TpaCadMd

2.2.0

Modelling in Tpacad

Tecnologie e Prodotti per l'Automazione

Table of Contents

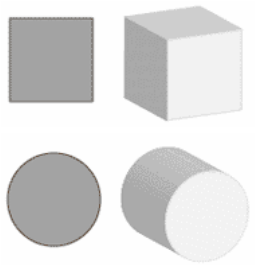
1 Overview

Help contains the description and the instruction of all the characteristics of the TpaCAD program, concerning the use of the following functionalities:

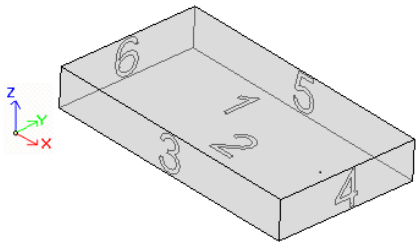
- Modelling a piece
- Creating curved faces
- Managing surfaces.

2 Modelling in TpaCad

TpaCAD allows you to create the three-dimensional modelling of a piece, using the extrusion technique.
The extrusion technique creates a thickness by lengthening bi-dimensional objects along the depth axis.
Typical applications concern the machining of pipes, bars, section bars, plates.



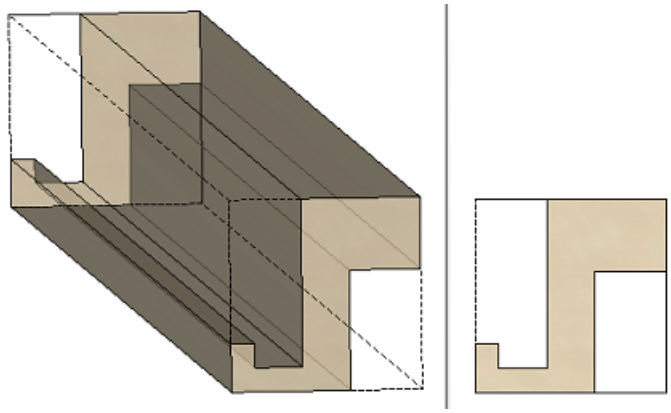
In this figure we can see how
- the extrusion of a square leads to a cube or a parallelepiped;
- the extrusion of a circle leads to a cylinder.



In TpaCAD, the starting piece is always a parallelepiped, so assigned on
three dimensions: length, height, thickness
six faces that are automatically numbered from 1 to 6.
The modelling process starts from one of the six faces of the piece and the extrusion is carried out along the depth axis of the face up to the opposite face.

Back to the cases of a square and a circle,
- the extrusion of the face 1 lengthens both the objects up to the face 2;
- the extrusion of the face 3 lengthens both the objects up to the face 5;
- the extrusion of the face 4 lengthens both the objects up to the face 6

and viceversa.



As a first significant aspect, modelling allows you to get a more real view of the face. An example in the figure
- on the left: an oriented view of the piece;
- on the right: a front view of the section.

In the illusstrated case, the extrusion face can be the 4 or the 6.

A second remarkable point concerns the definition of oriented faces and their programming, as a consequence of an assigned modelling or as a definition of the same. In the figure, up to 10 working plans may be assigned, each with a XT plan assigned along the normal cut section.

2.1 Enabling in configuration

The use of the modelling functionalities needs a specific must be specifically enabled from HW key besides the *Professional* functionality.

TpaCAD controls different modes to model a piece.

1. Modelling profiles

The first mode is to assign specific information to profiles that are programmed in a specific face of the piece. This is the mode that provides more flexibility because it can take advantage of all the processes of assigning a profile available to the application. Modelling derives from programming *modelling profiles.*

This modelling system totally works as an alternative to both the following systems.

2. Modelling section

A second mode is to enable a specific modelling section, through sections of assignment that can be activated for the piece. Modelling *sections* is managed according to specific rules and in no way depends on the workings programmed on the piece.
This modelling system totally works as an alternative to the previous system but not to the next.

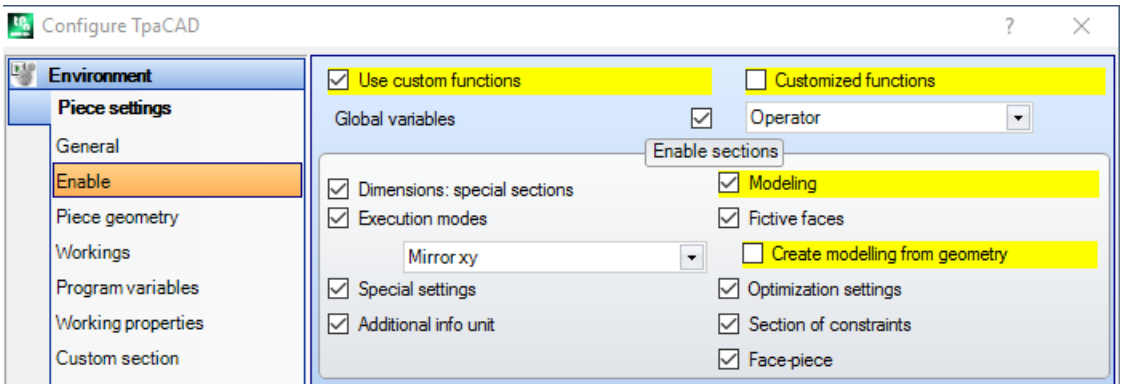
3. Modelling applied to fictive faces

A third mode is the ability to assign the modellings directly in the section of the fictive faces.
This modelling system totally works as an alternative to the first mentioned, but to to the previous one.

Let's see how to activate the Modeling functionalities. In TpaCAD, start the command Configure TpaCAD, from the menu .

Create modelling from dedicated section and/or variable geometries

In Piece Settings of the configuration window, select Enable:



In the figure we have captured the upper part of the page of the group enabling the sections. The most interesting entries are two:

- ✓ *Modelling*: select to enable the piece modelling section
- ✓ *Create modelling from geometry* : select to enable the possibility to assign some modellings directly in the section (that must be enabled) of the fictive faces.

They are both quite separated from each other and can coexist.

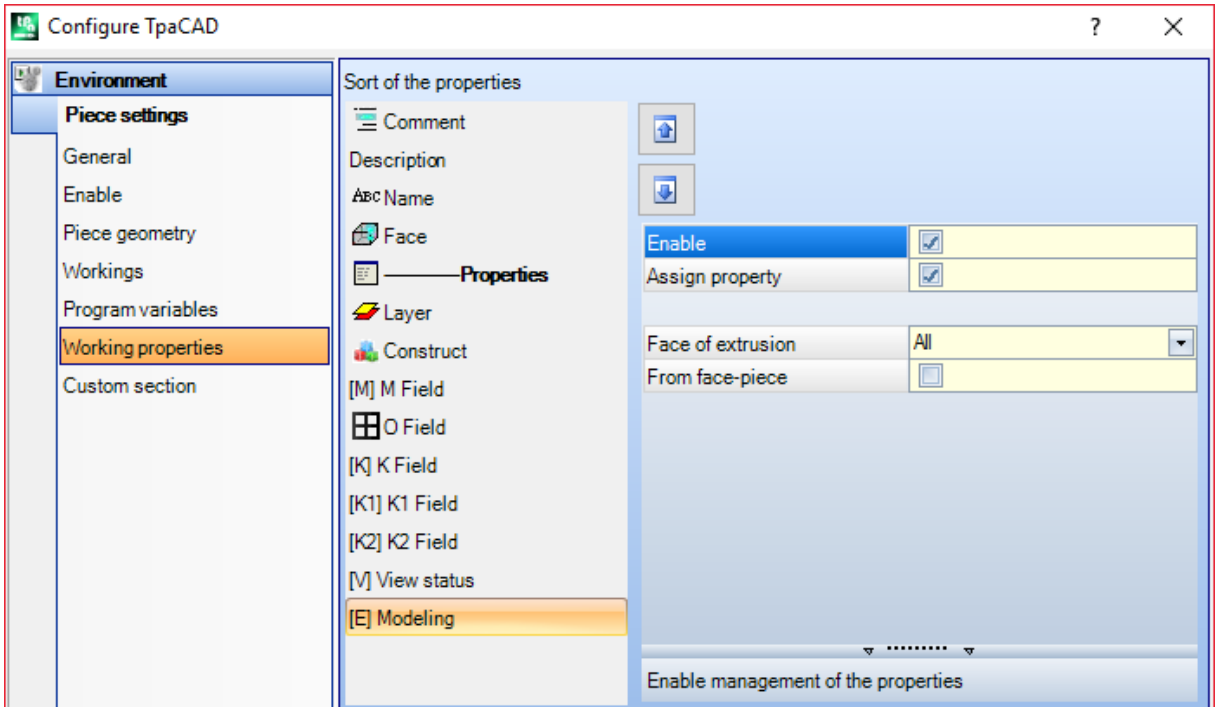
If the only entry *Create modelling from geometry* is selected, you must generally configure a particular field in the *Special Settings* custom section:

If you check *Create modelling from geometry* only, you can assign the modelling face in the unit of piece dimensions.

Create modelling from programmed profiles

In the configuration window select Enabled from Piece settings and make sure the the the items *Modelling and Create modelling from geometry* are not selected.

Remaining in Piece setting , now select Working properties:



The concerned property is shown as Modelling (field E).
This field is managed in the workings of Profile setup

- header is *Extrusion profile*
- the value assigned can be chosen from one of the following entries:
  - *none*: no useful assignment for modelling.
  - *external*: assigns as an external modelling profile
  - *internal*: assigns as an internal modelling profile

Let us see the field that can be set for the property:

- Enable: enables the property management
- Assign property: this option enables the management of the corresponding command for the overall assignment in the Assign property group of the Edit menu.

