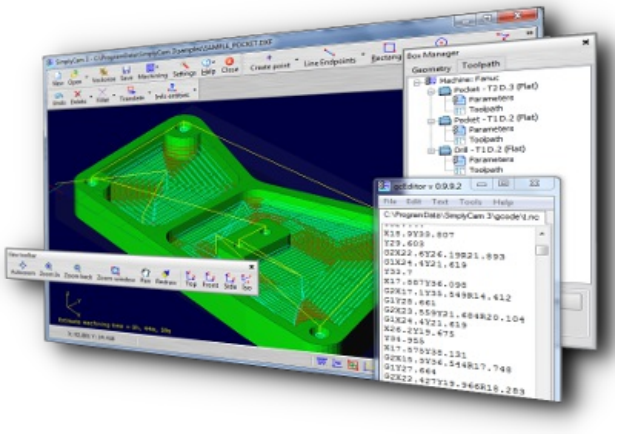
SimplyCam V3. Documentation

Introduction to SimplyCam

SimplyCam is a simple Cad/Cam system that creates toolpaths (G-Code). The geometric part can be created internally, with the available tools or imported from external files.

The main features of SimplyCam are:



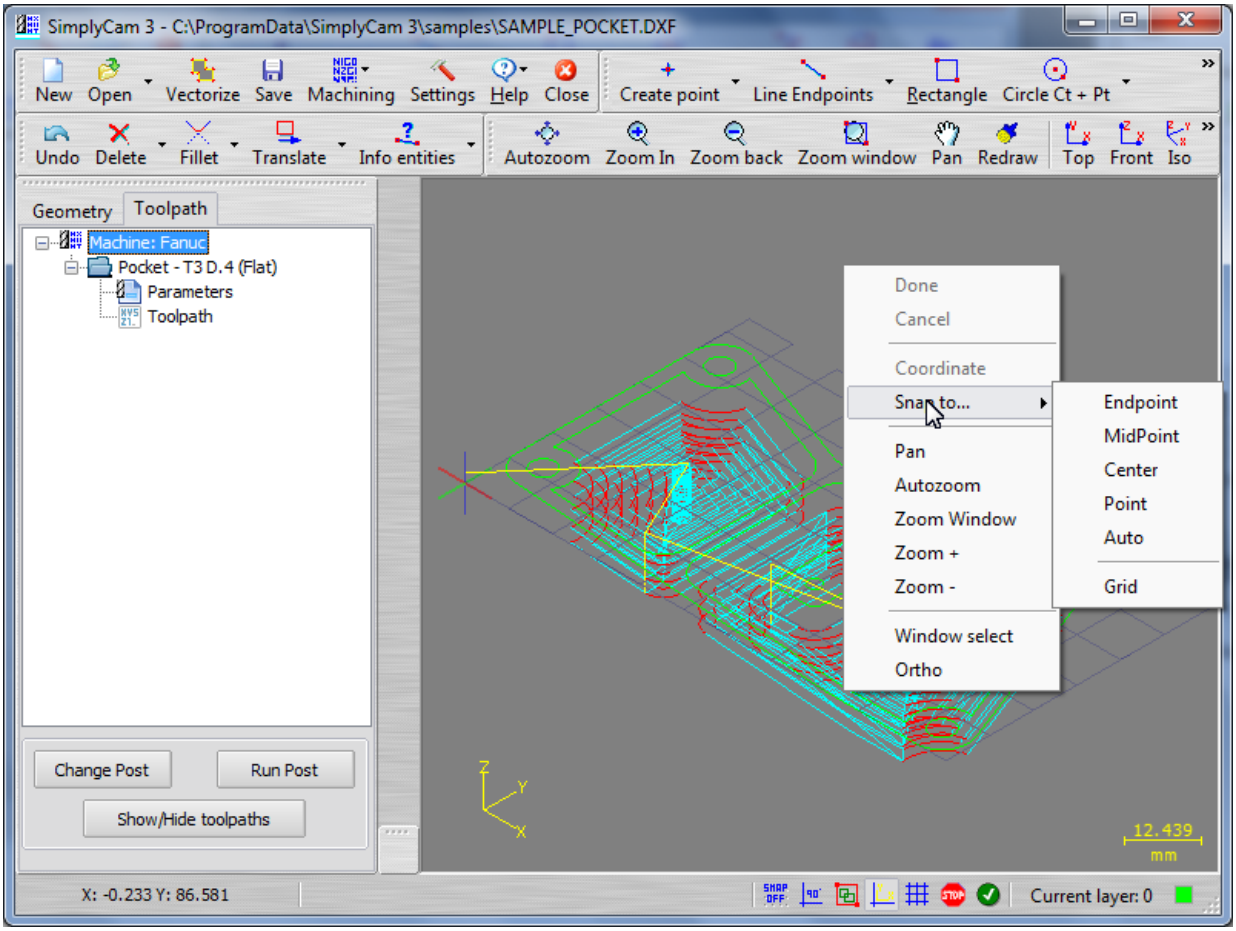
- Opening, creating, editing and saving geometry in industry standard DXF format.
- Opening Svg file.
- Vectorization with median line or outer edges mode of the raster images (Bmp and Jpeg).
- Creating geometry with the Line, Rectangle, Arc, Circle, Ellipse, Polyline and Spline functions.
- Changing the geometry with the Fillet, Trim, Break, Merge, Split, Extend and Explode block functions.
- Transform entity with Scale, Rotate, Mirror, Offset and Translate commands.
- Conversion of any TrueType font into vector.
- Generation programs for CNC machines such as milling machines, Router, Water Jet, Laser, Plasma and Lathe.
- Operations Manager to control all elements of the toolpath
- Great number of post-processor modifiable included, as for example:
  AutoGrav, Fanuc, Fadal, Haas, Heidenhain, Iso, Mach2 / 3, MaxNc, Selca, Siemens, ShopBot, Tecno-Isel, TurboCnc, ...
- Opening, editing and simulation of the programs in G-Code format.
- Opening/Saving geometry and toolpath with modification and recalculation.
- Interface with toolbars and panels draggable.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

SimplyCam interface.

Below the screen of SimplyCam. Click on the screen to the interface description.



Graphic area

This is the area where SimplyCam displays the images, the geometry and the toolpaths.

Toolbars

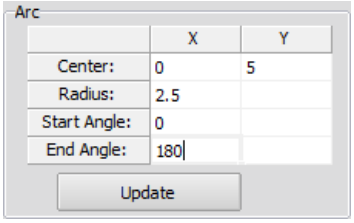
The Toolbars are a collection of functions accessible through the buttons.
You can undock, dock, move and hide the SimplyCam toolbars.
More information ....

Status Bar

The Status bar is located at the bottom of the Simplycam window.
It provides quick access to: Cursor position info, Prompt area, Snap, Ortho mode, Windows select, Axis, Grid, Layers.
More information ....

Geometry Manager

List each geometric entity of the current file. Use the Geometry Manager to edit the entity.



Toolpath Manager

List toolpaths of the current file. Use the Toolpath Manager to create, edit, simulate and post each operation.
More information ....

Right-click menu

The click of the right mouse button displays a drop-down menu. For example, during the execution of a drawing command, pressing the right mouse button displays the menu of Snap with the available options.

Layer

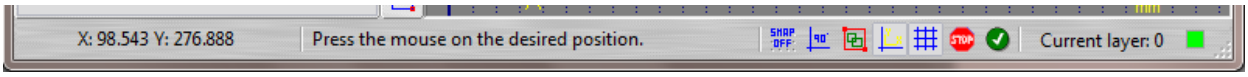
The Layers are used to separate and simplify access to geometric entities in a complex design.
To access the "Layers" dialog, click on "Layers" button in the status bar.
More information ....

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Status Bar

The Status Bar is located at the bottom of the SimplyCam application window.



Cursor Position

This area display the (x,y) position of the cursor in world coordinates.
If Snap options is active, the coordinates reflect the closest snapped point relative to cursor position.

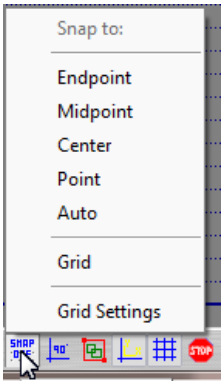
Prompt area

This area shows the messages and requests that SimplyCam sends to you.

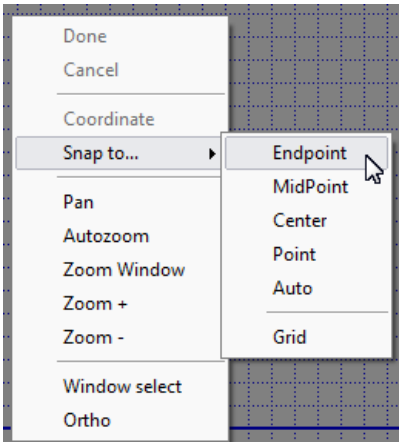
Snap (capture, hooking)

This tool ensures greater accuracy in defining the coordinates when creating entities.
The Snap tool recognizes the geometric characteristics of the entities closest to the cursor and captures the coordinates as End Point, Midpoint, Center, Point and Grid.

To set the Snap Mode press the "Snap" in the Status Bar.



You can also set the Snap Mode from the drop-down menu of the right mouse button.



SimplyCam identifies the snap point captured, in a dynamic way, using the distinctive symbols:

Snaps (capture) the nearest Endpoint of a line, an arc or a vertex of a polyline.

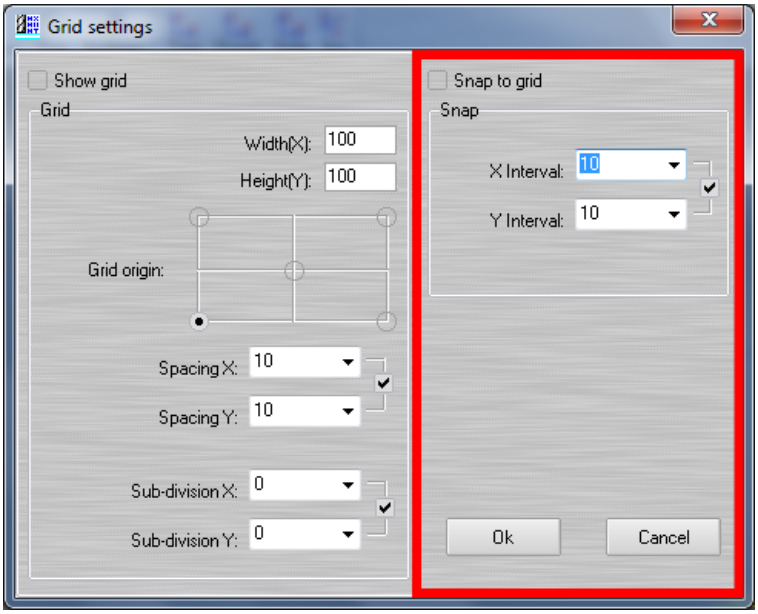
Snaps (capture) the nearest Midpoint of a line or arc.

Snaps (capture) the nearest Center point of an arc or a circle.

Snaps (capture) the coordinates of the nearest geometric entity Point.

Activating the Snap Grid option, the cursor can only select coordinates that belong to the range of the grid.
Spacing X and Spacing Y specify the snap spacing.

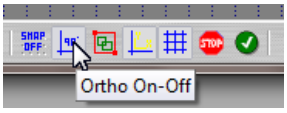
Mouse click on the "Grid Settings" item to define the spacing for the "Snap to Grid".



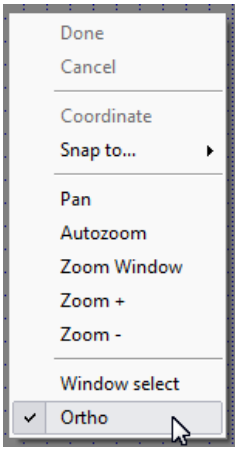
Ortho mode

The ortho mode (short for orthogonal) active, limits the movement of the cursor only in the horizontal direction and in the vertical direction.

Toggle On-Off mode by pressing the "Ortho" button in status-bar.



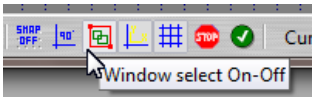
Also toggle On-Off ortho mode from the drop down menu of the right mouse button.



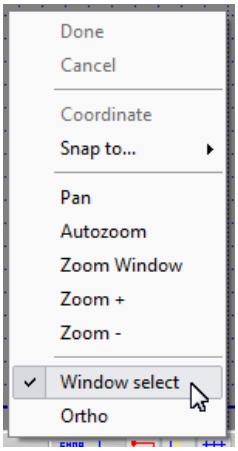
Windows select

The "Windows Select" allows you to select one or more entities within a rectangular area specified by two opposite vertices with the mouse.

Toggle On-Off mode by pressing the "Window Sel." button in status-bar.



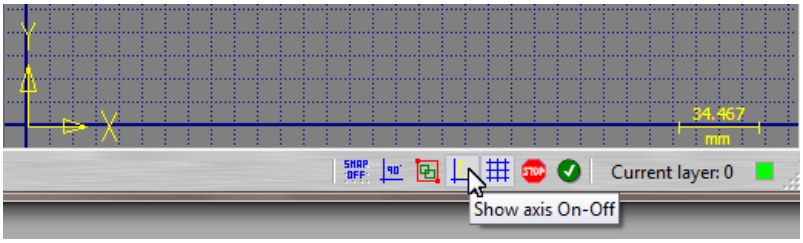
You can toggle On-Off "Window Sel." mode, also from the drop down menu of the right mouse button.



Axis button

This option show/hide in the bottom of viewport the following:

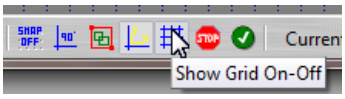
- The system origin
- The positive direction of X and Y axis
- The current scale video and the system unit



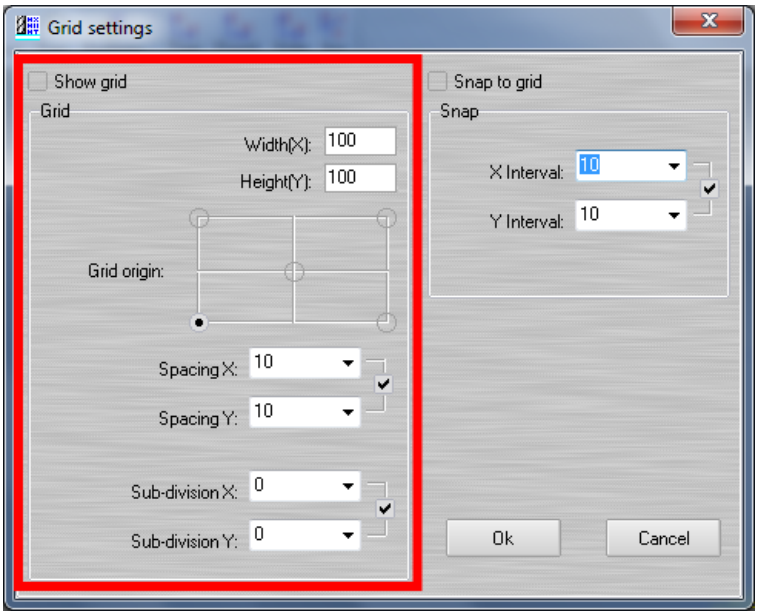
Visible Grid

This option draw a 2D rectangular grid on the viewport.
To use the Visible Grid:

Toggle On-Off mode by pressing the "Grid" button in status-bar.



Mouse click on "Grid Settings" item to define the visible grid spacing.

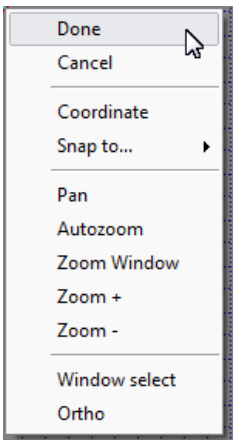


Done / Cancel

Many operation required the following user commands:

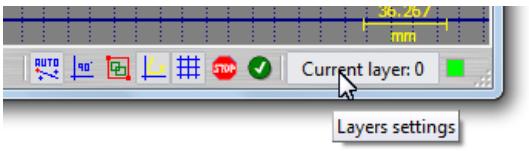
- Done: The operation is ended and accepted.
- Cancel: The operation is aborted and ignored.

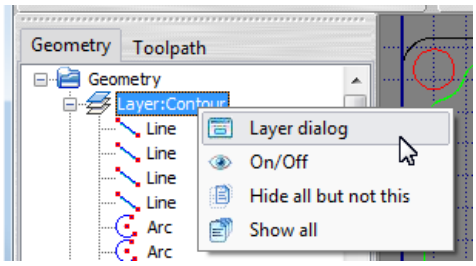
The action "Done / Cancel" can be quickly validated from the drop-down menu of the right mouse button.



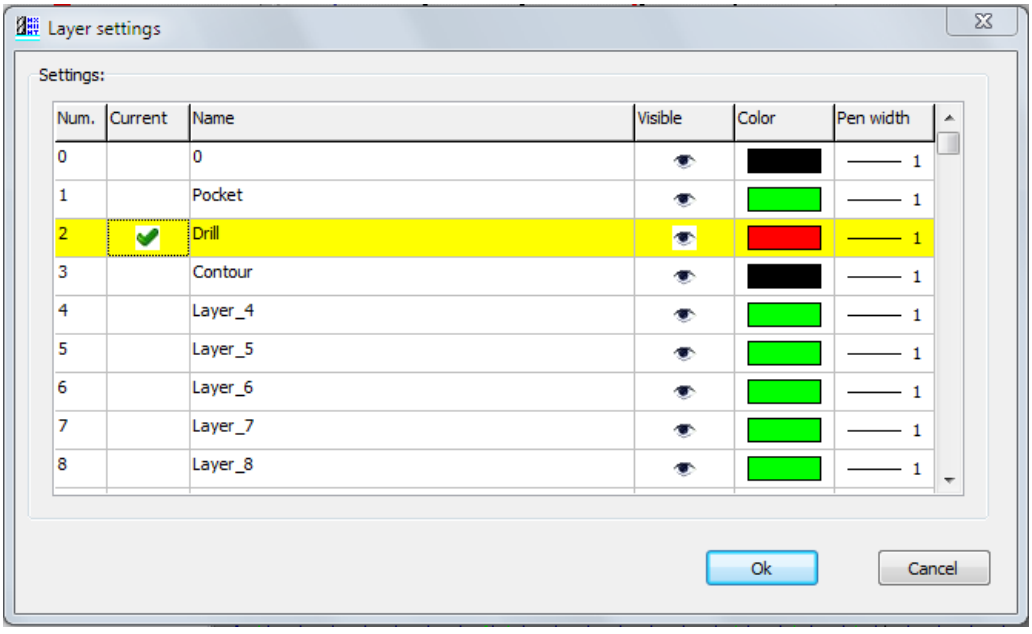
Layer

The Layers are used to separate and simplify access to geometric entities in a complex design.
Access the "Layer" dialog by clicking the "Layer Settings" button in the status bar or by clicking on the "Layer Dialog" item in the drop-down menu of the Geometry Manager.





