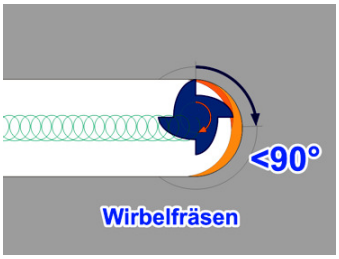
High chip volume with Trochiadal Speed Cutting

While in the case of conventional milling of a keyway the wrap angle has an amount of 180°, it can be limited to <90° depending on the spiral feed (horizontal spiral rise). This is made possible by a smaller diameter, which is swirling at high speed.



The horizontal infeed occurs in loops along a center path. The chip profile at the cutting edge is relatively equal and independent of horizontal milling direction changes. This results in a lower machine and tool load, which makes it possible to drive significantly higher cutting values.

Because of the smaller wrap angle, there is less heat generation. In addition, the chip load is reduced during Trochiadal Speed Cutting. Since the milling cutter does not completely fill the keyway, a chip accumulation and re-cut is almost impossible. This is particularly significant for keyways which are deep in relation to their width. For conventional milling, a max. Cutting depth of 1..1.5 * FrD (FrD = tool tip diameter) is customary. Different in Trochiadal Speed Cutting. By avoiding of a full cut are cutting depth of > 2 x FrD reachable, even without specially developed cutters.

During Trochiadal Speed Cutting, machining can be carried out with a high cut depth and a high cutting speed, and the roughing process can be accelerated. For the user, this means shorter machining times, better tool utilization and a noticeably lower tool wear. Since only one tool different keyway with or pocket sizes produces, the tool has to be exchanged less frequently.

Trochiadal Speed Cutting brings cost and profitability advantages:
- The whole cutting edge length is engaged, with this higher cutting effort and a higher chip volume;
- the tool life is increased;
- less vibrations; the machine mechanics and the workpiece are conserved;
- the processing times are shortened.

In the **CAM**, complex milling functions are offered under '**Whirl/Spiral milling cycles**'. Available are a Trochiadal basic motion ('**Whirl path**') and several Trochiadal - contour adaptations ('**Whirl internal cut**', '**Whirl mill around**' '**Whirl pocket**').

Trochiadal Speed Cutting only makes sense for full cut (keyways). During Trochiadal Speed Cutting, the tool moves spirally sideways into the material (see above). The disadvantage is the relatively large idle cut of approx. 50% on the backward spiral movement. For the surface clearing (in pockets), it is therefore useful to produce only the first keyway by Trochiadal Speed Cutting. An island which has already been milled around can be processed more economically with the previous clearance methods (for example contour-parallel).

The functions '**Whirl internal cut**', '**Whirl mill around**' and '**Whirl pocket**' respect this and only perform the first full cut with the Trochiadal Speed Cutting. The contour then can be cleared and smoothed. To start the Trochiadal Speed Cutting a vertical starting spiral can be selected into the material.

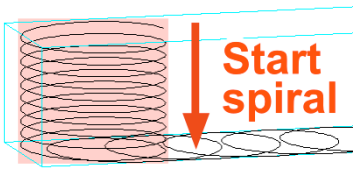
Something out of the frame falls the '**Spiral pocket**'. This is a highly efficient pocket processing with a starting spiral, clearing spiral and finishing path. Because of the special motion, this function is only suitable for simple, regular geometry's.

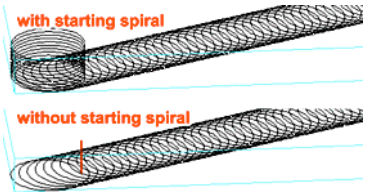
Whirl/Spiral milling cycles	Higher chip volume with Trochiadal Speed Cutting!

With these cycles, the innovative and highly productive Trochiadal Speed Cutting can be used on all milling machines for economical machining with large cutting depths and in difficult-to-process materials. The milling path calculations for the effective machining are always carried out only for a route or a contour. The milling paths are stored together with an starting spiral and the tool data in the specified target layer.

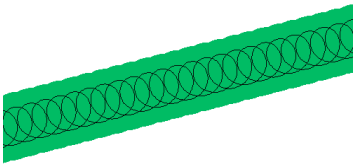
The functions '**Whirl internal cut**', '**Whirl mill around**' and '**Whirl pocket**' are designed for simple contours without nesting or overlapping. Complex contours must be entered manually or conventionally edited.

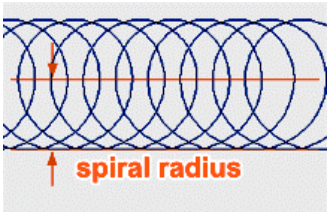
The functions use several individual processes, which are combined to complex milling cycles. All movements are executed with the same tool.