The last two fields are remarkable for the property:

- Face of extrusion: assigns the face of extrusion, choosing from a list of item:
  - *All*: does not make any pre-selection of a face (from those real of the piece)
  - *Face 1*: recognise the modelling from face 1 only
  - *Face 2*: recognises the modelling from face 2 only
  - ..
  - *Face 6*: recognises the modelling from face 6 only

ATTENTION: : the number of the faces are to be considered in absolute numbering

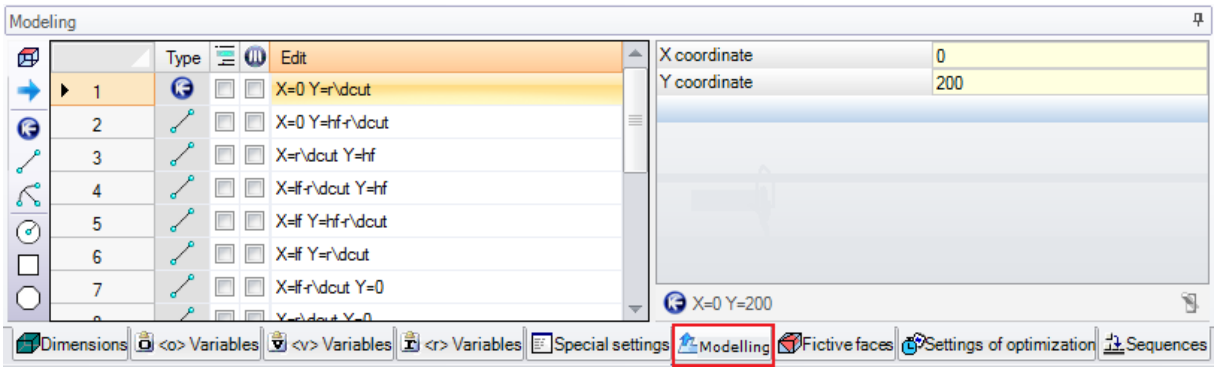
- From face piece: assigns the recognition criteria of the piece-face modelling profile.
  - *non-active selection:* : this option recognise the modelling only if not resulting from the face piece
  - *active selection:* : this option recognise the modelling only if resulting from the face piece.

Let us see some typical situations:

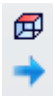
- Extrusion face=4, from piece-face= OFF: assignment of modelling profiles will be possible in the face view 4 only
- Extrusion face=4, from piece-piece = ON: assignment of modelling profiles will be possible in the view of the piece-face and programmed face 4 only
- Extrusion face= all, from piece-face= ON: assignment of modelling profiles will be possible in the view of piece-face only. The face is not previously fixed and will be assigned in the piece programming in the unit of Dimensions always only of the six real faces of the piece.

2.2 Modelling section

This section is added to other enabled for the piece, to the position before *Fictive faces*



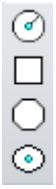
The figure shows the section wit already assigned settings: the left table shows the assigned geometric elements and the area on the right presents the element selected in the table.
Command bar in the section:



The first button shows the icon of the *extrusion face* (as in the figure: face above)
The second button (arrow) assigns the extrusion face to the next face, where rotation is possible on the six faces of the original parallelepiped, including possible non-managed faces.

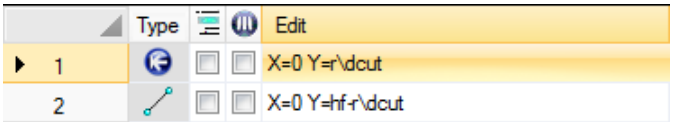
The three buttons control the insertion of a polyline, respectively:

- the initial point of the polyline
- a linear segment
- a curved segment (arc).



The three buttons control the insertion of closed and geometric isolated figures, respectively:

- circle
- rectangle
- POLYGON

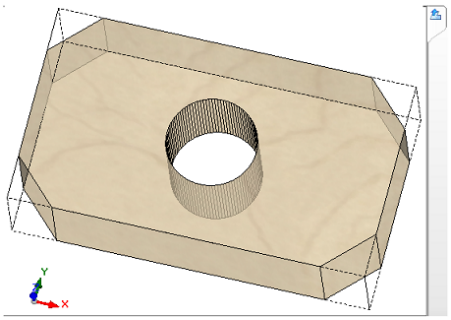


Now, let's see how the table of the element is arranged:

Type: shows the type of the element (toolbar icon)
: selected excludes the interpretation of the element
: selected, marks the element as *Internal profile* (significant for this kind of elements: initial point, circle, rectangle, polygon).
Edit: resulting settings of the element (as in the figure, also parameter settings), as assigned in the right list (see later).

You can also insert up to 300 elements.

Let's see how a graphic representation can appear:



the icon of the section is shown next to the graphic area (just like for the fictive faces, for example).

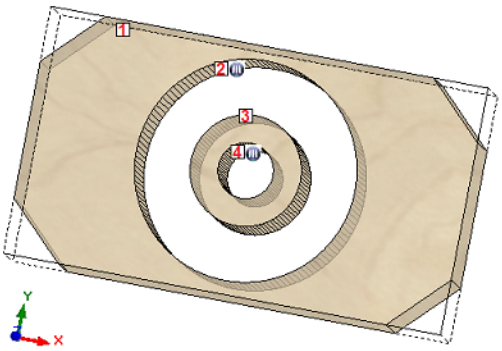
The piece is represented only in 3D view, without fictive faces and workings: the only elements shown correspond to the programmed extrusion.
Like in the figure:

- the extrusion face is the upper face (or lower);
- the edges of the original parallelepiped are dotted;
- the programmed geometries are two: an external polyline (of linear elements only) and a circle marked as an internal profile (*that subtracts* area from the external polyline, that is, it creates a *hole)*;
- the pattern (or colour) set to fill in the piece is now applied to the resulting piece without the modelling.

Let's now define each geometry inserted as a *polygon*,always interpreted as closed (in case of polyline: if it is not closed, a linear closing segment is automatically inserted): to all the polygons assigned those rules are applied that in a graphic context are called *tessellation rules*. The application of these rules leads to decide:

- the external overall dimension of the piece (as shown in the figure: the polyline among chamfered edges)
- the removed areas (as in the figure: the circle inside).

Let's see some particular aspects as follows:

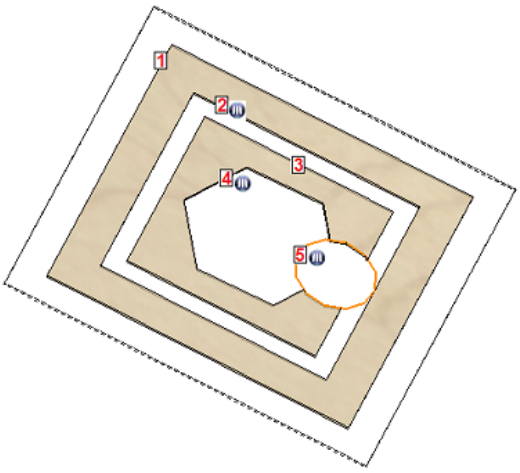


4 Polygons are assigned (pay attention to the assignment in the table):

1. external polyline
2. circle, where the *Internal Profilo* is active
3. circle, where the *Internal Profile is* not active
4. circle, where the *Internal Profile* is active

The polygon 2subtracts from the area of the polygon 1.

The polygon 3 assigns a full area.
The polygon 4subtracts from the area of the polygon 3.



5 Polygons are assigned (pay attention to the assignment in the table):

1. external rectangle
2. rectangle, where the *Internal Profilo* is active
3. rectangle, where the *Internal Profile is* not active
4. polygon (6 sides), where the *Internal profile is* active
5. polygon (12 sides), where the *Internal profile is* active

The polygon 2subtracts from the area of the polygon 1.

The polygon 3 assigns a full area.
The polygons 4 and 5 subtract from the area of the polygon 3.

REMARK:
for a correct interpretation, a polygon marked as *Internal profile* must be totally inside or almost intersect another polygon not marked as such. If the first listed polygon is assigned as *Internal profile*, it is automatically a *hole* of the extrusion face.

The definition of the composition rules for the modelling polygons follows these three points

1. each main polygon subtracts the corresponding areas from its own internal polygons. Therefore, one or more full areas can be found where possible empty areas are inside them;
2. all polygons corresponding to full areas are summed in consideration of their intersection, so one or more combined full areas of all the modelling are identified;
3. all polygons corresponding to empty areas are summed in consideration of their intersection, so one or more combined empty areas of all the modelling are identified.

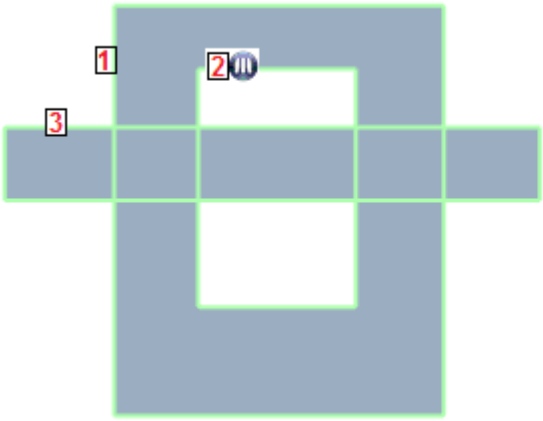
The application of the composition rules of the modelling polygons includes also any polygons assigned in section of fictive faces (see later).

