Workshop

AutoForm – DieDesigner 3.03

AutoForm-DieDesigner

AutoForm-DieDesigner – V3.03 Overview

AutoForm-DieDesigner is a tool intended for interactive and rapid design of draw dies. The procedure for designing draw dies and the resulting geometric entities have been designed to facilitate easy and rapid verification and optimisation of the dies and process parameters using incremental simulations. Therefore AutoForm-DieDesigner V3.03 has to fulfil certain requirements which are reflected in the new functionality:

- Automatic, variable filleting
- Determination of stamping direction (die tip)
- Automatic filling in of holes and smoothing of part edges
- Automatic, interactive generation of binder surface
- Automatic, interactive generation of addendum
- 2D-simulation for pre-optimising addendum profiles
- Fully parametric implementation of all die geometry

The main objectives in the development of AutoForm-DieDesigner were ease of use, flexibility and speed in generating and modifying die faces. As is the case with all AutoForm modules, many software features have been automated. However, when the design requirements are complex, the user may have to work interactively with the software to override the automatic parameters when necessary. This was also taken into account during the development of AutoForm-DieDesigner. In particular, for complex part geometries, the profiles used in addendum generation have to be carefully determined by the user.

AutoForm-DieDesigner is fully parametric and all input values, as well as profile parameters are saved. Changing these parameters may modify any profile. Furthermore, data structures within AutoForm-DieDesigner are fully parameterised. As a result, when the binder surface is modified, all profiles of the addendum are automatically adjusted to the new binder surface by simply clicking on "Update". This parametric functionality is sufficiently advanced that the entire part may be replaced by a new one and the entire addendum is adjusted to this new data. Clearly, this is limited to similar parts, e.g. those representing different stages of change and during the development life cycle.

An important aspect of parametric addendums is fully automatic optimisation of the addendum. This feature has been fully implemented in the current version.

AutoForm-DieDesigner is most appropriate for tooling concepts and addressing formability issues and has not been designed, to model a series production tool. The finer details of a die face are still best developed in a CAD-system, since the modelling of such details using profiles in DieDesigner may be tedious or infeasible. To address this requirement, AutoForm-DieDesigner V3.0 has the necessary interface to CAD systems (IGES, VDA-FS) as a Beta Version implementation.

Areas for further improvement in V3.03 that are being currently worked on for future releases include:

- interactive generation of additional faces on the part
- generation of addendum for double-attached parts
- inner binders
- automatic flange layout
- complex profiles
- local modification (morphing) of tool faces.

The integration of AutoForm-OneStep and Drawbead-Generator has been completed and these functions are available in V3.03.

This manual is written in the form of a tutorial workshop for AutoForm-DieDesigner. Several examples are used to illustrate how to work with the program to generate tool geometry. It also introduces the new line editor, and illustrates the generation of combined lines and their global modification.

The examples used in Chapter 2 are provided on your CD-ROM (tar-file: *training.tar*). Directories example data for *lesson1...* can be extracted from this tar-file. Each directory contains the necessary input data discussed in the following chapters. In most cases you will also find a corresponding *simulation (.sim) file* that may be created by following the procedure outlined in the relevant chapter.

It is recommended that the user review all of the included lessons before proceeding to use AutoForm DieDesigner on actual part geometries. This is the ideal way to get familiar with the functionality of the software as well as its capability and limitations.

Manual Usage Note:

This manual is structured as a workshop with several examples to help you fully understand the functionality as well as underlying concepts of AutoForm DieDesigner.

Working with AutoForm-DieDesigner

2.1   Lesson 1 – Basic Functions

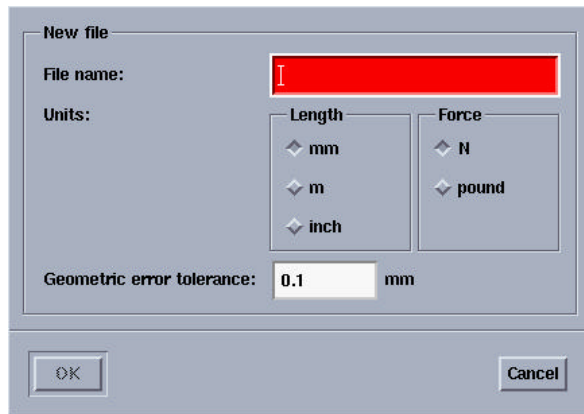**Simulation > New   (Opens the following window)**

Fig. 2.1.1 New model page

These units are used for post-processing and for setting realistic default values. Error tolerance determines the global chordal deviation used in all algorithms.

In the red box titled File name, type "lesson1" and click OK. This will open the Geometry generator window shown below at the default Prepare page.
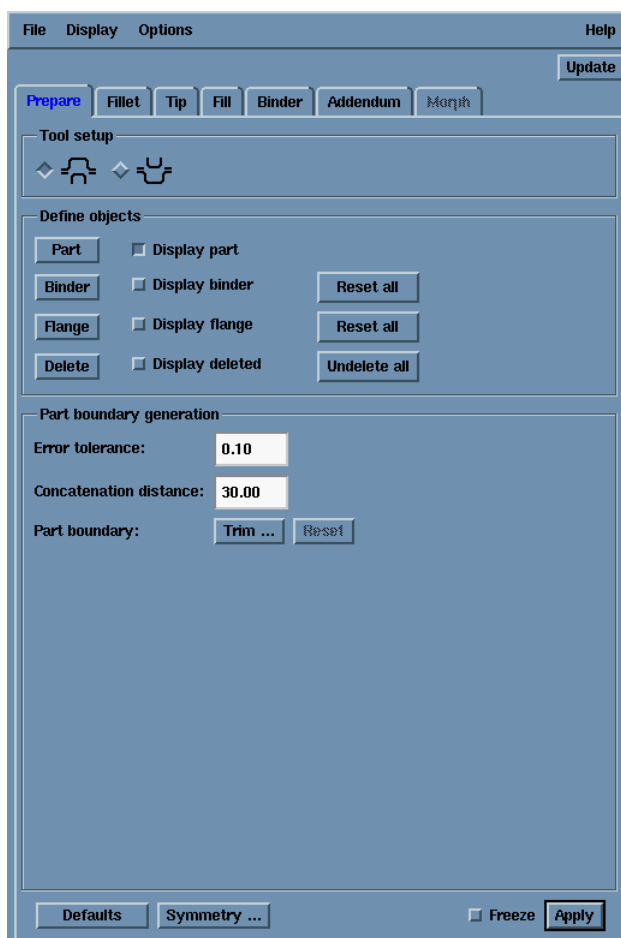
Fig. 2.1.2 Prepare page

Tool setup defines the geometric position of tools relative to each other. The first button (on the left) means that the die is above the punch and binder The second (right) buttonmeans that the die is below the punch and binder. The buttons named **Part, Binder, Flange** and **Delete** refer to specific faces of the part and tool, respectively.

- *Part **denotes part geometry to be formed in first draw.***
- *Binder **refers to the binder surface,***
- *Flange **allows working with the flange area.***
- *Delete **contains all deleted or unnecessary faces.***

V3.03 allows editing of the part boundary using the **"Part boundary: Trim"** function. The points can only be moved on the part patches in order to cut details of the part boundary. In order to activate this function the Apply button has to be used first in order to automatically define the part boundary.

Symmetry allows symmetry line to be defined, e.g., to mirror geometry about a reference plane.
Select File > Import > af > OK > Select > lesson1.af > OK

The part geometry is now shown in main graphics window.

Click on Prepare/Apply: the part boundary is automatically generated.
On this page, click on the bottom half of the vertical line (with left mouse button) and move it to approximately 50 mm, or use the dialog box to set the PO width **to 50.**
Click on Binder/Select Auto/Apply: This generates the initial binder surface.

Activate Main Display **and press the "c" key. The part and binder are now shown in different colors.**

Click on the Addendum Tab to open the Addendum profiles page (Fig 2.1.3)
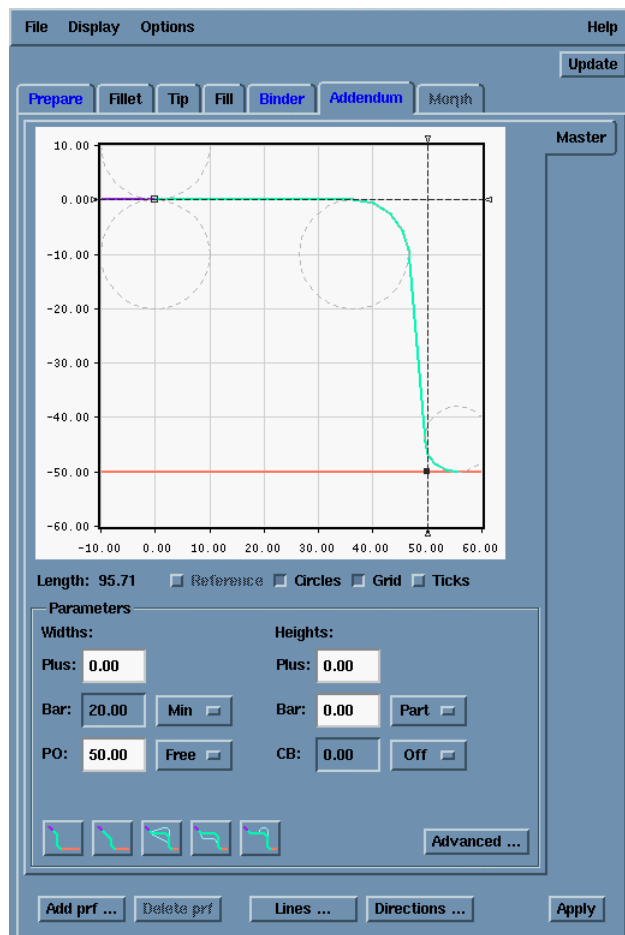

Fig. 2.1.3 Addendum page with master profile
Explanations for use of Option Menus for Bar Width, PO Width, Bar Height and CB Height as well as for the 5 Icons **can be found in chapter 3: Input pages.**

The master profile is used as the primary reference profile for the global addendum geometry. Here, global settings for parameters such as Width and Height, radii and angles can be done which are used as default by all other profiles.
The displayed profile does not represent the real geometry at every position on the addendum because the distance between part and binder varies. The displayed master profile is a schematic drawing with respect of the settings for width, height, radii and angles.

The dotted circles represent various radii parameters that can be interactively changed. Using the left mouse button, click and drag the circle perimeter. The dimension of the radius is shown on the upper left corner.

Click on Apply (The *Main Display* **should now contain the tool.**)

It is necessary for the binder surface to be larger.

Binder tab: Adv > Min flange width 200 > Apply

The binder surface is now larger, the **Addendum** button, however, is red because the addendum has not yet been connected to the new binder surface.

Press Addendum > Apply **and the tool is updated with the new binder.**

Changing the die radius:

At this point we will modify the die radius in the lower corner (geometry viewed from +z). On the *Addendum page* select **Options>Show All** to display all available profiles. In the following example, profile numbers may differ slightly from those on your screen; Rather than use the exact profile number, it is more important to select the profiles in the lower left corner area where the die radius is to be changed.

Use right mouse button to click on   or use the **Add prf** button

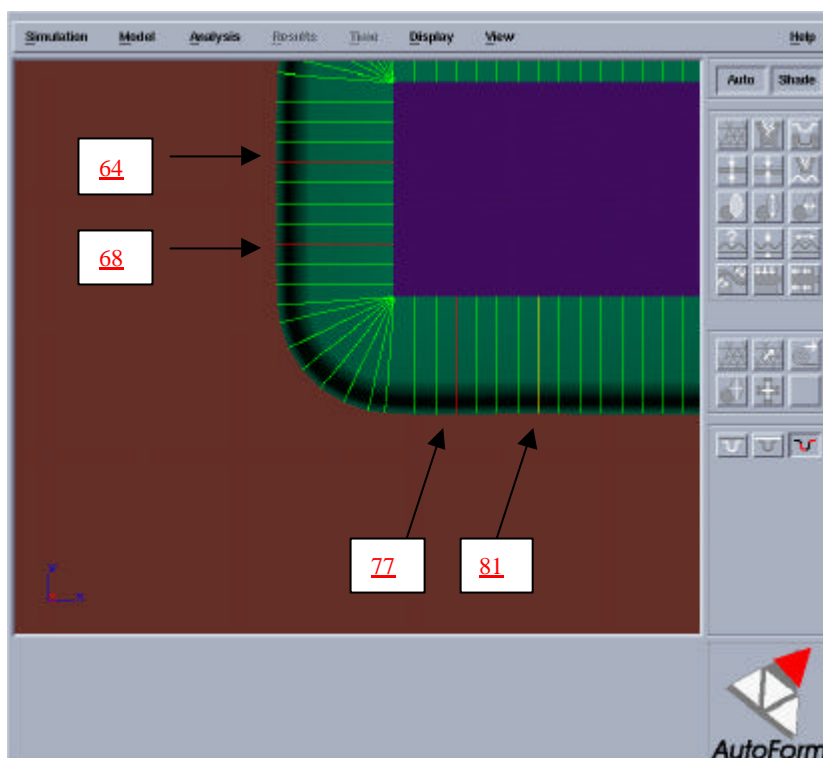|	
Profile 64 > Add	64 > Add
Profile 68 > Add	68 > Add
Profile 77 > Add	77 > Add
Profile 81 > Add	81 > Add

The model should now look similar to the following picture:

Fig. 2.1.4 Profiles selected for changing the die radius

Hint: Previously defined profiles can be copied. Select source profile with the right mouse button and move the profile with the mouse button pressed. Release the

mouse button at the destination profile (which is blue). The settings for the source profile have been copied to the destination profile.

On the *Addendum* page, activate profile 77. Click on **Advanced and** change the **die radius** to 10 mm. Activate profile 68 and again change the die radius to 10 mm and press **Apply.**

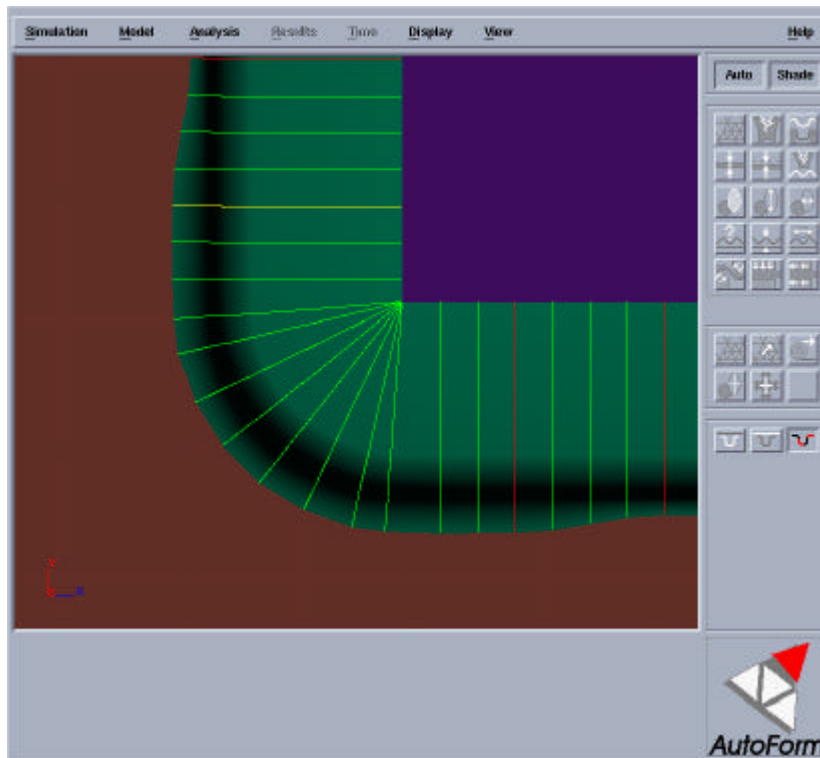These changes should lead to a result like in **Fig. 2.1.5.**

Fig. 2.1.5 Die radius after profiles are modified

Hint: The radii can now be changed with the mouse. Click on the circle line with the left mouse button and move the mouse with the mousebutton pressed. The dimension of the radius is shown on the upper left corner.