Dialog Layer Setting:



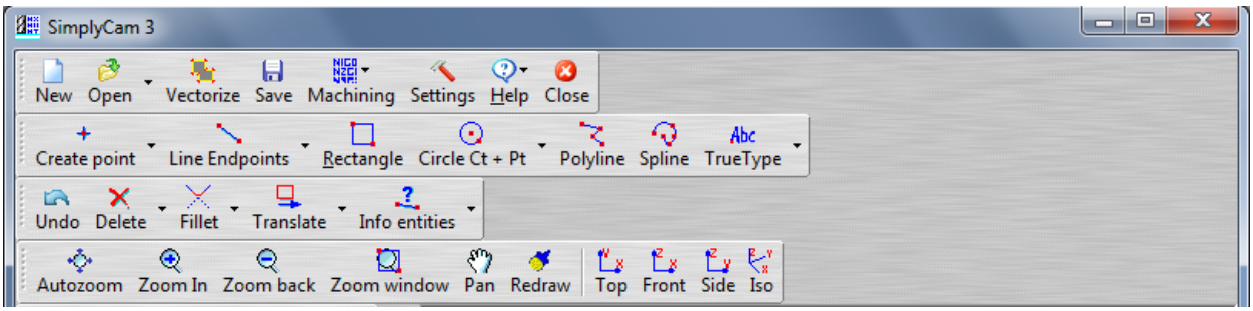
- **Current Layer:** Is the current working layer. Each geometry created is always placed on this layer In the Layers dialog, the current layer is highlighted in yellow.
- **Name:** Defines the name that identifies the contents of the layer.
- **Color:** Sets the color attribute for all entities belonging to its level.
- **Thickness:** Sets the line width in pixels for all entities belonging to its level.
- **Layer Visible:** Shows / hides all entities belonging to its layer.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Toolbars

The Toolbars are a collection of functions accessible through the buttons.
You can undock, dock, move and hide the SimplyCam toolbars.
The toolbars can be undocked by double clicking or dragging its separator bar.
To hide an undocked toolbar, click the X button in the upper right corner.
Select Settings, Toolbars, Visible to show/hide the toolbar.



File toolbar

Create toolbar

Edit toolbar

View toolbar

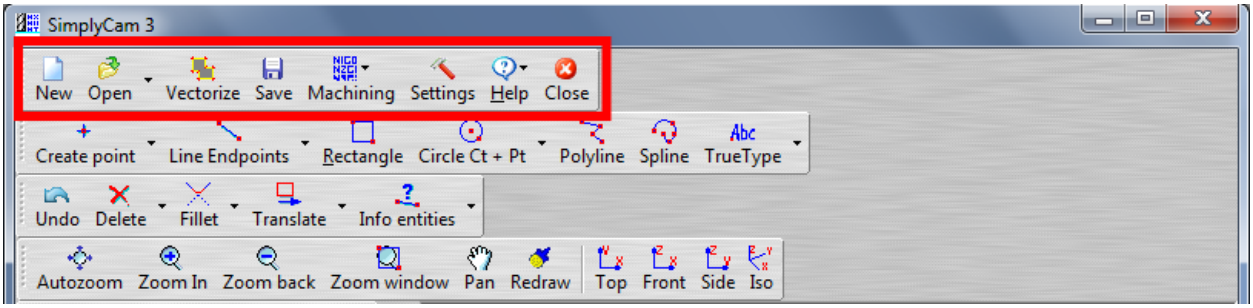
Status Bar

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

File Toolbar

The File toolbar allows you to open and save files in different formats. You can also run vectorization and machining commands, editing NC files and configure the SimplyCam environment.



New

Initializes SimplyCam eliminating all the geometry and, optionally, all the toolpath operations.

Open

The Open option allows you to open raster files (jpg or bmp), vector files (dxf, svg) and vector + raster + toolpath files (SNC) created in SimplyCam version 2 and version 3. It also closes the current file and removes all the entity from the graphics window.

Vectorize

The function Vectorize can convert a raster image (orthogonal grid of pixels), in vector graphics (composed by a set of graphic primitives, such as lines, arcs and polylines) that can be used in machining operations.

More information ....
See the vectorization example...

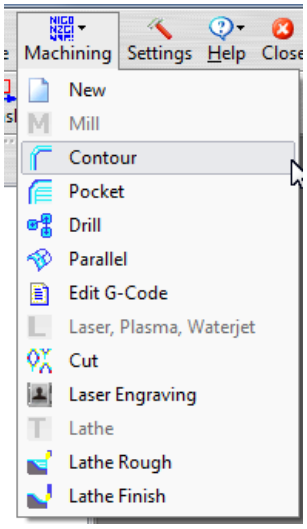
Save

This feature allows you to save the current geometry and toolpath operations.
The files in this format have the extension SNC (SimplyCam Numeric Control files).
You can also save only the geometry in any of the following file types:

- Dxf
- Hgl
- Txt

Menu Machining

The Machining menu gives options for creating the toolpath operations
To access a function, select it from the drop-down menu.



New

Initialize all the toolpath operations.

Mill

Milling Toolpaths

Contour

Contour toolpaths remove material along one or more profiles. You can select an unlimited number of profiles for each toolpath.
More information .....

Pocket

The pocket toolpaths clean out material from an closed boundary. You can select an unlimited number of profiles for each toolpath.
More information .....

Drill

The Drill toolpath will allow you to drilling holes selecting entities point, arcs or circles center.
More information .....

Parallel

Parallel
More information.....

Edit G-Code

The internal editor "gcEditor" can be used to view and edit files G-Code. It is also possible to define an external editor for this purpose. "GcEditor" provides advanced editing functions including CNC graphic simulation of CNC code.

Laser

Laser, Plasma, WaterJet Toolpaths

Cut

This option creates a contouring Cut toolpath
More information.....

Engraving

This option creates a Engraving toolpath
More information.....

Lathe

Lathe Toolpaths

Lathe Rough

This option creates a Lathe Rough toolpath
More information.....

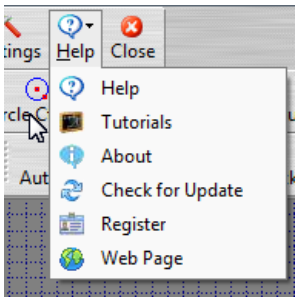
Lathe Rough

This option creates a Lathe Rough toolpath
More information.....

Settings

Allow to configure the SimplyCam environment.
More information .....

Help menu



Help

Call this help file.

Tutorial

Call the "Tutorial" section of this help file.

About

Provides information about the current version and contact methods.

Check for Updates

This command provides a way to manually check for updates.

Register

Opens the dialog box for entering the registration data.

Web Page

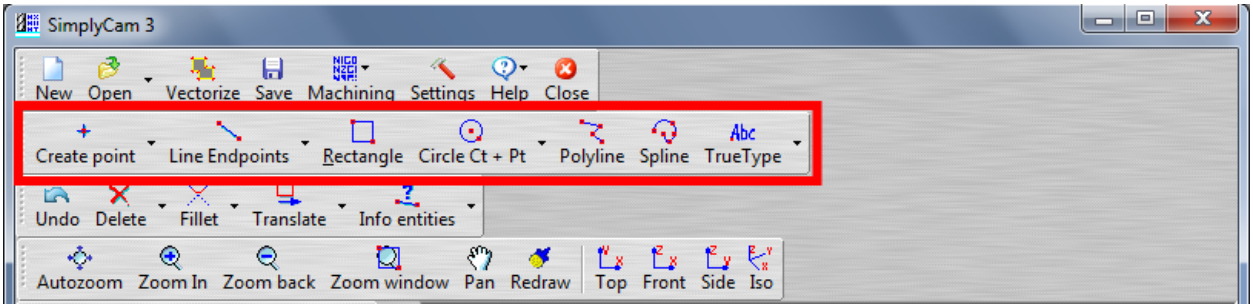
Connecting to www.mr-soft.net website.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

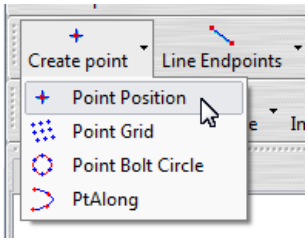
Create Toolbar

The following SimplyCam functions allow you to create basic geometry dynamically, ie, by moving the mouse and clicking in the graphics window. Basic geometry includes points, lines, arcs, polylines, splines, TrueType and SingleLine fonts.



Point menu

Creates points entities in different ways.
To access a function, select it from the drop-down menu.



Point position

Create a point at any position in the graphics area.
You can enter points using one of three methods: Sketch, Snap and Coordinates.

- Sketch: This option lets you enter a point freehand, using the cursor to pick a position in the graphics window.
- Snap: This option allows you to capture the significant points of entities such as End Point, Midpoint, Center ....
- Coordinates: This option allows you to define the point by entering from the keyboard the X and Y value.

Point Grid

Create points on a grid pattern.

- Set the parameters in the Point Grid dialog.
- Enter the position of the lower left corner of the grid.

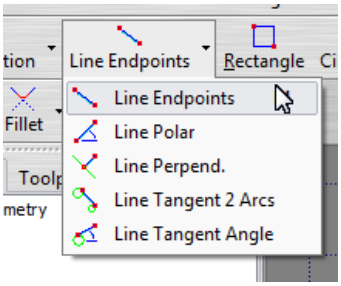
Points Bolt Circle

Create points in a bolt circle pattern.

1. Set the parameters in the Bolt Circle dialog.
2. Enter the position of the center of the bolt circle.

Line Menu

Create line entities in different ways.
To access a function, select it from the drop-down menu.



Line Endpoints

Click on the viewport to select the start point, move the mouse to position the end point and click.
You can enter points using one of three methods: Sketch, Snap and Coordinates.

- Sketch: This option lets you enter a point freehand, using the cursor to pick a position in the graphics window.
- Snap: This option allows you to capture the significant points of entities such as End Point, Midpoint, Center ....
- Coordinates: This option allows you to define the point by entering from the keyboard the X and Y value.

Line Polar

Create line with angle, length and starting point. In the dialog area:

1. Enter the angle of the line.
2. Enter the length of the line.
3. Select the starting point in the graphics area.

Line Perpendicular

Create a line perpendicular to another entity directly to a point.

Line Tangent 2 Arcs

Create a line tangent to two arcs. Select both arcs close to the point where you want to create the tangent line.

Line Tangent Angle

Creates a line tangent to an arc with a given angle.

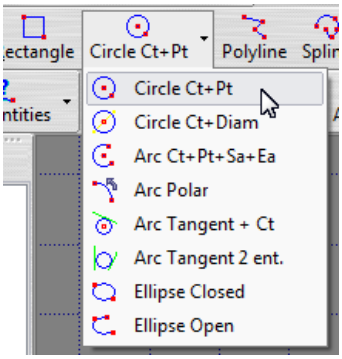
Rectangle

To create a rectangle:

1. Click in the graphics area to select the first vertex of the rectangle.
2. Move the mouse and click to define the second corner of the rectangle.

Arc menu

The Arc menu offers functions for the creation of entities arc (including circle and ellipse).
To access a function, select the item from the drop-down menu.



Note

SimplyCam calculates arcs in a counterclockwise direction.

Circle Ct + Pt

Create a circle by defining the center and a point on the circumference.

1. Select the centrer point of the circle.
2. Select a point that lies on the circumference of the circle.

Circle 3 Pt

Create a circle by defining three points on the circumference.

1. Simply define, select or capture (Snap) three points to create the circle.

Circle Ct + Diam

Create a circle by defining its center and diameter.

1. Enter the diameter of the circle in the box.
2. Define the center point of the circle.

Arc Ct+Pt+Sa+Ea

Create a arc with defined center point, edge point, start angle and end angle with mouse sketch.

1. Select on the graphics area the point that defines the center.
2. Move the mouse to define the radius/diameter and click.
3. Move the mouse to define the starting point and click.
4. Move the mouse again to set the end point and click.

Arco Polar

Creates an arc by defining the center, the diameter, the start angle and the end angle.

1. Enter the diameter in the dialog area.
2. Enter the start angle of the arc in degrees.
3. Enter the end angle of the arc in degrees.
4. Move the mouse and select the center point of the arc.

Arc 3 Pt

Creates an arc by defining three points on the circumference.

1. Simply define, select or capture (Snap) three points to create the arc.

Arc Tangent + Ct

Create an arc with tangency condition of an entity with definite center.

1. Select a line or arc in which the arc that will be created must be tangent .
2. Select the center point of the arc that will be created.

Arc Tangent 2 ent.

Create an arc with tangency condition in the two entities.

1. Enter the radius / diameter in the dialog area.
2. Select two lines, two arcs or line / arc.

Ellipse Closed

Create an ellipse closed with two opposite corners.

1. Click with the mouse in the graphics area to select the first vertex.
2. Move the mouse and select the second vertex.

Ellipse Open

Creates an ellipse by defining open the two opposite vertices, the start angle and the end angle.

1. Click with the mouse in the graphics area to select the first vertex.
2. Move the mouse and select the second vertex.
3. Move the mouse to define the starting point.
4. Move the mouse again to define the end point.

Polyline

A Polyline is an entity created by connecting a set of points.
To create a Polyline:

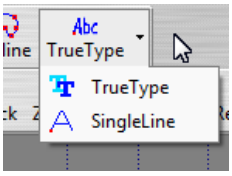
1. Move and select with mouse all points Polyline.
2. In the dialog area press the Close button to connect the first and the last point to create a closed polyline.
3. Or, press the "Delete Last Pt" button to remove the last point in a wrong selection.
4. Or to end, click with the right mouse button and select Done or Cancel from the drop-down menu.

Spline

A spline is an entity created by connecting a set of control points.
To create a spline:

1. Move your mouse and select all the control points of the spline.
2. In the dialog area press the Close button to connect the first and the last point to create a closed Spline.
3. Or, press the "Delete Last Pt" button to remove the last point in a wrong selection.
4. Or to end, click with the right mouse button and select Done or Cancel from the drop-down menu.

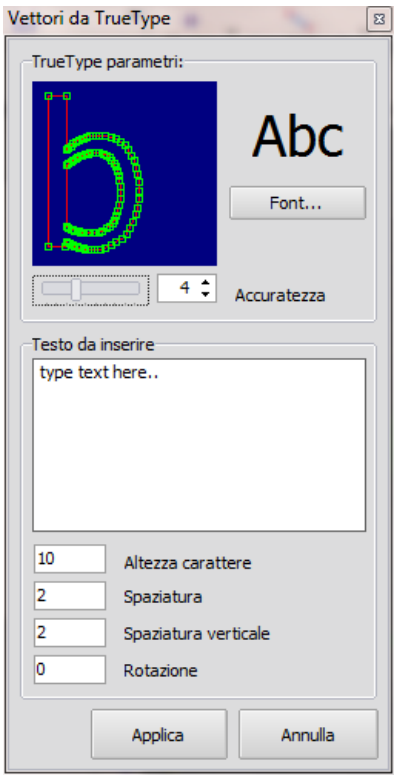
Vector Font



Vector Fonts (TrueType)

This function create a polylines from any TrueType fonts installed on your system.
To create a set of vectors from TrueType fonts:

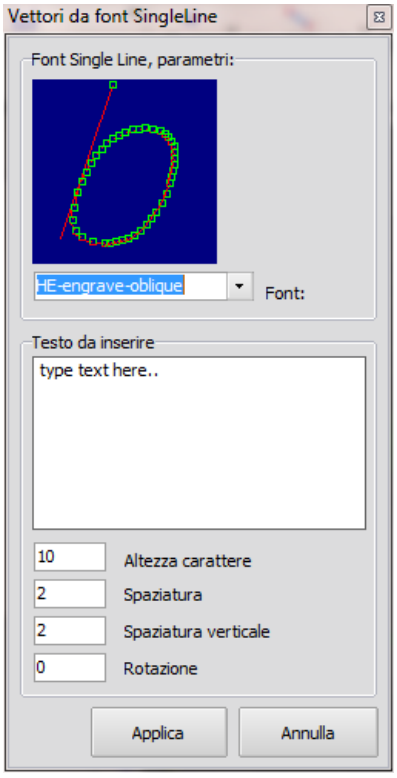
1. Type letter in the text box under "Text to insert".
2. Select the font with the "Font" button.
3. Change the "Accuracy" slider control.
4. Set the character height, spacing and/or text rotation.
5. Press the "Apply" button.
6. Select on the viewport the start point of the text.



Vector Font (SingleLine)

This function create a polylines from real internal Single Line Fonts. The Single Line fonts are special fonts that can be used for engraving text and part numbers.
To create a set of vectors from Single Line fonts:

1. Type letter in the text box under "Text to insert".
2. Select the font.
3. Set the height, spacing and/or text rotation.
4. Press the "Apply" button.
5. Select on the viewport the start point of the text.

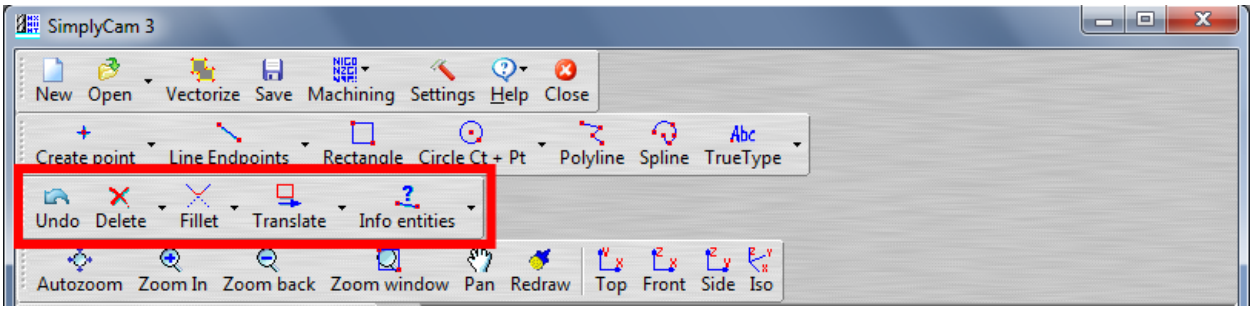


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Edit Toolbar

The following SimplyCam functions allow you to edit existing entities.

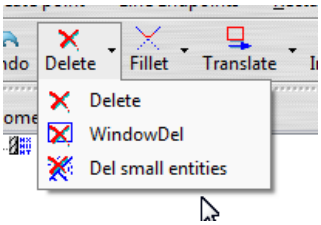


Undo

The Undo command allows you to cancel the last command executed and restore the previous state.

Delete menu

The Delete menu gives options for deleting entities and recovering deleted entities.
To access a function, select it from the dropdown list.



Delete

Use this function to remove one or more selected entities from the graphics window.

To delete entities:

1. Mouse click on the entities you want to delete.
2. Right mouse click, the dropdown menu show up.
3. Click on Done.

WindowDel

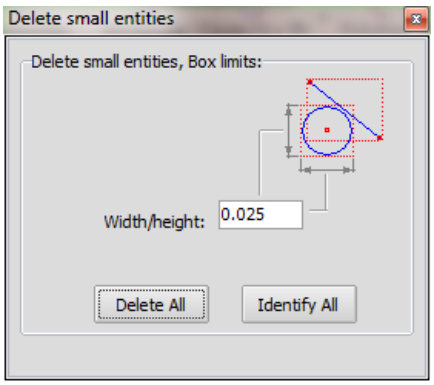
Use this function to remove one or more entities inside the defined window.