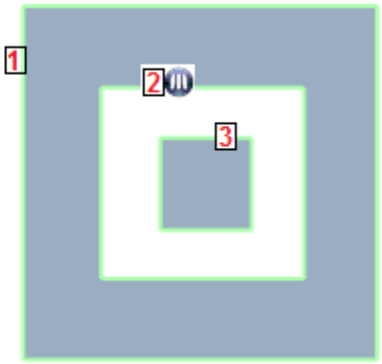
It is realised that the above mentioned result of the composition rules can determine a visualisation that, also considerably, does not not correspond to the original polygons, that the user can display selecting the button in the command bar:
once selected, the display of the original profiles is required. the representation colour is assigned at the option *Modelling* in the list of Graphics colours (command: Customize→ Colors).

Another button allows you to choose between two different rules of union of polygons ( ):
(button non selected) chooses the rules called of transparency
(selected button) chooses the rules called of opacity
The terms "transparency" and "opacity" used herein correspond to the need to facilitate understanding and application of the selectable criteria for the union of the polygons.

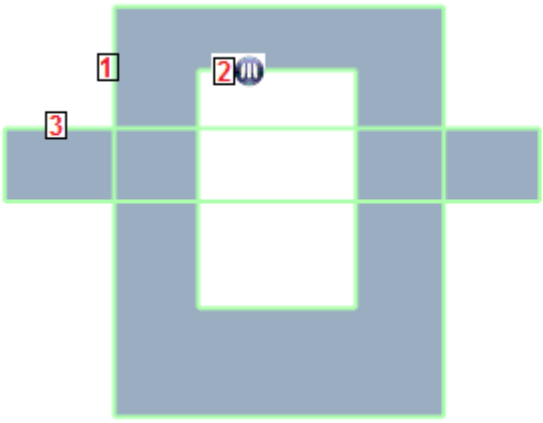
The difference between the two rules can be explained by the following example.
Both the following figures show two examples of modelling, solved with:

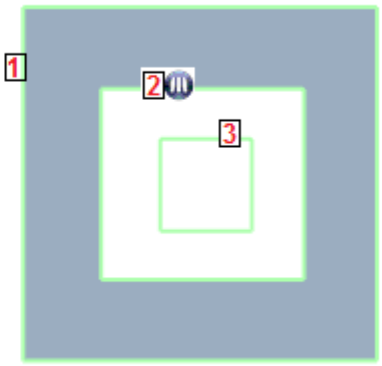
- Transparency: you can see how the polygon 3 in both the cases determines a full area within the internal profile 2 (let's say that the area subtraction determined by the profile 2 is applied in transparency)





- Opacity: you can see how the polygon 3 in both the cases does not fill the hole created by the internal profile 2 (let's say that the area subtraction determined by the profile 2 is applied in transparency)





Let's see now the detailed settings of the elements that can be inserted:

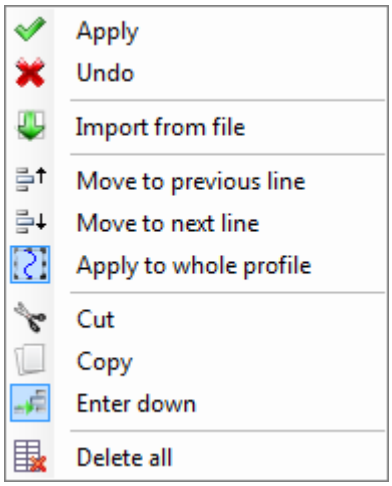
	Beginning of the polyline	X coordinate: X coordinate of the point; Y coordinate: Y coordinate of the point;
	line	X coordinate: X coordinate of the segment; Y coordinate: final Y coordinate of the segment; As an alternative: Length: it sets the length of the segment assigning a direction that continues from the previous segment (0° if the element starts a polyline or if it precedes an element of the beginning of the polyline) If the element starts a polyline, before a circle, a rectangle or a polygon, the segment starts from the centre assigned for the element before. If, though, the element is the first in the list, the same starts from the point assigned to (0:0) coordinates.
	arc	X coordinate X coordinate of the segment; Y coordinate final Y coordinate of the segment; Anticlockwise rotation select to set anticlockwise rotation of the arc;

		Radius radius of the arc (not less than 10.0*epsilon quote); As an alternative: X Centre X coordinate of the centre of the arc (normally incremental with respect to the starting point; absolute, if it starts with "a;"); Y Centre Y coordinate of the centre of the arc (normally incremental with respect to the starting point; absolute, if it starts with "a;"). As an alternative: (radius and centre not assigned) the assigned starting direction of the arc continues from the previous segment (0° if the element starts a polyline or if precedes an element of the beginning of the polyline) Number of sides: sides into which the arc is fragmented (maximum value accepted= 99) The fractionation of an arc interprets the arc as a part of a polygon, but with the possibility of an addition into a broken line. The linearisation of the arcs comprises the need to create working plans on the different facets of the arc. Anyway, a minimum 5.0° angle splitting is managed. If no number of split sides is assigned (or it is anyway=0), you need to distinguish two cases: 1. If the configuration does not manages the assignment of the curved faces, a default splitting - each 10°- is assigned: 2. otherwise, the arc is not split, allowing the creation of one only curved work surface. If the element starts a polyline, before a circle, a rectangle or a polygon, the segment starts from the centre assigned for the element before. If, though, the element is the first in the list, the same starts from the point assigned to (0:0) coordinates.
	circle	X Centre: X coordinate of the centre of the circle (in absolute system); Y Centre: Y coordinate of the centre of the circle (in absolute system); Radius radius of the circle (not less than 10.0*epsilon quote); Angle rotation angle with respect to the XY coordinated axes of the extrusion face: it assigns the starting point of the circle. If the configuration does not manage the assignment of the curved face, the development of the circle applies a default splitting - each 10° -. otherwise, the circle is not split, allowing the creation of one only curved work surface. if you need to force a different splitting, use a polygon.
	rectangle	X Centre: X coordinate of the centre of the rectangle (in absolute system); Y Centre: Y coordinate of the centre of the rectangle (in absolute system); Radius: fillet radius of the vertices (not less than 10.0*epsilon quote); Length dimension of the rectangle along the X axis (not less than 10.0*epsilon quote); Length dimension of the rectangle along the Y axis (not less than 10.0*epsilon quote); Angle rotation angle of the figure with respect to the XY coordinated axes of the face (0 if the rectangle is aligned to the axes).
	polygon	X Centre: X coordinate of the centre of the polygon (in absolute system); Y Centre: Y coordinate of the centre of the polygon (in absolute system); Radius radius of the circumcircle (not less than 10.0*epsilon quote); Number of sides sides of the polygon (between 3 and 99); Angle rotation angle of the figure with respect to the XY coordinated angles of the extrusion face (0 if the first vertex of the polygon is along the x positive semi-axis of the extrusion face).
	oval	X Centre: X coordinate of the centre of the oval (in absolute system); Y Centre: Y coordinate of the centre of the oval (in absolute system); Radius: radius of the curve along the major axis (not less than 10.0*epsilon quote); Length: dimension of the overall rectangle along the X axis (not less than 10.0*epsilon quote);

		Height: dimension of the OVERALL rectangle along the Y axis (not less than 10.0*epsilon quote); Angle: rotation angle of the figure in respect of the XY coordinated angles of the extrusion face (0 if the length dimension is along the x positive axis of the extrusion face).

WARNING: the coordinates of the elements concern the XY system of the extrusion face.
WARNING: parametric programming are admitted, including the dimension of the extrusion face (lf, hf, sf).

The local Menu of the section presents the usual commands to work the Section of the piece:



Apply: it applies the changes to the piece
Cancel : it cancels the changes and restores the section and the piece reading

Import from file: it reads the modelling section from a file

Move to the previous/next line : they move the current line to the previous or to the next position. The functioning of the commands depends on the status of the *Apply to whole profile* button

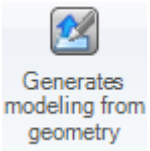
If not selected, the previous commands affect only the current working, even if it belongs to a polyline. Moreover, the current element is moved to a position in the list, even though it breaks a profile before or after the profile;
if selected: if the current element belongs to a polyline, the previous commands move the whole polyline Moreover, if a polyline is found before or after, this one is considered as a whole and it is not broken by the movement to the list.

Cut: it deletes the selected elements or the current one, with storage in the clipboard.
Copy: it copies the selected elements or the current one, with storage in the clipboard.
Enter down: it selects the insertion point Above/Below with respect to the current line
Delete all : it resets the section

Cut: it deletes the selected elements or the current one, without storage in the clipboard.

Click on a graphic area, next to a polygon, to move to the corresponding line in the table.
Click and hold down the CTRL key to change the selection status of the corresponding line in the table.