Generating a Drawbar:

As in the procedure to modify die radius, 4 profiles have to be selectedThe two outer profiles are left unchanged and the two inner ones are modified.   Profile 81 having been selected earlier, only three need to be identified now. The inner profiles are modified as follows:

Use right mouse button to click on   or use the **Add prf** button

Profile 85 > Add   85 > Add
Profile 97 > Add   97 > Add
Profile 101 > Add   101 > Add

Activate profile 97 (Use the corresponding tab on **Addendum** page, or, in *Main Display,* click on profile with the right mouse button). The row of 5 icons at the bottom of the Addendum page represent available "templates" for the profiles (e.g., Minimum PO, Step, etc.). Select the right-most Icon (Bar) from these templates. Next,

to change the profile bar height, left-click on the right hand segment of the horizontal black line and move it to a height of approximately 10 mm. You can also use the Bar dialog box in the *Heights* column to type in a value of **10.** Repeat this process for profile 85 and press **Apply**.

Changing the profile of the Drawbar

Activate profile 97, select the fourth Template Icon (Step). Repeat this process for profile 85 and press **Apply**.

Changing the profiles

Under the *Widths* column, click on the drop-down menu beside **Bar** and select the *Free* option. Similarly, under *Heights* click on the drop-down menu beside **CB** and select the *Free* option. There are now four different dashed lines that determine the position of the circles. These lines or circles may also be modified using the corresponding dialog boxes:

Widths:
Bar : Determines the width of the drawbar
Options in *Bar* drop-down menu:
Min: the drawbar will be of minimal width
Max: the drawbar width will be maximum (e.g., Step template)
Free: the drawbar width is user defined

PO :   Determines position of punch opening line (PO line)
Options in *PO* drop-down menu:
Min: Min. PO line dependent on selected radii and angles
Free: PO line user defined

Heights:
Bar : Determines height of draw bar
Options in Bar drop-down menu:
Part: Height is measured from part
Bndr: Height is measured from binder
Tngl: Height is measured from tangential elongation of part

CB :   Determines position of CB height line (Counter bar) with respect to Part boundary (**Caution**: In version 3.01 the height was measured with respect to bar height)
Options in CB drop-down menu:
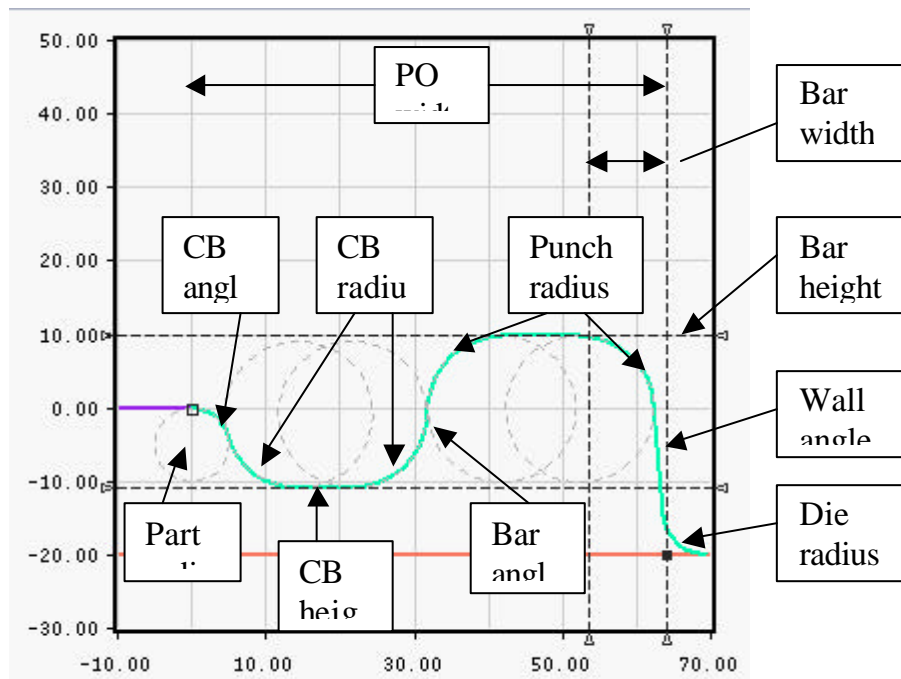Free = Counter bar user defined
Off = no Counter bar

Fig. 2.1.6 Various addendum parameters

You should try to create different profiles using these lines to develop a feeling for modifications that are permissible and that are not. In some cases the lines cannot be moved further because input for circle radii, wall angles and tangent face (plus) prohibit further movement. This constraining feature ensures that it is not possible to create extreme or infeasible addendum designs. You should also try to change these parameters and understand their effect on the profile.

CLARIFY: Tip: If no smooth surfaces result, it is in most cases because of the fact that here the radii and angle specifications are not realizable with the given punch opening line. It in most cases helps to modify the punch opening line e.g. to increase PO.

Changing the punch opening line (PO line)

Now, let us change the punch opening line using the **PO width**. This line is initially automatically generated along the edge of the running profiles.

Click on Lines > PO width > Edit ...

Right-click on the center of the upper side of the PO line thus creating a new point and drag this new point up. This changes the punch opening line. This method allows you to add, delete or move points until the line meets your requirements. Pressing **OK** accepts the changes, and the profiles are adjusted automatically to the new PO line which then results in an automatic update to the addendum surface (see Fig. 2.1.7)
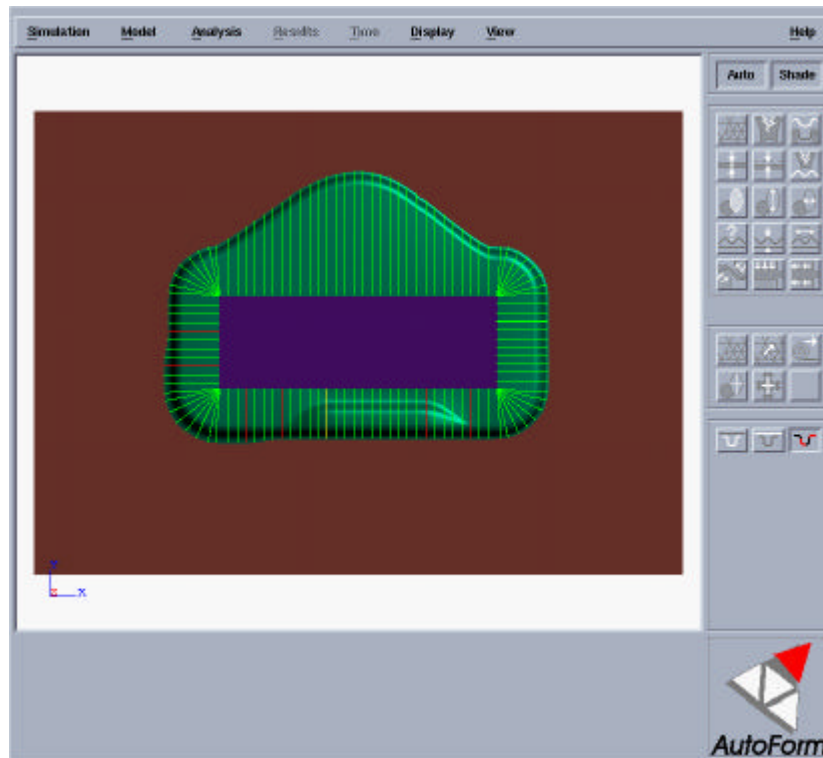
Fig. 2.1.7 Modified punch-opening line (PO width)

Besides these local changes to the punch opening line, global changes are also possible:

Click on Lines > PO width > Edit... > Global mod

Fig. 2.1.8 Global parameters for changing lines

Moving the **Expand** slider to the right uniformly increases the size of the PO line. Moving the **Smooth** slider to the right smoothes the line. Move the **Convex** slider completely to the right. The PO line gets a convex shape. Finally, press **OK**.
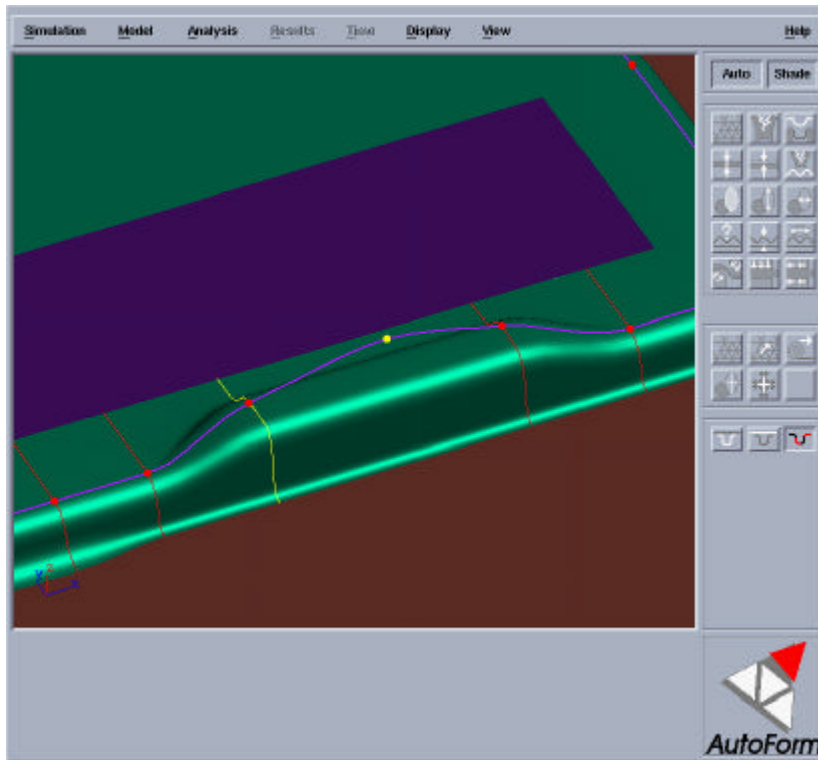
Manual editing of the bar height line:

Fig. 2.1.9 Editing the Bar height line

The **Bar height** line passes through the highest point of the drawbar.

Select **Options>Show Profiles** (shows only the modified profiles)

Select Lines > Bar height > Edit...

Click approximately at the center between points at either end of the drawbar (right mouse button), and drag the point (only movement in the z-direction is possible).

Move the point to generate a smooth profile for the drawbar. Then press **OK.** Addendum is now automatically adjusted to the new **Bar height** Line.

Editing the CB height line (only if **CB** option is **free**):

The **CB height** line describes the position of both left profile circles in Figure 2.1.6.

Select Lines > CB height > Edit ...

Rotate the model in the main display window to see a good view the CB height line. You can see that the profiles between part and drawbar are no longer even (flat) but vary in inclination due to the change of the **Bar height** line. Delete all nodes between the start and end profiles for the modified bar height region and the line becomes straight once again. To delete a point, select it with the right mouse button (turns yellow) and press the **Delete** key and click on **OK**. Once the intermediate points have been deleted, the addendum becomes even along the length of the drawbar.

Click on **Dismiss** in characteristic lines menu

Bar height Line and **CB height line** are used to smooth selected regions of the addendum in order to ensure uniform punch contact.

Changing the drawing depth:

Click on the *Binder* tab. Adjust the value of **Shift Binder** to 40 mm. Go back to the *Addendu*m tab and press **Apply**. You can notice that the distance between the binder and part has increased and the addendum surface connecting part to binder also correspondingly increases.

The default value for binder shift is 20 mm; this may be adjusted to the tool and the specific process.

This example demonstrates the underlying parametric link between the various functions available in Die-Designer V3.03.

Export of Addendum Surface:

In V3.03, the addendum surfaces can be exported to CAD systems in IGES or VDAFS format.

File>Export Addendum Surface
Note: This option is implemented as a test version in V3.03

Saving the simulation

In main menu Simulation > Save Writes the simulation file to the hard disk.

2.2 Lesson 2 – A simple tool

Simulation > New > File name: lesson2 > OK

In *Geometry generator*:

File > Import > af > OK > File: lesson2.af > OK

Prepare page: Symmetry > x-z plane 0 > OK   then Apply

Fill page: Outer bndry fill > 500   then Apply

Binder page:Auto > Uniform   then Apply

Addendum page: On Master Profile**, drag the vertical line to 50 mm (PO width
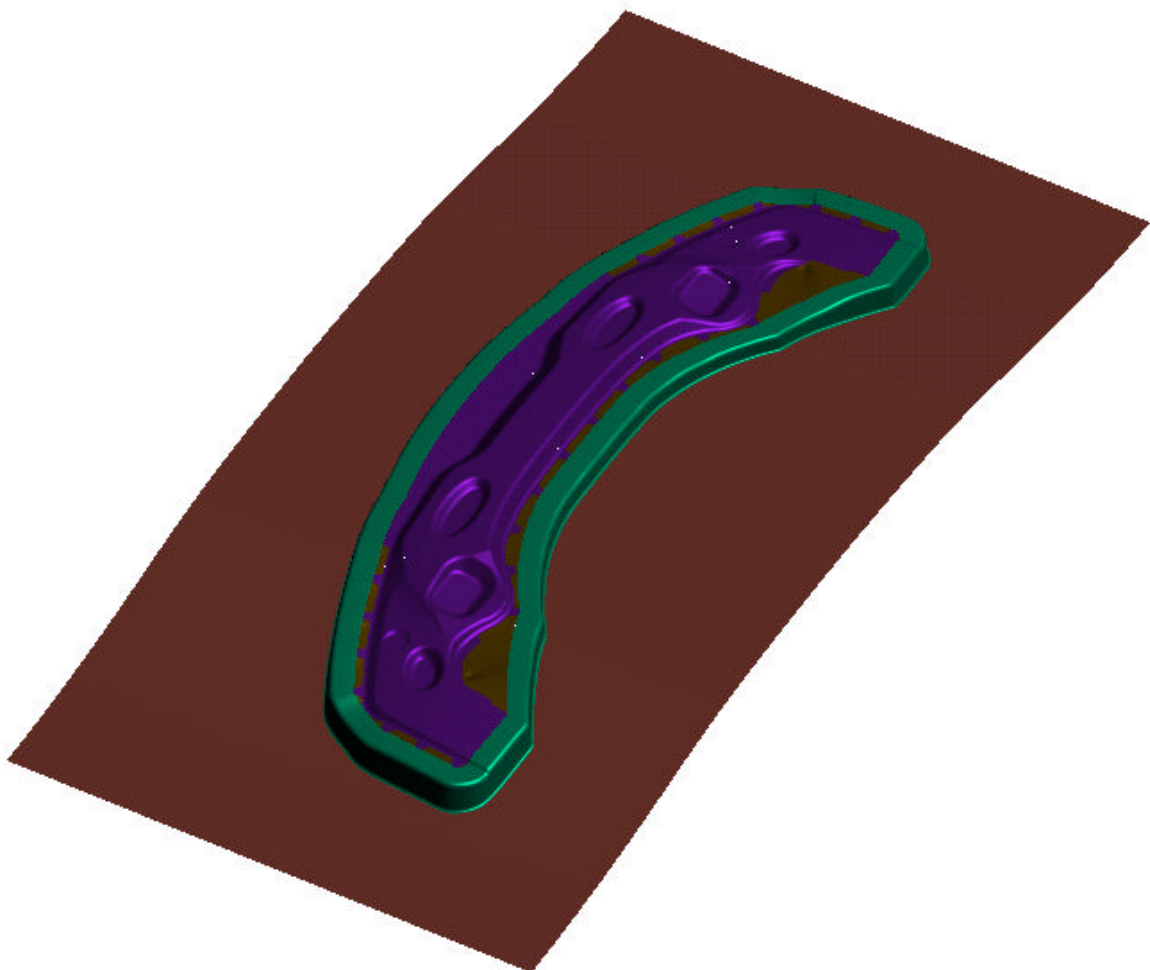50**)   **then** Apply

Fig. 2.2.1 Automatically generated tool

2.3 Lesson 3 – Significance of profile orientation

Simulation > New > lesson3 > OK

Geometry generator: File > Import > af > OK > lesson3.af > OK

Symmetry > x-z-plane y=0 > OK

Select the faces describing the license plate bay (shift + right mouse button)

Fig 2.3.1 Faces of the license plate bay are highlighted when selected

Hint: The delete register will not delete the faces finally, but they will not be used in the following steps

Press Delete. The selected faces disappear. Now click on Apply.
Select Fill Page: Hole
Right-click on the generated hole boundary. Press Apply.
Activate the Main Display **and press "c", to turn the colors on or off.**

On the Binder Page, select the following:

Auto > Minimum > Max bndry slope 10 > Angle diff 10
Click on the Adv tab and select Min flange width 250 > Apply
On the Addendum Page: Select PO width 140 > Apply

Fig 2.3.2 Profile directions are orientated perpendicular to the part and do not follow the feature-lines

It should be obvious that the orientations of the feature-line change directions because, by default, the created addendum profiles are perpendicular to the part edge. For this, as well as other parts, it is important to modify the profile orientation such that the addendum feature-lines are aligned properly with the part feature-lines. This is accomplished using the **Directions** ...button.