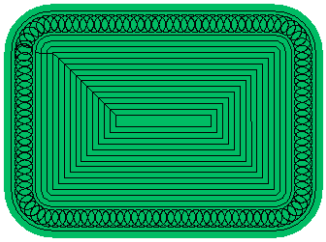


Starting spiral (approach movement, dip spiral).
For easier dip in the solid material a 3D start spiral can be selected.

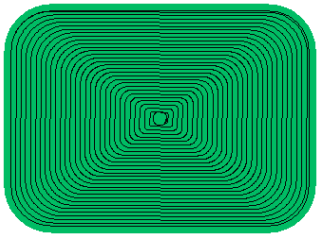




Whirl spiral (Trochiadal Speed Cutting).
For the first keyway, the Trochiadal Speed Cutting is used.

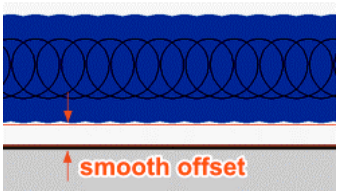


Wirbel Tasche



Spiral Tasche

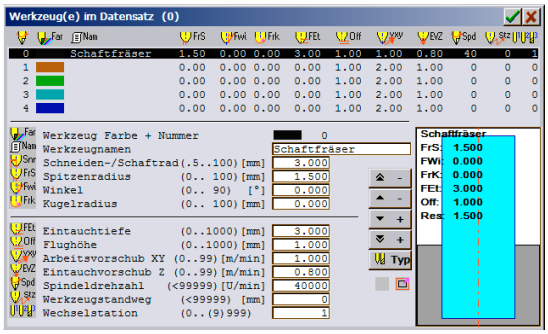
Clearing.
Pockets are cleared by conventional procedures (exception: 'Spiral pocket').



Contour smooth.
The workpiece is sized to the final dimension.

Tool input (see 'General instructions manual').
The Trochiadal Speed Cutting are performed with a cylindrical tool in 2D (exception the starting spiral). Each tool requires a geometry (Cutting radius, tip radius) and valid technology date (dip depth, safety height, cut feed rate XY, cut feed rate Z, spindle speed and tool changer #). If the tool data is incomplete or incorrect, wrong milling data can be generated.

All milling paths are executed with a tool and stored into the specified target layer.

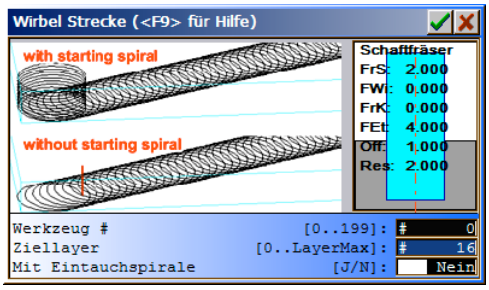


Tool

Target layer

Store milling objects into a milling layer (target layer) (see 'General instructions manual').
The calculated milling paths are stored in the specified target layer. The milling order should be respected. The milling objects in the target layer can be edited later with 'CAM. CAM Edit'. This technology is ideal for extensive, complex milling work and the combination of all milling processes. The milling work is completely prepared at the programming station. With the functions 'Cam . Norm/Zoom projection' and 'Volume aspect' the whole work can be controlled.

Whirl path (milling keyways)



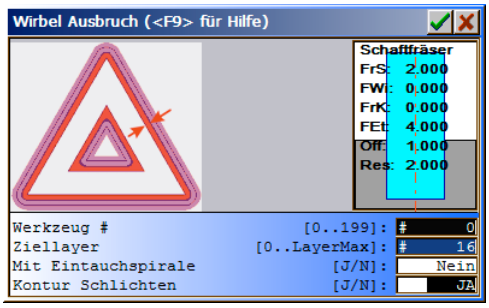
The '**Whirl path**' is a basic function for milling keyways. Caused by the milling process no smooth keyways are archived. For smooth keyways the required finishing paths must be entered conventionally / manually.

Tool # [0..199]:
Selection of a milling tool for complete machining.

Target layer [0..layer max]:
The milling path is saved into the selected target layer.

With starting spiral [Y/N]
To start the material a start-up spiral can be generated. The data of the starting spiral ('spiral radius', 'vertical spiral incline') are defined in the 'Whirl Parameter'.

Whirl internal cut (contour inner machining)



The contour is cut out with the whirl milling function and smoothed at the end.