Creating modelling from geometry

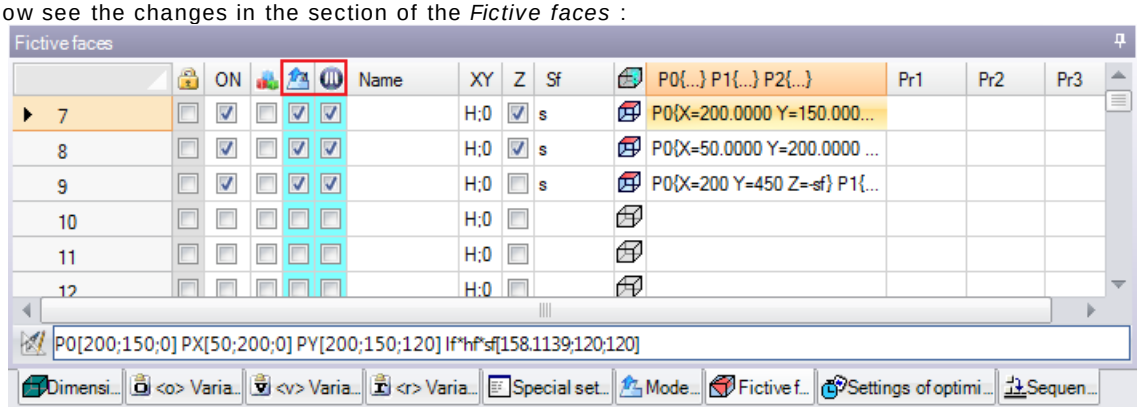


The *Creating Modelling from geometry* command is called in the group of *Advanced* in the *Apply* tab, if the enabling of the Modelling section is active.
This command is managed in face view with non-empty face program and current working of a simple profile (made of elements of arc or line type).
This command is not active in view of curved face or surface.

This command adds a polyline in the modelling section at the current profile and always until the maximum number of the available elements that can be entered in the section.
Assigned arcs in a plane different from xy or path elements become linear segments again.

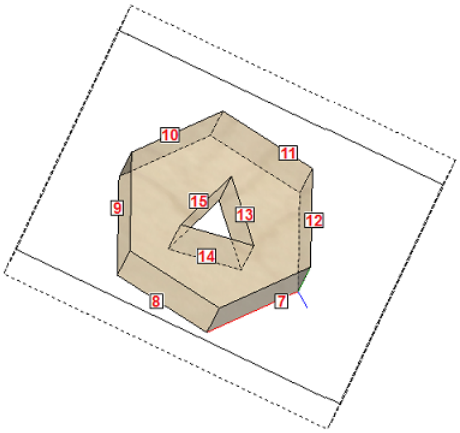
2.3 Modelling applied to fictive faces

Let's now see the changes in the section of the *Fictive faces* :



The table shows two new selection columns:
select to indicate that the fictive face is a modelling element
selected marks the element as a *Internal profile.*

Additional modelling polygons can be here possibly defined to those already defined in the Modelling section.
To this purpose, select the case to mark the modelling elements.



The *extrusion face* is assigned:
- in the field *Extrusion face* of the Special settings or, (if not assigned)
- in the Modelling section or (if not managed)
- always in Face 1.

Also in this case you need to assign closed polygons. The polygons are now identified in consideration of the countersigned modelling faces in assignment sequence (from the number 7):
- geometrically consecutive fictive faces define a profile;
- with interrupted geometric consecutiveness a new profile starts.

The geometric consecutiveness is calculated when the P0 and P1 points of the face are compared and converted into the reference system of the extrusion face and considering the (x;y) coordinates only.

Non-closed polygons are automatically closed.

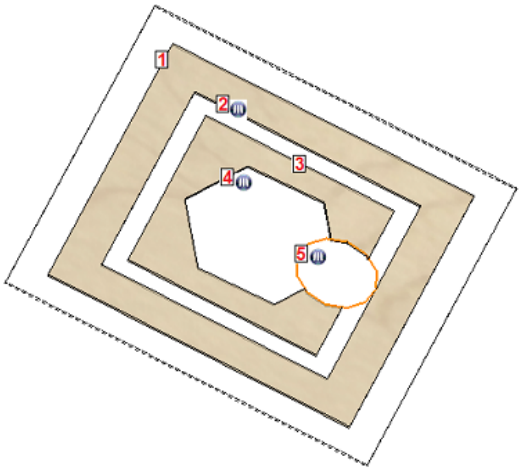
Modelling polygons here assigned are shown also in the modelling graphic representation.

NOTE:
for a correct interpretation, a polygon marked as *Internal profile* must be totally inside or almost it must intersect another polygon not marked as such. If the first polygon in the list is assigned as an *Internal profile* and if no polygons are assigned from the modelling section, the polygon is automatically a *hole* of the extrusion face.

Creating fictive faces from modelling elements

As it is evident, you may need to generate some work plans (fictive and automatic faces) that correspond to a modelling, as assigned in its section.
In general, it is obviously possible to create the faces in a direct way, using perhaps the same parametric forms used in the modelling assignment. However, the result of a modelling cannot always be easily predicted: as we have seen, the original elements can be modified after the intersection between the polygons.
Then, you may face the problem of how to access these elements, generally modified.

Let's consider the already suggested example, in which 5 polygons are assigned:



The situation corresponds to modelling polygons that do not correspond to the original ones any more, but are a more or less complex processing.
To assign some work plans on real boundaries, it may be necessary to enter these processed profiles.

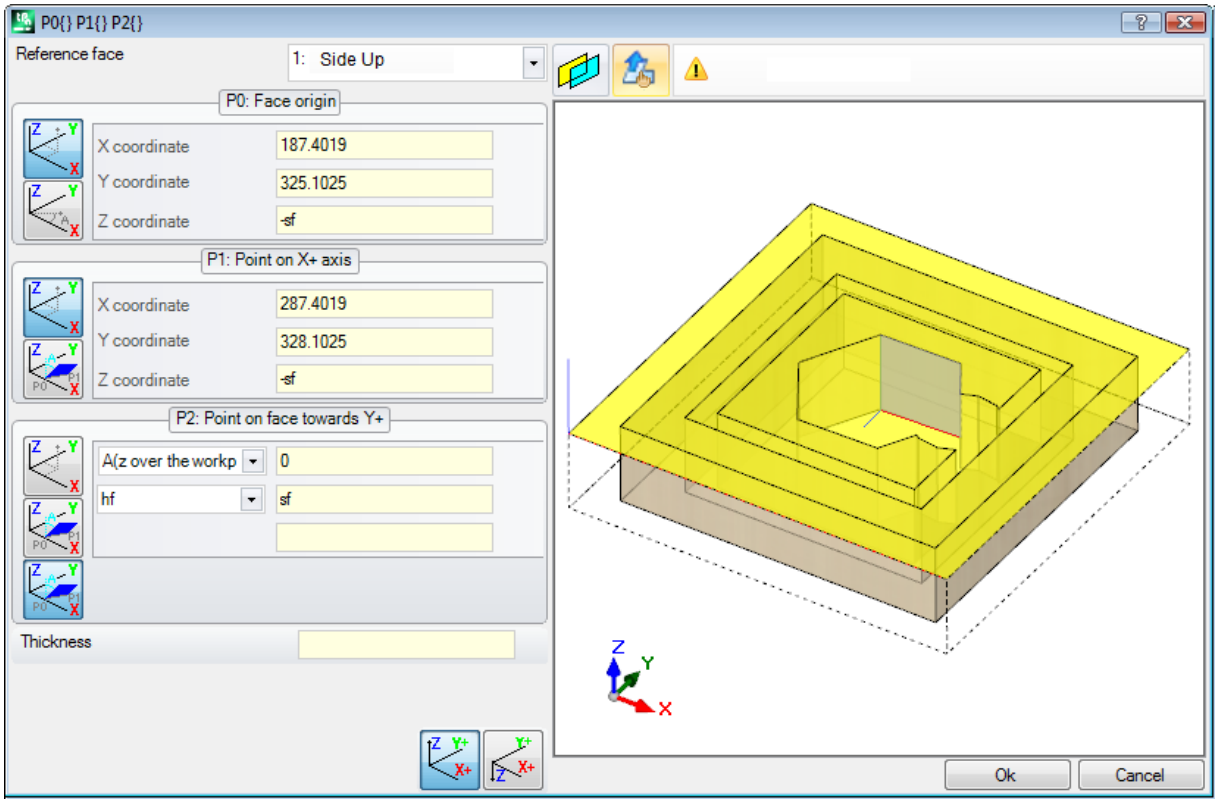
The set of processed profile is now accessible through the window of the fictive face assignment, whose button is shown here

- select the button to activate the *snap* functionality on the individual elements of the modelling;
- reset the selection to activate the functionality.

Anyway, the modelling polygons that are generated directly from fictive faces are neither represented nor accessible here.

Selecting the button,
- the reference face is set on Extrusion face (in the figure: Figure 1)
- then, click on the graphic area to assign the settings to the values read by the modelling element to correspond with the click.

As in the figure, the settings have been acquired from a wall of the innermost hole. The settings are always assigned as numerical coordinates, besides the depth coordinates.



Continue normally, making the changes required and quit, confirming or canceling.

If the TpaCAD configuration manages the assignment of the curved faces, you can select also a curved element. In this case the acquired settings show directly the assignment of a curved face (see following paragraphs).