In V3.03, profiles that undergo a change in direction are automatically fixed (and shown in red). The intermediate profiles are oriented with respect to the fixed ones.
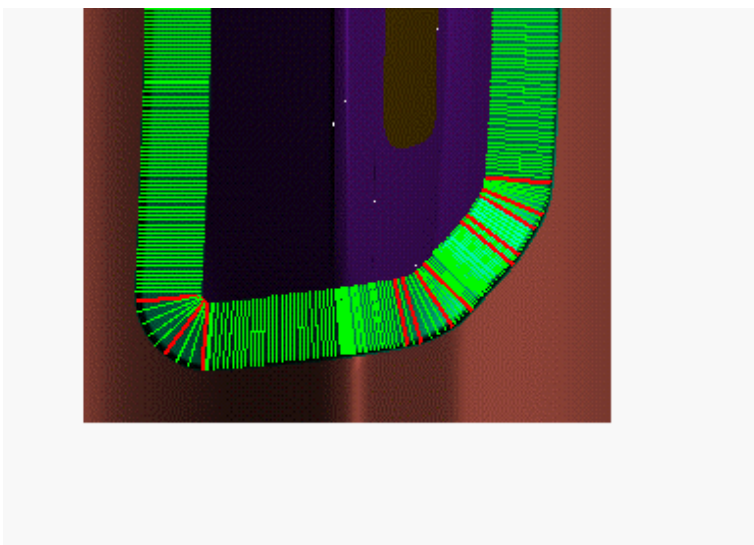
Fig 2.3.3 Automatically fixed profiles (shown in red)

The fixed and free profiles shown above can be selected with the right mouse button. For the current example, it is sufficient to rotate and re-align three profiles.
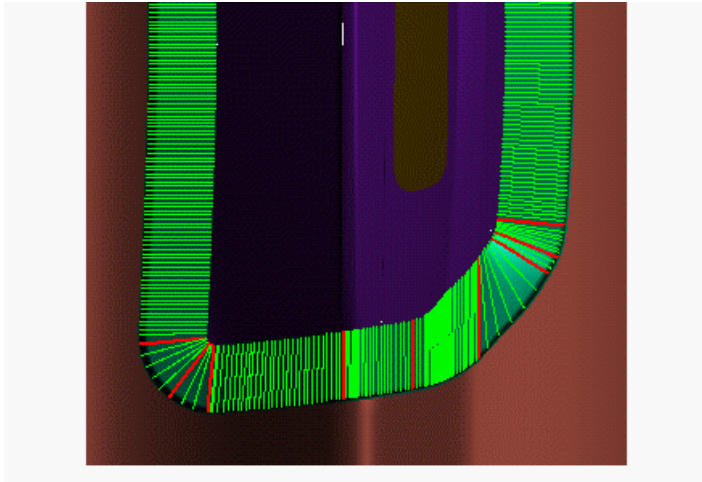
Fig. 2.3.4

Profile orientations are adjusted to part feature lines orientation

The directions of the three profiles in the middle are changed (from left to right in this example). The intermediate profiles will then be oriented such that they are either parallel or regular interpolated in case of a change of directions. It is important that profiles do not intersect one another. It is also important that they do not converge towards one another when moving away from the part because this can result in intersecting profiles when the PO line is expanded. In most cases this will result in absurd faces.

Press OK**. The addendum should look like *Fig. 2.3.5*.**
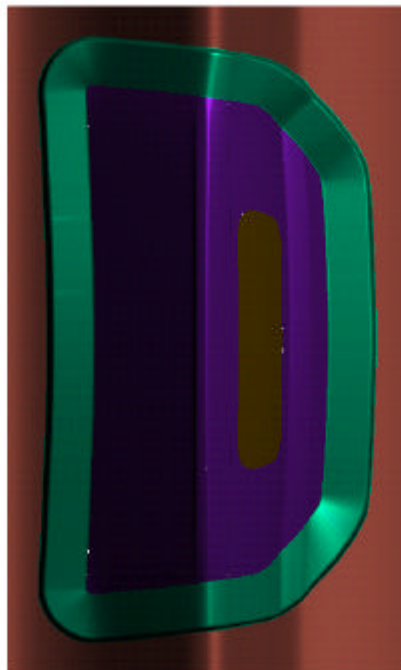
Fig 2.3.5

Addendum after modification of profile orientation

In the next step the punch opening line will be modified

Lines > PO width > Edit

Modify the punch opening line using the techniques described earlier according to Fig. 2.3.6.
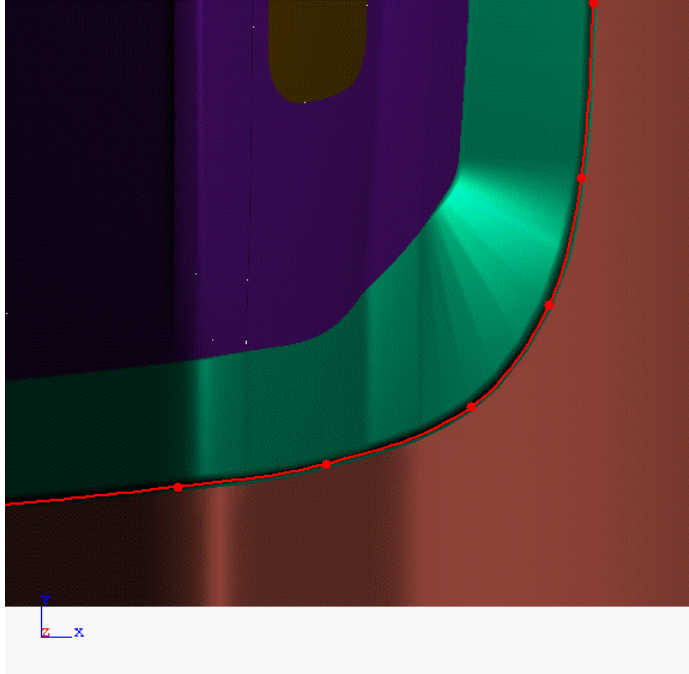

Fig. 2.3.6 Addendum after modified punch opening line

Now change the Master Profile**. Use the middle mouse button to click in the profile window.** This adjusts the profile window to full view (**Fit to Window**). **Choose the right template icon (Bar). Then click with the left mouse button on the right side of the dashed horizontal line and drag this up to approximately 20 mm. In the Heights column, change the** Bar **drop-down menu option to** Bndr.

Now, the height of the draw bar will be measured with respect to the binder surface). **Set the draw bar height to 80. The profile should look as shown in *Fig. 2.3.7.***
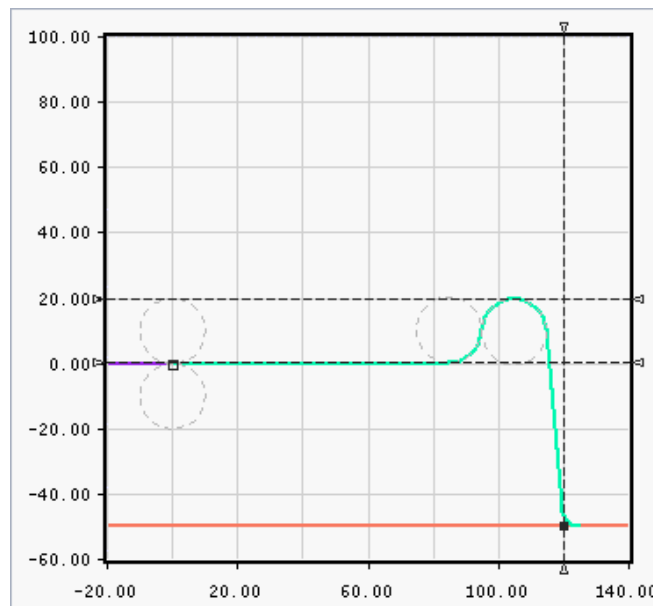

Fig. 2.3.7 Adjusted master profile

Click on Advanced and enter the following values:

Die radius	**8**
Wall Angle	**15**
Punch Radius	**35**
CB radius	**30**

Press the **Return** key and observe the effect on the profile.

Dismiss

The radii can now be changed with the mouse.

Left-click and drag on the circle line. The radius is shown on the upper left corner.

Drag the lower horizontal line downwards as far as possible.

The profile should now look like the following:
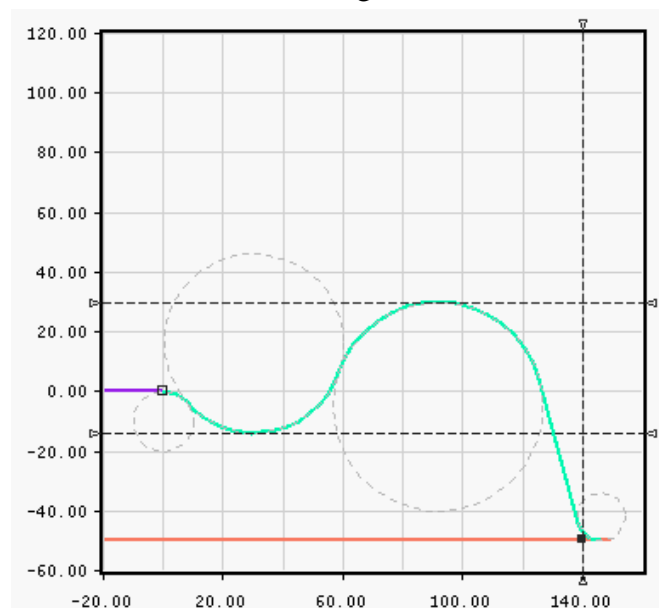
Fig. 2.3.8 New master profile

Press Apply**. The blended profile leads to the following addendum**
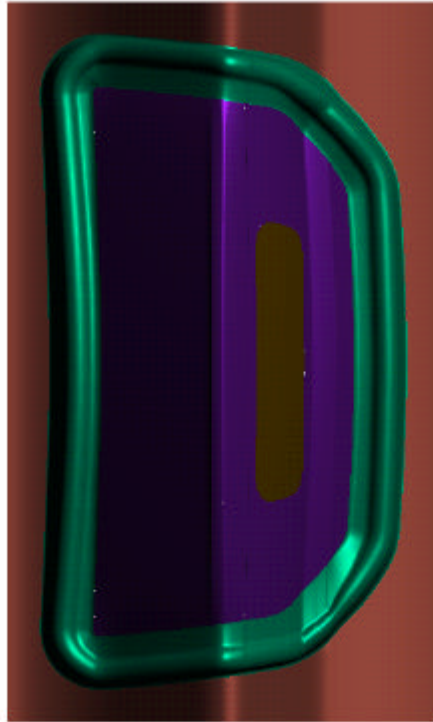
Fig. 2.3.9 Addendum generated by one blended profile

Editing the Plus parameter

Plus is defined as that part of the addendum that contains the laid out flanges. The **Plus** region should not be used to model beads within the addendum. This could lead to strange and unusable results when AutoForm interpolates between the profiles. The **Plus Parameters** are located in the *Advanced Menu* and have the following meaning:

Edge radius: Radius of the part, default value 3 mm
Edge angle: Angle with respect to z-direction, default value 5°
Flange radius: Tangential radius of run off at edge of part, default value 5 mm
Flange angle(incr): Angle with respect to tangent at edge of part, default value 0°
Min. Flange angle: Min. flange angle with respect to horizontal direction
Max. Flange angle: Max. flange angle with respect to horizontal direction

Min. Flange angle and **Max. Flange angle** are limits, in order not to exceed the max/min cutting angle.

There are two additional parameters

Plus width: Width of Plus region, default 0 mm
Plus height: Height of Plus region, default 0 mm

These values depend on one another to a certain extent in the calculation of minimum values. *Fig. 2.3.11* explains the meaning of these parameters.
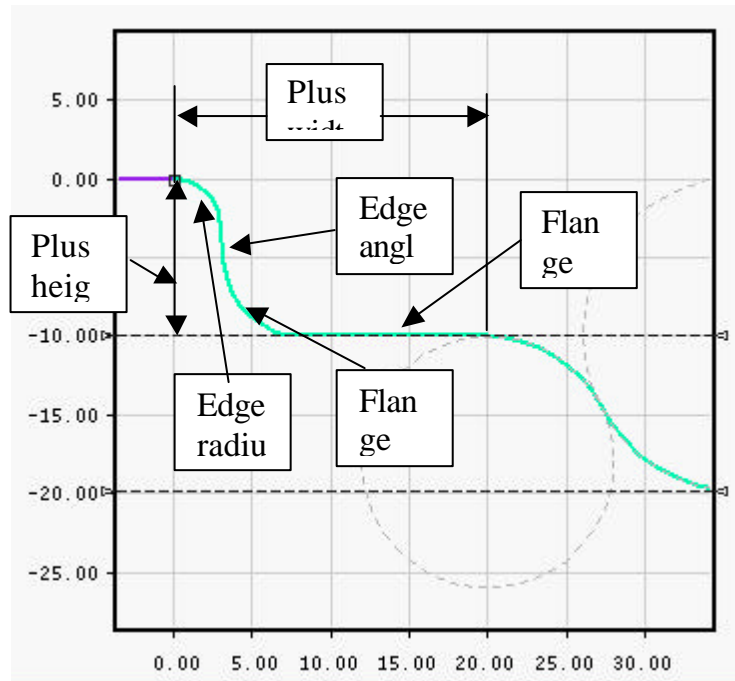
Fig. 2.3.11 Plus parameters

Make the following changes for this part:

Plus width=10 and Plus height=5 > Apply

The profile directly adjoining the part is changed, and there is a new edge within the addendum.

Adding and editing profiles

Select profiles at the areas of the addendum where you would like to create a new profile that is different from the master profile. Use:

Add prf   >   1 >   Add
Add prf   > 414 >   Add
Add prf   > 401 >   Add
Add prf   > 398 >   Add
Add prf   > 381 >   Add
Add prf   > 349 >   Add
Add prf   > 309 >   Add
Add prf   > 223 >   Add

Fig. 2.3.12 Profiles selected on the addendum

A second possibility to do the editing is to display all profiles **(Options All)** and to select profiles with the right mouse button.

A third option is to copy or move profiles: Click on a predefined profile with the right mouse button and drag it to another profile. You now have the possibility to choose between copying or moving the original profile. In case you have made a wrong selection use the **Cancel** button.

Dragging the profile outside the part area deletes it.

Editing profiles

Select the profile to be edited. You can graphically change (mouse click+drag) values for **Punch radius** and **CB radius**.

Profile 1:   as Master

Profile 414: as Master

Profile 401: CB radius 52   Punch radius 52

Profile 398: CB radius 55   Punch radius 60

Profile 381: CB radius 39   Punch radius 44

Profile 349:   Punch radius 40

Profile 309: as Master

Profile 223: drag CB radius down as much as possible (lower horizontal line)

Apply

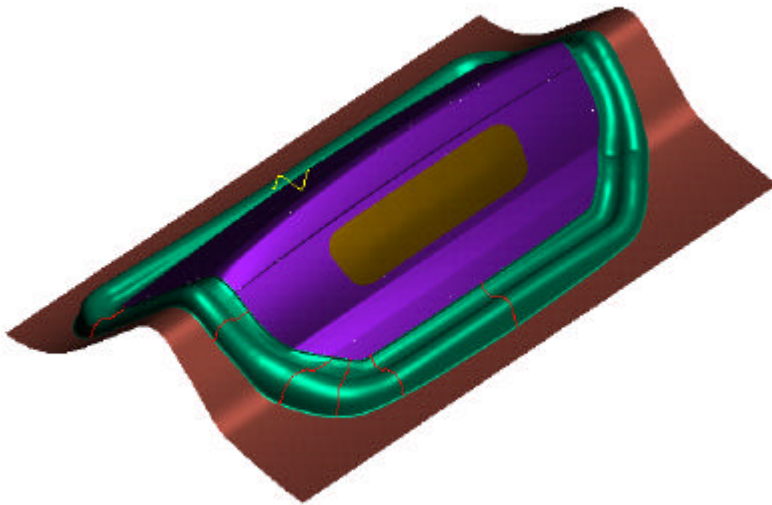The resulting addendum is shown in *Fig 2.3.12*

Fig. 2.3.12

New addendum generated using modified profiles

Smoothing the Drawbar:

The feature-lines of the part run through the entire addendum. To smooth them, the **Bar height** line is used
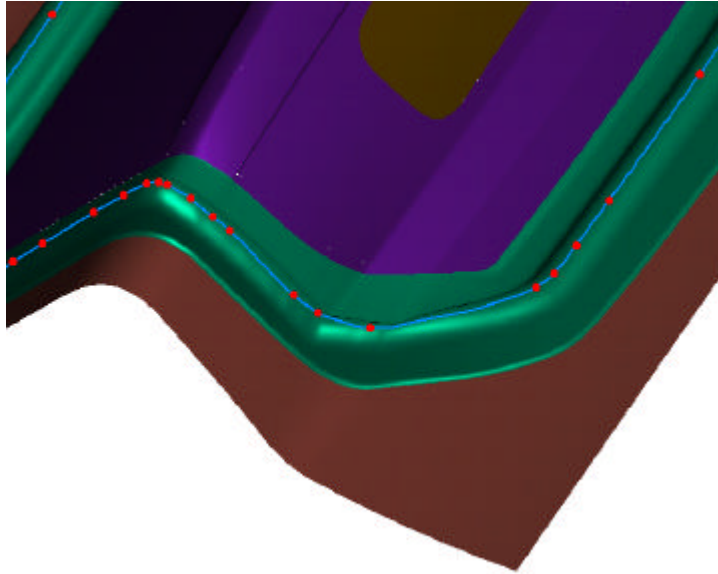
Lines > Bar height > Edit...