To delete multiple entities within window:

1. Use the mouse to sketch the window corners.
2. All entities within window are deleted.

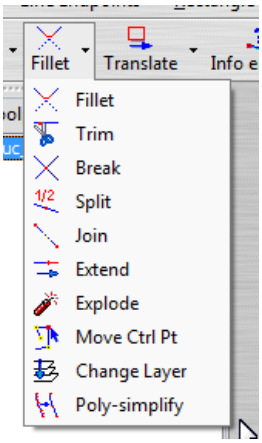
Delete Small entities

Allow to delete small entities (ex. noise of vectorize process).



Edit menu

The Edit menu gives options for modifying entities.
To access a function, select it from the dropdown list.



Fillet

The fillet function create an arc of a defined radius tangent to the entities.
To create fillet:

1. First set the fillet radius on the fillet dialog.
2. Select two entities (linee and/or arcs) close to the fillet zone.
3. The selection point determine where is createt the fillet.

The lines are trimmed to tangent filled position, when the arcs are breaked to tangent fillet position (you can delete the unused portion).

Trim

The trim function gives you options for trimming entity to one another.
To trims two entites to their intersection:

1. Select a line or arc as the first entity.
2. Select a line or arc as the second entity.
3. The selection point on the entity determines which part will be maintained.

Break

The Break function gives you options for breaking entities at their intersections with other entities.

Extend

The Extend function gives you options for extending or trimming an entity by a defined length.
To extend an entity:

1. First set the value for length on the extend dialog.
2. Select a line or arc close to the endpoint that you want to extend or trim.

Note:A positive value extends the entity, a negative value decrease the entity.

Explode

The Explode command allows to change a grouped item into its individual elements, so it can be edited or machined.

You can Explode a Block into its base entity, a Rectangle and Polyline into Linee.

Move Control Point

This function is used to move the control points.
To move the control points of an entity:

1. Select the entity you want to edit.
2. Drag with the mouse one or more of the control points.
3. To end, right mouse button and press Done or Cancel from the dropdown menu.

Change Layer

The Change Layer function gives you options for moving entities between layer.
To change the layer of entity:

1. First set the target layer in the dialog.
2. Select one or more entities to move to the target layer.

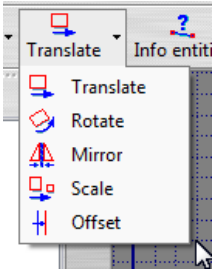
Poly Simplify

Polyline simplify reduce the number of vertices, saving storage and subsequent processing costs.
To simplify a polyline:

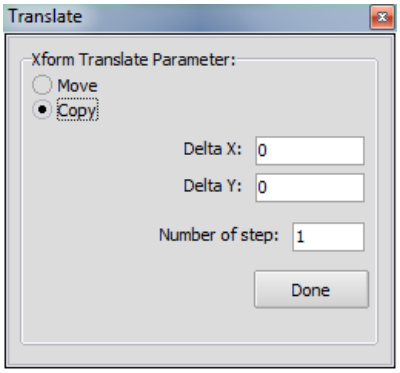
1. First set the tolerance value in the dialog.
2. Select one or more polyline.

Xform menu

The Xform menu gives options for translating, rotating, mirroring, scaling and offsetting entities.
To access a function, select it from the dropdown list.



Translate



To translating entities:

1. Select one or more entities you want to move and press Done from the drop down menu. In alternative, use "Window select" with related status-button activated.
2. Select Move or Copy in the "Translate" dialog box.
3. Enter a value for number of steps in the dialog box.

Dynamic option:

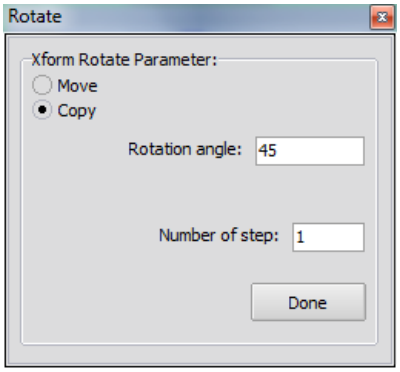
4. Snap with the left mouse the "Base point".
5. Drag with the mouse to define the translation vector (from "Base point" to "New point") and confirm with the click of a mouse.

Static option:

4. Define the translation vector in "DeltaX" and "Delta Y", and then press the "Done" button in the dialog box.

To interrupt the Translate operation, click the right mouse button and select "Cancel" from the drop-down menu, or press Esc keys.

Rotate



To rotating entities:

1. Select one or more entities you want to rotate and press Done from the drop down menu. In alternative, use "Window select" with related status-button activated.
2. Select Move or Copy in the "Rotate" dialog box.
3. Enter a value for number of steps in the dialog box.
4. Snap with the left mouse the center of rotation.

Dynamic option:

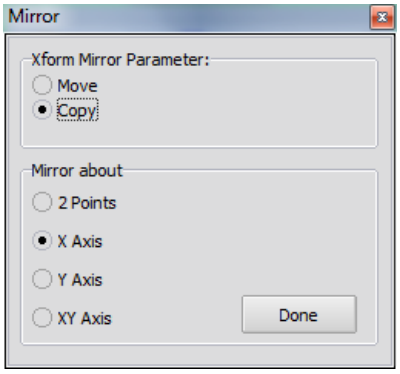
5. Move the mouse to define the angle of rotation and confirm with the click of a mouse.

Static option:

5. Insert the angle of rotation, and then press the "Done" button in the dialog box.

To interrupt the Rotate operation, click the right mouse button and select "Cancel" from the drop-down menu, or press Esc keys.

Mirror



To mirroring entities:

1. Select one or more entities you want to mirror and press Done from the drop down menu. In alternative, use "Window select" with related status-button activated.
2. Select Move or Copy in the "Mirror" dialog box.

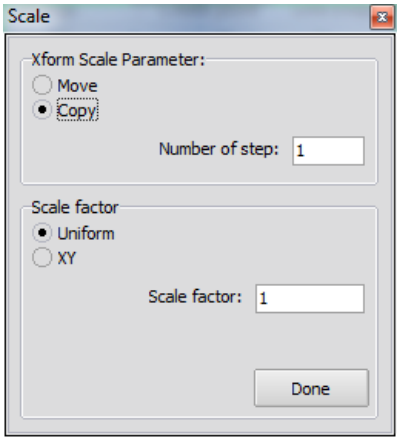
Axis X, Axis Y or Axis XY option:

3. Set the mirror axis, and then press the "Done" button in the dialog box.

"2 Points" option:

3. Snap with the left mouse the first point of mirror axis.
4. Snap with the left mouse the second point of mirror axis.

Scale



To scaling entities:

1. Select one or more entities you want to scale and press Done from the drop down menu. In alternative, use "Window select" with related status-button activated.
2. Snap with the left mouse the reference point. The entities will be scaled relative to this point.
3. Select Move or Copy in the "Scale" dialog box.
4. Enter a value for number of steps in the dialog box.
5. Define the type of scaling (Uniform or different factors along the X and Y).

Dynamic option:

6. Move the mouse to start the scaling or define the scale factor.

Static option:

6. Insert the scale factor, and then press the "Done" button in the dialog box.

To interrupt the Scale operation, click the right mouse button and select "Cancel" from the drop-down menu, or press Esc keys.

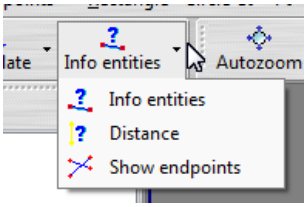
Note:
A scale factor greater than 1 enlarge the entities.
A scale factor less than 1 shrink the entities.
A scale factor equal to 1 produce no change in size.

Offset

Offset is the function to create entities by a distance in a perpendicular direction.
To offsetting entities:

1. Select Move or Copy in the "Offset" dialog box.
2. Set the offset distance in the offset dialog.
3. Select entity you want to offset.
4. A red offset entity with arrow direction, is show up in viewport.
5. Use "Flip dir" button to change the direction.
6. Click on "Done" button to accept the offset result.

Info menu



Info entities

This command is used to get info from entities.
The dialog window show first the limits and the count of all entities.

Distance

This command is used to analyzing the distance and angle between two points.

Show endpoints

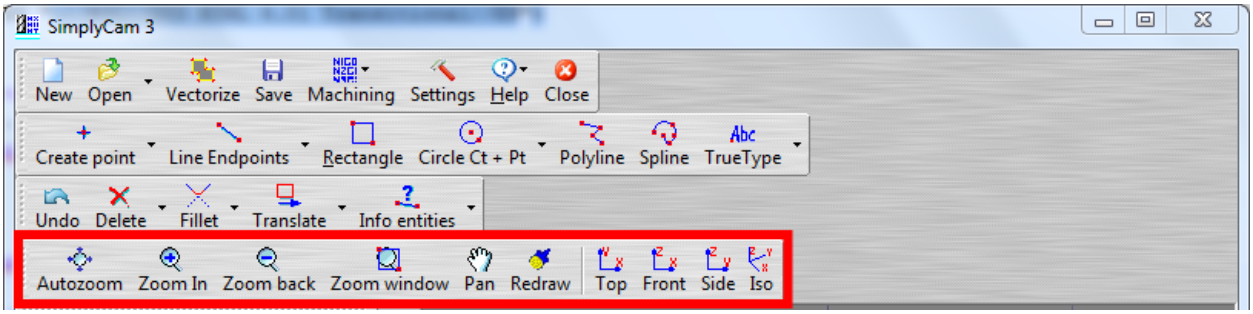
This command is used to view the endpoints of all visible entities. This is useful for determining the boundaries of an individual entity.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

View Toolbar

The View Toolbar contains buttons to change your view of entities in the graphics window.
SimplyCam also provides other methods for viewing the geometry and toolpaths in the graphics window.



Autozoom

The Autozoom command allows you to see the entire geometry in the viewport.
You can access this function from the toolbar and from the dropdown menu.

Zoom in

The ZoomIn command enlarges the view at each successive use, by a 0.5 factor.
You can access this function from the toolbar and from the drop-down menu.

Alternative methods:

- You can use the mouse wheel any time, without selecting a function, to "Zoom In" and "Zoom Back" the contents of the graphics window
- With the mouse in the graphic area, a quick mode to "Zoom In" is: press CTRL + left mouse button and move up the mouse.

Zoom back

The ZoomBack command is the opposite of the ZoomIn command.
It reduces the view at each successive use, by a 0.5 factor.
You can access this function from the toolbar and from the dropdown menu.

Alternative methods:

- You can use the mouse wheel any time, without selecting a function, to "Zoom In" and "Zoom Back" the contents of the graphics window
- With the mouse in the graphic area, a quick mode to "Zoom In" is: press CTRL + left mouse button and and move down the mouse.

Zoom window

The Zoom Window command allows you to zoom into a rectangular area that you specify by opposite corner points with the cursor.
To use Zoom window:

1. Left mouse button to select first corner of the area you wish to zoom into;
2. Left mouse button to select the opposite corner of this area.

You can access this function from the toolbar and from the right-click menu.

Pan

The Pan command move the visible portion of the drawing.
The Pan command can be used along with another executing command. For example, you can use the Pan command during the Line command.

Alternative method:

- With the mouse in the graphic area, a quick mode to "Pan" is: press SHIFT + left mouse button and move the mouse.

Redraw

Redraws and cleans up the drawing.

Top

Change the graphics view a predefined Top view

Front

Change the graphics view a predefined Front view

Side

Change the graphics view a predefined Side view

Iso

Change the graphics view a predefined Isometric view

Alternative methods:

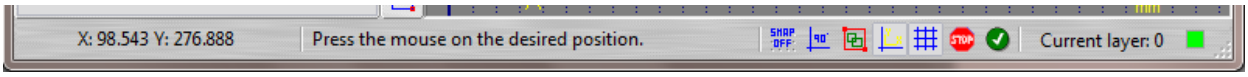
- With the mouse in the graphic area, a quick mode to "Rotate the view" is: press ALT key + left mouse button and dragging the mouse.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Status Bar

The Status Bar is located at the bottom of the SimplyCam application window.



Cursor Position

This area display the (x,y) position of the cursor in world coordinates.
If Snap options is active, the coordinates reflect the closest snapped point relative to cursor position.

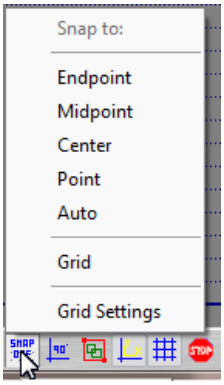
Prompt area

This area shows the messages and requests that SimplyCam sends to you.

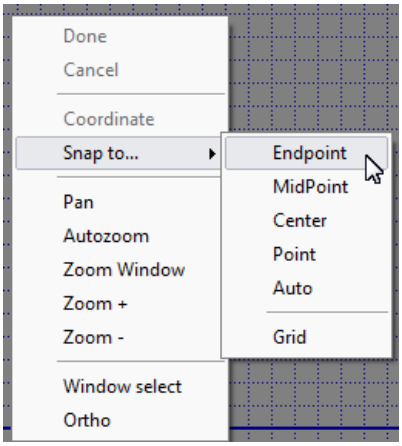
Snap (capture, hooking)

This tool ensures greater accuracy in defining the coordinates when creating entities.
The Snap tool recognizes the geometric characteristics of the entities closest to the cursor and captures the coordinates as End Point, Midpoint, Center, Point and Grid.

To set the Snap Mode press the "Snap" in the Status Bar.



You can also set the Snap Mode from the drop-down menu of the right mouse button.



SimplyCam identifies the snap point captured, in a dynamic way, using the distinctive symbols:

Snaps (capture) the nearest Endpoint of a line, an arc or a vertex of a polyline.

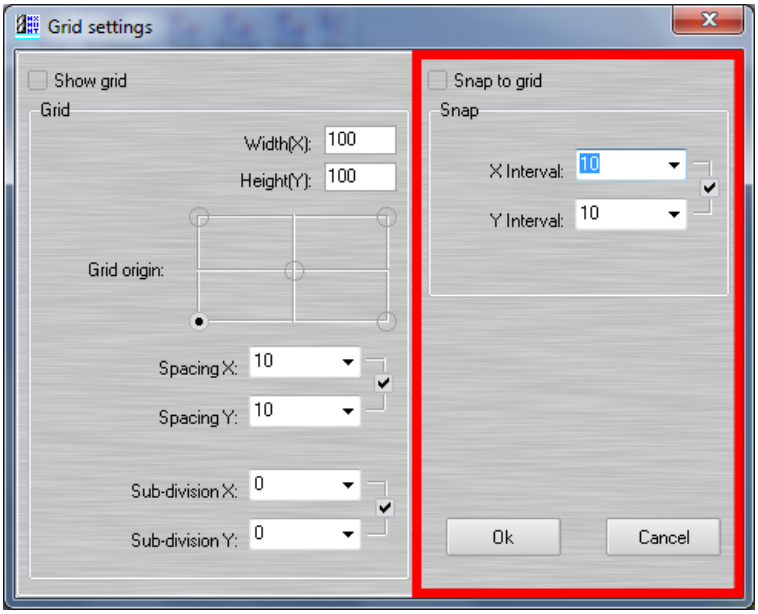
Snaps (capture) the nearest Midpoint of a line or arc.

Snaps (capture) the nearest Center point of an arc or a circle.

Snaps (capture) the coordinates of the nearest geometric entity Point.

Activating the Snap Grid option, the cursor can only select coordinates that belong to the range of the grid.
Spacing X and Spacing Y specify the snap spacing.

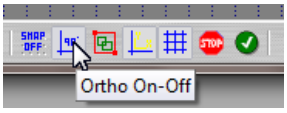
Mouse click on the "Grid Settings" item to define the spacing for the "Snap to Grid".



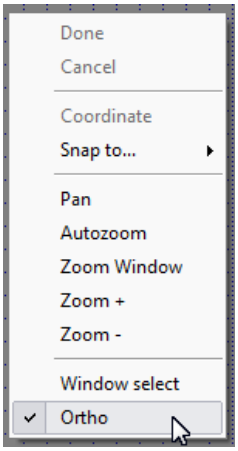
Ortho mode

The ortho mode (short for orthogonal) active, limits the movement of the cursor only in the horizontal direction and in the vertical direction.

Toggle On-Off mode by pressing the "Ortho" button in status-bar.



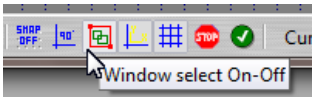
Also toggle On-Off ortho mode from the drop down menu of the right mouse button.



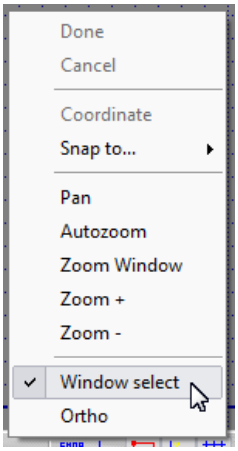
Windows select

The "Windows Select" allows you to select one or more entities within a rectangular area specified by two opposite vertices with the mouse.

Toggle On-Off mode by pressing the "Window Sel." button in status-bar.



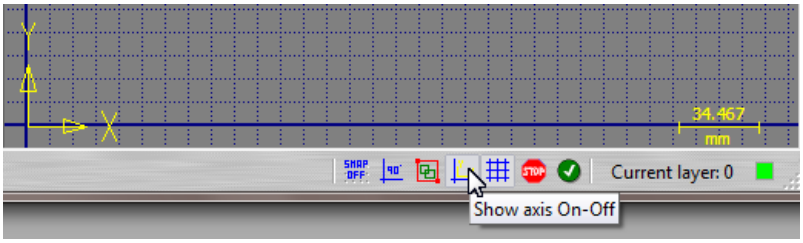
You can toggle On-Off "Window Sel." mode, also from the drop down menu of the right mouse button.