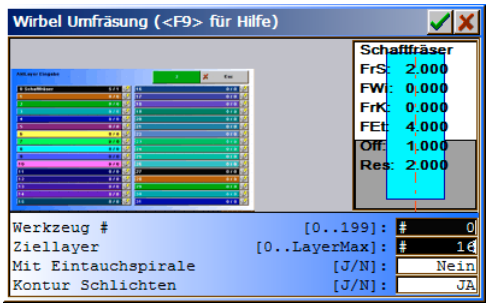
Tool # [0..199]:
Selection of a milling tool for complete machining.

Target layer [0..layer max]:
The milling path is saved into the selected target layer.

With starting spiral [Y/N]
To start the material a start-up spiral can be generated. The data of the starting spiral ('spiral radius', 'vertical spiral incline') are defined in the 'Whirl Parameter'.

Contour smooth [Y/N]:
The cut out will be smoothed.

Whirl mill around (contour outer machining)



The mill around is performed with the whirl milling function and smoothed at the end.

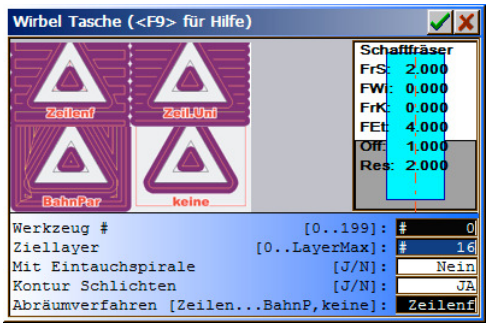
Tool # [0..199]:
Selection of a milling tool for complete machining.

Target layer [0..layer max]:
The milling path is saved into the selected target layer.

With starting spiral [Y/N]
To start the material a start-up spiral can be generated. The data of the starting spiral ('spiral radius', 'vertical spiral incline') are defined in the 'Whirl Parameter'.

Contour smooth [Y/N]:
The cut out will be smoothed.

Whirl pocket	(contour inner machining with clearing)



The contour is cut out with the whirl milling function, cleared and smoothed at the end.

Tool # [0..199]:
Selection of a milling tool for complete machining.

Target layer [0..layer max]:
The milling path is saved into the selected target layer.

With starting spiral [Y/N]
To start the material a start-up spiral can be generated. The data of the starting spiral ('spiral radius', 'vertical spiral incline') are defined in the 'Whirl Parameter'.

Contour smooth [Y/N]:
The cut out will be smoothed.

Clearance procedure [Line..ContPar.,None]:
The resulting areas will be cleared.

line formed: Create clearance lines with the entered angle. The lines are connected as far as possible.

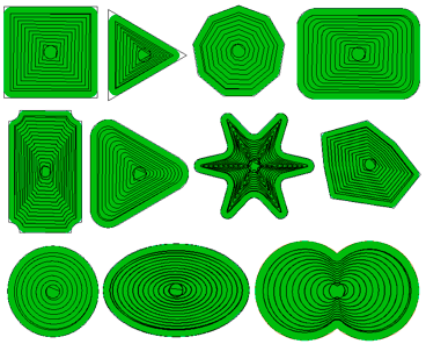
line single dir.: Create clearance lines with the entered angle. The lines always trend in one direction.

contour parallel rad.: A clearance path is produced by equidistant milling contours (radius - island engraving).

none: No clearing paths are calculated.

Spiral pocket	(contour inner machining with clearing)

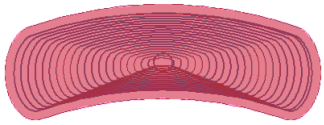
1



This function is only suitable for simple shapes whose center (start of machining) lies in the shape center (e.g. triangle, rectangle, circle, ellipse, etc.). Irregular shapes result in unfavorable spirals. If the center is outside the area, no clearing spiral can be calculated.

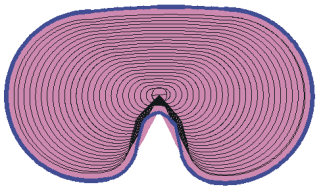
1: Suitable forms for 'Spiral Pocket'.

2



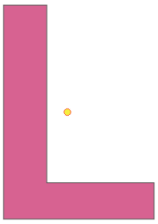
2: Unfavorable spiral distribution, but still without error.

3

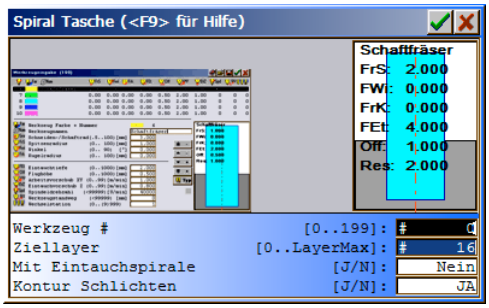


3: Unsuitable: Contour is demaged.