Fig. 2.3.9 New Bar height line

Edit the line by deleting and moving the nodes until you get a result similar to that shown in Fig. *2.3.9*. The nodes can only be moved in z-direction.

Deleting points:

Click points (yellow) and press Delete with active *Main Display*

Adding points:

Click line with right mouse button (not on existing point)

Moving points:

Right-click (point turns yellow) and drag points to desired location.

Simulation > Save

2.4   Lesson 4 – From DieDesigner to simulation

Simulation > New > lesson4 > OK

File > Import > af > OK > lesson4.af > OK

Prepare Page: Apply

Fill Page: Outer bndry fill > Smooth radius 100 > Apply

Binder Page: Auto > Uniform > Angle 90 > Apply
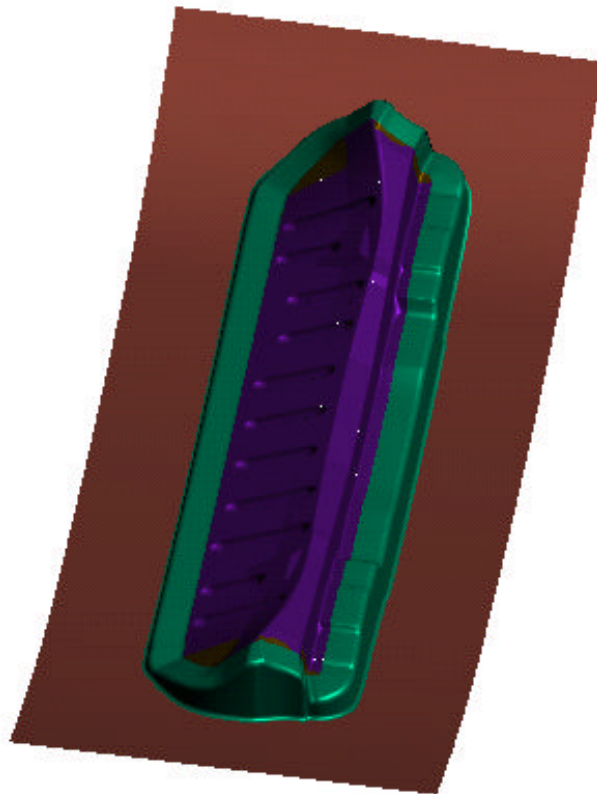
Addendum Page: PO width 50 > Apply

Figure 2.4.1 First model for Addendum

Changing the Directions (Profile re-orientation)

Change profile orientations using Directions (Addendum page) as shown in Fig. *2.4.2*.
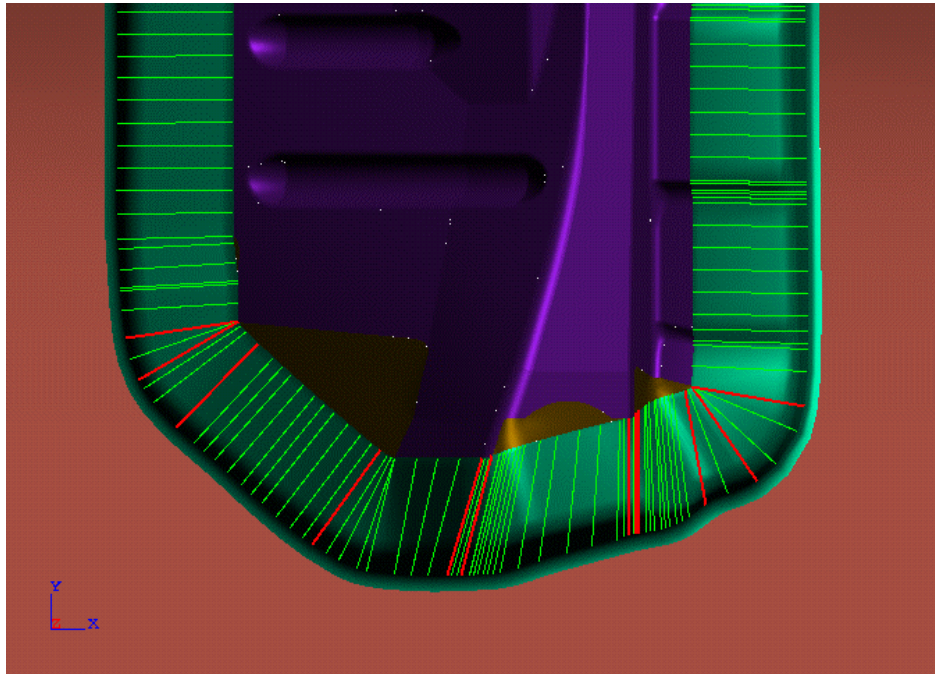
Fig. 2.4.2 Modified profile orientation

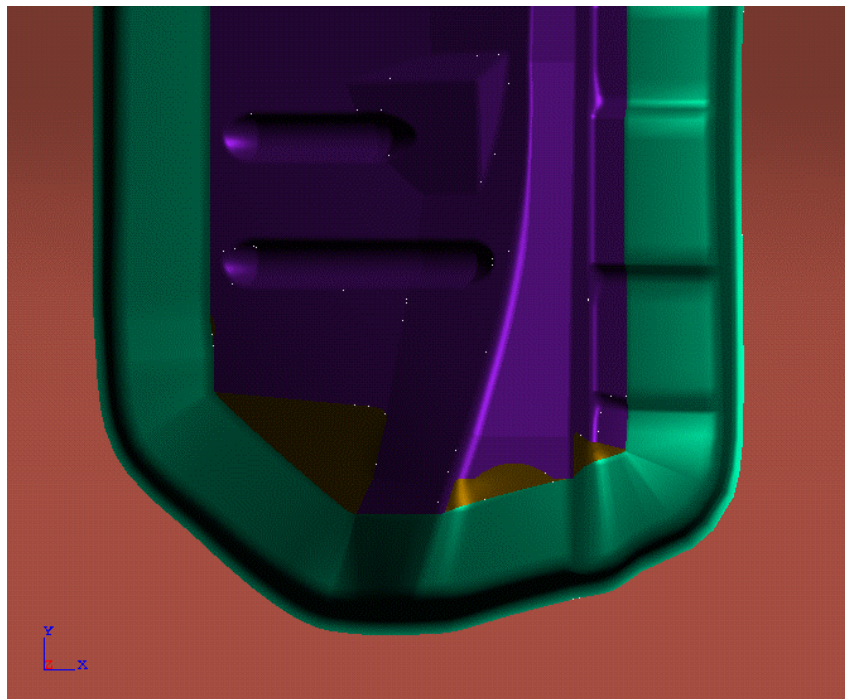The addendum should be similar to the one shown below (Fig. 2.4.3)

Fig. 2.4.3 Addendum after changing the profile orientation (Directions)
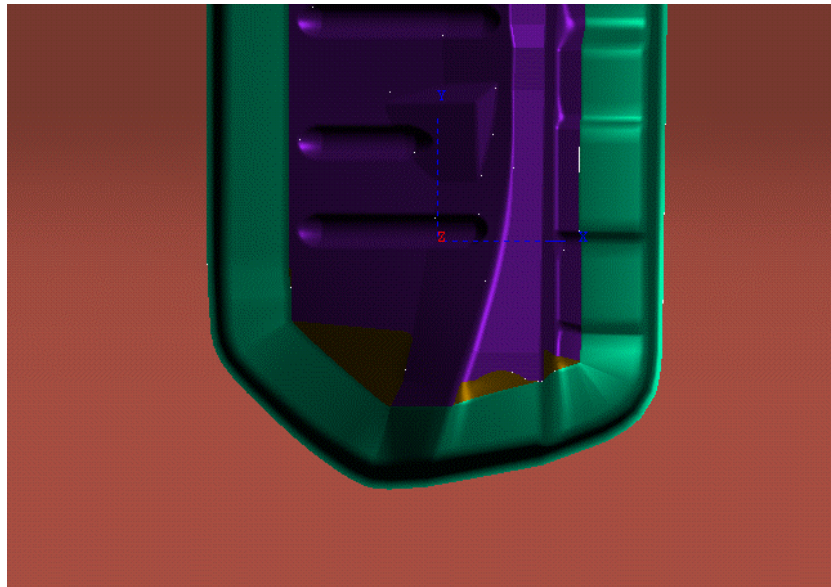
Changing the punch opening line

Fig. 2.4.4 Addendum after modifying punch opening line (PO width)

Lines > PO width > Edit > Global mode > Convex = 1 > OK

Reading a PO line from file

Model > Curve manager > File > Import > af > lesson4_pol.af > OK >Dismiss
Lines > PO width select
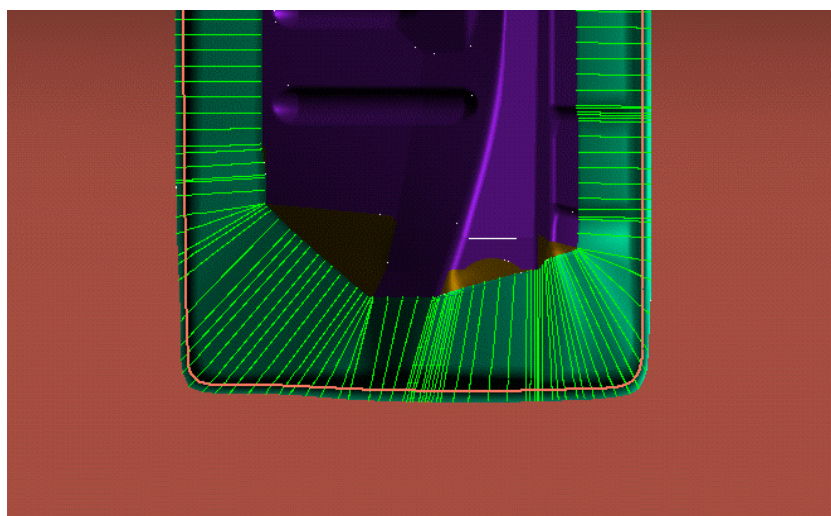**Select the line with the right mouse button or click on Line 1, then OK.**

Fig. 2.4.5 Addendum with imported PO line.

Model > Curve manager > line 1> clear > Dismiss

Smoothing the punch radius

Rotate the part around as shown in *Fig. 2.4.6*
Lines > Bar height Edit
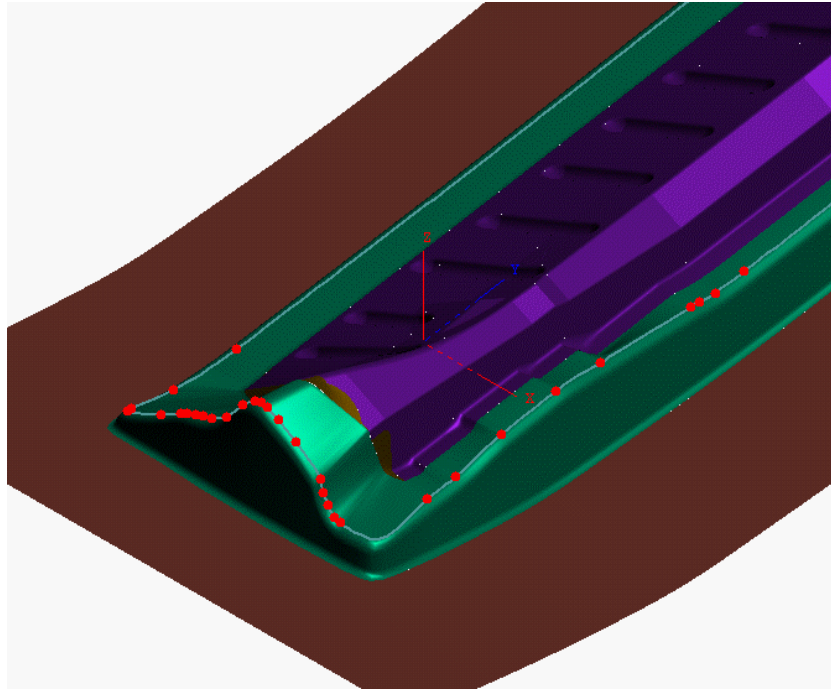Change the Bar **height** line according to *Fig. 2.4.6*.

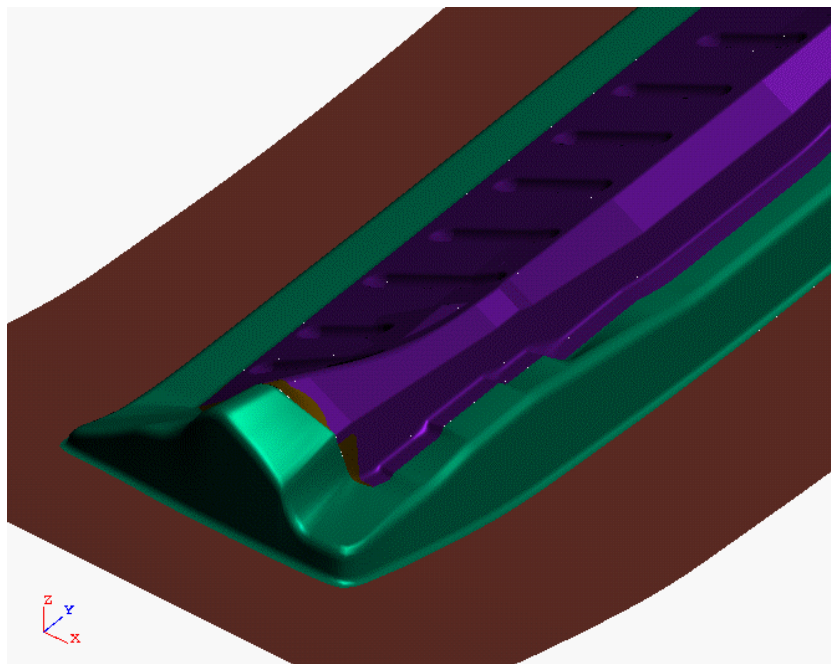**Fig. 2.4.6 Editing the Bar height line**

OK

Fig. 2.4.7 Addendum after smoothing the Bar height line

Dismiss in **Characteristic Lines**.

Reading a binder from file

Binder > Delete > Delete

Import > af > OK > lesson4_binder.af > OK
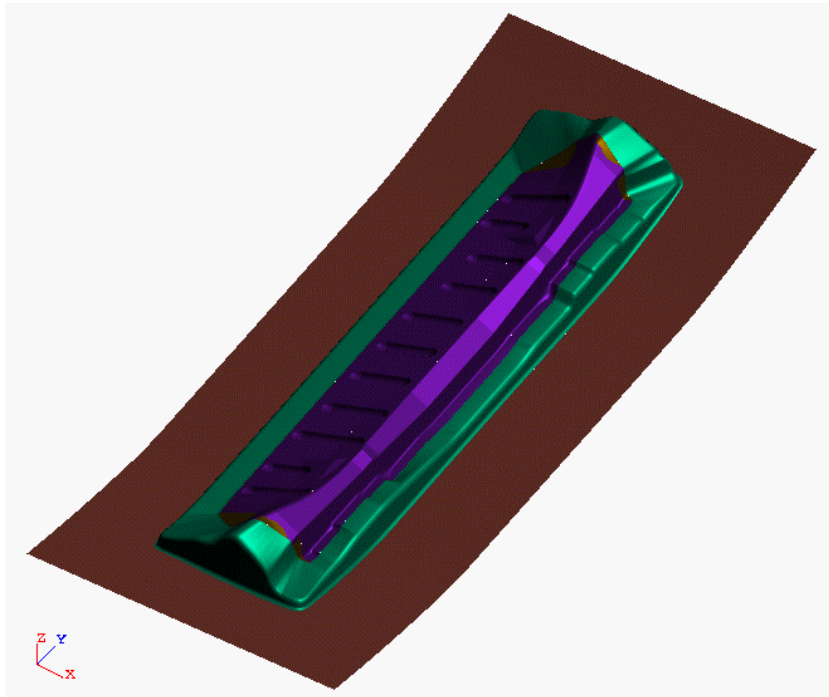
Addendum > Apply

Fig. 2.4.8

Addendum with imported binder surface

Starting a simulation:

Model > Input generator > OK

*Note: Stamping direction/tool setup can be modified only on **Prepare page**)*

Title: Defined by default in this version

Tools: Geometry is already defined. DieDesigner distinguishes between *Part Geometry*, *Fill Geometry*, *Addendum,* and *Binder*. The separation follows the following order:

          Die: Part + Fill + Addendum + Binder
          Punch: Part + Fill + Addendum
          Binder: Binder

Select Blank > Input and press Ctrl-Z to re-orient the view to the Z axis.