Axis button

This option show/hide in the bottom of viewport the following:

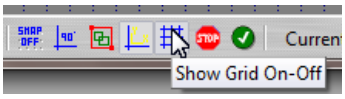
- The system origin
- The positive direction of X and Y axis
- The current scale video and the system unit



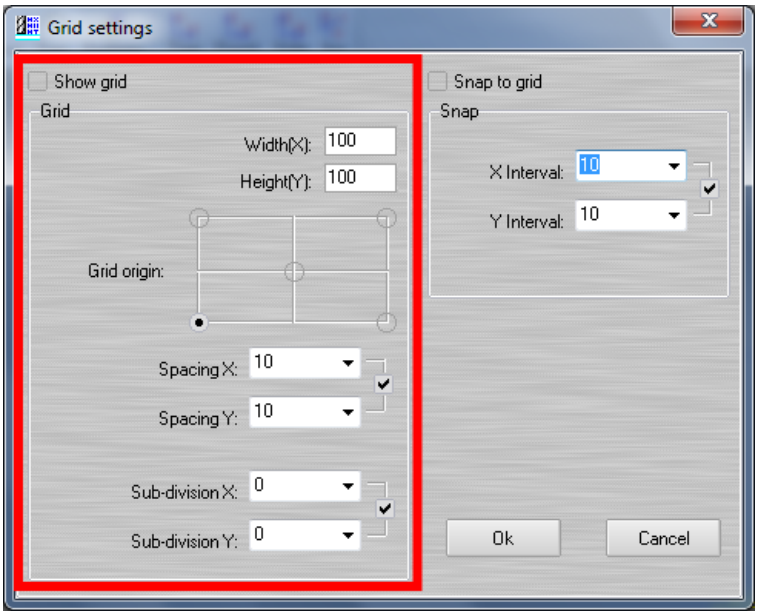
Visible Grid

This option draw a 2D rectangular grid on the viewport.
To use the Visible Grid:

Toggle On-Off mode by pressing the "Grid" button in status-bar.



Mouse click on "Grid Settings" item to define the visible grid spacing.

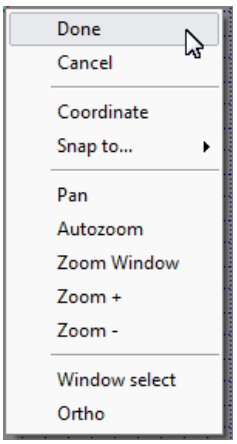


Done / Cancel

Many operation required the following user commands:

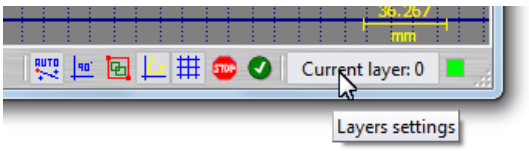
- Done: The operation is ended and accepted.
- Cancel: The operation is aborted and ignored.

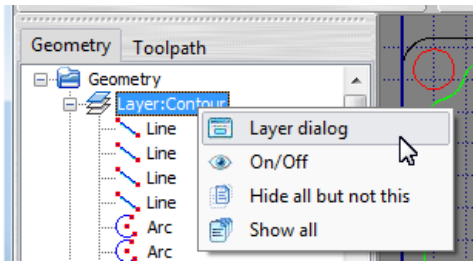
The action "Done / Cancel" can be quickly validated from the drop-down menu of the right mouse button.



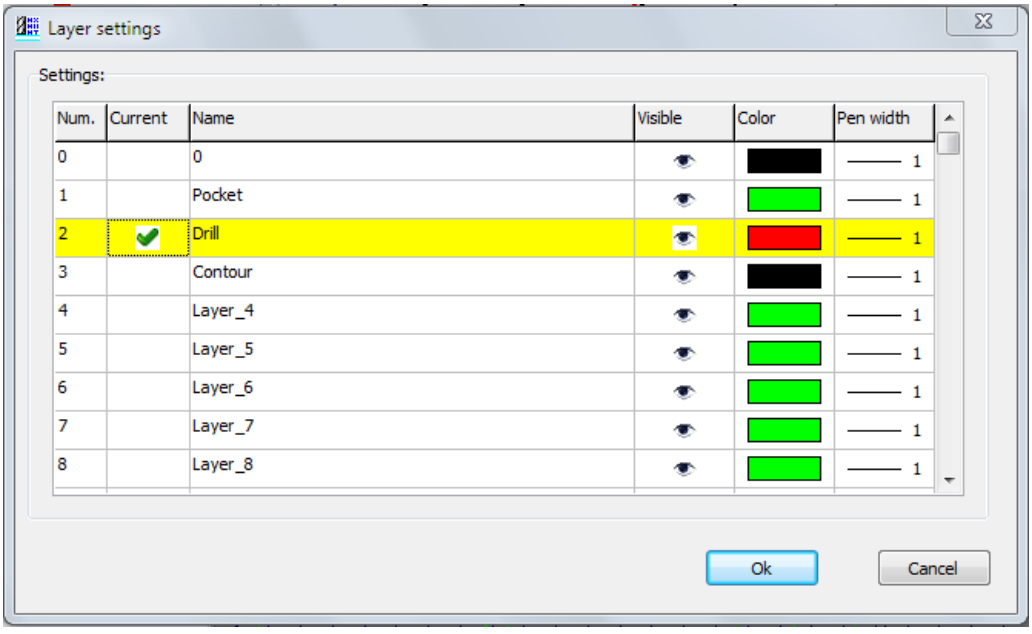
Layer

The Layers are used to separate and simplify access to geometric entities in a complex design.
Access the "Layer" dialog by clicking the "Layer Settings" button in the status bar or by clicking on the "Layer Dialog" item in the drop-down menu of the Geometry Manager.





Dialog Layer Setting:



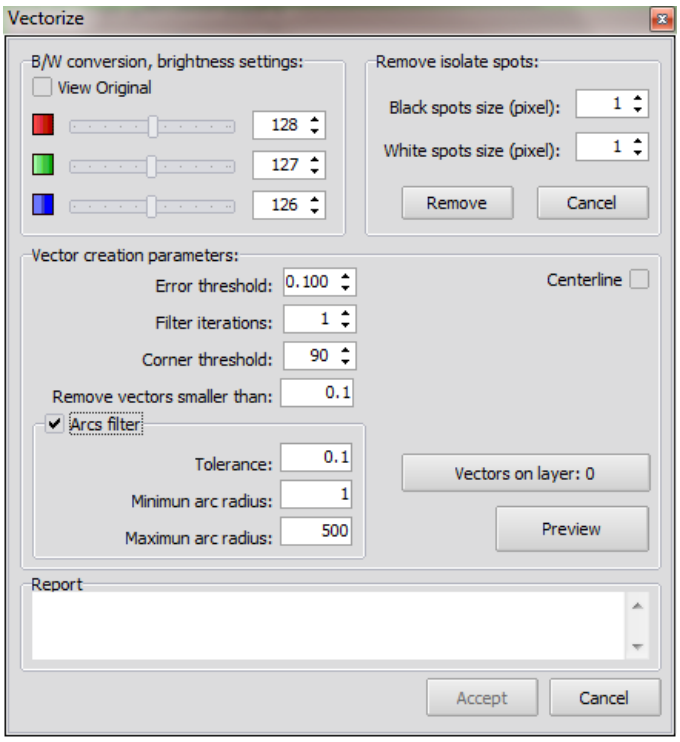
- **Current Layer:** Is the current working layer. Each geometry created is always placed on this layer In the Layers dialog, the current layer is highlighted in yellow.
- **Name:** Defines the name that identifies the contents of the layer.
- **Color:** Sets the color attribute for all entities belonging to its level.
- **Thickness:** Sets the line width in pixels for all entities belonging to its level.
- **Layer Visible:** Shows / hides all entities belonging to its layer.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Vectorize

This function allows for quick and easy vectorizations from graphic images, whether black and white, grayscale or 8, 24 or 32 bits color depth.
The vectorization is a process that converts a raster image (bmp or jpg) into vector graphics (composed of a set of graphic primitives such as lines, arcs, and polylines) that can be used in machining operations.
The vector format can be imported into other CAD/CAM applications .



Black and White conversion

Each image is made up of millions of pixels and each pixel has its own color.
Move the slider in order to reduce the color information present on each pixel from the image information only to two: white and black.

Colored Image

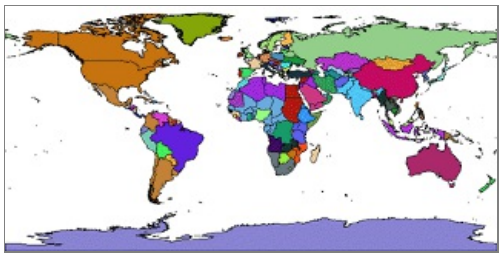
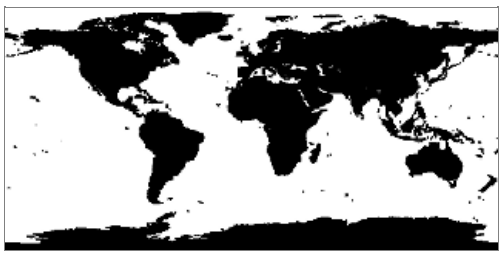


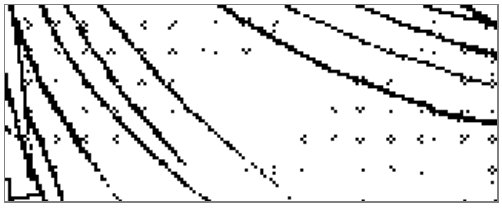
Image converted to black and white



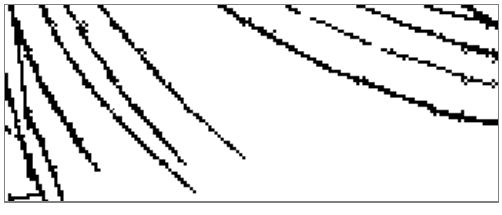
Removing spots or isolated pixels

This section allows you to choose the maximum size of black and white spots and remove them from the image raster before performing the vectorization process.

Before removing



After removing

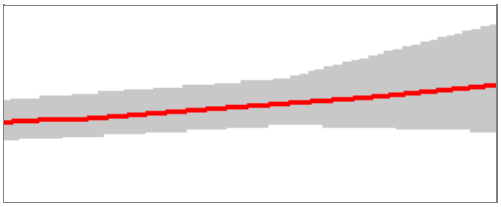


Vectors creation parameters

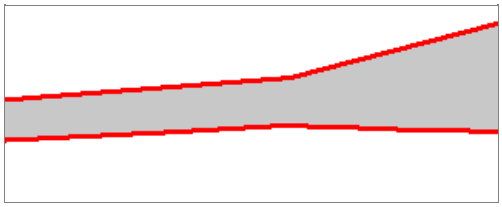
Centerline

With SimplyCam you can opt for two different modes of vectorization: "Centerline" and "Outline".
If you choose the option "Centerline", the vectorization generates vector along the center of the raster elements, otherwise the vectorization generates vector along the edge of the raster elements.

Centerline



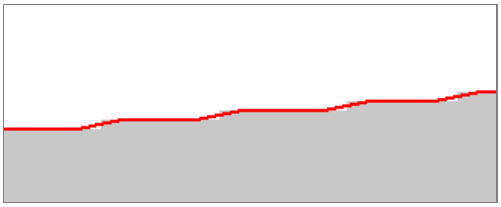
Outline



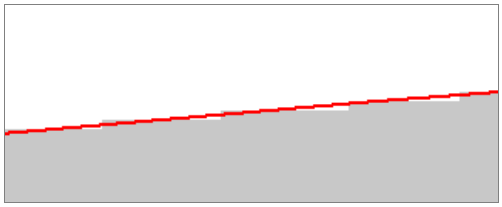
Error threshold

This parameter determines how the curves will be created to fit the raster pixels.

Error = 0.1



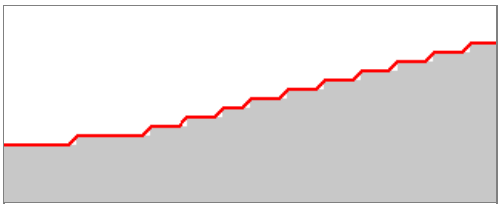
Error = 1.0



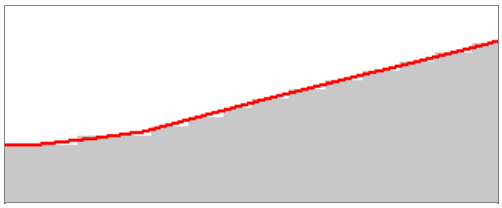
Filter iteration

Sets the number of filtering or smoothing that are applied to created curves.

Filter iteration = 0



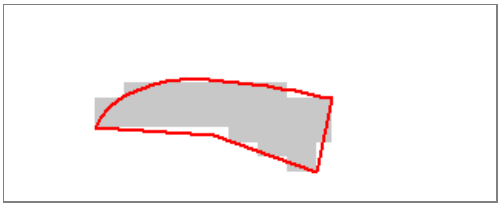
Filter iteration = 3



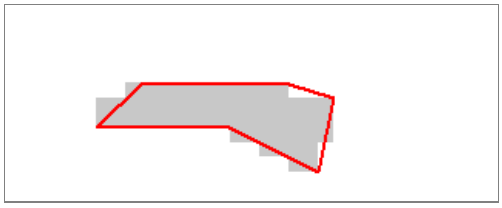
Corner threshold

The set value (degrees), determines the maximum angle between a series of preceding pixels and a series of subsequent pixels to form an corner.

Corner threshold = 160



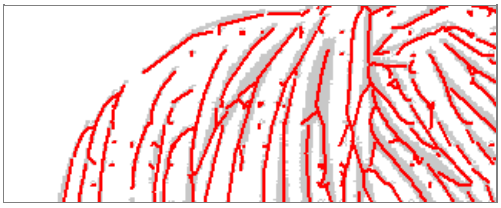
Corner threshold = 90



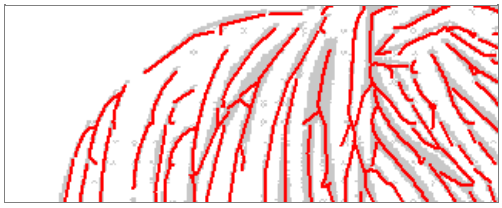
Removing vectors smaller than

The vector length smaller than the set value are not created by the vectorization process.

Removing vectors smaller than 0.0



Removing vectors smaller than 0.5



Vectors on the layer

The vectors created by the vectorization process are positioned on the selected layer and inherit their attributes (color, thickness).

Preview

Vectors created by the vectorization process are shown in the graphic area without being stored in the database.

Arcs filter

This option allows to program that replaces a series of linear vectors with arcs whenever possible.

- Tolerance:

This parameter sets the tolerance of the arcs recognition.

- Minimum radius arc:

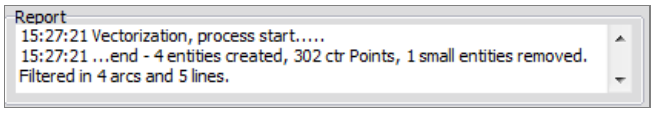
Specify the minimum permissible arc radius.

- Maximum radius arc:

Specify the maximum permissible arc radius.

Report

Show information about the vectorization process.



Accept

The vectors created by the vectorization process are stored in the database.

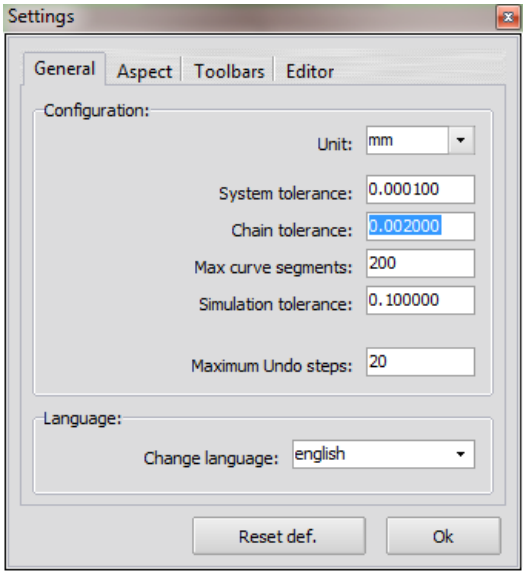
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Settings

Here are displayed the dialogs and the options that allow you to set SimplyCam operating.

General



Unit

This option sets the unit of measurement used in SimplyCam ((English=inch or Metric=mm)).

System Tolerance

This value defines the maximum distance between two points to be considered coincident.

Caution: The setting of a high tolerance value, can be generates unexpected results.
Please use this feature sensibly and with care

Chain Tolerance

This value defines the maximum distance between two points to be connected in the chaining profiles process.
It also defines the tolerance used in the toolpath calculation.
Smaller tolerance provide more accuracy but can result in slower performance.

Caution: The setting of a high tolerance value, can be generates unexpected results.
Please use this feature sensibly and with care

Maximum curves segmentation

This option sets the maximum number of segments for the curves representation in the graphics area.
Increase this value to have a more accurate representation.
Increase this value, if when opening a dxf file with spline, they are incomplete.

Caution: A high value segmentation may increase the time to display a complex drawing.

Simulation tolerance

This option sets the tolerance used in the toolpath simulation.
Decrease this value to have more accurate simulation of small objects.

Caution: A low tolerance may increase the time for the simulation of complex toolpaths.

Maximum Undo steps

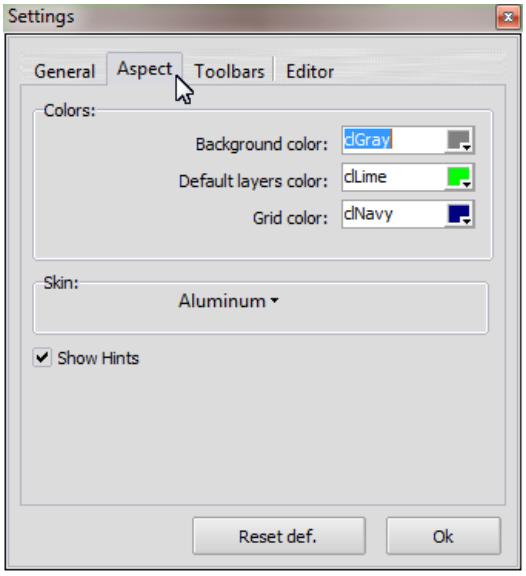
This value sets the number of Undo that SimplyCam "keeps track".

Caution: A high value of "Maximum Undo steps" absorbs more memory.

Language

This option allows you to select different languages for the interface.

Aspect



Background Color
Set the background color of the graphics area.

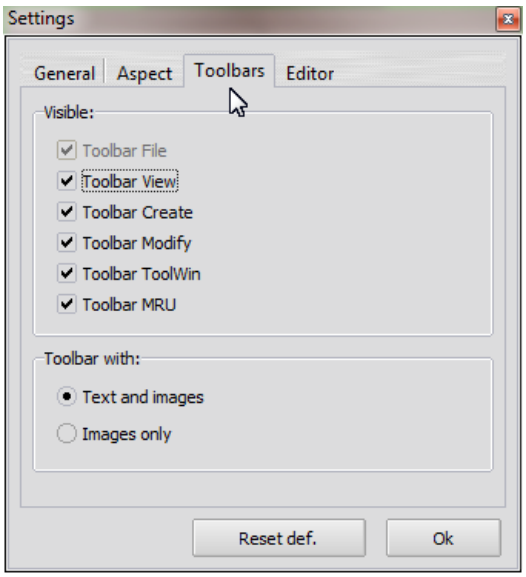
Default layers color
Sets the default color assigned tonot defined layers.

Grid color
Sets the color of the grid in the graphic area.

Skin
Allows you to change the graphical appearance of the SimplyCam interface.

Show Hint
Enable / disable the box that contains an explanation of the command that appears when the mouse passes over them.