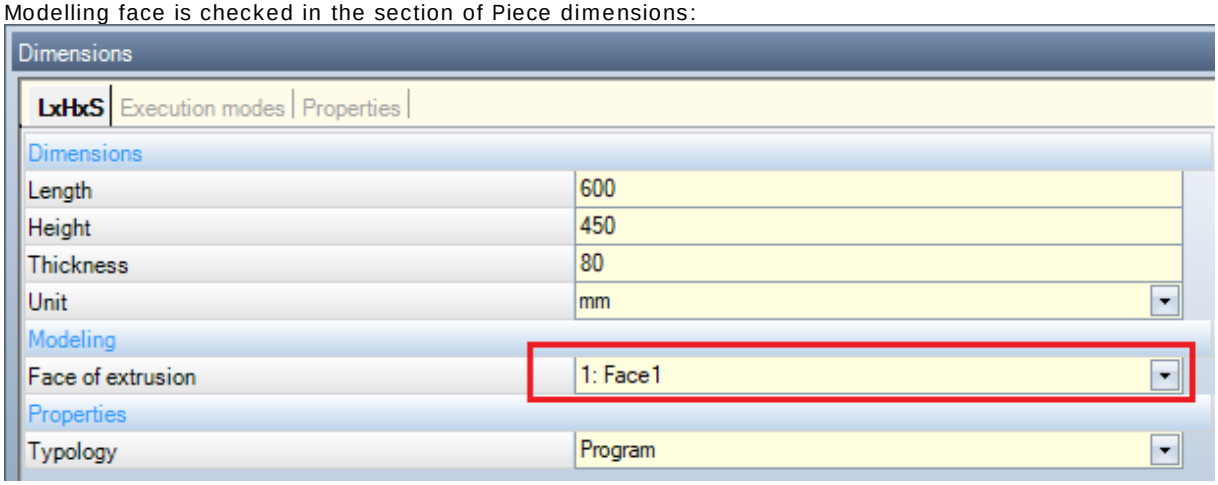
Creating automatic faces from modelling elements

Also during the assignment of automatic face it is possible to identify the face from a modelling element. In this case, a new work plan will be always assigned. It corresponds to a modelling anyway assigned, both from the modelling section and from the variable geometries.

2.4 Modelling from programmed profiles

A Setup of profile assigned as a modelling setup will actually be useful if followed by elements of profile: a modelling polygon is now defined directly in the programming phase of the piece.

Modelling face is checked in the section of Piece dimensions:



If you have assigned the modelling face during the Configuration of TpaCAD, the field cannot be changed. If you also enable the management of piece-face, the field will be displayed, if the modelling is recognized by piece-face.

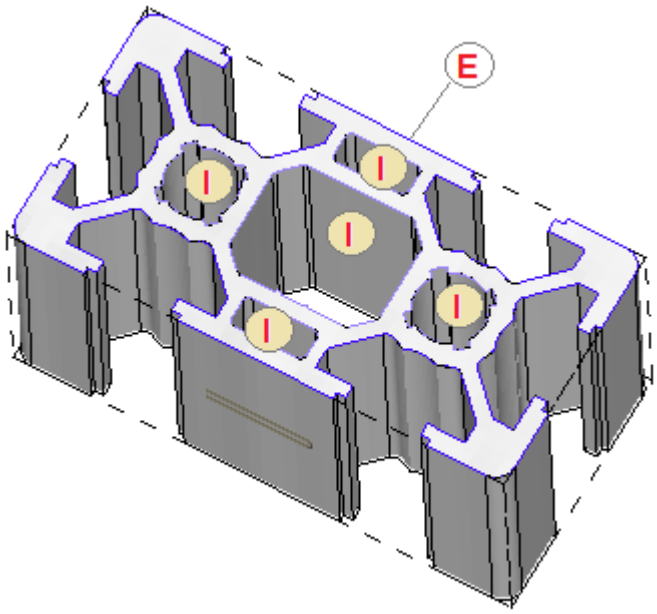
A modelling profile is always interpreted closed, with possible automatic closing with a linear segment. Assigned arcs in a plane different from xy or path elements become linear segments again.

Without any information added, assigning the modelling, all profiles externally marked are applied first and after all the internal ones, each one is numbered in progressive programming order. This means that it does not recognize any selective allocation between individual profiles. This operation fulfils for instance the case of bar section modelling, in which external polygons have no overlapping area.

Below, two typical examples of bar sections:

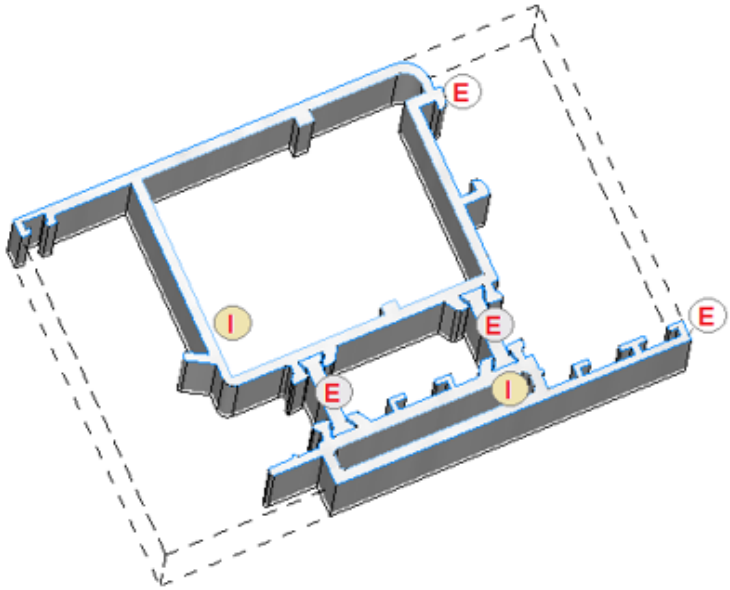
a first example with 6 modelling profiles:

- an external profile (marked by E)
- 5 internal profiles (marked by I)



a second example with:

- 4 external profiles (marked by E)
- 2 internal profiles (marked by I)

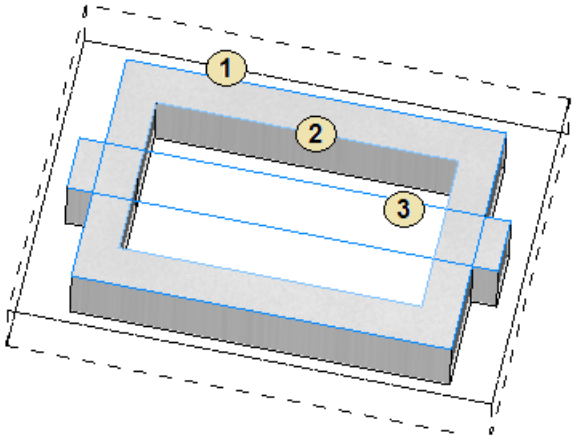


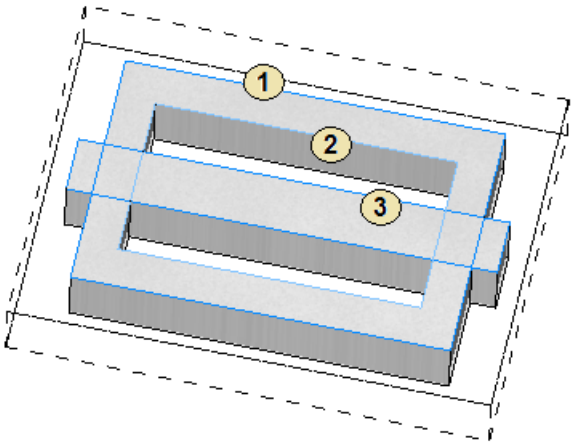
If you need to acknowledge some selective attributions between individual profiles you can program different field values L (level). In the case of use of level values 0 and 1, all external contours are first applied and then everyone with level 0, later all those with level 1.

The figure shows an example of 3 profiles:

- 1 and 3 are external profiles
- 2 is an internal profile.

In the case on the left all profiles have the same Programmed level.
In the case on the right, the profiles (1,2) have level 0, while the profile 3 has level 1.





Modellings generated by programmed profiles can be used to create fictive faces, and find an individual element, as mentioned above in operating section of *Fictive faces.*
Instead, it is not possible to use the modelling when the automatic face is assigned.

When a macro-program is assigned, no modelling is applied.

2.5 Modelling: options of graphic representation

Commands for specific activations relating to the modelling process are grouped in View on of the Customize views group.

	Modelling: enables or disables the graphic representation of the modelling
	Modelling transparency: check this entry to get the modelling representation in the foreground, with opaque effect on the other programmed entities. Uncheck the entry to see the modelling with graphic transparency effect, so as to keep the visibility of all the programmed entities.

3 Curved faces

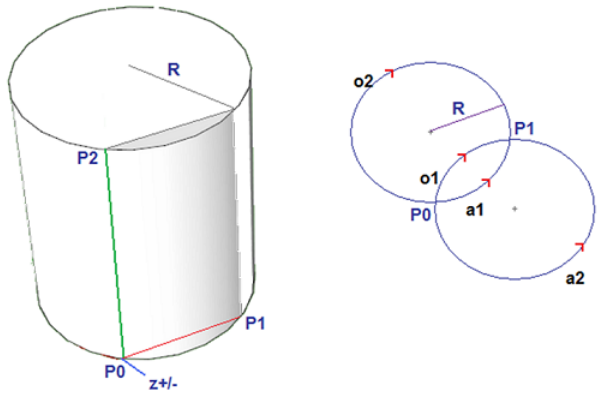
As you know, TpaCAD allows the creation of work faces anyway oriented, both fictive or automatic. They are normally faces assigned in the plane, with the direct or indirect identification of three distinct and not aligned points.

- ✓ P0: origin of the axes
- ✓ P1: extreme point of the face towards X+
- ✓ P2: extreme point of the face towards Y+

in addition to the right/left-handed coordinate system for the Z axis.
The P0-P1 distance is the length of the face.
The P0-P2 distance assigns the height of the face.

For the assignment functionality of a fictive face, please look up the manual of TpaCAD (chapter: Piece; paragraph: Assignments →Fictive faces).

Let's now consider a generic fictive face (plane) and let's examine the figure:



The plane face is assigned with three points P0-P1-P2 and the Z axis (see the figure: right-handed coordinate system).
Suppose now we need to find a non-plane face, but a face defined from the surface of the cylinder whose radius is R, that in the figure is represented from the overall rectangle of the plane face.
After having assigned the radius R of curvature of the cylinder, you can find two cylinders and 4 arcs, passing through the Points P01 and P1 (left side of the figure):

- two arcs shown with Cw rotation ((o1, o2)
- two arcs shown with Ccw rotation (a1, a2).