To define the blank outline use the **Ctrl** + **right** mouse button to sketch the line segments from point to point. Click on the first point of the curve in order to close the line.

Once a line has been closed, you cannot add additional points AFTER the last one. However, points can be added and moved on already defined lines.
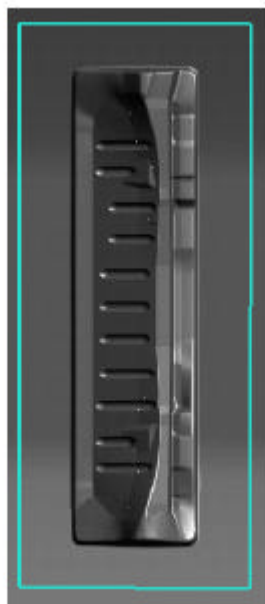
OK **when the blank matches** *Fig. 2.4.9*.

Fig. 2.4.Blank

Process > Gravity > Upwards > (Die) stationary

Job > Start

2.5   Lesson 5 – Manual Binder

The Initial steps here are similar to those in Lesson 3:

Simulation > New > lesson5 > OK
Geometry generator: File > Import > af > OK > lesson5.af > OK
Symmetry > x-z-plane 0 > OK

Select the faces describing the license plate bay (**shift** + **right mouse button**)

Press the Delete key. The selected faces disappear from the main display window
Click on Apply.

Fill Page>Hole: Select the boundary of the generated hole with the right mouse
button. Press Apply.

Activate the *Main Display* **and press** c**.**

Binder Page: Auto > Minimum > Max bndry slope 10 > Angle diff 10
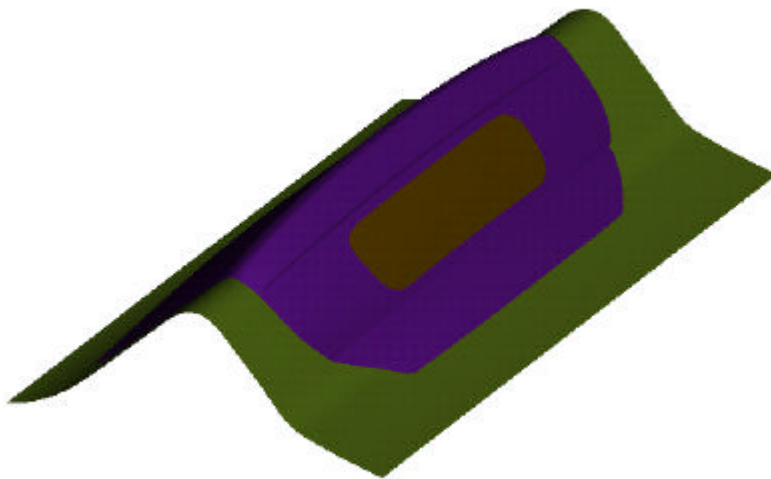**Adv >   Min flange width 250 > Apply**

Fig. 2.5.1

**Automatically generated
binder surface**

Upon selecting the *Binder* Page the drawing depth is shown as a color contour map
with a legend at the bottom of the main display window. This post variable may be
switched on or off in the menu **Display > Drawing depth.** The range may be edited in
Display>Ranges ... For this part, set:

Display > Ranges > Manual 0 80 > Return > Dismiss

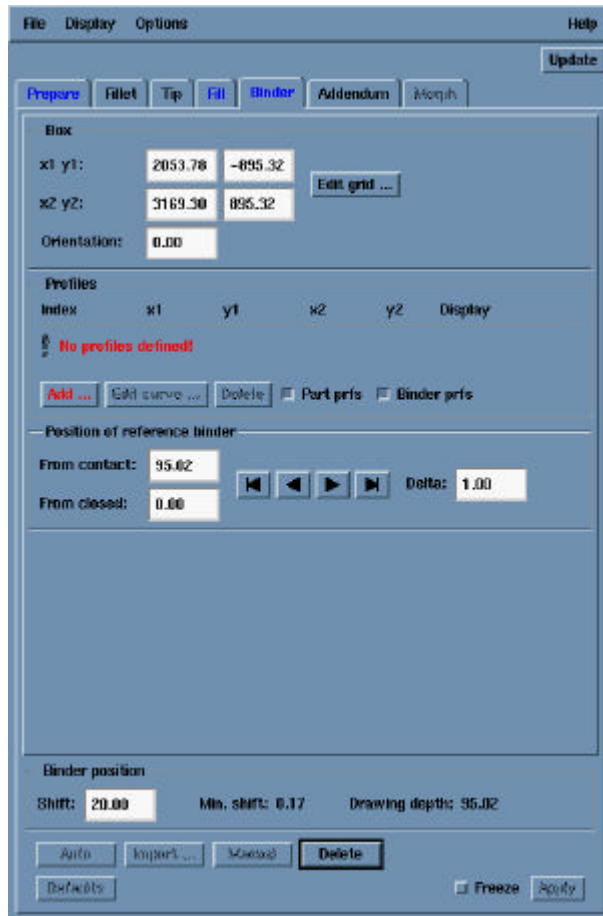The binder surface may now be edited manually. To do so, select **Manual** on *Binder page*:

Fig. 2.5.2

Input menu for manual editing of the binder surface

For manual editing of the binder surface you have to create lines (profiles) that can be modified using the curve editor. To create profiles, use the red **Add** button.

Add: Right-click in the middle of the topmost part boundary curve and release the mouse. Now, move the cursor towards the opposite (lowermost) part-boundary curve and press the right mouse button once again. Repeat this with procedure going from the left to right on the part boundary curves.

Hint**: Use Shift + right mouse button to obtain horizontal or vertical lines**

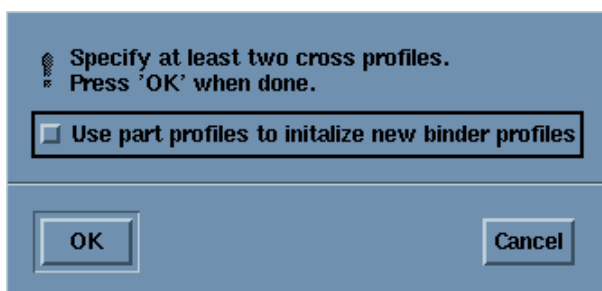To modify the binder surface you need at least two intersecting lines, Fig. 2.5.4.

Fig. 2.5.3 User-prompt when adding cross profiles

Note: Selecting "Use part profiles" will **take the part geometry as a template for the binder profile. This option** has to be selected **before adding new profiles.**

A manually edited binder is no longer parametric. After using the profiles you can no longer obtain the original binder, i. e. not all lines can be deleted. If you wish to do so, the entire manual binder needs to be deleted. After deletion, the original binder can be re-created using **Auto > Apply** or **Import**.
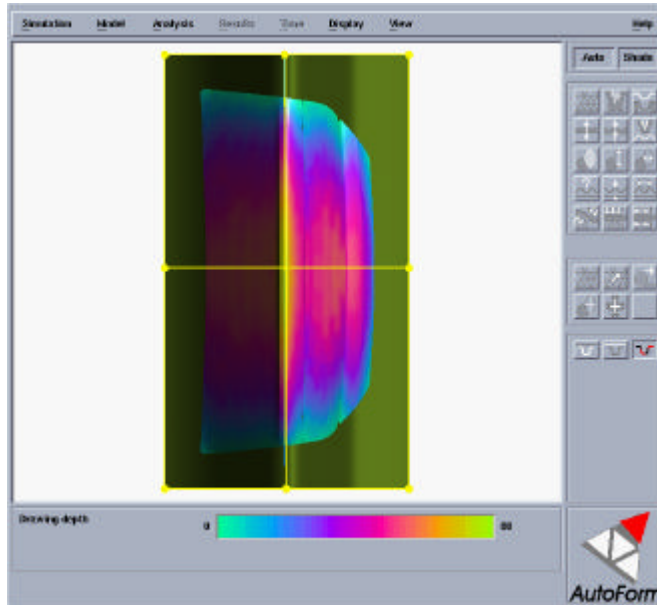

Fig. 2.5.4

At least two intersecting lines are required to modify the binder

OK > OK

Upon rotating the model out of the top view in *Main Display* you will see blue and yellow lines:

Blue lines: Profiles of the binder
Yellow lines: Profiles of the part or the selected profile of the binder.

The profiles of the binder can be selected by clicking the left mouse button on the *Binder* page or clicking the right mouse button in the *Main Display* (The profile will be highlighted with yellow color).

Modifying the binder:

Select one of the binder profile curves and press **Edit curve...**

Clicking on an existing point activates it. It is possible to drag the activated point by holding the right mouse button pressed. Clicking at a point on the line creates a new point, which can also be dragged with the right mouse button. Press **OK** if the line meets your needs.

The binder is immediately updated and you can see the new "Drawing depth" (that is, the distance between binder surface and the part) in color. Binder profiles at intersections are adjusted to the edited curve. This requires a mesh of curves to fix the binder edges.
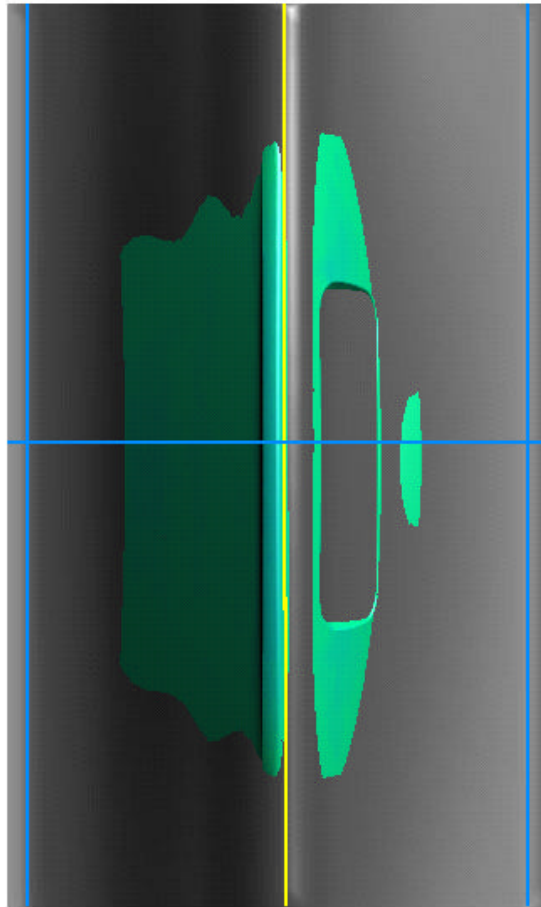

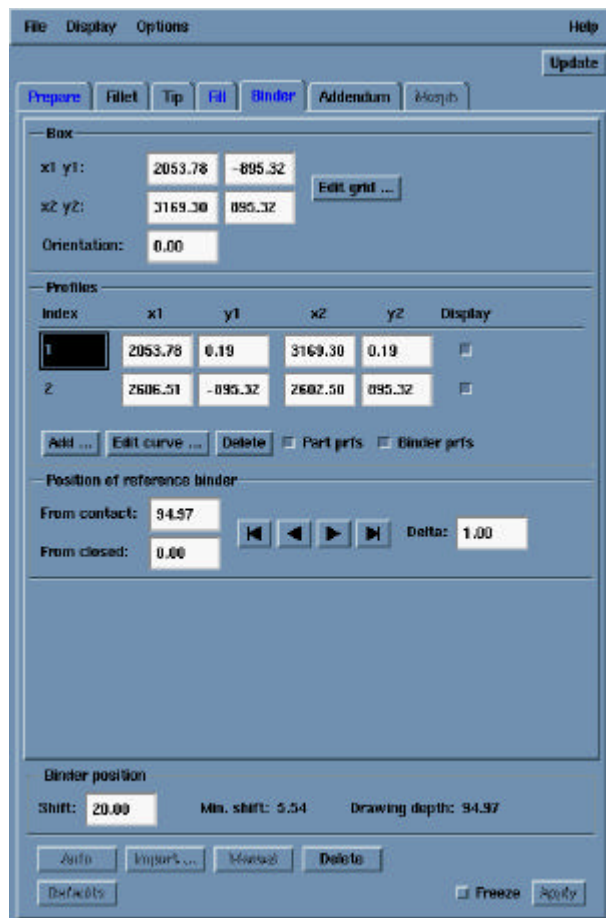**Fig. 2.5.5 Binder with additional profiles along the edge**

Fig. 2.5.5 Manual Binder Page

Other options on the *Manual Binder* Page:

Edit grid... here can you change the size of the binder.

Position of reference binder shows the theoretical maximal drawing depth. Using the arrow buttons, you can move the binder surface with respect to the part. This feature can be only used to check the contact of the punch. **From contact** and **From closed** shows the distance from contact and from contact position respectively.

Binder position shows the position of the binder for the generation of addendum (**Shift**).
Min.shift **shows the minimal value, for which the part geometry is positioned entirely beneath/above the binder.**

Drawing depth shows the drawing depth with respect to the first theoretical contact position.

2.6   Lesson 6 – Tip angle / Definition of drawing direction

The Tip page provides an easy way to modify the stamping direction of a part to a more suitable direction. This feature has been considerably enhanced in V3.03.

Simulation > New > lesson6 > OK

Geometry generator: File > Import > af > OK > lesson6.af > OK
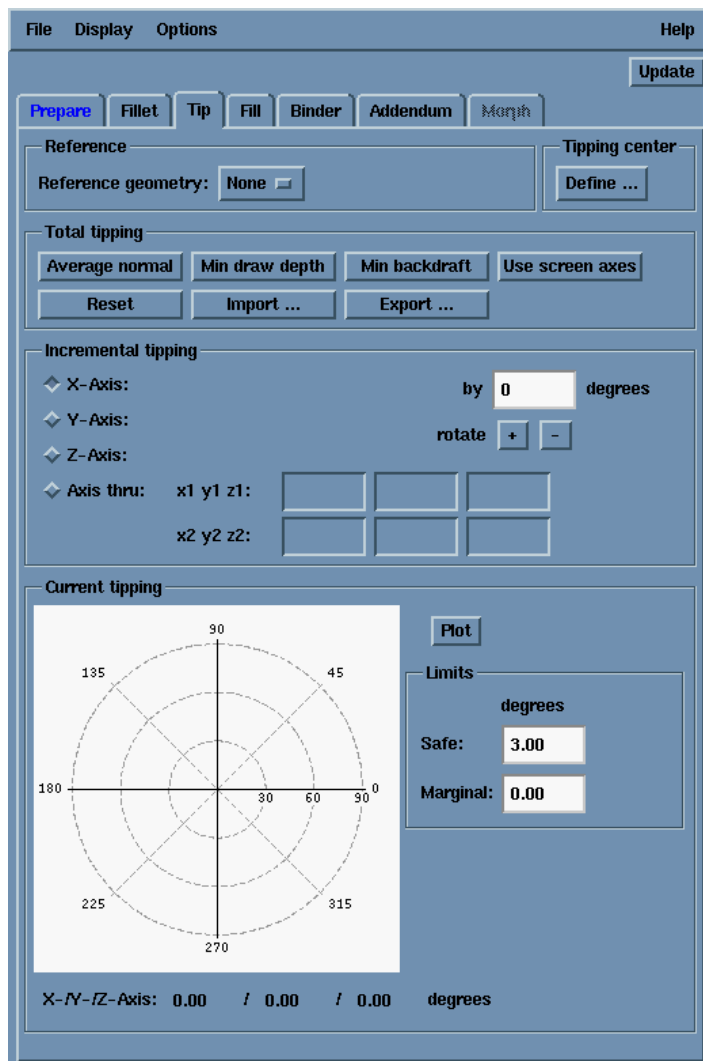
Symmetry > x-z-plane 0 > OK

Fig. 2.6.1 Tip Page

Reference geometry allows the import of a previous rotated geometry and maintains this rotation
Define... defines a local axis system
Average normal defines the normal (perpendicular) vector as stamping direction
Min draw depth chooses the drawing direction with a minimum drawing depth.
Min backdraft minimizes the undercuts
Use screen axes defines the direction of the perpendicular to the current orientation of part on the screen as stamping direction.
Reset discards all changes and uses the original CAD z-axis for stamping direction.
Import... allows the user to read in a rotation matrix from file
Export... enables you to write a rotation matrix to file in VDAFS-, IGES-, or AF-Format
Plot shows all possible stamping directions in one 2D diagram. The vertical axis represents the possible rotation around the y-axis and the horizontal axis represents the rotation around x-axis.