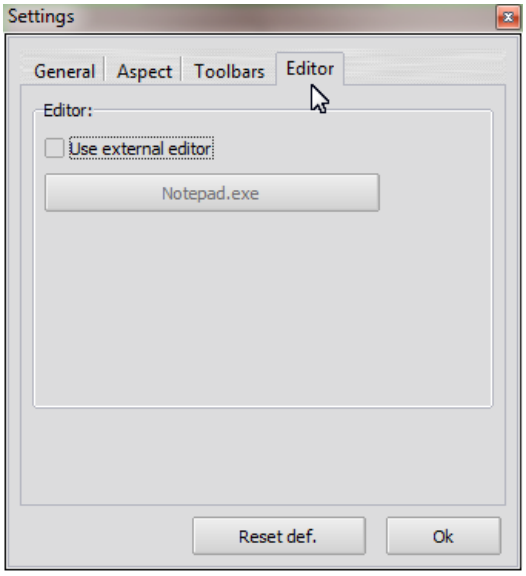
Toolbars



Visible
Each Toolbar can be made "Visible / Hidden" with the click on the relative box of this dialog.
Or each Toolbar can be hidden by clicking the X button in the upper right corner of the toolbar.

Toolbars with
Allows you to modify the appearance of toolbar items: Text and Icons together or only Icons.

Editor



Use External Editor

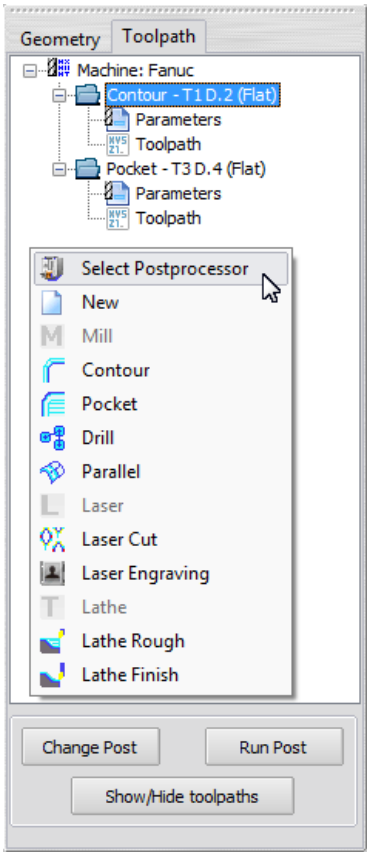
Allows you to use an external editor to view, edit and print the G_Code files generated in SimplyCam.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

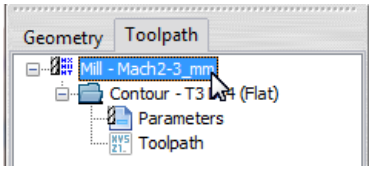
Toolpath Manager

Use the Toolpath Manager to create, manage, edit and view the operations of toolpath.
A single geometric part can have one or more operations.
One or more operations can be postprocessing (ie converted) into a single G-Code file, specific to your CNC machine Vs.
More operations, with different tool numbers, produce the G-Code file, the instructions for tool change (eg T3 M06).
The following figure shows how each operation is displayed in Toolpath Manager:

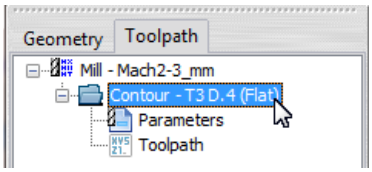


Treeview description

- Type of machine and current postprocessor active.

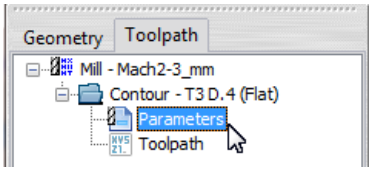


- Operation



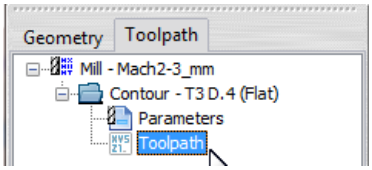
Each operation has a name that describes the type of toolpath and the tool used.
Each operation has two sections:

- Parameters



Contains all the information of the toolpath, such as the tool, feedrates, the size of cuts, selected geometry, etc..

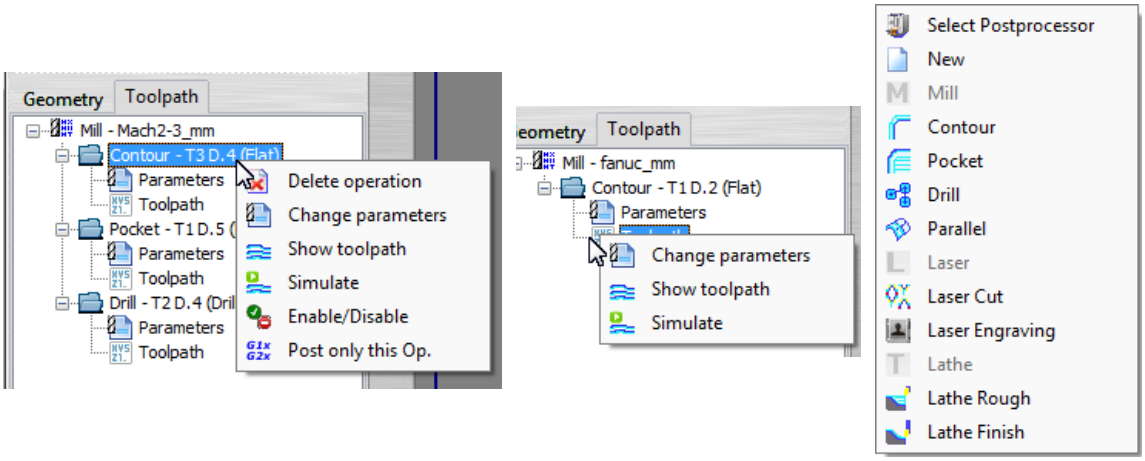
- Toolpath



The file contains all the interpolations of the toolpath. The post-processor uses this file to create the G-code specific to your Cnc machine.

Right-click menu

By right-clicking in the area of â€‹â€‹operations the menu are displayed in context to the clicked position.



Change Post

To change the current post-processor, click on the "Change Post" buttonor select the same function from the drop-down menu of the right mouse button.
More information ....

New

Initialize all the toolpath operations.

Mill

Milling Toolpaths

Contour

This option creates a new contouring machining.
More information ....

Pocket

This option creates a new pocketing machining.
More information ....

Drill cycle

This option creates a new drilling machining.
More information ....

Parallel

This option creates a 3D Parallel toolpath on Stl models.
More information....

Laser

Laser, Plasma, WaterJet Toolpaths

Cut

This option creates a contouring Cut toolpath
More information....

Engraving

This option creates a Engraving toolpath
More information....

Lathe

Lathe Toolpaths

Lathe Rough

This option creates a Lathe Rough toolpath
More information....

Lathe Finish

This option creates a Lathe Finish toolpath
More information....

Delete Operation

The selected operation is removed from the list of toolpaths created earlier.

Change Parameters

The selected operation is open and its dialogs shown. All parameters, tools, chains, etc.. can be modified and the machining recalculated.

Show Toolpath

Displays the toolpath of the selected operation.

Simulate

This opens the dialog to simulate the toolpath of the selected operation.
More information ....

Enable / Disable

The selected operation is Disabled / Enabled. If the operation is disabled, when you run the postprocessor, the toolpath is not included in the G-Code file.

Post only this operation

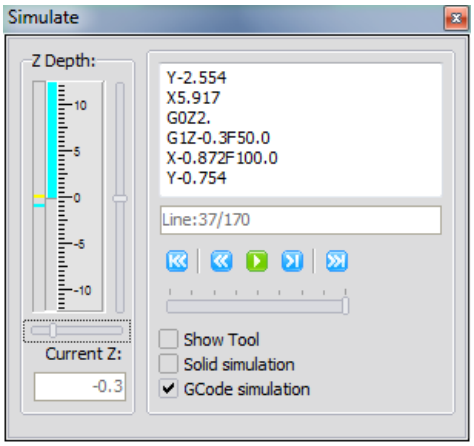
This option creates the G-Code file with the current postprocessor, converting only the selected operation.
More information ....

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Simulation

This dialog allows to perform the simulation of the tool path so as to be able to recognize errors in the program, before performing the machining of the part.
At the end of the simulation the approximate time of processing is shown at the bottom of the screen.



Z Depth

This section shows the current position of the Z axis The yellow color identifies rapid traverse (G0), while the cyan color identifies the feed move (G1).
The horizontal scrollbar increase or decrease the scale factor of the representation of the Z axis
The vertical scrollbar moves the zero point of this representation.

Current Z

Here is shown the actual numeric value of the Z axis.

Area G-Code

In this box you will see the conversion to G-Code, of the current position in course of simulation.
Instead, use the SimplyCam editor and its corresponding section to simulate / edit the complete G-code created by the postprocessor or opened from an external file.

Riga .. / ..

Report in the current line number simulated.

Simulation Toolbar

- Previous operation:
Move the pointer of the simulation at the beginning of the previous operation, if available.

- Rewind:
Move the pointer of the simulation at the beginning of the current operation.

- Play / Stop:
Start / Stop the continuously simulation.

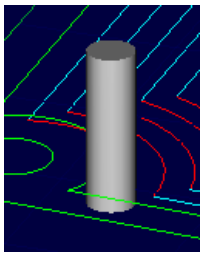
- Step to step (F5):
Executes the simulation of one instruction at a time.

- Next operation:
Move the pointer of the simulation at the beginning of the previous operation, if available.

- Slider Slow / Fast:
Set the speed of the simulation in Play Mode.

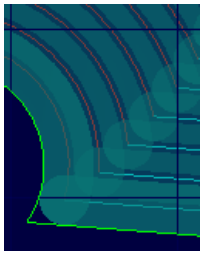
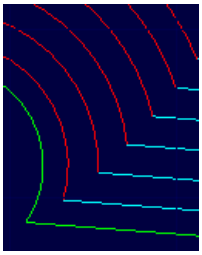
Show Tool

If active, during the toolpath simulation, it is also displayed the 3D tool.



Solid simulation

If active, the movements are displayed showing the track that simulates the tool diameter.
If off, the movements are displayed in wireframe showing the tool center.

Simulation G-Code

If activated, the graphic simulation is also converted in the corresponding G-Code.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

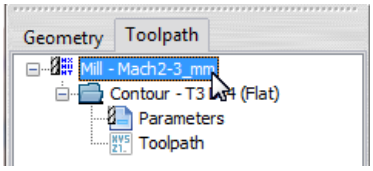
SimplyCam V3. Documentation

Postprocessor

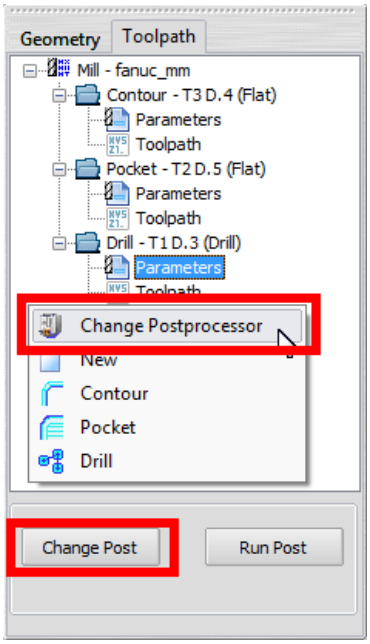
The post processor converts the Toolpaths Manager operations in a G-Code file compatible with your CNC machine.

Current Postprocessor

The current postprocessor, used in the conversion into G-code format, is shown at the top in the first node of the operations.

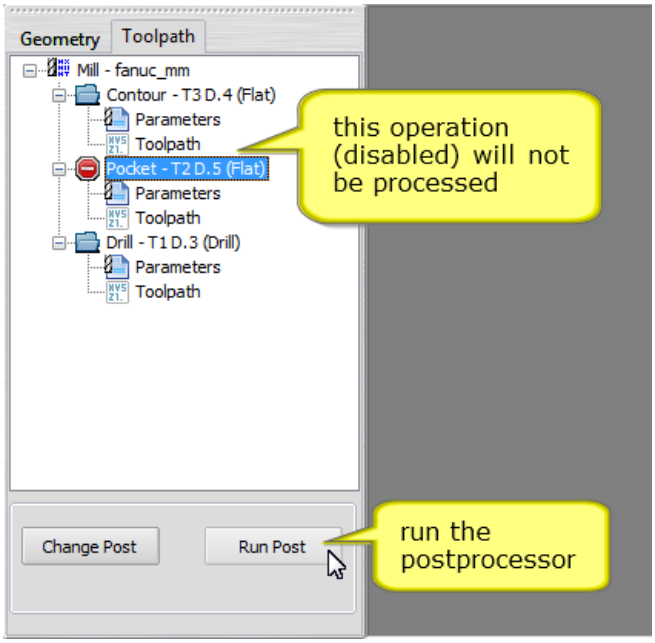


To change the current post-processor, click on the "Change Post" buttonor select the same function from the drop-down menu of the right mouse button.



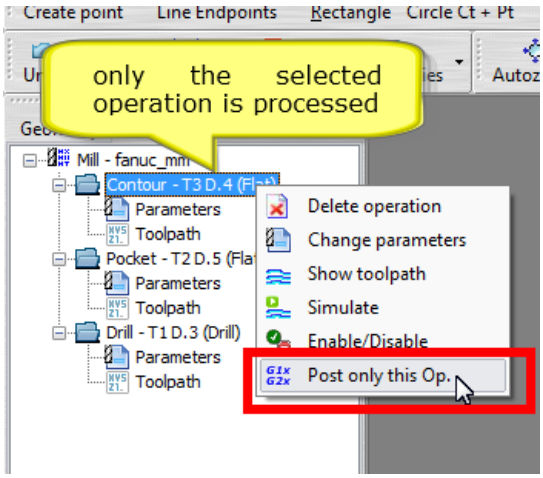
Run Post

This option creates the G-Code file with the current postprocessor, converting all the **active** nella lista delle lavorazioni.



Post only this operation

This option creates the G-Code file with the current postprocessor, converting only the selected operation.



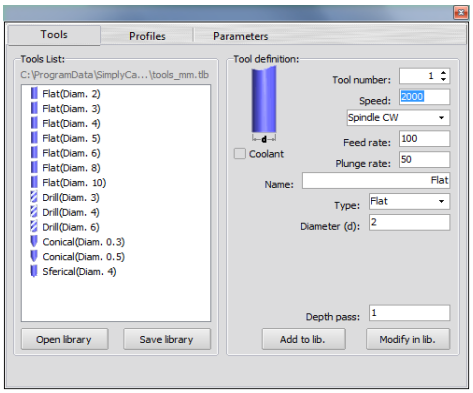
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

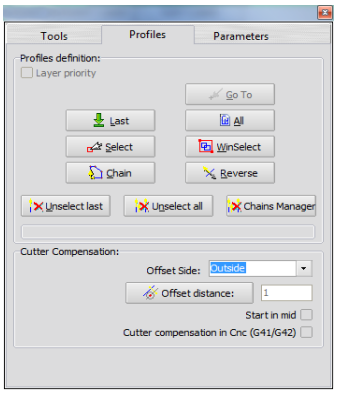
SimplyCam V3. Documentation

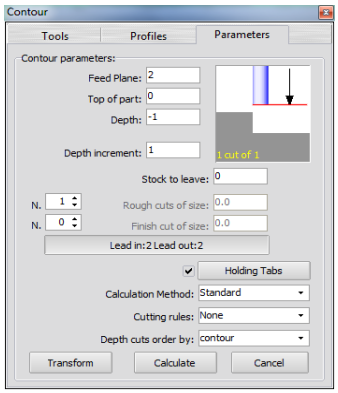
Contour Toolpath

The Contour toolpaths remove material along one or more profiles. You can select an unlimited number of profiles for each operation, but all the selected profiles, they will use the same parameters of the operation.
The selected profiles can be offsetting internally/externally (left/right for the open profiles) to compensate the tool diameter.

Dialogues of Contour Toolpath







Dialog 1 - Tool Definition

This dialog sets the technological parameters of the tool. These parameters directly affect the G-code generated by the postprocessor.
More information ....

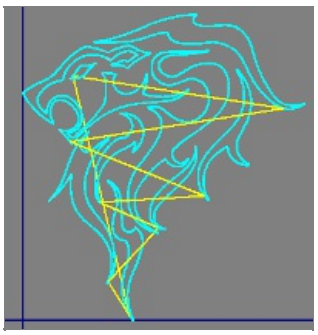
Dialog 2 - Defining Profiles

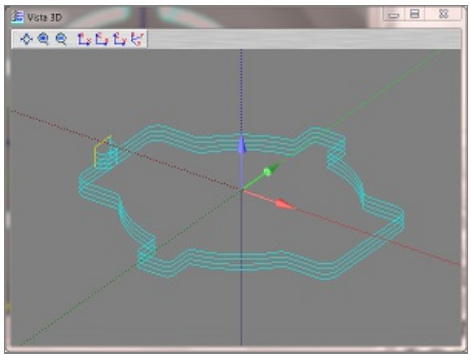
Use this dialog to define and edit the profiles for the selected operation.
This box is also used to define the tool compensation and the lead in /lead out.
More information ....

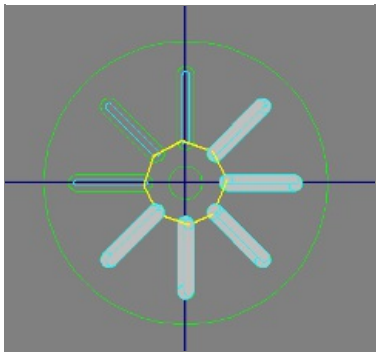
Dialog 3 - Contour Parameters

This dialog contains the parameters specific for the Contour machining.
More information ....

Examples of machining Contour





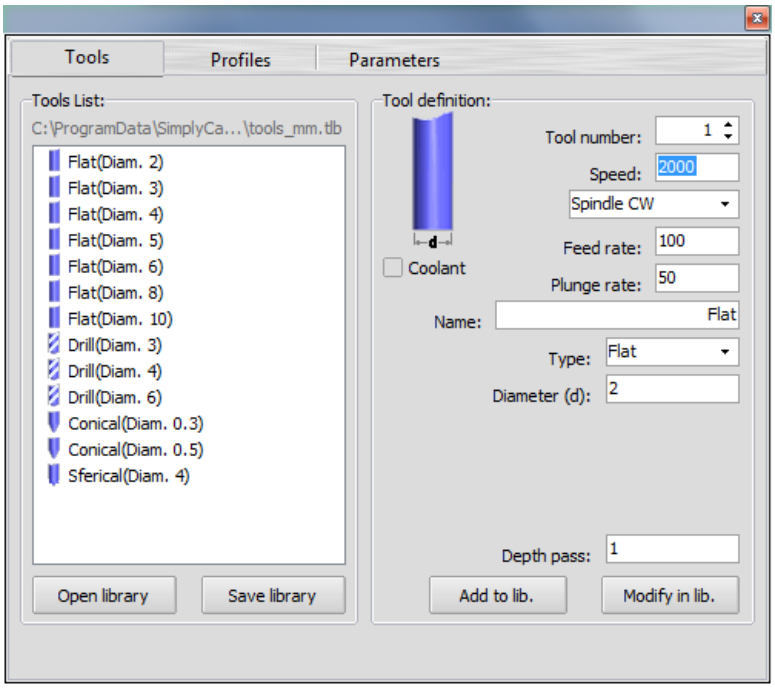


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Tools

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values â€‹â€‹that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Speed

This parameter sets the number of revolutions of the spindle.
SimplyCam normally calculates this value in revolutions / min (rpm) and produces the G-Code file the statement S (eg S3000)

Spindle: Off, CW, CCW

Sets the direction of rotation of the spindle. (Ex: M03)

Feed rate

This value controls the velocity (usually expressed in millimeters / minute or inches / minute) feed to the axes X and Y.

