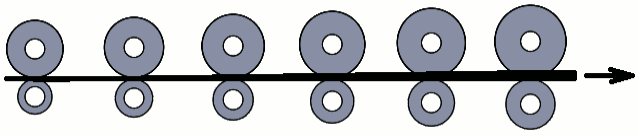
Calling the function

When you are going to modify the working diameter, select in Settings, 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 wan 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.

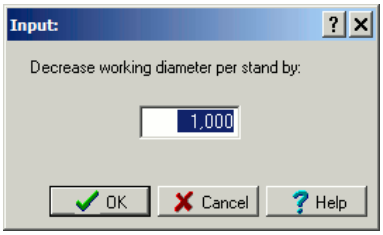


After calling the function the input window appears with the question **New Working Diameter?** In case of **Modify working diameter** the topic working diameter is shown and can be modified. In case of **Calculate working diameter from transm. ratio** the result of the calculation is shown and can be confirmed or modified. Furthermore the topic setting of the switch **Changing Machine Data**, **Keep Roll Data** in Settings, Rolls is shown and can be modified.



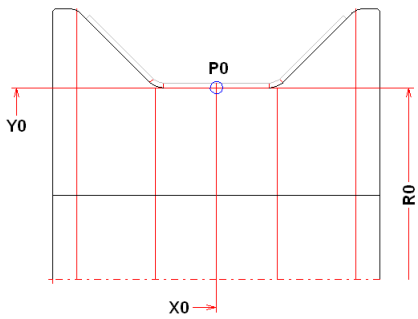
Lead (Strip tension):
It's a common practice to keep the strip under tension by gradually increasing the roll pitch diameters. Thus also the circumferential speed of the rolls is increased from stand to stand. This

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). This function can also be called via the machine window main menu **Rolls, Lead (Strip tension)** alternatively.



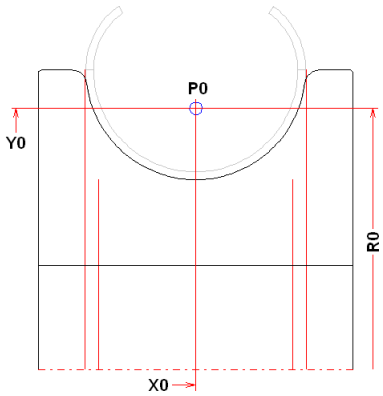
A window opens and asks **Decrease working diameter per stand by?** The diameters of all previous stands are decreased by the given value, thus the diameter increase in sheet running direction.

3.3.10.12 Reference Point



Pic. 1: The roll reference point is identical to the profile reference point

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.

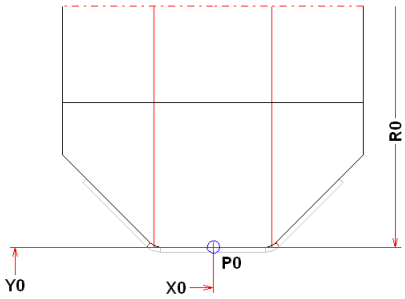


Pic. 2: The roll reference point is in the centroid of the profile

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

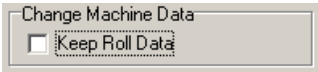
For top rolls it makes sense to set the roll reference point **P0** to the top side of the profile, i.e. to the point that is moved upwards from the Reference Point X0/Y0 by the sheet thickness (see pic. 3). Thus the Working Diameter **D0** (=2x working radius **R0**) can be measured at the roll. But 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 Settings, 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.

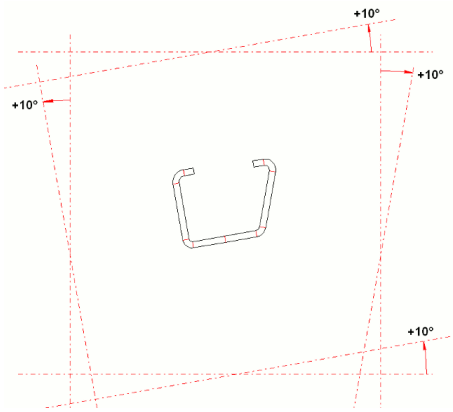
Principle of operation

- **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.