All potential backdrafts may now be calculated for each drawing direction and displayed in color.

Display > Backdrafts in the geometry generator. (Default values for the Tip page).

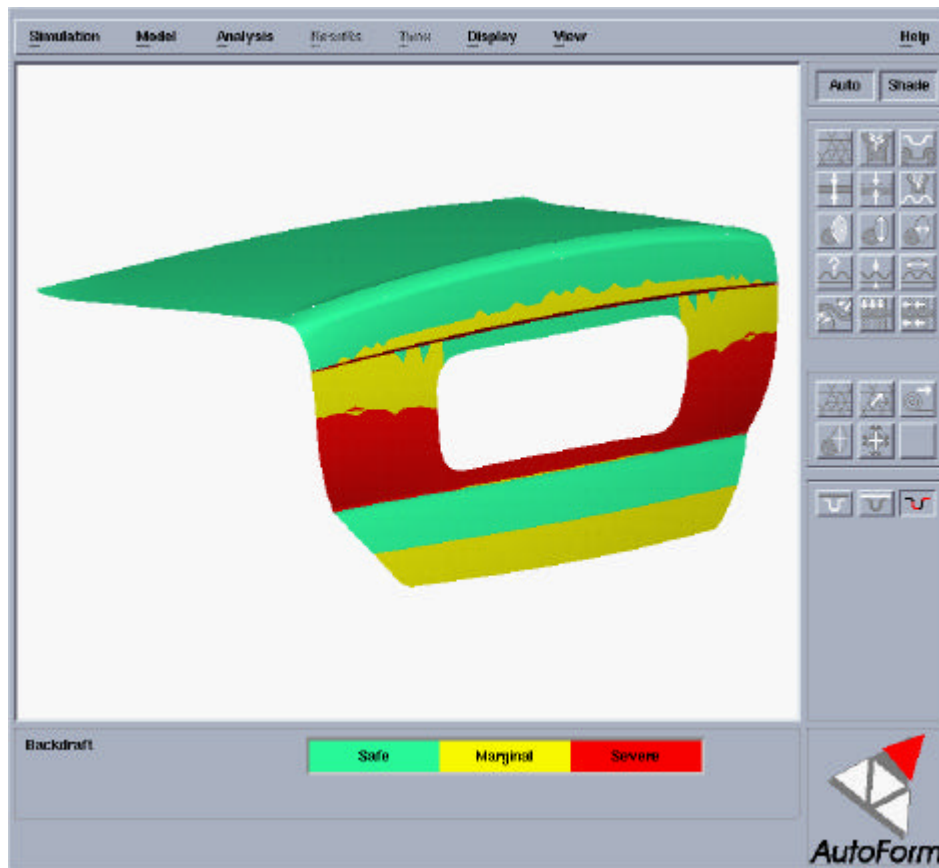Fig. 2.6.2 shows the result for the imported geometry.

Fig. 2.6.2

Representation of backdrafts

Safe: > 3°

Marginal: 0°<3°

Severe: < 0°

The delimiting angles severe/marginal (default 0º) and marginal/safe (default 3º) can be user defined.

In order to determine a suitable stamping direction it is recommended to start with the option **Average normal** and then look at other alternatives using the options **Min draw depth** and **Min backdrafts**. Then activate the 2D-diagram with **Plot.** In case none of the options lead to an appropriate stamping direction you can manually rotate the part until all backdrafts (red and/or yellow regions) disappear.

Average normal	Rotation and update of backdraft faces are carried out immediately.
Min draw depth	stamping direction having minimal drawing depth
Min backdrafts	stamping direction having a minimized backdraft
Plot	2D-diagram to illustrate the possible stamping directions free of backdrafts
Y-Axis > by 1 degrees > -	stamping direction of the hood without backdrafts
X/Y/Z-Axis	Definition of the rotation angle around each axis. It is necessary to follow the exact order of the rotation (i.e., rotation around x-axis, then y-axis then z-axis)

2.7 Lesson 7 – Automatic and variable filleting

The present version allows filleting of all sharp edges using a global radius. In addition, the user can select edges to be filleted with a constant or variable radius.

Simulation > New > lesson7 > OK
File > Import > afb > OK > lesson7.afb > O
Fillet Tab : Check: Checks geometry for sharp edges
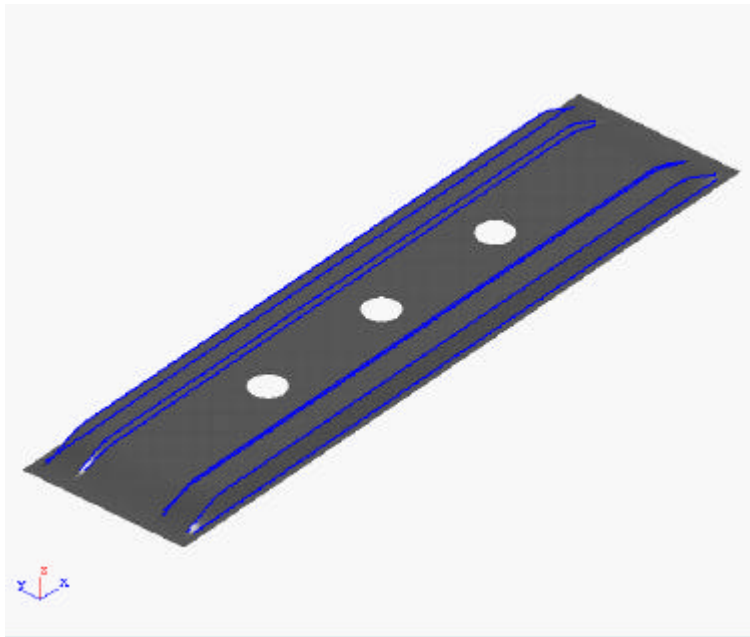**OK: Acknowledges sharp edges**

Fig 2.7.1 Sharp edges

      Global Radius=**3**   Specifies a global fillet radius of 3 mm.
        Apply   Runs filleting to fillet all edges with a 3 mm radius.

In case it is desired to fillet individual edges with radii different from the global setting, it is necessary to specify both the edges and radii values to be used. Use the **Add line...** button at the bottom of the page to achieve this.

Add line ...   Opens a window with the pop-up box   *"Mark radius control edge*
                                                            Finish with double click"

You may now click on the desired edge using the right mouse button. Release the button and move the mouse along the edge. The edge is highlighted in yellow. Double click with the right mouse button to finish. You may now enter values for different radii.

Constant   Constant radius along the edge

Variable   Click points along the edge for which a specific radius is desired (right mouse button), then press **OK**. Radii are entered in a table.

Note: When working with long curved edges, the direction of the curved edge may not always be detected correctly. If this happens, it usually helps to click on the edge with the right mouse button.
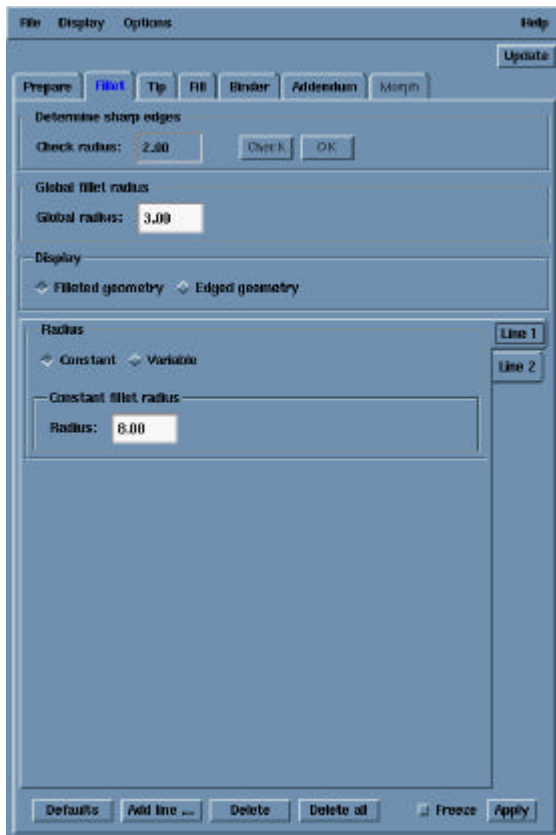

Fig 2.7.2 Fillet page

Example definition for an edge to be filleted with a constant radius.

Filleted geometry shows the filleted part.

Edged geometry shows the original, unfilleted part with the sharp edges found.

The above display buttons are not active until after **Apply** is pushed for the first time.

Fig 2.7.3 Definition of an edge with variable radii

Variable radii are defined by means of a table. The entries in the table represent points along the edge, which have to be defined first. Points can be added to the edge after pushing the **Add point ...** button to the right of the table. Clicking the edge with the right mouse button sets the points. Once all desired points have been set, push the **OK** button and enter the radius for every point into the table.

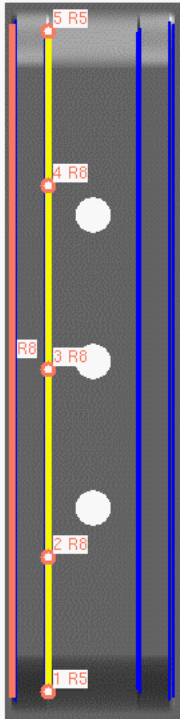Two edges of the following part receive radii other than the global filleting radius.

Fig 2.7.4   Edges with constant and variable radii

The left edge receives a constant radius of 8 mm and the right edge (yellow) a variable radius starting at 5 mm that rises to 8 mm and then drops again down to 5 mm.

Chapter 3 AutoForm-DieDesigner – Input Pages

3.1   Prepare Page

Fig. 3.1 Prepare page

The Prepare page is used to work with the imported CAD geometry.

Tool setup defines the geometric position of tools. The left button means die is above, punch and binder are below and the right button means die is below and punch and binder are above.

Buttons **Part, Binder, Flange** and **Delete** refer to specific faces of part and tool, respectively. **Part** denotes part geometry to be formed in first draw. **Binder** refers to the binder surface, and **Flange** allows working with the flange area. **Delete** contains all deleted or unnecessary faces.

The definition is done by selecting the patches (clicking with right mouse button) and pressing the register button (left mouse button).

Error tolerance determines the chordal error for generation of part boundary and **Concatenation distance** is the distance in which points can be merged.

The Part Boundary can be edited now. The points can only be moved on the part patches in order to cut details of the part boundary. The name of the function is therefor **Trim**. To activate this function the button **Apply** has to be used first in order to get the part boundary.

Symmetry allows defining a symmetry line.
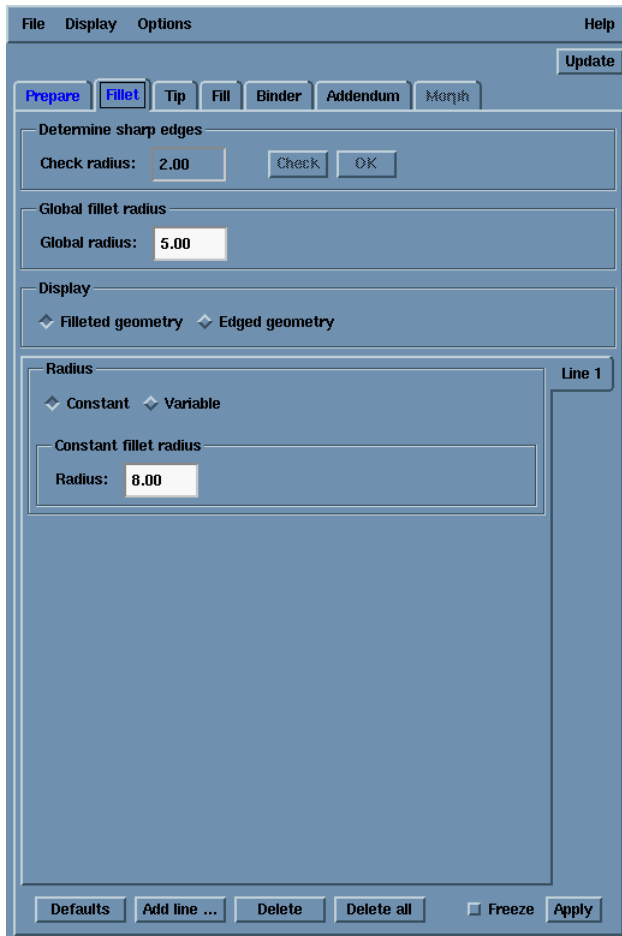
3.2   Fillet Page

Fig. 3.2.1 Fillet Page

Check checks geometry for radii smaller than **Check radius**. **OK** accepts the result.

Filleted geometry shows the part with fillets. **Edged geometry** shows part without fillets but with detected sharp edges.
These buttons are active after **Apply**.

Constant or **Variable** is the option for specifying radii for selected lines. The line has to be defined by **Add line ...** before. (After pressing **Add line...** the starting point of the line can be selected with right mouse button in Main Display, release mouse button and follow the line which should be selected with the mouse. Finish with double click of right mouse button.

The specification of variable radii is done in a table. At first all points for which a radius should be specified have to be defined on the line. This is done with the **Add point ...** button next to the table. The definition of the points is done by clicking on the line with the right mouse button. When all points are defined, press **OK** and specify the radii in the table.

Delete deletes active line.

Delete all deletes all lines.

Tip: At a "T-junction", it is possible that the directions of the line are not recognised. Normally it helps to click once on the line with the right mouse button. Of course the clicking must be done when the selection of the line is still the right direction.
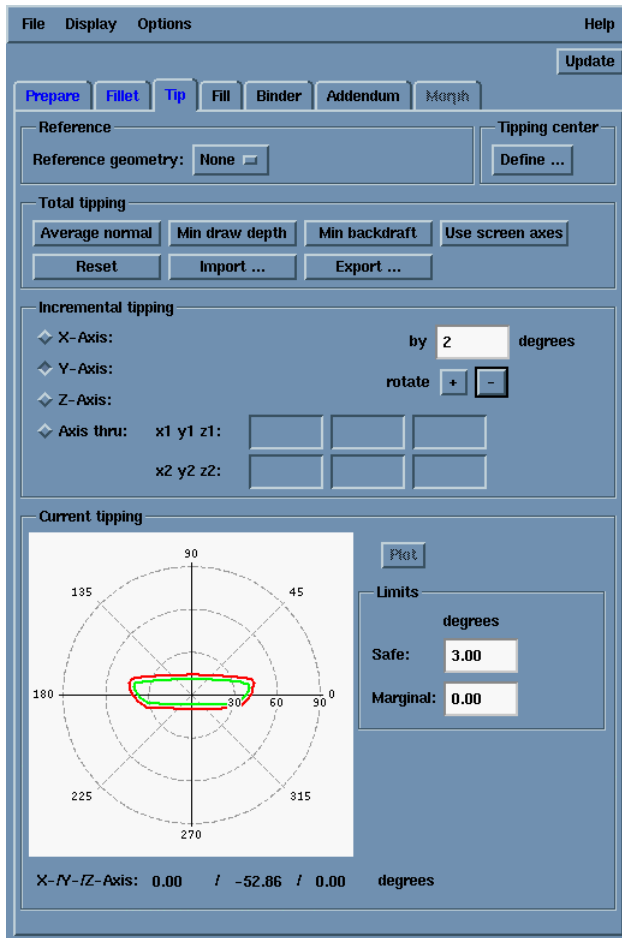
3.3 Tip Page

Fig. 3.3.1 Tip Page

Reference geometry allows the import of a previous rotated geometry and maintains this rotation

Define... defines a local axis system

Average normal defines the normal (perpendicular) vector as stamping direction

Min draw depth chooses the drawing direction with a minimum drawing depth.

Min backdraft minimizes the undercuts

Use screen axes defines the direction of the perpendicular to the current orientation of part on the screen as stamping direction.