Plunge rate

This value controls the velocity of movement of tool penetration. The Plunge rate is only applies to tool Z movement in the negative direction (usually expressed in mm / min or inches / min).

Coolant

Turn on / Turn off the coolant.

Name

This field allows you to assign an alphanumeric string to identify the tool.

Type

This option allows you to choose the type of tool proposed by SimplyCam.

Diameter

This value sets the diameter of the SimplyCam used to calculate the toolpaths and tool compensation.

Depth pass

This value is not currently used in the calculation of the processes. Use it as a reminder.

Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

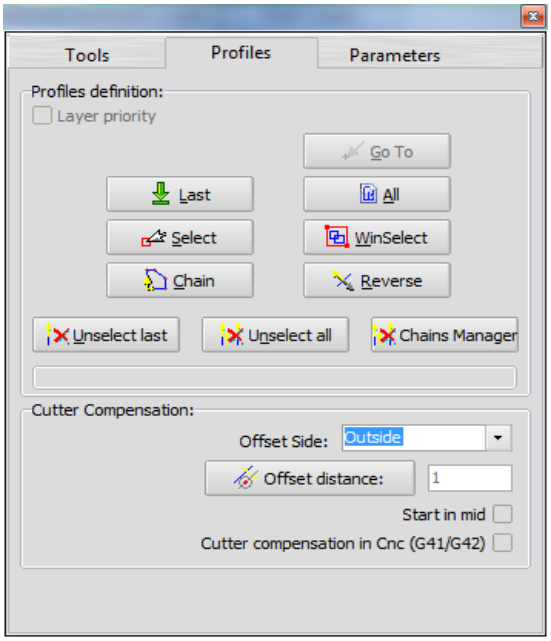
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

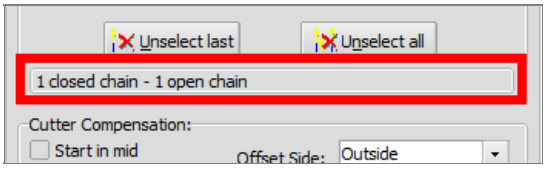
Profiles

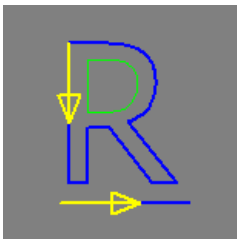
This dialog allows you to define the profiles to be machined.
It also allows you to define the tool compensation.



Profiles Definition section

Each profile defined, it is displayed on the screen with an arrow indicating the direction of the cut and the point of start.
It also increased the indicator in bottom section, showing the number and type of defined profiles.





Last
The chain defined in the last preceding operation is selected.

All
All entities on the active layers (visible) are selected and chained automatically, if possible.

Select
The selected entities, one by one, at the end are concatenated to create one or more profiles (chains).

Window Select
All entities lying inside the rectangular window are selected and concatenated to create one or more profiles (chains).

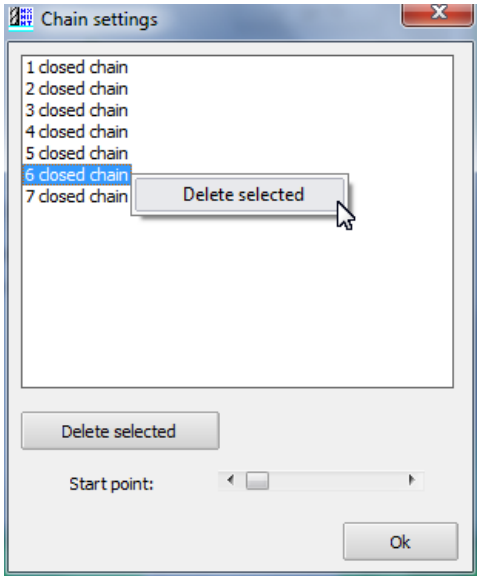
Chain
Clicking on an entity, all entities connected to the respective end points, are concatenated to form the profile (chain).

Reverse
Reverses the direction of the chains.

Unselect Last
The last defined chain is removed from the selection list.

Unselect All
All defined chains are removed from the selection list.

Chains Manager
This parameter open the dialog box containing the list of all chains with this options:



- Move the order in which they chains are performed in the machining
- Delete one or more chains
- Move the starting point each single chain.

Cutter Compensation section

Offset Side

This parameter determines the offset direction, ie the direction of the tool relative to the chained profile.
This direction is displayed on the screen with a small arrow, perpendicular to the cutting direction, which indicates the offset direction.

- Off:
No compensation, the tool is placed directly on the chained profiles.

- Outside
The tool is placed externally respect to the concatenated profile. This rule applies only to closed contours, while the open profiles this rule is not valid; normally the tool is placed to the right of the open profile.

- Inside
The tool is placed internally respect to the concatenated profile. This rule applies only to closed contours, while the open profiles this rule is not valid; normally the tool is placed to the left of the open profile.

Offset distance

This value corresponds to the tool radius. Pressing the button refers to the dialog of selection tool.

Start in mid

Setting this option, the starting point of the toolpat is moved to the middle of the first entity in the chain.
Useful to avoid having the attack on an inner corner of the path.

Cutter compensation in Cnc (G41/G42)

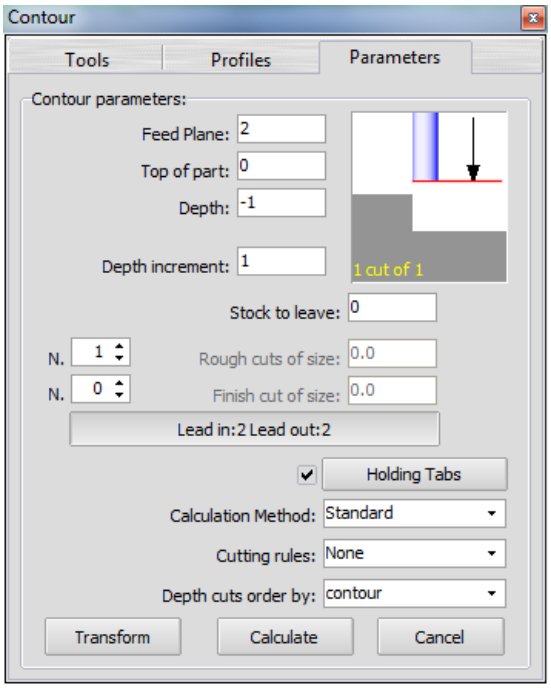
Setting this option, SimplyCam inserts at the beginning of profile G41 (left compensation in control) or G42 (right compensation in control), and at the end of the profile G40 (compensation off)

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Contour Parameters

This dialog contains the parameters specific for the Contour toolpath.



Feed Plane

The parameter set the Z coordinate at which the tool move at rapid traverse (G0).
From this position, the tool move in Z to enter into the workpiece, with feedrate (G1).
The "Feed Plane" parameter also set the Z coordinate at which the tool retract, after the processing, or before performing the move rapidly between the various profiles of the working or between the different operations (absolute).

Top of Part

This parameter set the Z coordinate of the upper surface of the workpiece/material (absolute).

Depth

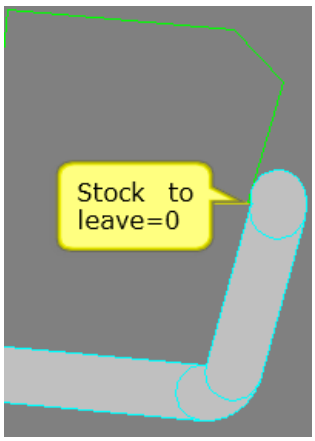
This parameter set the final machining depth (absolute).

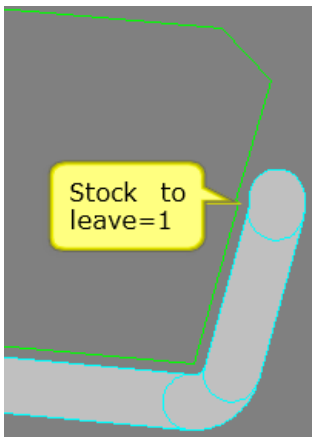
Depth Increment

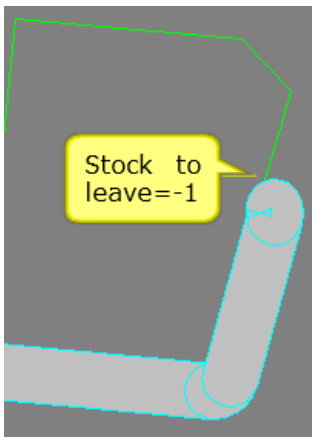
Set the maximum value of material removed for each Z cut.

Stock to leave

Sets the value of material to leave (or remove) on the profile; example, to perform a subsequent finish pass with another tool.
A positive value leaves the material.
A negative value removes material.

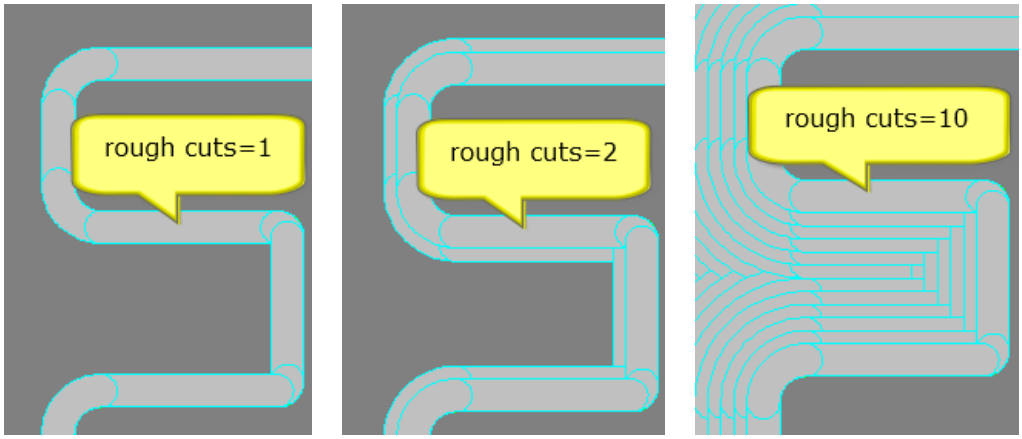






Number of rough cuts / Rough cut size

SSet the number of rough passes and the amount of material to remove for each rough pass.

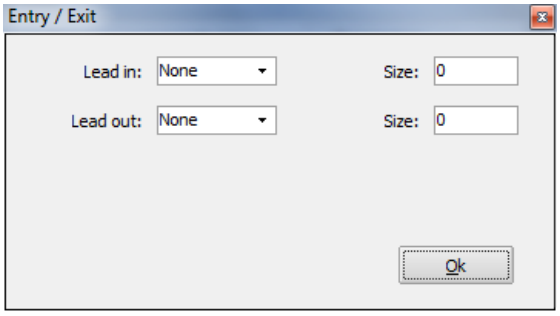


Number of finish cuts / Finish cut size

Same to the previous parameter "Rough cuts". Allows stock removal different between roughing and finishing cuts.

Lead In / Lead Out

This dialogs controlling how the tool approaches and retracts to the part at the start and at the end of profile.
More information....



Tab

This parameter allows, in case the machining removes all the material around it, to set the supports in the chained geometry, to avoid the detachment of the part.
More information ....

Method of calculation:

Defines the algorithm used in the calculation of contour machining. There are three modes:

- Standard:

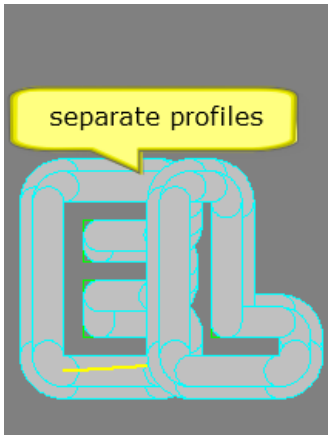
Each profile is calculated individually.
Fast calculating with complex contours by a lot of information.
Are allowed Tab
Is allowed the starting point.
Use the machining tolerance for the calculations.
Only one profile at a time is controlled and maintained intact.

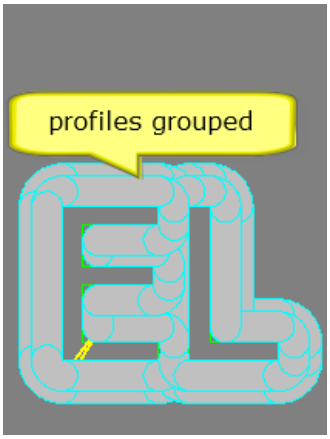
- Group profiles:

All profiles defined for the machining, are grouped as a single object before the calculation.
The adjacent profiles defined in the selection, are controlled and maintained intact.
Fast calculating with complex contours by a lot of information.
Use the machining tolerance for the calculations.
Are still not allowed Tab
We currently do not use the starting point defined in the chain.

- Pairwise Algorithm:

Very precise toolpath generation, does not use the approximation of the machining tolerance.
Each profile is calculated individually.
Are allowed Tab
Is allowed the starting point.
Slower in the calculation with complex contours by a lot of information.
Fails with lots of small entities, overlapping or self-intersecting.
Only one profile at a time is controlled and maintained intact.





Cutting Rules

This parameter determines the type of optimization in the execution of the cuts, when multiple profiles are defined.

- None

No optimization is applied to the toolpath.

- Shortest path:

With this option, SimplyCam creates the shortest path between the various profiles.

- All inside first:

SimplyCam performs before cuts inside profiles and then cuts outlines. Useful when the whole piece, composed of more than one profile, is cut and would lack external support.

- Layer priority:

SimplyCam performs the cuts, in the increasing order, with which the geometric profiles are stored on graphic layers. First will be cut profiles lying on Level 0, then the profiles lying on level 1, etc.. etc..

Depth cuts order by:

This parameter determines the Z cuts order, when multiple profiles are defined.

Contour

Removes all material of a profile (performs all cuts in depth), then proceed with the next profile (all the cuts in depth), repeated for all profiles.

- Depth:

SimplyCam removes all material from all the profiles to the current Z depth, then proceeds with the removal of all the material to the next Z depth.

Transform

This option allows you to arrange multiple parts on the same workpiece.

Calculate button

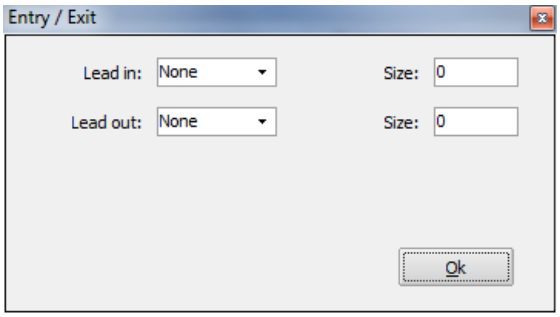
Performs the calculation process of the toolpath, using the chained profiles and the current parameters defined.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Lead In / Lead Out

This dialogs controlling how the tool approaches and retracts to the part at the start and at the end of profile.



Lead In / Size

This option adds a line or arc the beginning of each rough cut and / or finish cut of the profile.

- Line:
A perpendicular line is added to the start of the pass.

- Arc:
An tangent arc is added to the start of the pass.

- Size
If the lead in is a line, the value refers to the length, however, if the lead in is an arc, the value refers to the radius.

Lead out / Size

This option adds a line or an arc at the end of each rough cut and / or finish cut of the profile.

- Line:
A perpendicular line is added at the end of pass.

- Arc:
An tangent arc is added at the end of the pass.

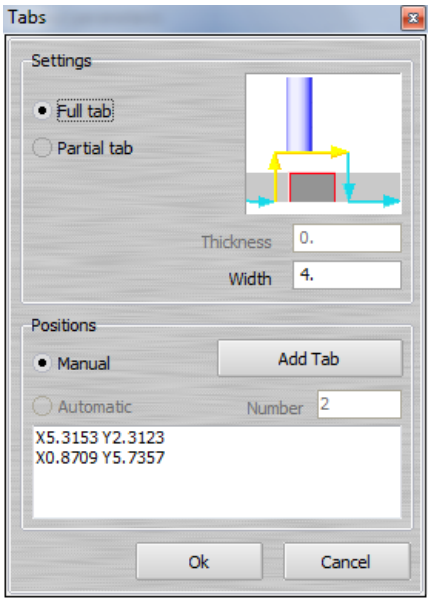
- Size
If the lead out is a line, the value refers to the length, however, if the lead out is an arc, the value refers to the radius.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Tabs

This dialog allows you to insert a 'holding tab' on the part at a location of your choosing.



Tab Full / Partial

Select Full for tabs that equal the full thickness of the material.
Select Partial for tabs less than the full thickness of the material. For partial tabs, enter the Tab thickness.

Width

Enter the tab width.

Tab Manual / Automatic

Select Manual, then choose the Position button to place each tab on the part.
Select Automatic, then enter the number of tabs. SimplyCam distributes the tabs evenly around the part.

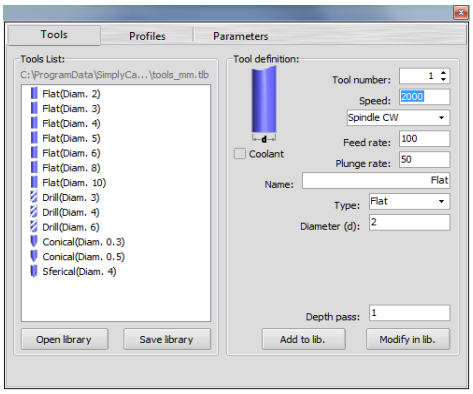
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

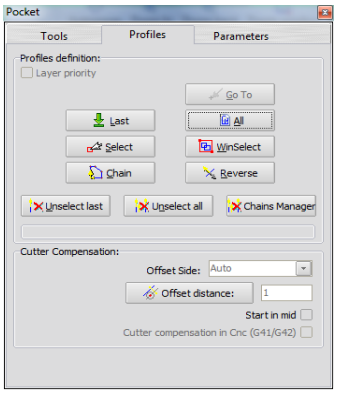
SimplyCam V3. Documentation

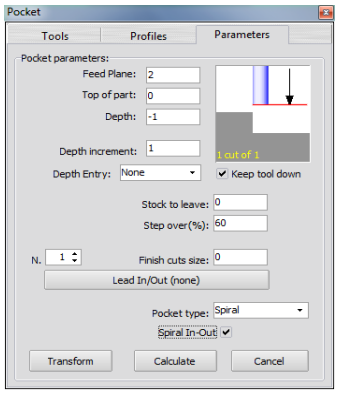
Pocket toolpath

The Pocket toolpat removes with roughing and finishing all the material inside to one or more closed contours.
The outer profile determines the "outer edge" of the pocket, while the profiles than are internal to this limit are called "islands".
You can select an unlimited number of profiles for each operation, but all the selected profiles, they will use the same parameters in the operation.