3.3.10.13 Inclination Angle



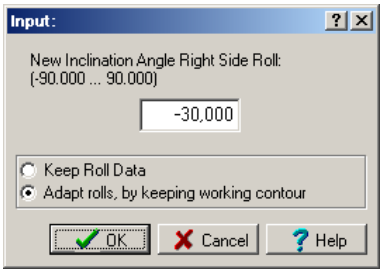
The inclination angle belongs to machine data and is displayed in the bottom part of the Machine Window for each shaft.

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 Settings, 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 Settings, Rolls is shown and can be modified.

3.3.11 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**. In case import from AutoCAD via ActiveX and import DXF file the desired block can be selected in the input field **Select Block**. 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.

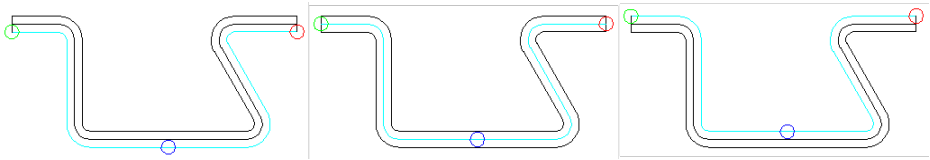
The read drawing contains the markers **Contour Start Point** (green) and **Contour End Point** (red) and - if you have called **Profile List Read CAD-Contour** - the additional maker **Reference Point x0/y0** (blue). Set the markers to the correct points of the contour by using the buttons in the button 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 nor find the desired contour: see topic **Controlling the contour tracking manually**.


Contour Start Point


Reference Point x0/y0

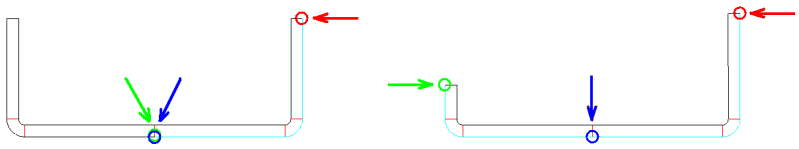

Contour End Point

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.

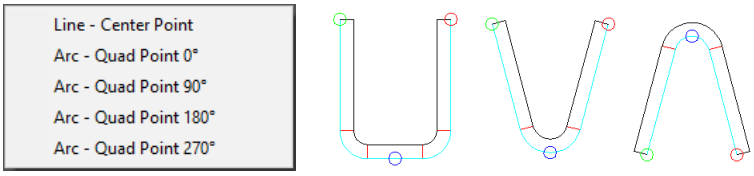


Symmetrical profile Asymmetrical profile

(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 unsymmetrical 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 **P**oint **S**ymmetrical) to the end of the profile list. This mirrors the right side to the left.

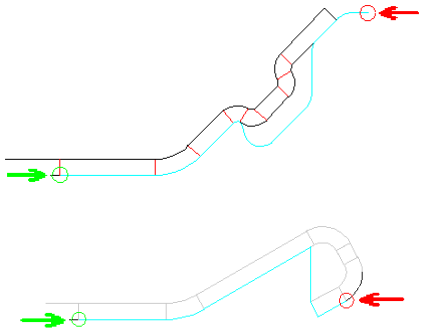
Snap points in the context menu:

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.



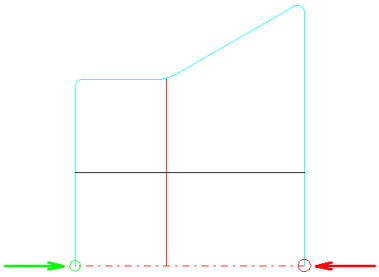
You can use further snap points by opening the context menu (right mouse button click), if the imported CAD contour has no element breakup at the desired position. The picture shows the typical applications: **Line Center Point** (left), **Arc Quad Point 270°** (center), and **Arc Quad Point 90°** (right). Furthermore possible: **Arc Quad Point 0°** and **Arc Quad Point 180°**. 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 that way the project file contains all data belonging to a profile design and can be archived and passed on to other users of PROFIL.

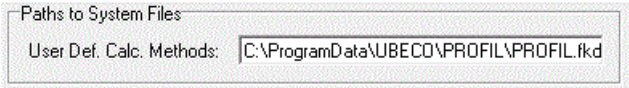
The project data are:

Customer	Date
Description	Name
Drawing No.	Revision
Material	Thickness
Machine	

If needed, the profile project name can be used as variable **$PR** for the automatic roll numbering, see Settings Rolls.