In our example, the assignment plan of the arcs is perpendicular to the XY plane shown for the face.

For each of the 4 arcs you can find a curved surface, that correspond to the part of the cylinder found by the arc. You should now choose the arc you need.
Because of the difficulty in defining a precise sense of rotation in the space, we do not take under consideration such a direct choice. Let's assign the choice criteria in two points:

1. development of the arc from the part of the Z axis (figure a1,a2) of from the opposite part (figure o1,o2)
2. arc with shorter (figure a1,o1) or longer length (figure:a2,o2).

According to this view, to create a curved face you should:

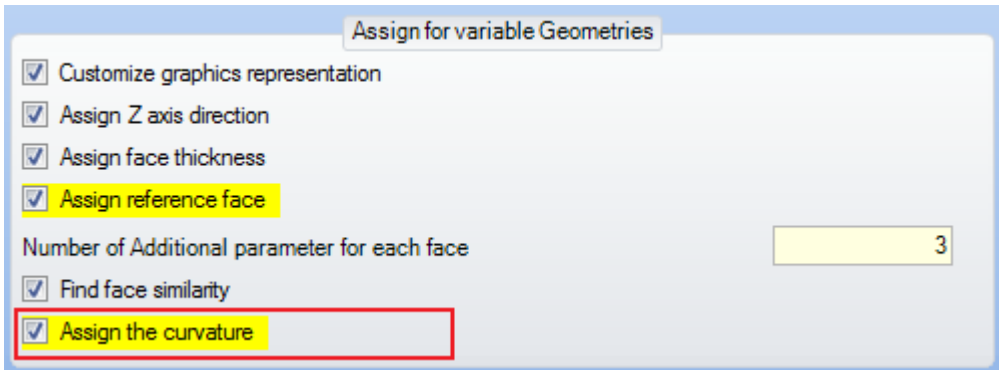
- define the usual Cartesian coordinate system (P0-P1-P3) and the direction of the Z axis, this whole as for a plane face;
- set a R radius of curvature: radius of the arc for the points P0-P1 (in the xz plane of the face);
- give a criterion to choose the arc among the 4 possible arcs: concordant/discordant with respect to the Cartesian coordinate system and the choice of the arc length.

A null R radius of curvature corresponds to the plane face.
If the R radius of curvature is non-null, the curved face is correctly assigned if (R*2.0)>=(P0-P1).
The plane face on which a curved face is constructed is shown as construction basis and provides the geometric solution of the face in case of assigned curvature incorrectly assigned

3.1 Enabling in configuration

The use of the functionalities concerning the management the curved faces must be specifically enabled from HW key besides the *Professional* functionality.

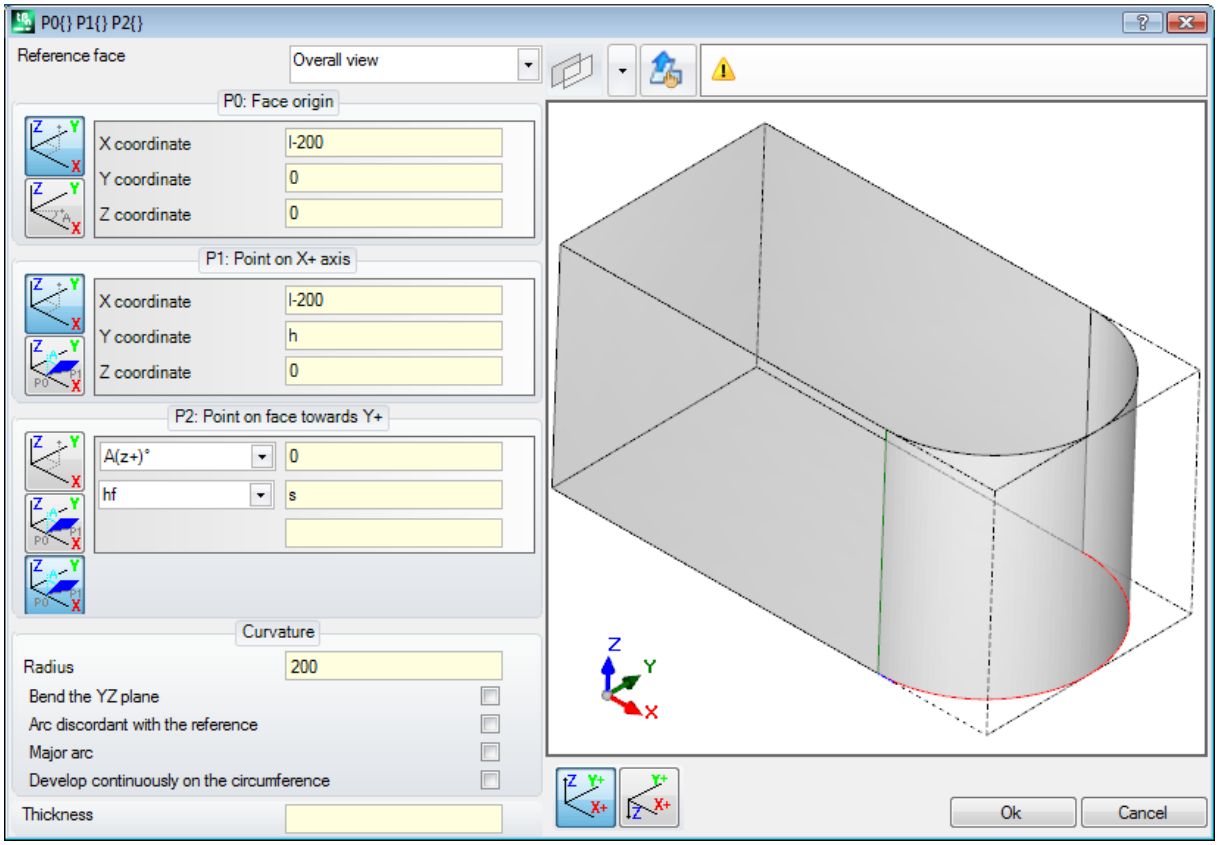
Let's see how to activate the functionalities. In TpaCAD, start the Configure TpaCAD command, from the menu , then select the Geometry piece page, in the Piece setting section:



In the lower part of the page, please see the option Assign the curvature.
The activation concerns the assignment of Fictive or automatic faces.

3.2 Creating curved faces

Let's see how the assignment window of the fictive faces changes.

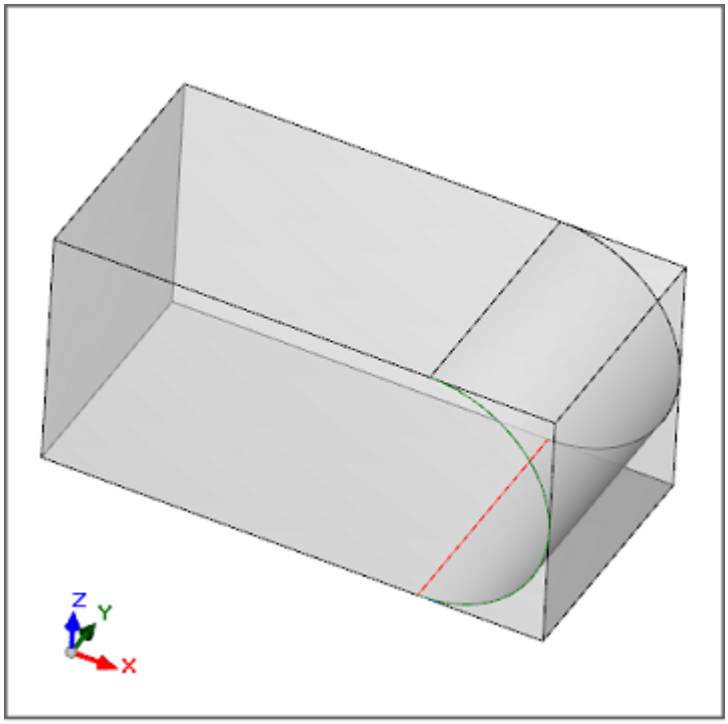


The fields relating to Curvature are shown in the lower part of the window:

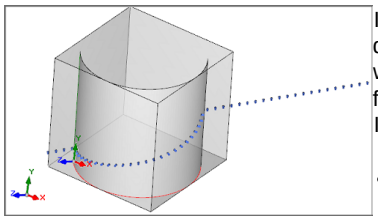
- Radius: it sets the radius of curvature and allows the parametric programming. Set the field and check select its associated box to enable the curved face recognition:

- Curve the YZ plane: select to assign the curvature on the YZ plane of the construction basis plane. The example in the figure assigns the curvature on the XZ plane; in this way the X axis in represented along the curved part (red axis), while the Y axis of the face (green) remains linear. The length of the face (If) is now calculated equal to the length of the determined arc.

The application of the curvature on the YZ plane brings the Y axis of the face along the curved part, while the X axis remains linear. The height of the face (hf) is now calculated equal to the length of the determined arc. The figure shows how the development of the face changes by selecting the option Curve the YZ plane:



- Discordant arc with respect of the Cartesian coordinate system: it selects the arcs that develop on the side opposite the direction assigned for the Z+ axis of the base Cartesian coordinate system;
- Major Arc: it selects the major developed arc;
- Continuous development on the circumference: select to extend the curved continuous development of the face. The figures show how the field status changes the face interpretation. More specifically: a hole repetition with inclination with respect of the X axis of the face, final and negative initial X coordinate higher than the linear length of the arc is programmed.