4



4: Not suitable: Center is outside.



The pocket is always cleared from the center with a spiral. This results in very economical milling paths, without empty paths. The pocket is cleared with the spiral milling function.

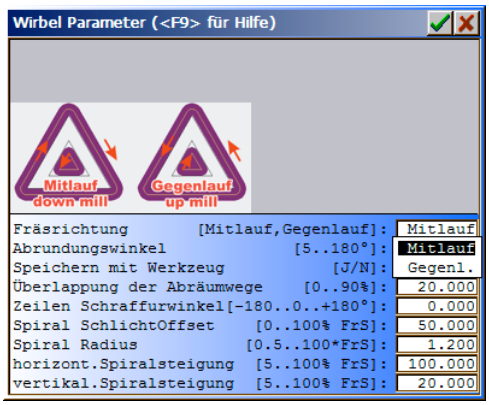
Tool # [0..199]:
Selection of a milling tool for complete machining.

Target layer [0..layer max]:
The milling path is saved into the selected target layer.

With starting spiral [Y/N]
To start the material a start-up spiral can be generated. The data of the starting spiral ('spiral radius', 'vertical spiral incline') are defined in the 'Whirl Parameter'.

Contour smooth [Y/N]:
The cut out will be smoothed.

Whirl parameter



Milling directn. [climb, up-milling]:
Set the milling direction for '**Whirl internal cut**' '**Whirl mill around**' '**Whirl pocket'.**
Down mill: Milling clockwise (standard). Typical setting for the majority of materials.
Up mill: Milling toward clockwise.

Round-off angle [5..180 °]:
An offset calculation is performed for all contours. Outside corners of the offset paths are rounded with the rounding angle.
Recommended setting 20..30 °

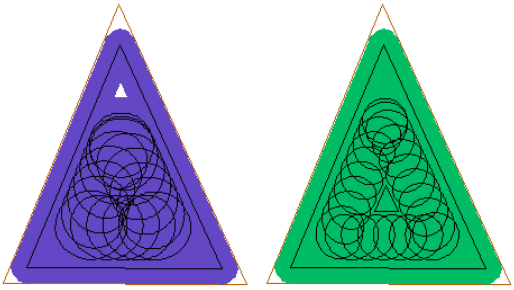
Save with tool [Y/N]:
'Yes': Saves the specified tool to the milling paths. If '**No**', the result is saved without tools.
Note: Milling data should always be saved with the tool.

Overlapping of clear. path [0..90%]:
Overlap of the clearance paths. In order for pockets to be cleared cleanly, the clearance paths should overlap. The input is in % of the tip radius. This setting is used only for the 'Whirl Pocket' and 'Spiral Pocket'.
Typical settings 20 .. 30%.

Line hatching angle [-180..0..+180°]:
Direction of the line-shaped hatching in the mathematically positive rotation. 0 degree gives horizontal, 90 ° gives vertical hatching lines. This setting is only used for the 'Whirl Pocket'.

Spiral smooth offset [0..100% FrS]:
For the whirl contour, a smooth offset (minimum distance to the milling contour) is redeemed. The final dimension is reached after smoothing.

Spiral radius [0.5..100*FrS]:
Radius of the whirl spiral. Since approximately half of the spiral path is performed without milling, the expansion of the spiral should be kept as small as possible. On the other hand, a larger spiral radius provides for a more favorable wrap angle. A factor (0.5..100) to the tool tip radius is always entered.



1) **Spiral radius** = 2.0 2) **Spiral radius** = 1.4

If the contour is smoothed with the same tool, care must be taken that the spiral radius for narrow points and narrow corners is max. 1 (* FrS). For rounded shapes (circle, ellipse, ..) without constrictions, the spiral radius can be set higher.
Typical values are 1..2 (* FrS).

In peak corners and in narrow places the spiral milling must be kept too far from the contour in case 1.
In the example, a tool radius FrS = 1.5mm was selected. In case 1, a spiral radius = 2.0 remains an unprocessed island. In case 2, the pocket is processed completely with spiral radius = 1.4.

Horizont. spiral incline [5..100% FrS] (spiral feed)**:**
Distance of the sequenced spirals. A percentage value of the tool tip radius is entered. Large values result in a large chip and are increasingly stressing the tool during material contact also result smaller empty paths of the spiral circles.
Typical values are 20..50% (FrS).

Vertical. spiral incline [5..100% FrS]:
Z - Step width of the dipping (starting) spiral. A percentage value of the tool tip radius is entered.
Numerous values result in a large chip load during the dip procedure.
Typical values are 10. 50% (FRS).
Note: The tool must be suitable for slanting dip procedures.