3.4.2 Factor File

In the factor file **PROFIL.FKD** you can define your own calculation methods by using the Factor Method. for the developed length. For parameterizing open the Developed Length Window.



Select the desired factor file by using the function Settings Calculate, **User Def. Calc. Methods**.

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.

For the **Factor method** there are now some lines with a leading **F** and the contents:

Column 1:	**F** 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.

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.

To define own methods modify the file, which contains only examples. You can modify the examples or append new methods.

3.4.3 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.

3.4.4 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, CADDA.. 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 unsymmetrical 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 unsymmetrical 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)

3.4.5 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 unsymmetrical 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 unsymmetrical 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.

3.4.6 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 Settings 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.

3.4.7 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 999 (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.

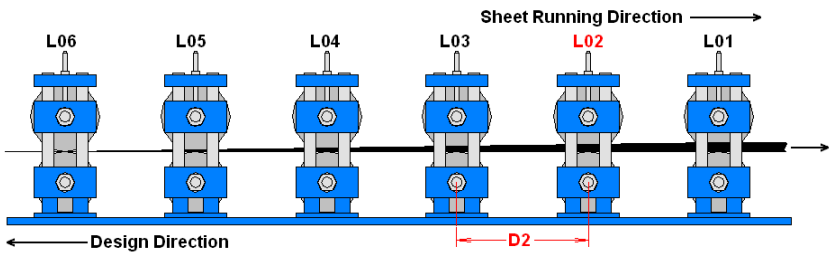
If additional profile lists are inserted in a profile project, the numbering in the field **Pass** is not renewed automatically. So it is necessary to renumber the pass number like this: Point to the arrow down key, select from the drop-down-list the desired numbering scheme and press ENTER. In order 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 Settings 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.

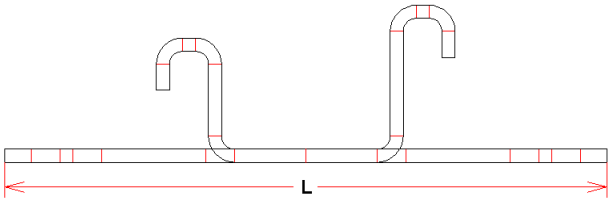
The distance between stands belongs to the Profile List data and is displayed in header of the the Profile List Window.

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.

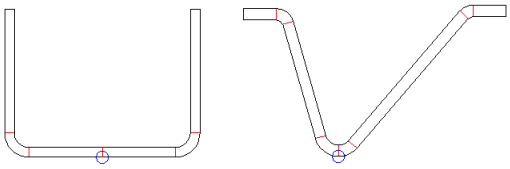
PROFIL contains different Calculation Methods to calculate the strip width.

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.

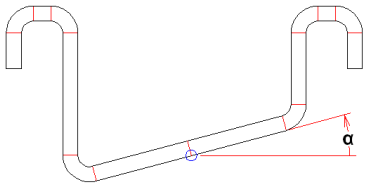
The reference point normally should be at or near the center of the profile's web on the sheet metal's bottom side (see left pic.). If the profile has not a horizontal web, the reference point can be set on the bottom point of an arc segment instead (see right pic.). Because both legs turn around the 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 α** 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 unsymmetrical 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 an unsymmetrical 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 from the discharged state, if you have selected a Material in the Project Data Window. Also the calculation of the discharged state out of the loaded state is possible.

The Spring Back is calculated from the Young's Modulus and the Yield Stress by using the **Kalpakjian** method.

Radius and angle are shown in the Profile List Window, if you have selected the layout **With Spring Back** in Settings 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 Settings 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 Settings 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 %.

The calculation is based on the ratio Ri/t (min. inner radius / sheet thickness) from the material data.

If values are greater than 100%, you must expect cracks. Furthermore the Statics is not valid because of the notch effect.

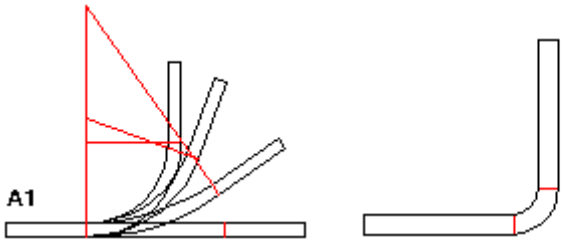
See also: Profile List.