Reset discards all changes and uses the original CAD z-axis for stamping

Import... allows to read in a rotation matrix from file

Export... enables you to write a rotation matrix to file in VDAFS-, IGES-, or AF-Format

Plot shows all possible stamping directions in one 2D diagram. The vertical axis represents the possible rotation around the y-axis and the horizontal axis represents the rotation around x-axis.

Safe and **Marginal** allow the specification of angles for the transition Severe/Marginal and Marginal/Safe.

X-/Y-/Z-Axis specifies the rotation around the specific axis. The sequence rotation around x then y then z must be kept.
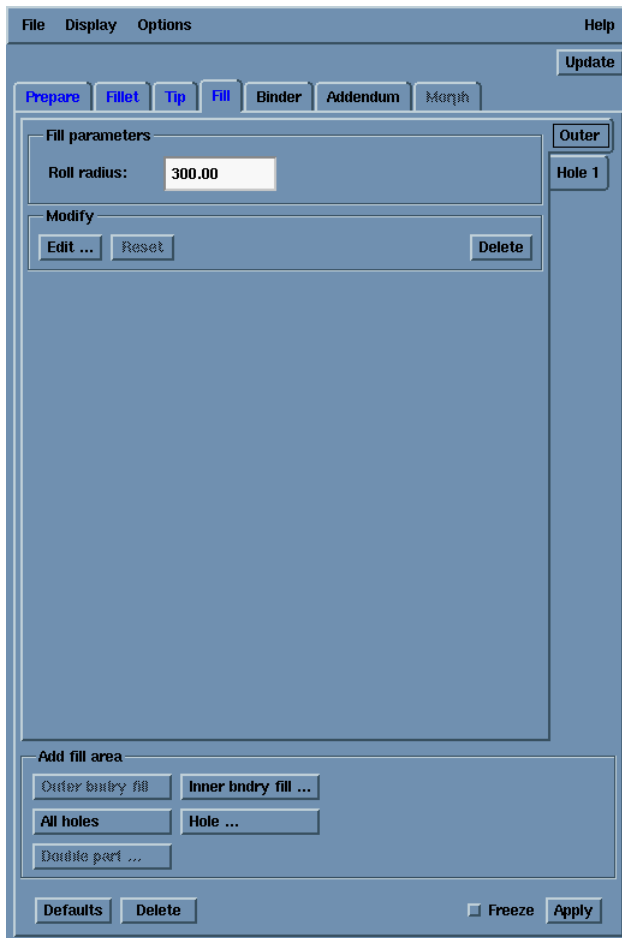
3.4   Fill Page

Fig. 3.4.1 Fill Page

Roll Radius is the radius of the cylinder, which rolls around the part boundary.

Edit edits the new boundary. To activate this button **Apply** has to be pressed.

Outer bndry fill generates a new smoothed outer boundary.

Inner bndry fill generates a new smoothed inner boundary.

All holes fills all holes in part geometry.

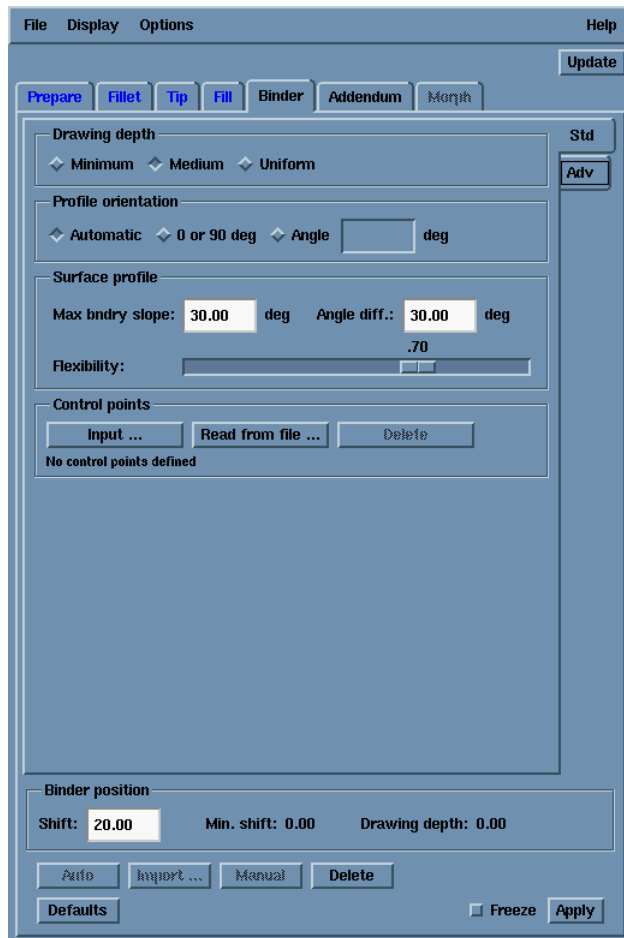**Hole ...** fills one selected hole in part boundary.

3.5 Binder Page

Fig. 3.5.1 Automatic Binder – Standard

These options are specifications for the optimization algorithm, which calculates the binder surface. Sometimes the algorithm can not fulfil all the specifications made by the user.

Drawing depth specifies the desired drawing depth (i.e., min, max or uniform).

Profile Orientation specifies the direction of the main curvature of the binder surface.

Max bndry slope specifies the slope at the boundary of the binder surface.

Angle diff specifies the max. difference between the slope angles of right and left boundary of the binder surface.

Flexibility specifies the number of splines used for the binder profiles (min: 3 and max: 20)

Control points allows the user to specify additional control points, in order to change the shape of the binder surface.

Binder position shows the displacement (**Shift**) of the binder surface for the generation of the addendum.

Min.shift shows the minimum shift for which the binder surface is totally below/above part geometry.

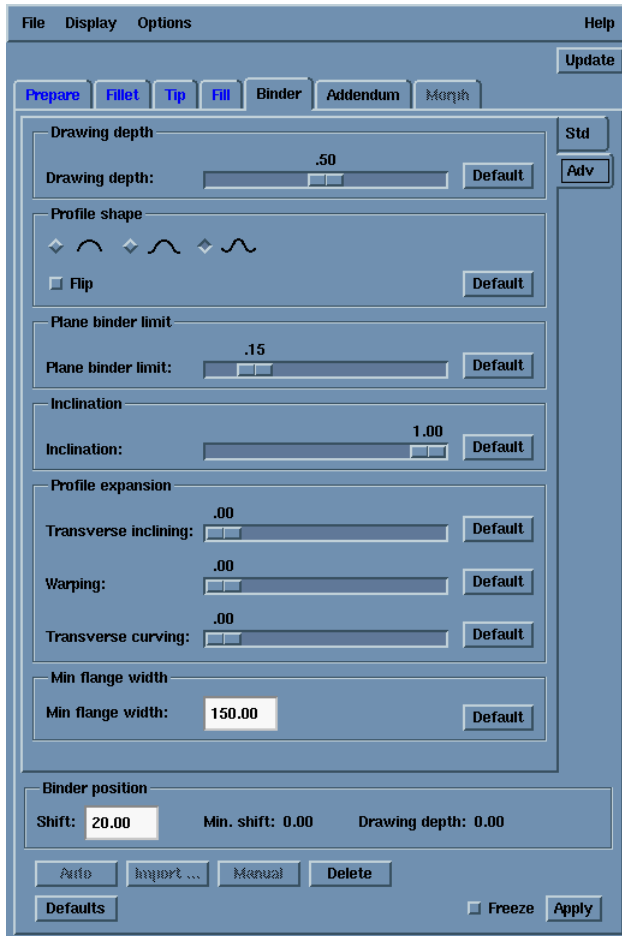**Drawing depth** shows the drawing depth with respect to the initial punch contact.

Fig. 3.5.2 Automatic Binder – Advanced Tab

Profile expansion specifies the slope (Transverse inclining), warping (Warping) and curvature (Transverse curving) for the second direction of the binder surface.

Min flange width specifies the size of the binder surface. The value is the size of the binder outside of part boundary.
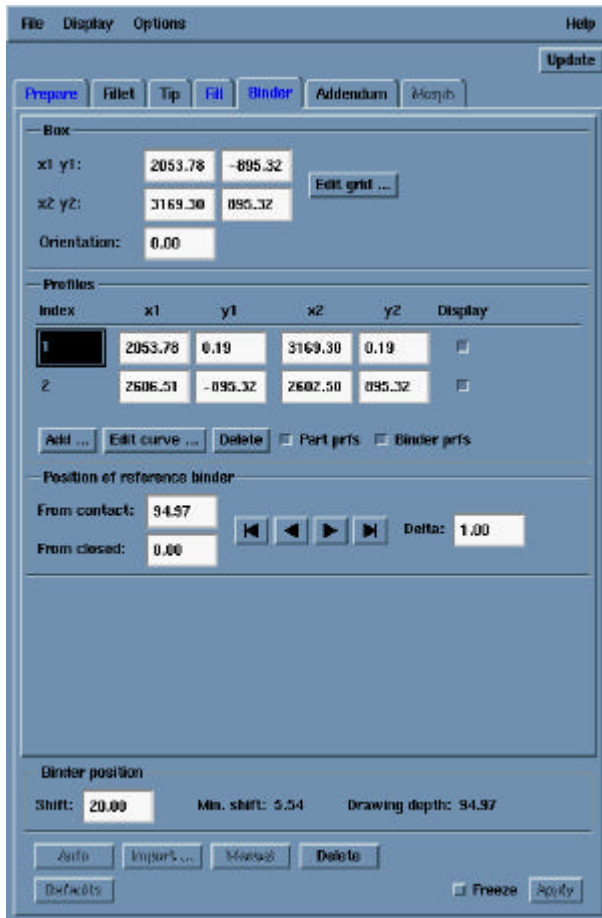
Fig. 3.5.5 Manual Binder Page

Edit grid... here can you change the size of the binder.

Position of reference binder shows the theoretical maximal drawing depth. Using the arrow buttons, you can move the binder surface with respect to the part. This feature can be only used to check the contact of the punch. **From contact** and **From closed** shows the distance from contact and from contact position respectively.

Binder position shows the position of the binder for the generation of addendum (**Shift**).

Min.shift shows the minimal value, for which the part geometry is positioned entirely beneath/above the binder.

Drawing depth shows the drawing depth with respect to the first theoretical contact position.

3.6   Addendum Page
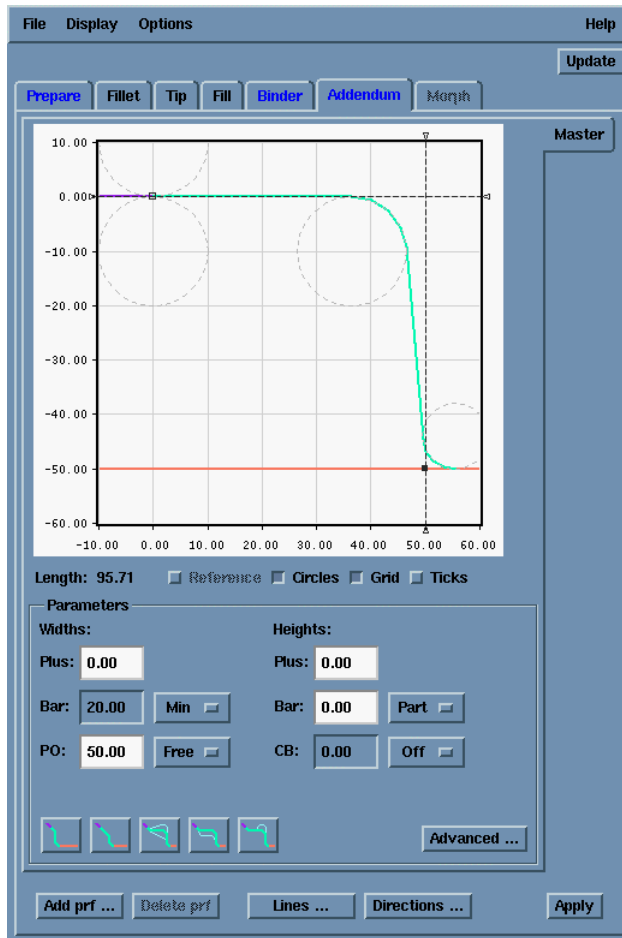
Fig. 3.6.1 Addendum page with master profile

The master profile is used as reference profile. Here, global settings for parameters Width and Height, radii and angles can be done which are used as default by all other profiles. The displayed profile is not the real geometry at every position of the addendum because the distance between part and binder varies. The displayed master profile is a schematic drawing with respect of the settings for width, height, radii and angles.

The radii can now be changed with the mouse. Click on the circle line with the left mouse button and move the mouse with the mousebutton pressed. The dimension of the radius is shown on the upper left corner.

Widths:
Bar: Determines the width of the drawbar
Options in Bar drop-down menu:
Min: the drawbar will be of minimal width
Max: the drawbar width will be maximum (e.g., Step template)
Free: the drawbar width is user defined

PO: Determines position of punch opening line (PO line)
Options in PO drop-down menu:
Min: Min. PO line dependent on selected radii and angles
Free: PO line user defined

Heights:
Bar: Determines height of draw bar
Options in Bar drop-down menu:
Part: Height is measured from part
Bndr: Height is measured from binder
Tngl: Height is measured from tangentiall elongation of part

CB :   Determines position of CB height line (Counter bar) with respect to Part boundary (**Caution**: In version 3.01 the height was measured with respect to bar height)

Options in CB drop-down menu:

Free = Counter bar user defined

Off = no Counter bar

Plus is defined as that part of the addendum that contains the laid out flanges. The **Plus** region should not be used to model beads within the addendum. This could lead to strange and unusable results when AutoForm interpolates between the profiles. The **Plus Parameters** are located in the *Advanced Menu* and have the following meaning:

Edge radius: Radius of the part, default value 3 mm

Edge angle: Angle with respect to z-direction, default value 5º

Flange radius: Tangential radius of run off at edge of part, default value 5 mm

Flange angle(incr): Angle with respect to tangent at edge of part, default value 0º

Min. Flange angle: Min. flange angle with respect to horizontal direction

Max. Flange angle: Max. flange angle with respect to horizontal direction

Min. Flange angle and **Max. Flange angle** are limits, in order not to exceed the max/min cutting angle.