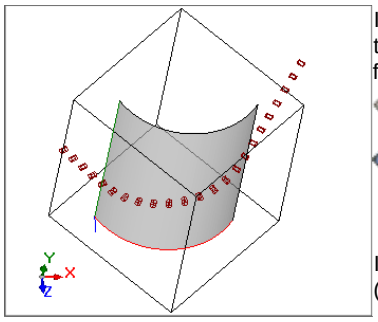
If the entry is not selecdted and if there are no further selections: the development of the curve is limited to the assigned arc. In the figure you will find a Cartesian coordinate system corresponding to the origin of the face.
In the figure (with curvature of the XZ plane):

- X coordinates from 0.0 to the length of the arc determine some positions within the curved development of the face;

- X negative coordinates determine some positions within the half-plane that continues the basic construction plane on the left side of the figure;

- X positive coordinates besides the length of the arc determine some positions within the half-plane that continues the basic construction plane on the right side of the figure;

That is, the face continues a plane development outside the development of the construction arc.



If the entry is not selected, it is possible to modify its behaviour outside the development of the construction arc, by operating on the following fields:

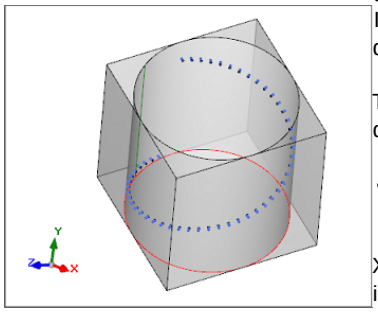
Entry tangent: selection of the entry half-plane of the curved face

Exit tangent: selection of the exit half-plane of the curved face

In the case represented in the picture both the entries are selected (always with curvature of the XZ plane):

- X-coordinates starting from 0.0 to the length of the arc find positions within the curved development of the face;
- X-negative coordinates find positions within the half-plane that continues in the part of the picture, now in entry tangency to the arc.
- X- positive coordinates beyond the length of the arc find positions within the half-plane that continues in the right part of the picture, now in entry tangency to the arc.

This means, that outside the development of the arc construction, the face carries on a plane development, that can be in continuity of tangency with the curved development of the face.



If the option is checked, the face determines a cylindrical surface; the development of the curvature is now extended to 360°.

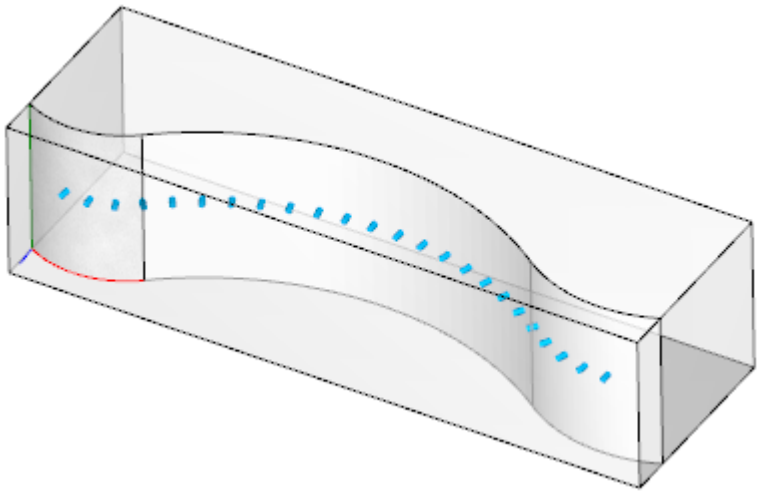
The figure marks the positioning of the holes on a continuous helicoidal development:

- the length of the face measures now the whole circumference (2*pi*radius);

X negative coordinates or coordinates beyond the length of the face are interpreted on a continuous circular development.

4 Surfaces

Consideration is being given to the possibility of creating a surface made of more elements similar to single, flat or curved faces, placed side by side along one direction.
Programming a working can be attributed to a composed surface.
In the picture you will see an example of a surface made of 3 curved elements placed side by side along the X axis of development of the surface: the repetition of the holes adapt to the resultant of the three elements.



The parts of which a surface is made verify a geometric continuity and can be reduced to a flat sheet:

- the direction of the three points is invariant;
- they must verify a geometric "continuity" along X or Y (in the picture: along X);
- it is possible to include flat and curved geometries. In case of curved geometry, the curvature must be assigned along the continuity axis;
- the development of the surface can close on the first element ("circular" continuation).

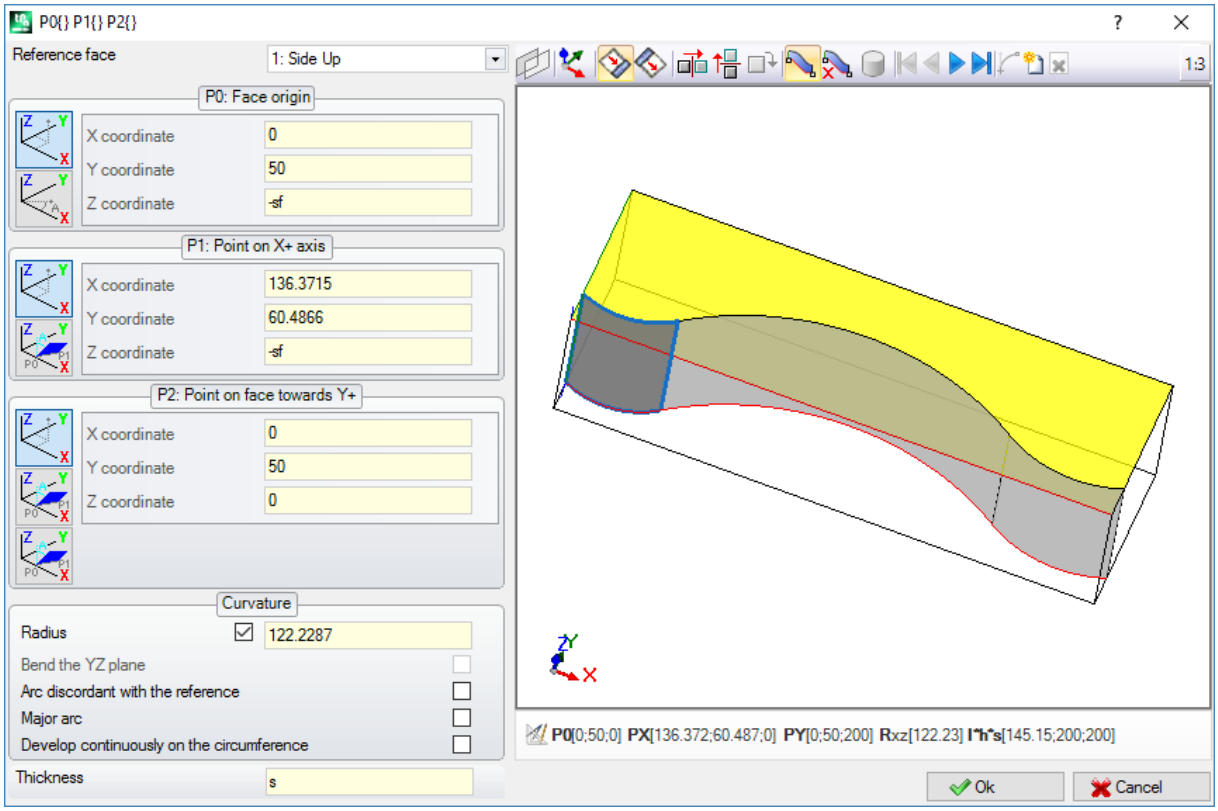
Maximum number of element that can be assigned in a surface: 300.

4.1 Enabling in configuration

The use of the functionalities requires a specific activation from HW key besides the Professional functionality.
To activate the functionalities, select from the menu the Configure TpaCAD command, then select the page Piece geometry , in the Piece settings section.
In the margin at the bottom of the page, please enable Surface.
The activation concerns the assignment of Fictive faces.

4.2 Creating surfaces

Let us see how the assignment window of the fictive faces changes:



The top right bar shows new buttons:

 : select to manage a surface. In this case the remaining buttons of the bar can seem to be enabled. If you deselect the button, a warning will inform that all the elements that follow the first one will be lost; confirmation or cancellation of the command will be required.
Select the button to modify the typology of the fictive face on the surface; it will be possible to add some element to the current one. By selecting the button, an element is automatically added, confirming the final position of the base element. If needed, we recommend to change the selection of the development axis before each other change of the elements of the surface.
If the surface is assigned, the three buttons of the tool cannot be used anymore.

The next button select the development axis of the surface, selecting between X and Y. The active selection is marked by the change of the image of the button:

 : the development axis is X (see the picture).

 : The development axis is Y.

 : this is a display-only button. If selected, it indicates that the surface is closed, that is that develops in a continuous way.

 : the group of the buttons allows you to scroll the elements of which the surface is made.

By positioning on the first element it is possible to carry out the general assignments.

- reference face
- x-y-z reference

- face thickness.

: select this button to keep the current element as tangent to the next one. . The current element is resolved by a plane face geometry, according to the data set in the window. If the solution is valid, the geometry is reevaluated in tangency continuity:
- if the element is plane, its orientation only is modified and its programmed dimension remains the same;
- if the element is curved, the radius is calculated automatically (the radius cannot be programmed).