3.5.6.10 PE

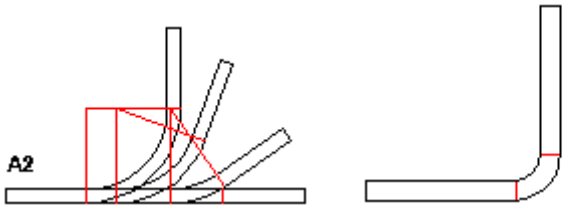
For future use only.

3.5.7 Arc Types

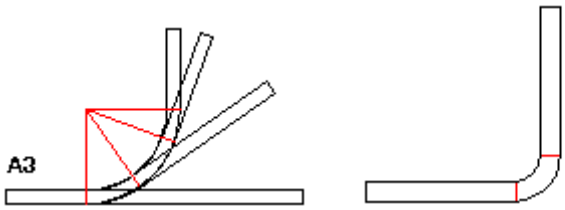
PROFIL knows 4 various arc types (bending methods), which are treated differently while bending:



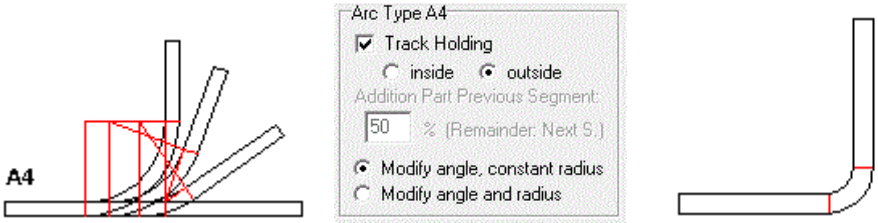
Arc type A1 has a constant straight length. By changing the bending angle the bending radius is calculated and modified. Also possible: by changing the radius the angle is calculated and modified. This method often is used in case of sharp bending (this means small inner radii).



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.



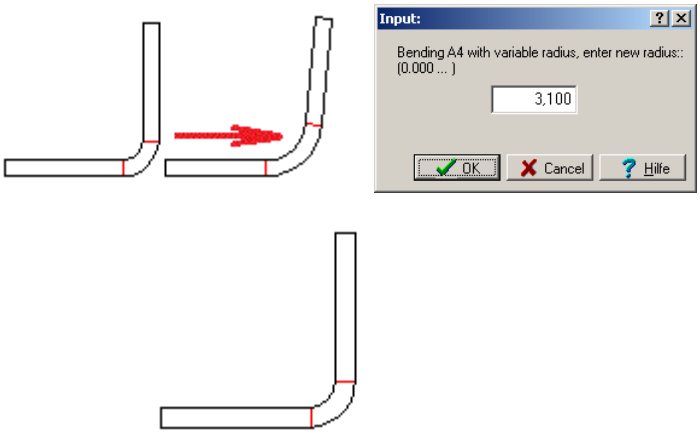
Arc type A3 is similar to arc type **A2**, but instead of the previous segment the next one is used for the length compensation. PROFIL calculates the new lengths. Also this method often is used for large radii.



Arc type A4 is a combination of **A2** and **A3**. Select in Settings 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 Settings 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 that 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 Settings 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 Settings 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.

3.6 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 Settings 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 Settings 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 Settings 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 Settings 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.

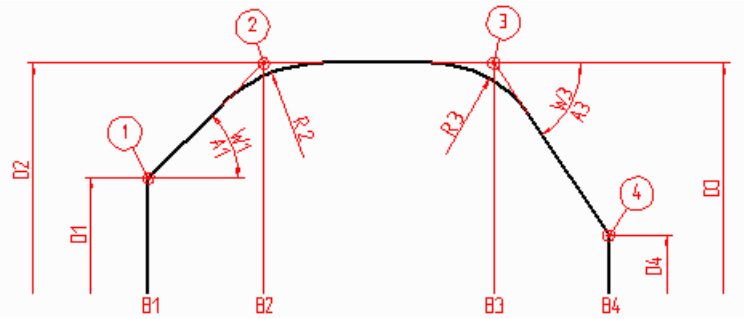
3.6.6 Spacer Roll

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. Dependent 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 Settings, 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 Settings, 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:

Width
Radius
Diameter
Angle

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 Settings 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.