Dialogs of Pocket toolpath







Dialog 1 - Tool Definition

This dialog sets the technological parameters of the tool. These parameters directly affect the G-code generated by the postprocessor.
More information ....

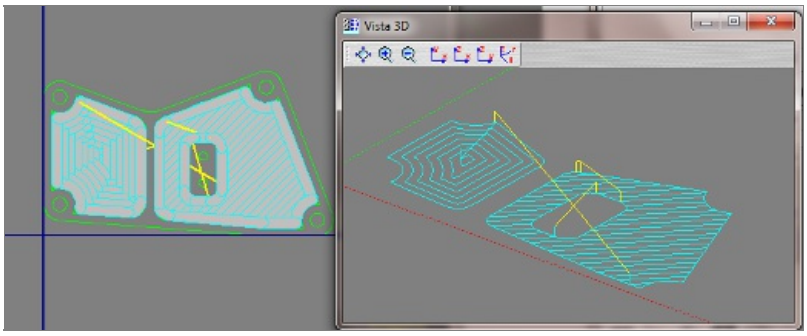
Dialog 2 - Defining Profiles

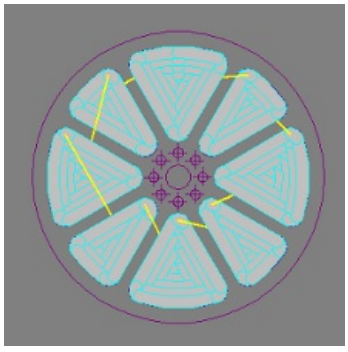
Use this dialog to define and edit the profiles for the selected operation.
This box is also used to define the tool compensation and the lead in /lead out.
More information ....

Dialog 3 - Pocket Parameters

This dialog contains the parameters specific for the Pocket toolpath.
More information ....

Examples of Pocket toolpath.



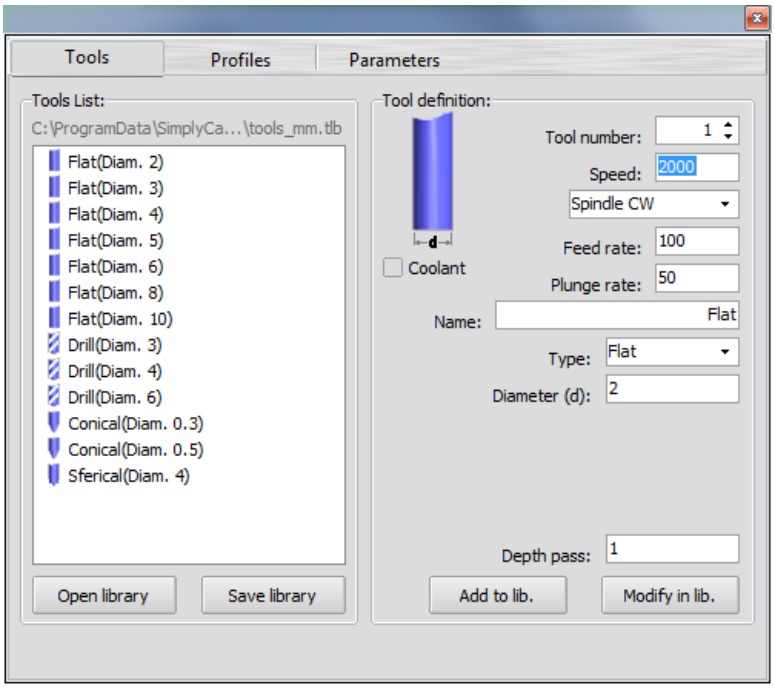


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Tools

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values â€‹â€‹that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Speed

This parameter sets the number of revolutions of the spindle.
SimplyCam normally calculates this value in revolutions / min (rpm) and produces the G-Code file the statement S (eg S3000)

Spindle: Off, CW, CCW

Sets the direction of rotation of the spindle. (Ex: M03)

Feed rate

This value controls the velocity (usually expressed in millimeters / minute or inches / minute) feed to the axes X and Y.

Plunge rate

This value controls the velocity of movement of tool penetration. The Plunge rate is only applies to tool Z movement in the negative direction (usually expressed in mm / min or inches / min).

Coolant

Turn on / Turn off the coolant.

Name

This field allows you to assign an alphanumeric string to identify the tool.

Type

This option allows you to choose the type of tool proposed by SimplyCam.

Diameter

This value sets the diameter of the SimplyCam used to calculate the toolpaths and tool compensation.

Depth pass

This value is not currently used in the calculation of the processes. Use it as a reminder.

Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

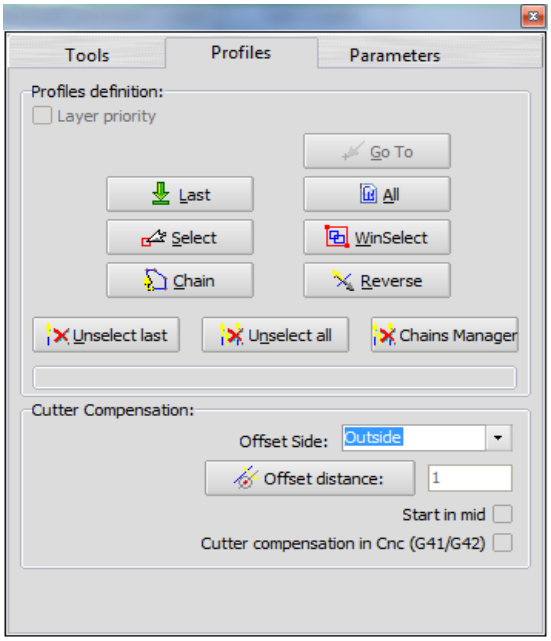
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Profiles

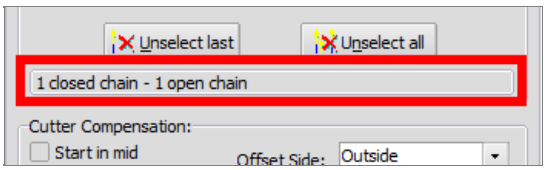
This dialog allows you to define the profiles on which to perform the pocketing machining.
It also allows you to define the tool compensation.

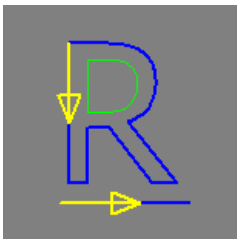


Profiles Definition section

Each profile defined, it is displayed on the screen with an arrow indicating the direction of the cut and the point of start.
It also increased the indicator in bottom section, showing the number and type of defined profiles.

Note: the Pocket toolpath does not allow open profiles.





Last
The chain defined in the last preceding operation is selected.

All
All entities on the active layers (visible) are selected and chained automatically, if possible.

Select
The selected entities, one by one, at the end are concatenated to create one or more profiles (chains).

Window Select
All entities lying inside the rectangular window are selected and concatenated to create one or more profiles (chains).

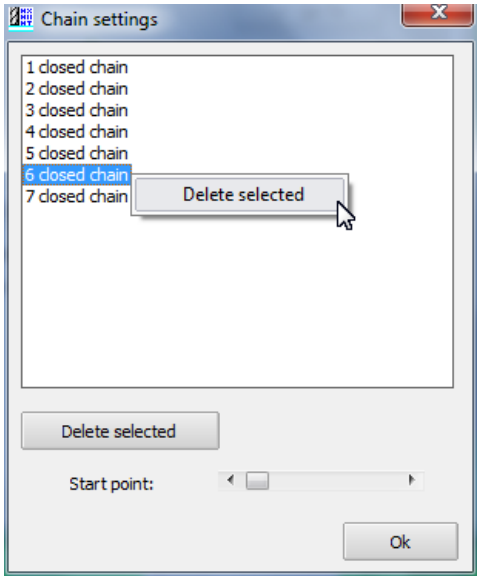
Chain
Clicking on an entity, all entities connected to the respective end points, are concatenated to form the profile (chain).

Reverse
Reverses the direction of the chains.

Unselect Last
The last defined chain is removed from the selection list.

Unselect All
All defined chains are removed from the selection list.

Chains Manager
This parameter open the dialog box containing the list of all chains with this options:



- Move the order in which they chains are performed in the machining
- Delete one or more chains
- Move the starting point each single chain.

Cutter Compensation section

Offset Side

This parameter, which determines the offset direction, ie the direction of the tool relative to the profile, is handled automatically by the pocket function.
The external profiles are offset towards the inside, while the inner profiles (islands) are offset towards the outside.

Offset distance

This value corresponds to the tool radius. Pressing the button refers to the dialog of selection tool.

Start in mid

With this option, the starting point of the finishing passes, it is moved to the middle of the first entity.
Useful to avoid having the attack on an inner corner of the path.

Cutter compensation in Cnc (G41/G42)

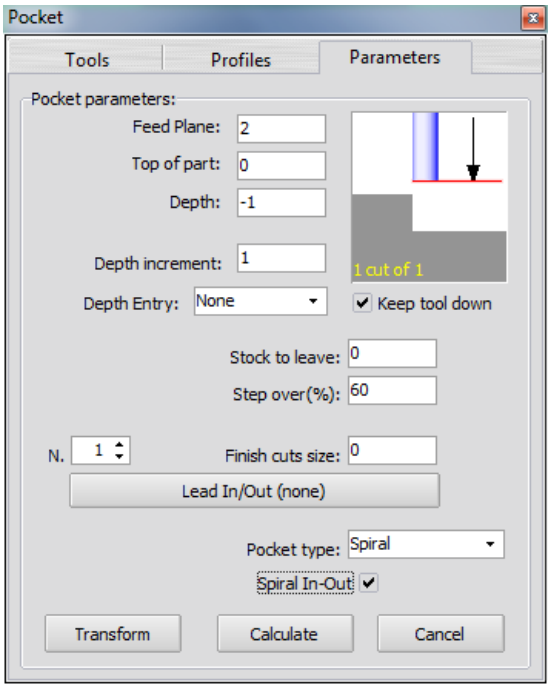
Setting this option, SimplyCam inserts at the beginning of profile G41 (left compensation in control) or G42 (right compensation in control), and at the end of the profile G40 (compensation off)

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Pocket parameters

This dialog contains the parameters specific for the Pocket toolpath.



Feed Plane

The parameter set the Z coordinate at which the tool move at rapid traverse (G0).
From this position, the tool move in Z to enter into the workpiece, with feedrate (G1).
The "Feed Plane" parameter also set the Z coordinate at which the tool retract, after the processing, or before performing the move rapidly between the various profiles of the working or between the different operations (absolute).

Top of Part

This parameter set the Z coordinate of the upper surface of the workpiece/material (absolute).

Depth

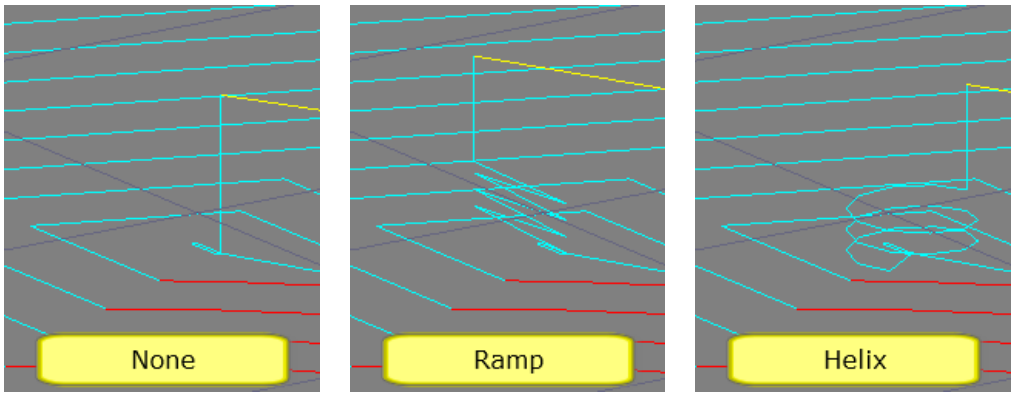
This parameter set the final machining depth (absolute).

Depth Increment

Set the maximum value of material removed for each Z cut.

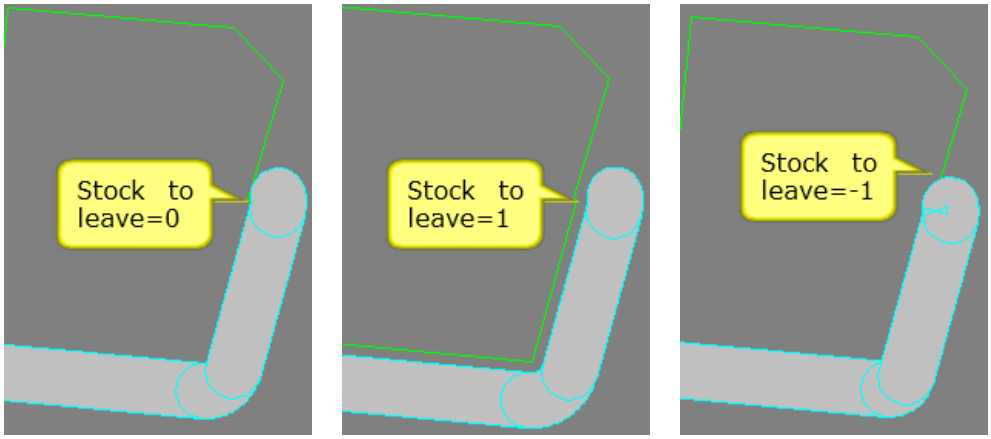
Depth Entry

The Depth Entry parameters contains options for adding ramp or helix to the pocket toolpath entry.



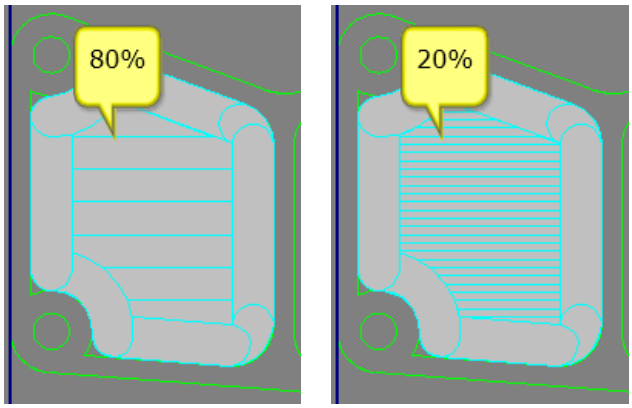
Stock to leave

Sets the value of material to leave (or remove from) the outer contour and the inner islands of the pocket, such as performing a subsequent finishing with another tool.
A positive value leaves the material.
A negative value removes material.



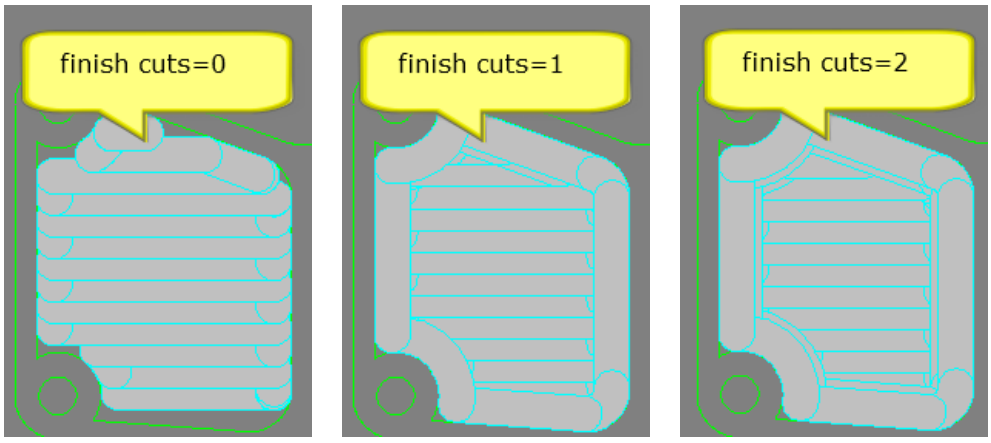
Step Over %

Set the spacing between each XY roughing pass.
This value is defined as a percentage of the cutter diameter.



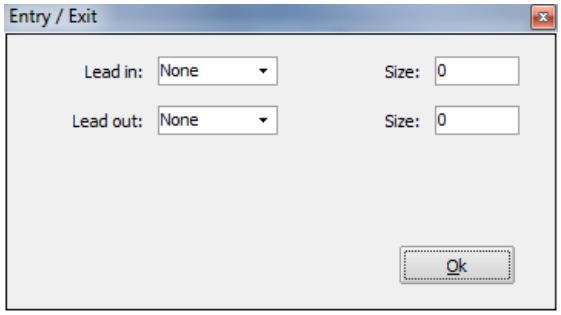
Number of finish cuts / Finish cut size

Set the number of finishing cuts and the removal value for each finishing cut on the outside walls of the pocket and on the walls of the islands.



Lead In / Lead Out

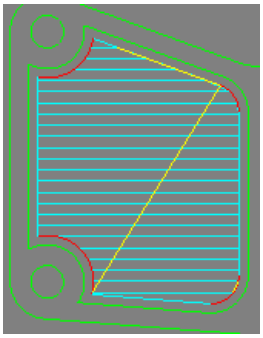
This dialogs controlling how the tool approaches and retracts to the part at the start and at the end of profile.
More information....



Pocket Type:

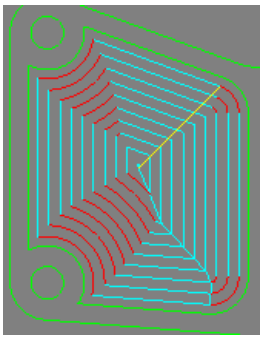
This parameter sets the roughing method of the pocket.

- Zig Zag:
Roughs the pocket in a linear motion according to the roughing angle



- Spiral:

Roughs the pocket in parallel profiles from inside to outside or vice versa, from the outside to inside.



Transform

This option allows you to arrange multiple parts on the same workpiece.

Calculate button

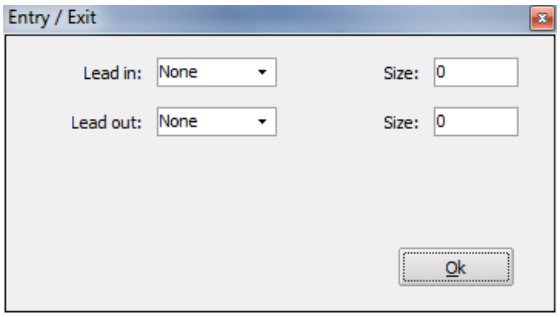
Performs the calculation process of the toolpath, using the chained profiles and the current parameters defined.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Lead In / Lead Out

This dialogs controlling how the tool approaches and retracts to the part at the start and at the end of profile.