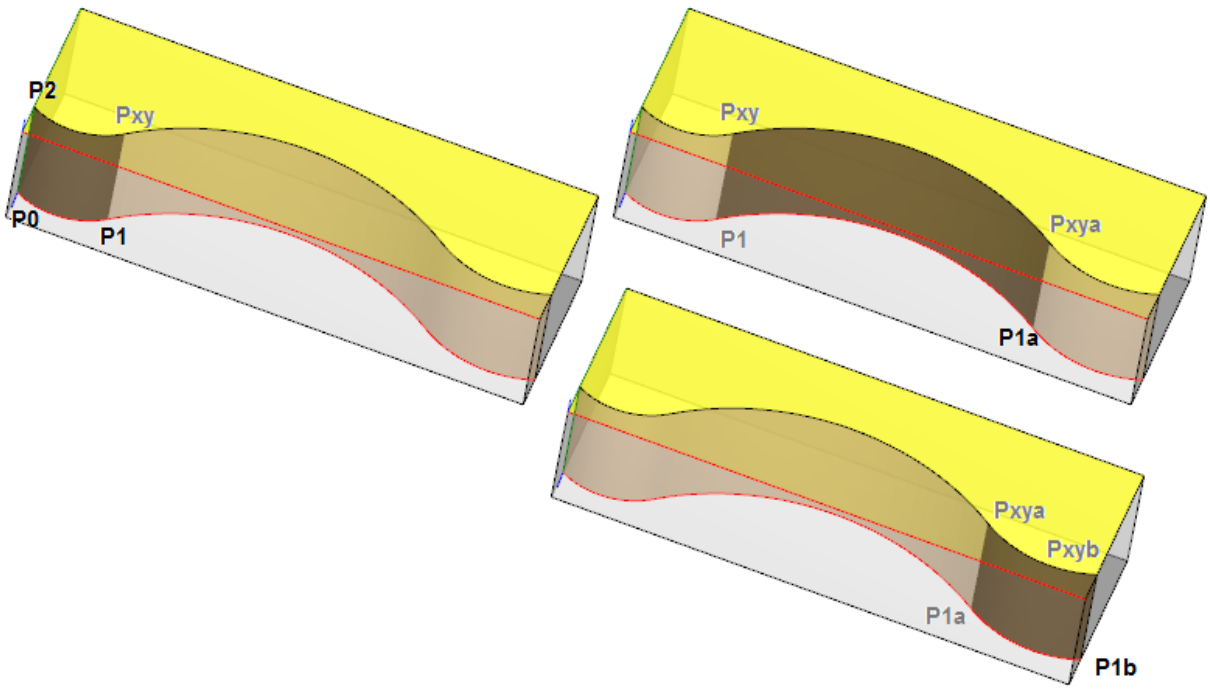
The button is not enabled on the first element of the surface.

Use the two buttons to modify the list of the elements, by adding or removing respectively the current element:
- an element is entered after the current element, never before the first element assigned
- the elimination affects the current element and in any case not the first one.

If the elimination of elements reduces the list to one element only, the selection of the Surface button is automatically removed,
1:3 the digits at the right end of the bar show the current element over the total (see the picture: ther first of three).

The setting of the first element of the surface correspond exactly to the assignment of a normal fictive face, while the following elements are partly bound by the geometry of the first element and by the development axis.

The picture shows the three elements of the example surface and the programmable points:
- the first element programs the three (P0, P1, P2) points. The Pxy points marks the XY plane of the element. The development axis is X; so, the surface continues along the X axis of the first element.
- the second element programs the P1a point, that defines the limits of the X axis. The origin of the segment coincides with the P1 point and the Y axis is determined by Pxy; the Pxya point marks the limits of the XY plane of the element. The P1a point must be assigned so that (P1-P1a) is perpendicular to the (P1-Pxy) segment;
- the third element programs the P1b point, that defines the limits of the X axis. The origin of the segment coincides with the P1a point and the Y axis is determined by Pxya; the Pxyb point marks the limits of the XY plane of the element. The P1b point must be assigned so that (P1a-P1b) is perpendicular to the (P1a-Pxya) segment;



The curvature of each element can be assigned along the axis only.
So, the XY plane of the surface is determined on the four extreme points, as follows:
- P0: surface origin
- P1b: extreme point along the X axis

- P2: extreme point along the Y axis
- Pxyb: extreme on the XY plane

The distance (generally curved) between the (P0-P1b) points corresponds to the length of the surface (If).
The distance between the (P0-P2) points corresponds to the height of the surface (If).

The behaviour before the point P0 and over the point P1b, fully in line with the case of an individual curved face, is assigned in accordance to the selections of the entries:

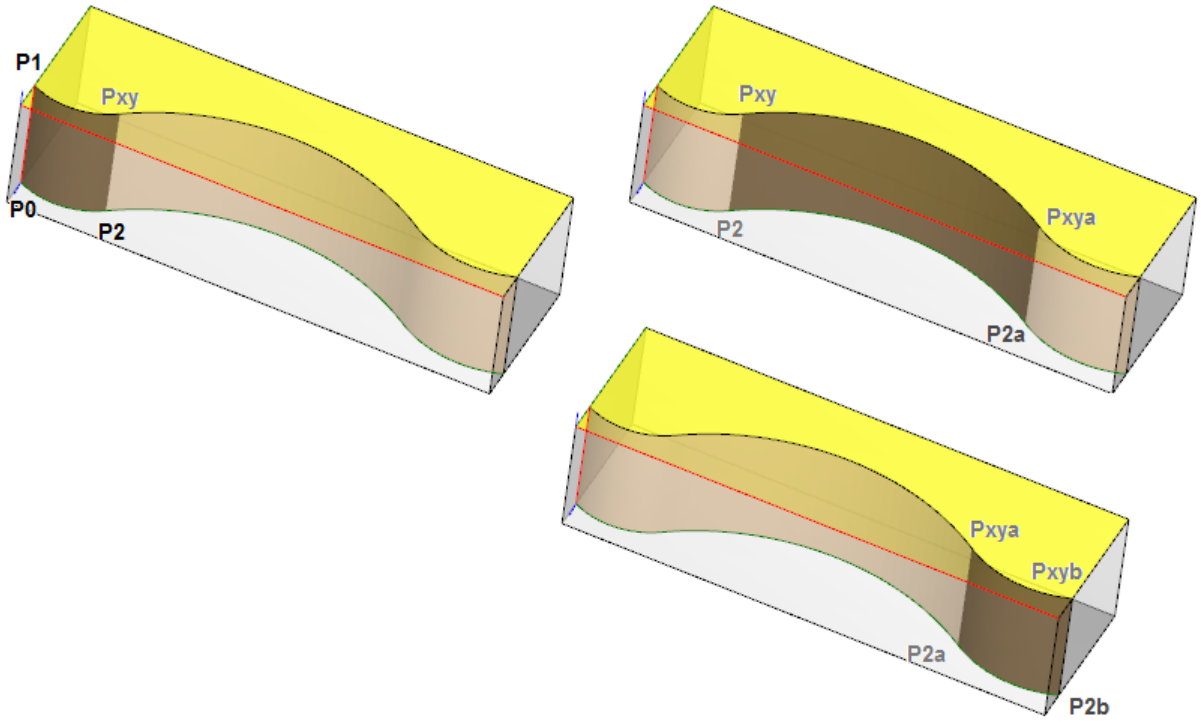
 Entry tangent : selection related to the ingoing half-plane of the surface (significant if the first element of the surface is not plane)

 Exit tangent : selection related to the outgoing half-plane of the surface (significant if the last element of the surface is not plane)

In the event of a curved element you cannot select Develop continuously on the circumference.

In the case of the axis with a development along Y, the elements next to the first one program the extreme point along Y.
The picture shows the three elements of the example surface, but with development along Y:

- the first element programs the three (P0, P1, P2) points. The Pxy points marks the XY plane of the element. The development axis is Y; so, the surface continues along the Y axis of the first element.
- the second element programs the P2a point, that marks the limit of the Y axis. The origin of the element coincides with P2 and the X axis is determined by Pxy; the Pxya point marks the limits of the XY plane of the element. The P2a point must be assigned so that (P2-P2a) is perpendicular to the (P2-Pxy) segment;
- the third element programs the P2b point, that marks the limit of the Y axis. The origin of the element coincides with P2a and the X axis is determined by Pxya; the Pxyb point marks the limits of the XY plane of the element. The P2b point must be assigned so that (P2a-P2b) is perpendicular to the (P2a-Pxya) segment;



The curvature of each element now can be assigned along the Y axis only.
The XY plane of the surface now is identified on the four extreme points, as follows:

- P0: surface origin
- P1: extreme point on the X axis
- P2b: extreme point along the Y axis
- Pxyb: extreme on the XY plane

The distance between the (P0-P1) points corresponds to the length of the (If) surface.

The distance (generally curved) between the (P0-P2b) points correspond to the height of the (hf) surface.

The behaviour before the point P0 and over the point P2b, fully in line with the case of an individual curved face, is assigned in accordance to the selections of the entries:

 Entry tangent : selection related to the ingoing half-plane of the surface (significant if the first element of the surface is not plane)

 Exit tangent : selection related to the outgoing half-plane of the surface (significant if the last element of the surface is not plane)

4.3 Creating surface from geometry

The Create surface from geometry command is recalled in the Advanced *group* in the Apply tab.
This command is managed in face view with non-empty face program and current working of a simple profile (made of elements of arc or line type).
This command is not active in view of a curved face or a surface.

This command adds a surface fictive face to the current profile.
Assigned arcs in a plane different from xy or path elements become linear segments again.
Circles in the xy-plane are split into two semicircles.

Tecnologie e Prodotti per l'Automazione

Via Carducci 221
I - 20099 Sesto S.Giovanni (Mi)
Tel. +390236527550
Fax. +39022481008

www.tpaspa.it

info@tpaspa.it

9/14/2017