

Automatic meshing

Table contains for each profile element in mm

- A: Real element width
- L: Limit for maximum element width

Table can be preset with default widths and can be modified manually. Table is saved in the project file.

Default widths: If the Table is empty in a new project, it is filled with these default widths dependent on the radius. After modifying the profile list sequence the table settings can be overwritten by the **Copy to table** button. The default values are saved in the Windows registry.

Element width ratio: For lines (L) this factor is used for increasing neighboring elements until the limit in the L line is reached. Ratio = 0: Manual meshing, from the L lines the real widths are taken.

Start, FEA Input/Output

Simulation Path: Path for the output of the simulation model. This is the network path of the LS-Dyna solver.

FEA-Project Name: Often several simulations are needed for the same project. Select a different FEA project name for each simulation for clear archiving.

Start Simulation (either - or):

- With flat sheet,
- With a proper simulation result after modifying a roll in the next stand.
- With a preformed (ideal) profile, that is taken form a certain pass of the lower pattern.

Rolling Speed: Is used when the lead end of the strip enters each stand.

Overdrive Factor: Otherwise the speed is increased by this factor.

Material

Young's Modulus = Inclination of the Hooke's line (elastic area)

Table of Stress-Strain-Curve points (plastic area, true stress):

- Point = Yield Stress with Strain = 0 (true stress)

True Stress, curve must increase continuously, i.e. eliminate decrease caused by constriction

Import TXT: Result from a Tensile Test

Guiding

Guiding the nodes at the profile reference point horizontally mandatory for symmetric profiles!

Guiding the first row of nodes at the profile lead end. Basis upon flower pattern and PSA model. Also at the profile tail end.

Guiding the nodes at the profile reference point vertically. Not for center line forming!

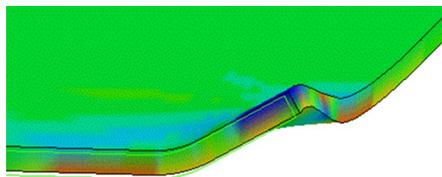
While entering the FEA parameter the user can see the slides of the FEA training directly in the software (examples)

What's New? – Rel. 5.6

Built-in Slides of the PROFIL FEA Training

While parameterizing the simulation model for the FEA system LS-DYNA, the corresponding slides of the FEA training can be displayed by pressing the help button. So the designer easily remembers the meaning of the input fields without consulting the manual each time. If the info is not sufficient, a further key-press opens the appropriate manual page.

Comparison Plan vs. Actual Analysis

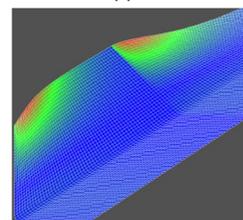


The passes of the designed flower pattern (light green in the front) are shown in the FEA result if desired. Because contact conditions are not defined, the designed passes have no influence on the simulation result. So the desired cross-section can be compared with the actual formed by the rolls. Unwanted deviation can be detected easily. After modifying faulty rolls the simulation is restarted at the certain stand.

Avoiding unwanted deformations

Stress related to the yield stress:		Material	3 FE P02 G 275 NA	Re = 380 MPa
		%	Ps	%
<input type="checkbox"/> Band edge only		73	1	73
<input type="checkbox"/> Center Line Forming		73	2	73
		112	3	112
		108	4	108
		10	5	10
		14	6	14
		8	7	8
		0	8	0

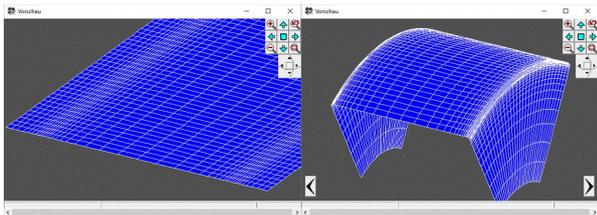
Edge waviness, bow, camber, twist, oil-canning, end flare: These are problems the designer of roll forming tools often is facing. Deformations mostly are caused by too high longitudinal stress. This is why the yield point of the material is exceeded. A valuable help is the calculation of stress and stain at the band edge. It is calculated by using 3D curves for the longitudinal fiber that approximate the real course of the band edge.



A very new feature is the calculation within the whole profile cross section. Thus too high strain also can be recognized if the maximum is not at the band edge, e.g. in case of bending upwards folded edges (hems). The switch **Band edge only** toggles between both methods.

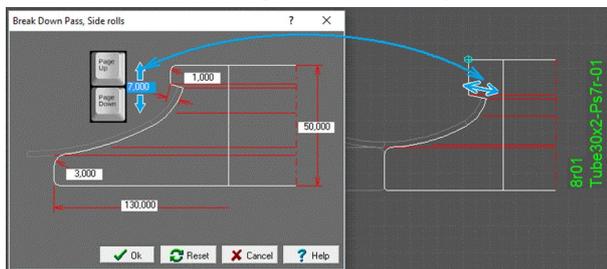
Another switch is useful for a quick check if **Center Line Forming** solves a problem of too high stress. This means the centroid of an area is constant in all passes.

Meshing Preview



When presetting the meshing parameter it is not necessary anymore to try, export the files, and check the meshing. A preview window shows the meshed sheet and the meshed rolls. So the user can see the effect of parameter changes to the FEA model directly during input.

Tool Boxes Tube Forming and Profile Design



The values in all input fields of the tool boxes (left) can be modified step by step by using the Pg Up/Dn keys. The step interval can be preset. At the same time, the effect of the modification is shown in a preview in the drawing area (right) simultaneously. This makes it easier to use the tool boxes for the design of tube forming passes and rolls and the final cross section of open profiles.

Further New Features

FEA Finite Element Analysis:

- Sometimes it is necessary to restart the simulation with an existing FEA result after modifying the count of stands. To enable this, the user is asked to confirm or modify the stand number for restart.
- Extended plausibility check: Mixed symmetric and unsymmetric passes are reported. Check, if the first profile list contains the flat sheet and not rolls.
- To enable the simulation of large projects with a large count of rolls, each stand now has an own roll file. This file contains rolls only that have contact to the sheet.

Profile Design:

- In the window stress of band edge the corresponding training slide can be opened for better understanding.

Tube Design:

- Calibrating the sheet width, if the sheet width is reduced in fin passes.
- In the tool box tube forming the calibration factor is considered in fin passes and in break down passes.

Machine:

- Lead (strip tension) now as a menu function.

CAD Interfacing:

- DXF import: Files with BLOCKS can be imported, too.
- ActiveX import from AutoCAD: BLOCKS and POLYLINES can be imported, too.

More info: www.ubeco.com