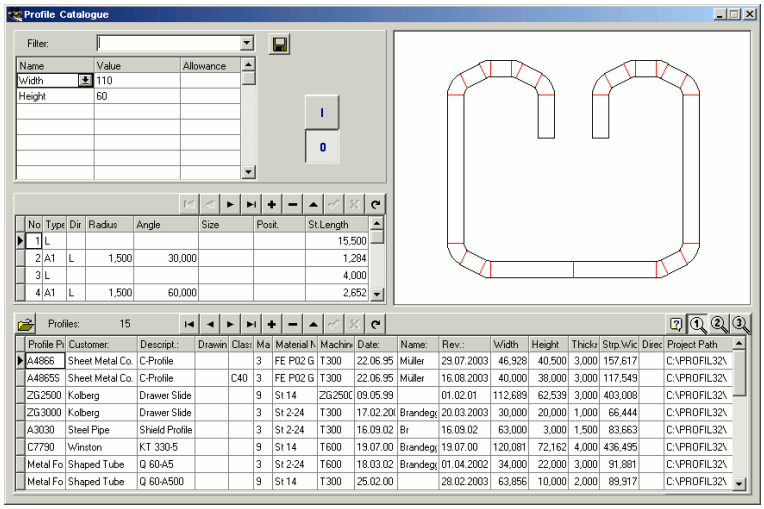
The angle is positive, if the next corner point has a larger radius, in the other case it is negative. The roll axle is the reference line for the angle. Thus a cylindric part of a roll has the angle 0°.

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.

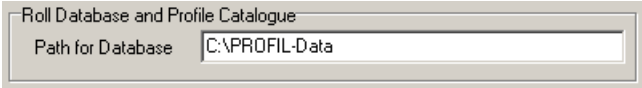
Profile Catalogue Save

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

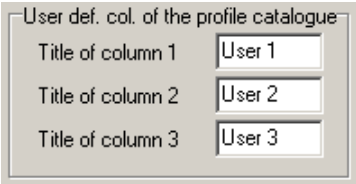
- **Current Project:** The currently opened project is stored.
- **All Projects of a Path:** The path selection window appears and all projects of the selected path are stored.

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 Settings Database.



Title of column 1..3: In Settings Database, enter the titles for the last 3 columns of the Profile Table. You can define these columns for your own needs.

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:

Customer	Date	
Description	Name	
Drawing No.	Revision	
Material		Thickness
Machine		

Further columns contain the profile data of the Profile List:
Strip width
Direction

From the profile geometry are calculated:

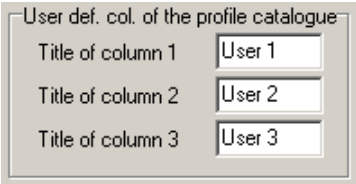
- **Width**: Total width of the profile
- **Height**: Total height of the profile

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 Settings 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 Settings 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.

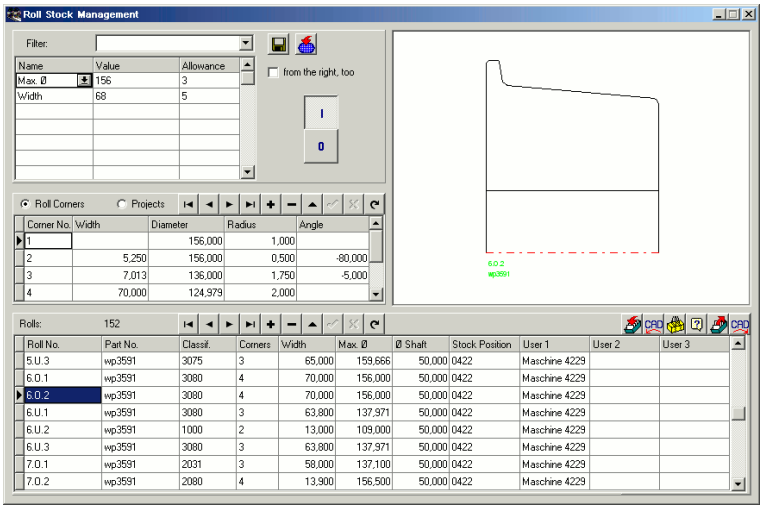
Buttons

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.