There are two additional parameters

Plus width: Width of Plus region, default 0 mm

Plus height: Height of Plus region, default 0 mm

These values depend on one another to a certain extent in the calculation of minimum values. *Fig. 3.6.2* explains the meaning of these parameters.
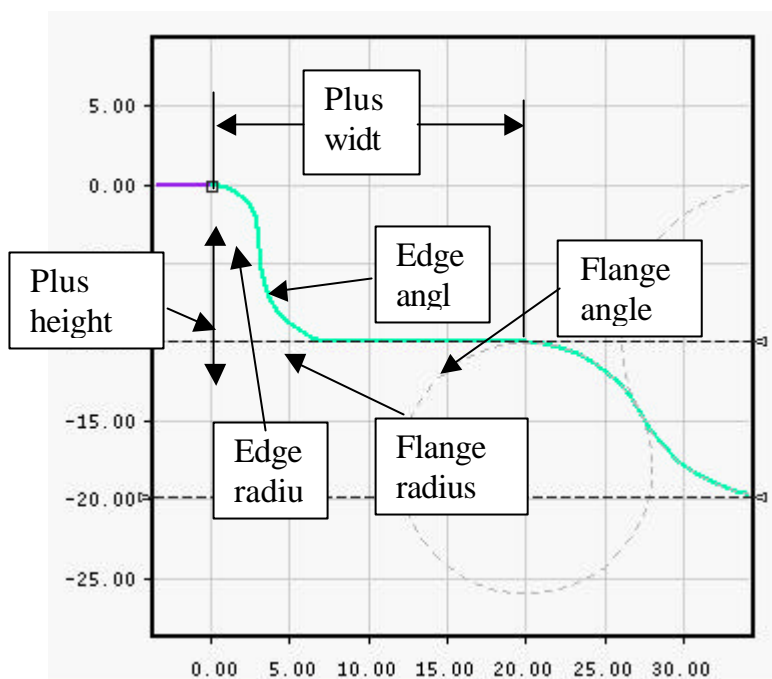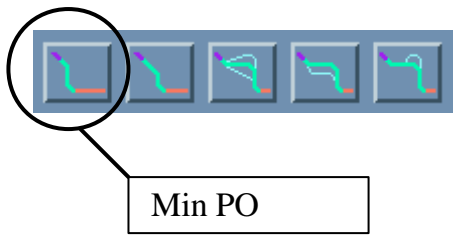

Fig. 3.6.2 Plus parameters

3.6.1 Profile Templates - Nr. 1: Min PO

Min PO

This template minimizes the distance between part boundary and punch opening line. Bar width, Bar height and CB height are not active. The punch opening line is determined by:

1. punch radius
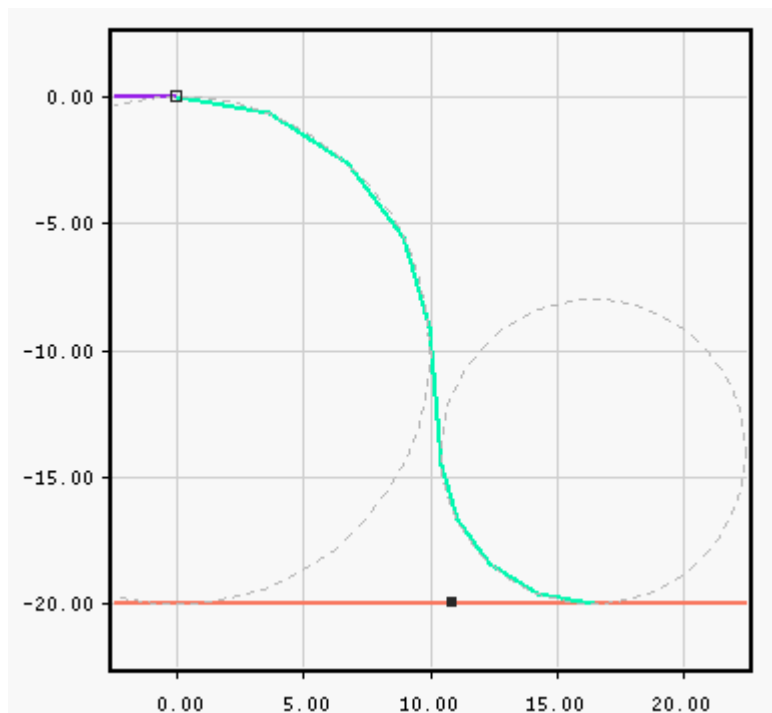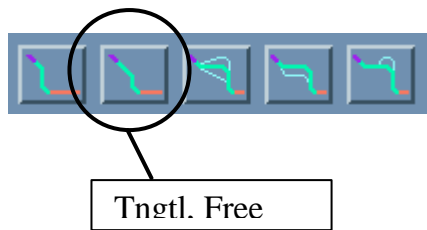2. die radius
3. Plus width (if not equal zero)

Fig. 3.6.3: Profil Templates – Min PO

3.6.2 Profile Templates – Nr. 2: Tngtl, Free PO

This template puts punch opening line to free, which means the user can define punch opening line without restrictions. The height of Bar will be measured from the tangential elongation of the part. Bar width is set to Min and is therefore not active. CB height is switch off (Off) and is therefore also not active.
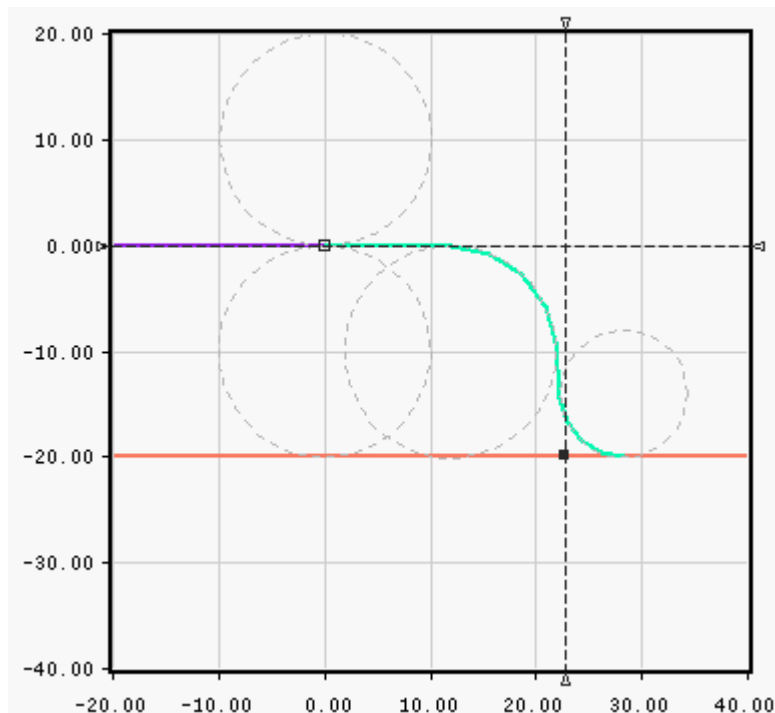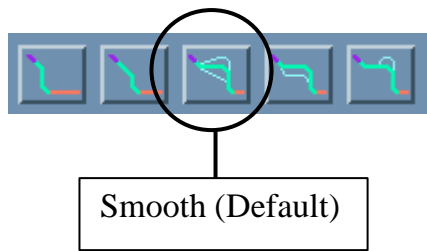

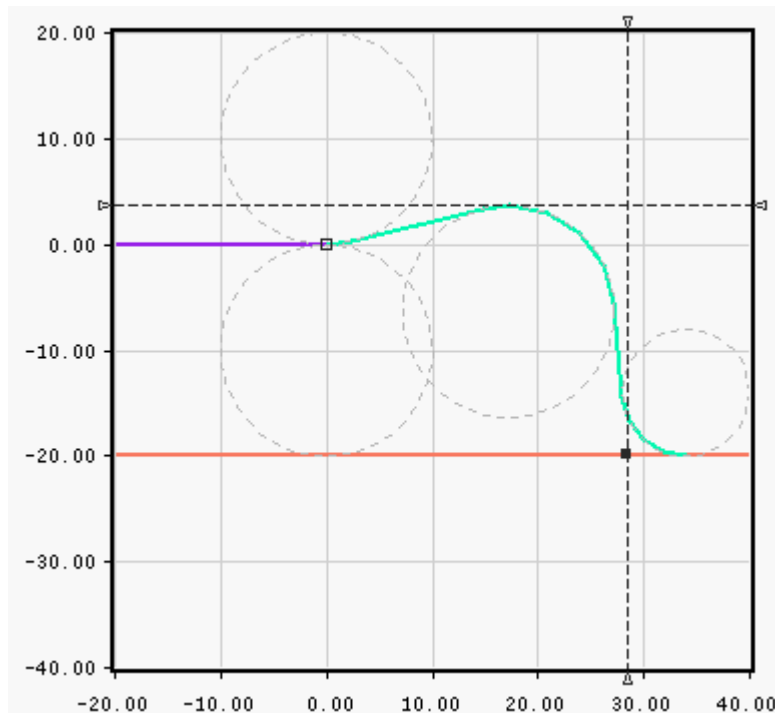**Fig. 3.6.4: Profile Templates – Tngtl, Free PO**

3.6.3: Profile Templates – No. 3: Smooth (Default)

Smooth (Default)

This template puts punch opening line to free, which means the user can define punch opening line without restrictions. The height of Bar will be measured from the part (Part). In many cases it makes sense to use option Bndr. In this case Bar height is measured from binder surface which makes it possible to achieve a uniform initial punch contact. Bar width is set to Min and is therefore not active. CB height is switch off (Off) and is therefore also not active.
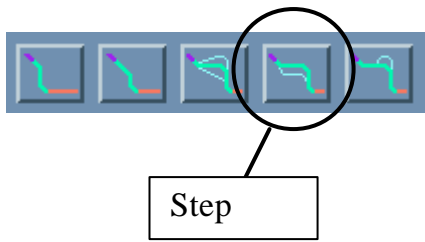

Fig. 3.6.5: Profile Templates – Smooth (Default)

3.6.4 Profile Templates – No. 4: Step

Step

This template puts Bar width to maximum (Max). This results in a step in addendum. Punch opening line is set to free, which means the user can define punch opening line without restrictions. The height of Bar will be measured from the part (Part). In many cases it makes sense to use option Bndr. In this case Bar height is measured from binder surface which makes it possible to achieve a uniform initial punch contact. CB height is switch off (Off) and is therefore not active.
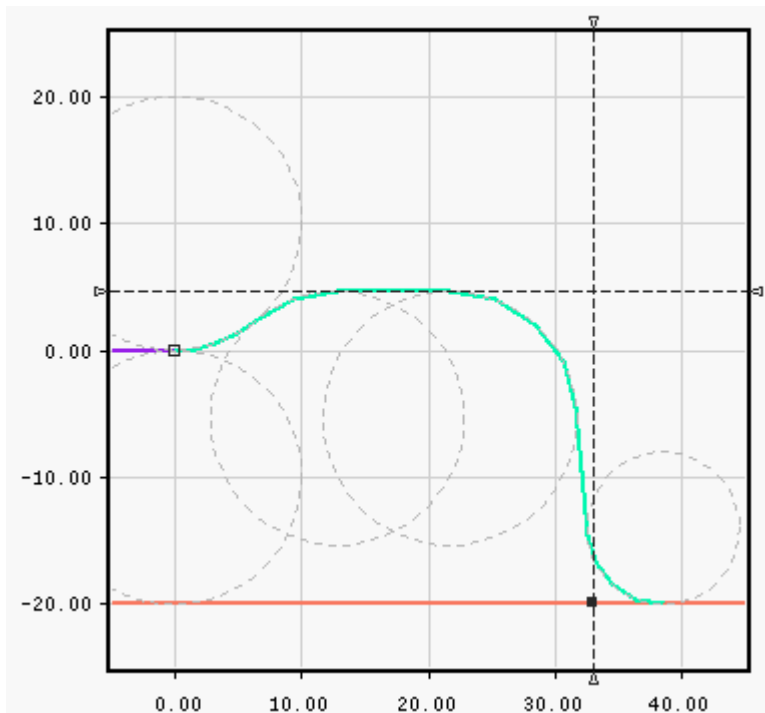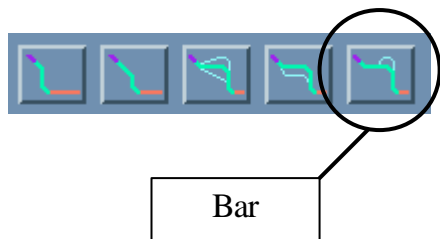

Fig. 3.6.6: Profile Templates – Step

3.6.5 Profile Templates – No. 5: Bar

Bar

This template activates the counter bar (CB height is set to free).
Punch opening line is set to free, which means the user can define punch opening line without restrictions. The height of Bar will be measured from the part (Part). In many cases it makes sense to use option Bndr. In this case Bar height is measured from binder surface which makes it possible to achieve a uniform initial punch contact. Bar width is set to Min and is therefore not active.

Caution: CB height is measured from part and not like in previous version with respect to Bar height. This means that for CB height = 0 the counter bar (CB) disappears. This change can lead to the effect, that the Addendum created in an older DieDesigner version looks different in version 3.03.
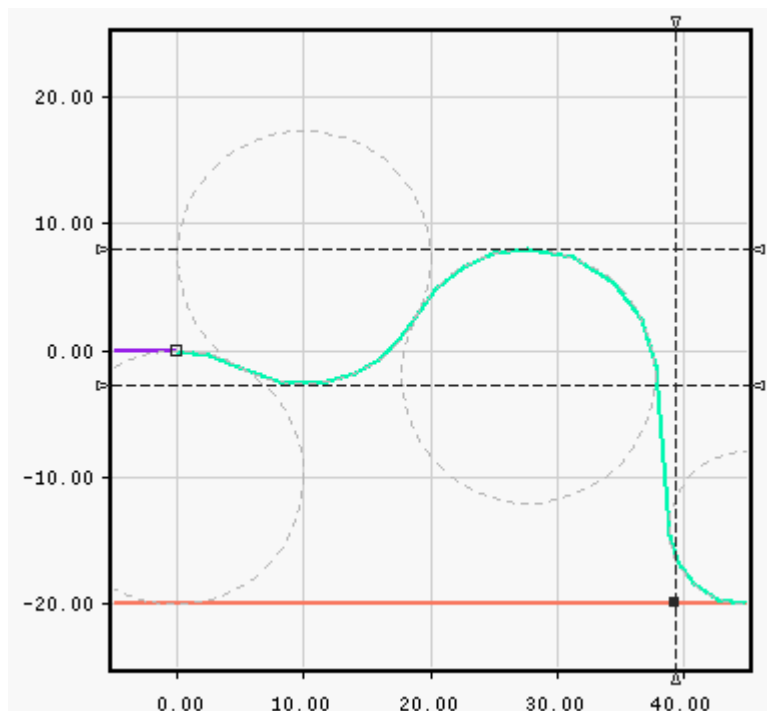

Fig 3.6.7: Profile Templates – Bar
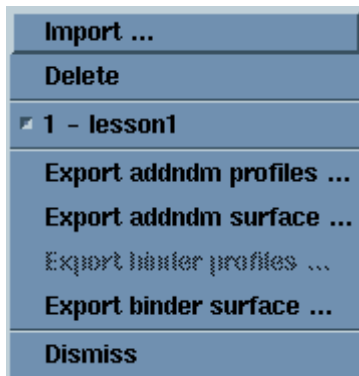
Menu Options

3.7.1 File Menu

Fig 3.7.1 File Menu

Import reads in geometries in IGES-, VDAFS-, af-, afb-, Nastran- or STL-format. The imported geometries are numbered sequentially as they are read in for easy reference. The active geometry is marked by the depressed button beside it (e.g., lesson1)

Delete deletes the current geometry.

Export addendum profiles ... exports profiles of addendum in IGES-, VDAFS- or af-format.

Export addendum surfaces ... exports addendum surfaces in IGES- or VDAFS-format.

Export binder profiles ... exports profiles of binder (for manual binder only) in IGES, VDAFS- oder af-format.

Export binder surfaces ... exports binder surface in IGES- oder VDAFS-format.
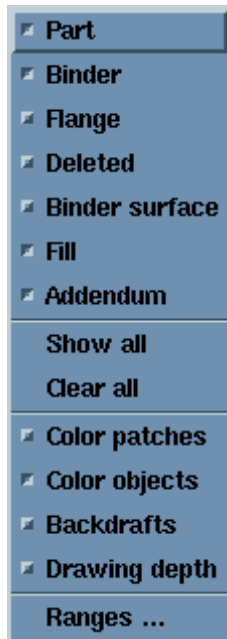
3.7.2 Display Menu

Fig 3.7.2 Display Menu

The upper half of the menu is for switching on or off, the display of the appropriate register (Part, Binder, Flange, Delete) and of the surfaces generated by AutoForm (Binder surface, Fill, Addendum).

Show all switches display of all surfaces to on.

Clear all switches display of all surfaces to off.

The next four options determine the display properties of the objects: **Color patches** colors the imported patches of CAD geometry. **Color objects** colors the different AutoForm objects (Part, Binder, Addendum, Fill...). **Backdrafts** shows the undercuts of the part with respect to the actual tip. **Drawing depth** shows the drawing depth as a color distribution on the part with respect to binder surface.

Caution: The default setting varies depending on which page you work. The settings can be adjusted at any time from this menu.
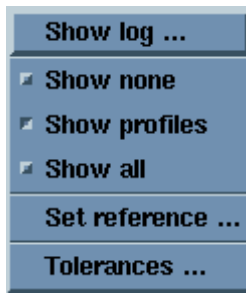
3.7.3 Options Menu

Fig. 3.7.3 Options Menu

Show log... shows the log-file.

Show none shows none of the addendum profiles.

Show profiles shows the defined profiles of addendum.

Show all shows all profiles of addendum.

Set reference... reads in a reference profile.

Tolerances ... opens tolerance menu for setting tolerances (Fig. 3.7.4).
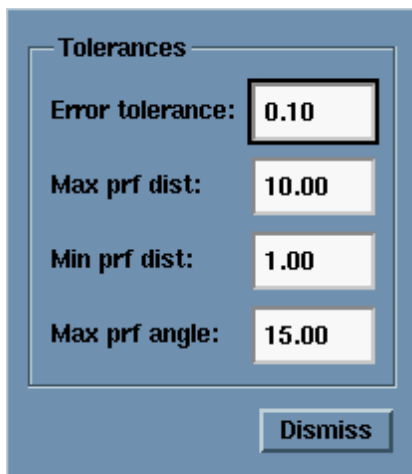
3.7.4 Tolerances

Fig. 3.7.4 Tolerances

Error tolerance: chordal error for meshing the addendum.

Max prf dist: defines maximum distance between profiles

Min prf dist: defines minimum distance between profiles

Max prf angle: defines maximum angle between profiles