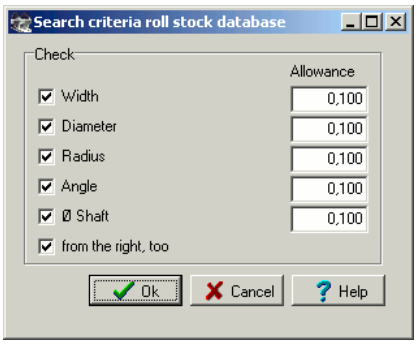
Calling the function

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 Search

Use this function to check if a designed roll (within your project) already exists in the roll stock. If a roll exists with same or similar properties, the designed roll can be replaced by the stock roll for re-use. For replacing, use the function **Replace roll in project by roll from roll stock** in the Roll Table.



Before calling this function, select the designed roll in the project that should be replaced by a stock roll. After calling this function the window **Search criteria roll stock database** opens with these entries:
Width: Valid for widths of all roll corner points.

- **Diameter:** Valid for diameters of all roll corner points.
- **Radius:** Valid for radii of all roll corner points.
- **Angle:** Valid for contour angles between all roll corner points to the next.
- **Ø Shaft**
- **from the right, too:** Means that also rolls are found that are saved mirrored.

Select by checking the boxes which search criteria should be used. For each criterion, you can enter an allowance. Avoid allowance 0, because unsafe results can occur. After pressing the **Ok** button, the Roll Stock Window opens and shows the rolls that fit to the given criteria. If no roll is shown, uncheck one or more check boxes or increase the allowances in order to make the search more tolerant. E.g. if you uncheck the criterion **Diameter**, rolls are shown that have the desired contour but have a different diameter. You can re-use this roll nevertheless by modifying the height adjustment of the shaft.

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.

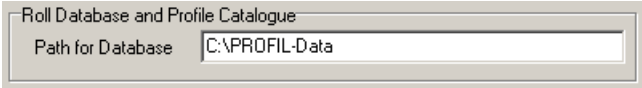
Stock Management Save

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:

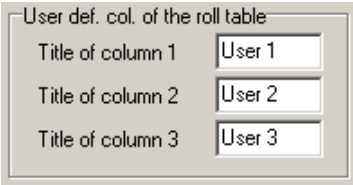
- **Roll:** The marked roll is stored.
- **Shaft:** All rolls of a shaft are stored. Define the shaft by marking any roll of the shaft.
- **Stand:** All rolls of all shafts of one stand are stored.
- **Project:** All rolls of all stands of the whole project are stored.

If the window Roll Stock Management is not yet open, it will be opened after calling one of these functions.

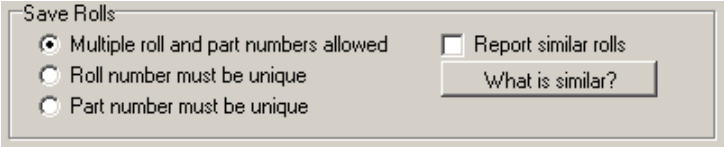
Setup



Set the path to the roll database in Settings Stock Management.



Title of column 1..3: Enter in Settings Stock Management the titles for the last 3 columns of the Roll Table. You can define these columns for your own needs.



Save Rolls: In Settings Stock Management, you can define wether multiple roll and part numbers are allowed or if the must be unique.Furthermore you can check if similar rolls already exist in the roll stock before saving a roll. In **What is similar?** you can enter the criteria for for searching similar rolls.

3.8.1 Roll Table

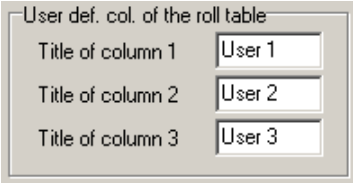
The lower area of the Roll Stock Management Window displays the table of the roll data. This table contains all rolls (if the filter is switched off) or a subset of the rolls (if the filter is switched on). Each row of the table displays the data of one roll. At first, there are the data from the top of the Roll Tool Window:

Roll Number	Width
Part Number	Max. Diameter
Classification	Diameter Shaft
Number of Roll Corner Points	

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 Settings 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.

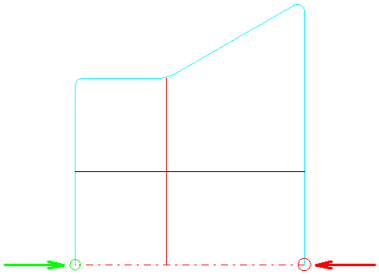
The top bar of the roll table contains these indicators and buttons:

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 Copypreviously.

 **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 or ZWCAD rel. 2021 or higher** (Precondition: In Settings 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 Settings 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 Settings Files **Output to CAD, A11, DXF or IGES** is selected): The KTR or DXF file is read that is set in Settings 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.

Replace roll in project by roll from roll stock
copies the selected roll from the database to the project, deletes the selected roll in the project and replaces it by the roll from the database. The clipboard is used for this transfer as well.

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:

Width	Diameter
Radius	Angle

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 Settings 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 that 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 Settings 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

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 UBECO 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 Settings Files:


CAD system **AutoCAD R14 and higher** or **SolidWorks 2003 and higher** or **SolidEdge ST7 or higher** or **BricsCAD release 15 and higher** or **ZWCAD release 2021 and higher**: Use the built-in ActiveX interface (Setting in Settings 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. You should have received these files in an e-mail download link. If a necessary file is absent please order it or download it from the web-site http://www.ubeco.com

- **profil64.exe** Setup procedure for the demo version 64 bit for WINDOWS 7/8/10/11 (Contains the ActiveX interface to **AutoCAD**, **SolidWorks**, **SolidEdge**, **BricsCAD** and **ZWCAD** 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

Please start the installation with Installing PROFIL.

4.2 Installing PROFIL

In order to install **PROFIL** for WINDOWS 7,8,10, 11 proceed as described:

- Installation: Start the setup program:

  PROFIL64.exe (64bit version)

  and follow the instructions of the setup procedure. Afterwards the **PROFIL** demo version with limited functions is installed.
- Login at WINDOWS as **Administrator**.
- 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. Check this in the initial window.

  copy ubeco.psf "c:\program files\UBECO\PROFIL" (Win 7/8/10)

  If you have more than 1 license, use the Personality File with the proper serial number that fits to the serial number on the hardlock lanyard.

PROFIL LT: The software now is fully installed for WINDOWS.

PROFIL Full Version: With it the **PROFIL** is installed and - in case you use **AutoCAD** or **SolidWorks** or **SolidEdge** or **BricsCAD** or **ZWCAD** - 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 7, 8 10 and 11. 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).
- Start **PROFIL** and check the initial window, your **PROFIL** option (DB, FEA), your serial number (identically with the number on the hardlock lanyard), and your company name must be displayed.

4.4 ActiveX Interface to AutoCAD, SolidWorks, SolidEdge, BricsCAD, ZWCAD


The interface to AutoCAD R14, SolidWorks, SolidEdge, and BricsCAD 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/ZWCAD 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 Settings 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:

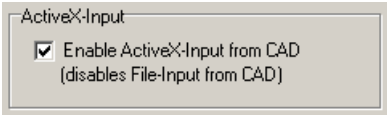
  AutoCAD 14: AutoCAD.Application.14
  AutoCAD 2000: AutoCAD.Application.15
  AutoCAD 2004: AutoCAD.Application.16
  AutoCAD 2005: AutoCAD.Application.16.1
  AutoCAD 2006: AutoCAD.Application.16.2
  AutoCAD 2007: AutoCAD.Application.17
  AutoCAD 2008: AutoCAD.Application.17.1
  AutoCAD 2009: AutoCAD.Application.17.2
  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
AutoCAD 2022: AutoCAD.Application.24.1 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 2015: SldWorks.Application.23
  SolidWorks 2016: SldWorks.Application.24 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
  BricsCAD Release 16: BricscadApp.AcadApplication 16.0
- **BricscadApp.AcadApplication**: Connects to the current BricscadApp release, this is the last installed one.

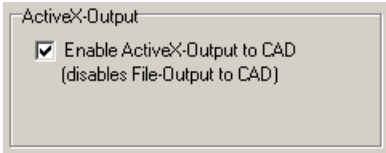
If the drop-down-list is empty, no AutoCAD or SolidWorks or SolidEdge or BricsCAD release is installed on your computer.

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 Settings 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 Settings 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. 5.3 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 PROFI.

Peculiarity of the BricsCAD interface:

Since PROFIL Rel. 5.3 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. Since BricsCAD Rel. 16 also 3D models can be transferred from PROFIL to BricsCAD.

Properties of the ZWCAD interface:

Since PROFIL Rel. 6.0.3 and ZWCAD Rel. 2021 this interface has been provided to transfer profile and roll tools drawings from PROFIL to ZWCAD and to transfer profile and roll contours from ZWCAD to PROFIL.

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 Settings 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 Settings 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 Settings 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 Settings 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 **UBECO**.

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