Lead In / Size

This option adds a line or arc the beginning of each rough cut and / or finish cut of the profile.

- Line:
A perpendicular line is added to the start of the pass.

- Arc:
An tangent arc is added to the start of the pass.

- Size
If the lead in is a line, the value refers to the length, however, if the lead in is an arc, the value refers to the radius.

Lead out / Size

This option adds a line or an arc at the end of each rough cut and / or finish cut of the profile.

- Line:
A perpendicular line is added at the end of pass.

- Arc:
An tangent arc is added at the end of the pass.

- Size
If the lead out is a line, the value refers to the length, however, if the lead out is an arc, the value refers to the radius.

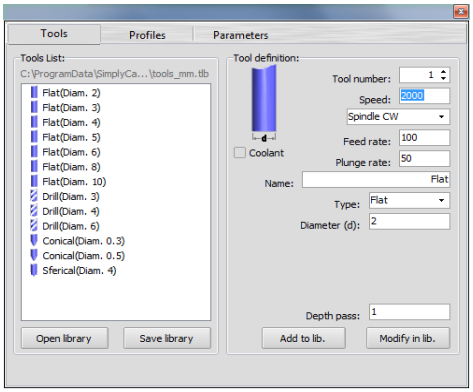
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

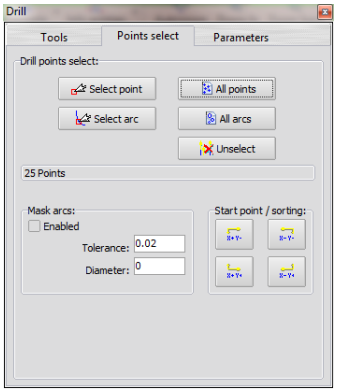
SimplyCam V3. Documentation

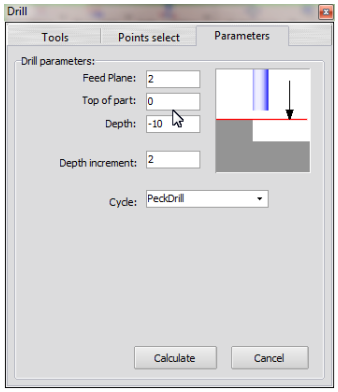
Drill toolpath

The Drill toolpath will allow you to drilling holes selecting entities point, arcs or circles center.
Running the postprocessor, SimplyCam translates the canned cycle in code G81, G82, G83, G84, G80. If the postprocessor does not support these codes, SimplyCam explodes the cycle in linear move G0 and G1.

Dialogs of the Drilling Cycle







Dialog 1 - Tool Definition

This dialog sets the technological parameters of the tool. These parameters directly affect the G-code generated by the postprocessor.
More information ....

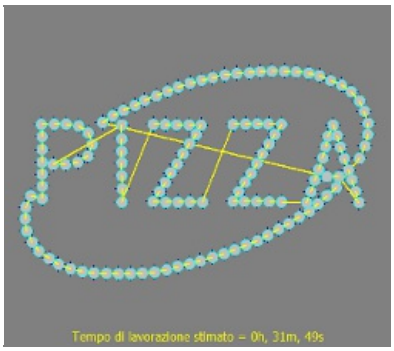
Dialog 2 - Points Select

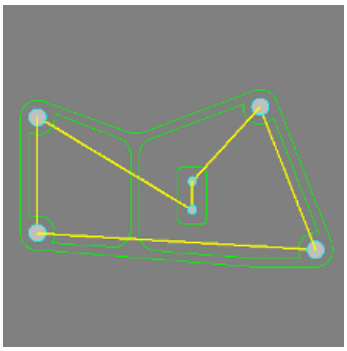
Use this dialog to define the points on which to apply the Drill Cycle.
More information ....

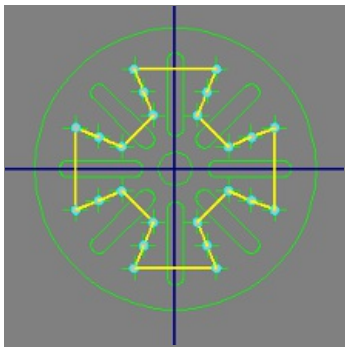
Dialog 3 - Drill Cycle Parameters

This box contains the parameters specific to the Drill Cycle toolpath.
More information ....

Examples of Drill Cycle toolpath.





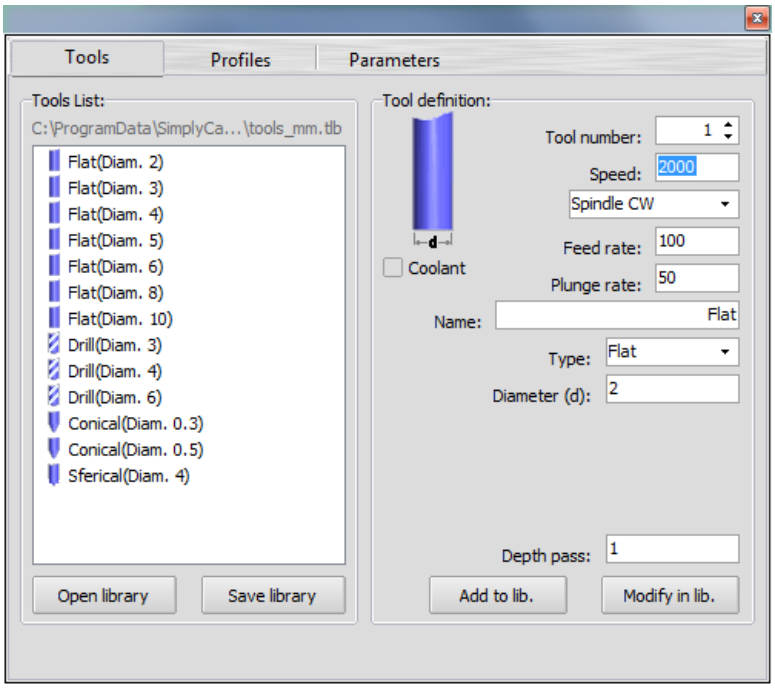


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Tools

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values â€‹â€‹that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Speed

This parameter sets the number of revolutions of the spindle.
SimplyCam normally calculates this value in revolutions / min (rpm) and produces the G-Code file the statement S (eg S3000)

Spindle: Off, CW, CCW

Sets the direction of rotation of the spindle. (Ex: M03)

Feed rate

This value controls the velocity (usually expressed in millimeters / minute or inches / minute) feed to the axes X and Y.

Plunge rate

This value controls the velocity of movement of tool penetration. The Plunge rate is only applies to tool Z movement in the negative direction (usually expressed in mm / min or inches / min).

Coolant

Turn on / Turn off the coolant.

Name

This field allows you to assign an alphanumeric string to identify the tool.

Type

This option allows you to choose the type of tool proposed by SimplyCam.

Diameter

This value sets the diameter of the SimplyCam used to calculate the toolpaths and tool compensation.

Depth pass

This value is not currently used in the calculation of the processes. Use it as a reminder.

Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

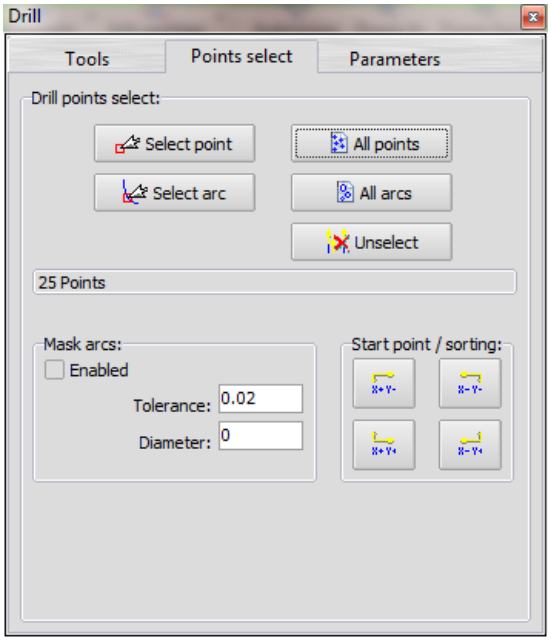
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

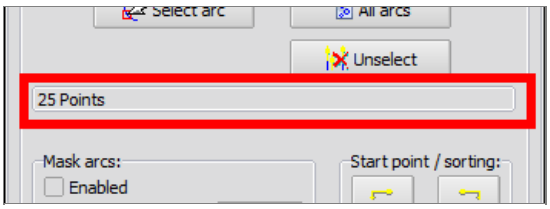
Points Select

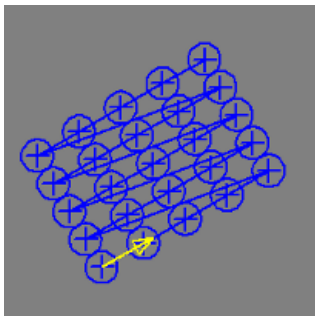
This dialog allows you to define the points on which they are executed the Drilling cycles.
You can also automatically select the center points of arcs with a specific diameter.



Drill Points Select section

Each point defined, it is displayed on the screen with a cross and a circle (equal to the tool diameter).
An arrow indicates the direction of path and the starting point.
It also increased the bottom indicator in section, showing the number defined points.





Select Point

Entities point, select one by one, are added to the points list of the drill cycle.

All Points

All points entities, on the active layers (visible), are added to the points list of the drill cycle.

Select Arc

The centers of the arc entity, select one by one, are added to the points list of the drill cycle.
If "Mask Arcs" is active, the selection is limited only to the arcs of defined diameter.

All Arcs

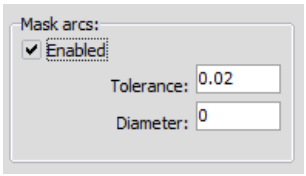
All the centers of the arcs entities, on the active layers (visible), are added to the points list of the drill cycle.
If "Mask Arches" is active, the selection is limited only to the arcs of defined diameter.

UnSelect

All the points are removed from the selection list.

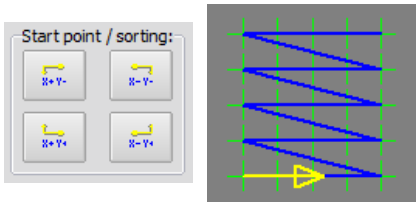
Mask Arcs section

If this section is active, the selection is limited only to the arcs of defined diameter (+ or - tolerance defined).

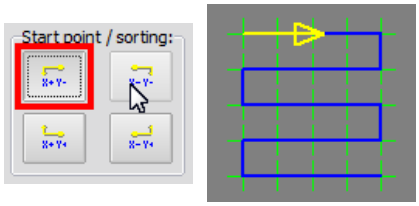


Start point / Sorting section

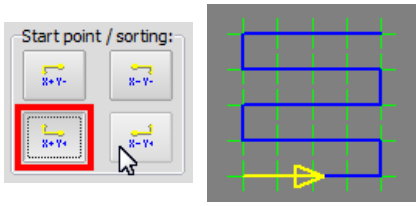
No sorting:



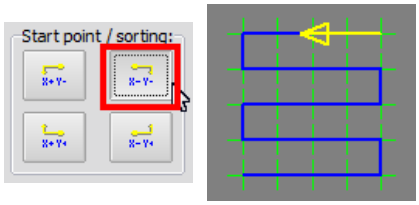
Sorting X + Y-:



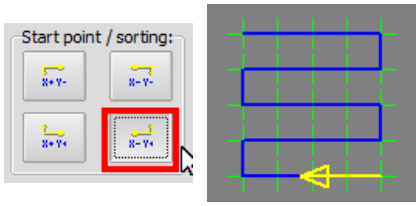
Sorting X + Y +:



Sorting X-Y-:



Sorting X-Y +:

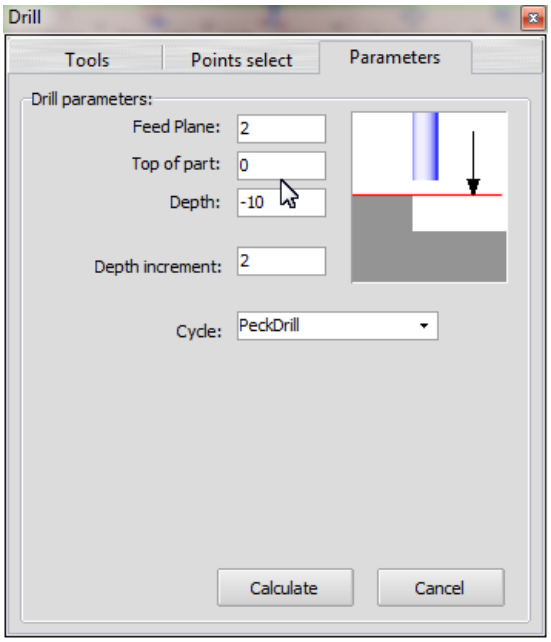


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Drill Cycle Parameters

This box contains the parameters specific to the Drill Cycle toolpath.



Feed Plane

The parameter set the Z coordinate at which the tool move at rapid traverse (G0).
From this position, the tool moves in Z to enter into the workpiece, with feedrate (G1).
The "Feed Plane" parameter also set the Z coordinate at which the tool retract, after the processing, or before performing the move rapidly between the various profiles of the working or between the different operations (absolute).

Top of Part

This parameter set the Z coordinate of the upper surface of the workpiece/material (absolute).

Depth

This parameter sets final drilling depth (absolute).

Depth Increment

Set the maximum value of material removed when the PeckDrill cycle is selected (intermittent cycle or cycle with chip removal).

Pitch

Sets the value of the thread pitch when the Tap cycle is selected (threading cycle).

Cycle

Defines the type of cycle applied to the selected points.

- Drill Cycle (Simple drilling):

Rapid move to the hole center with XY axis
Rapid Z to "Feed plane" (Reference height)
Feed Z down to the "Depth" position
Rapidly retract Z to "Feed plane" (Reference height)
Rapid move with the XY axes on the center of next hole
Repeat this sequence

- PeckDrill Cycle (drilling with chip removal):

Rapid move to the hole center with XY axis
Rapid Z to "Feed plane" (Reference height)
Move the Z axis at the feed rate by "Depth Increment"
Rapidly retract Z to "Feed plane" (Reference height)
Rapid Z down to clearance up from the previous drilled depth
New feed Z down with one "Depth increment"
Repeat the last 3 step until the drilling bottom is reached
Rapidly retract Z to "Feed plane" (Reference height)
Rapid move with the XY axes on the center of next hole
Repeat this sequence

- Tap Cycle (thread):

Rapid move to the hole center with XY axis
Rapid Z to "Feed plane" (Reference height)
Feed Z down to the "Depth" position
The spindle is rotated in the reverse direction
Feed retract Z to "Feed plane" (Reference height)
The spindle is rotated in the normal direction
Rapid move with the XY axes on the center of next hole
Repeat this sequence

Calculate button:

Performs the calculation process of the Drill toolpath, using the selected points and the current defined parameters.

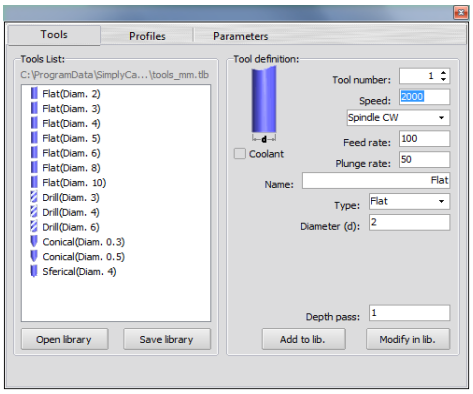
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

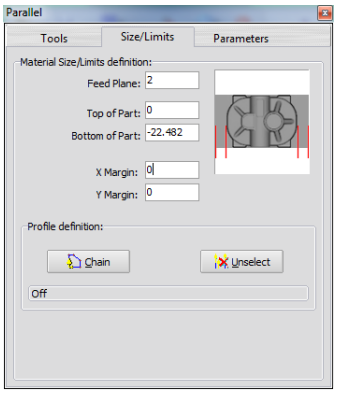
SimplyCam V3. Documentation

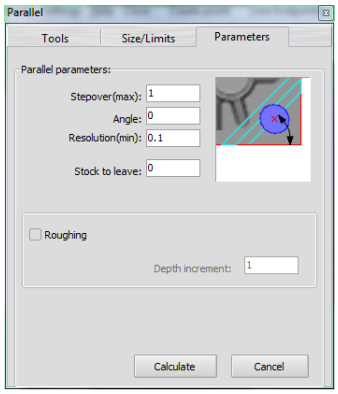
Parallel Toolpath

The Parallel toolpath (typically used for finishing) is similar to a 2D zig-zag pocket toolpath that has been projected down on to a 3D Stl part. This process can also be used as roughing, activating the option and entering the depth increment.

Dialogs of Parallel Toolpath







Dialog 1 - Tool Definition

This dialog sets the technological parameters of the tool. These parameters directly affect the G-code generated by the postprocessor.
More information ....

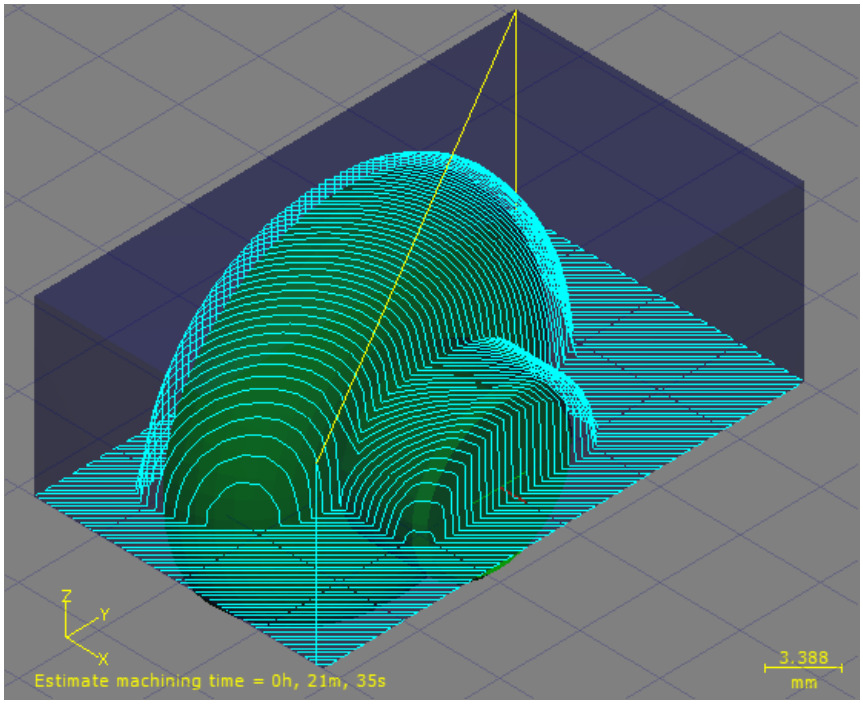
Dialog 2 - Size/Limits

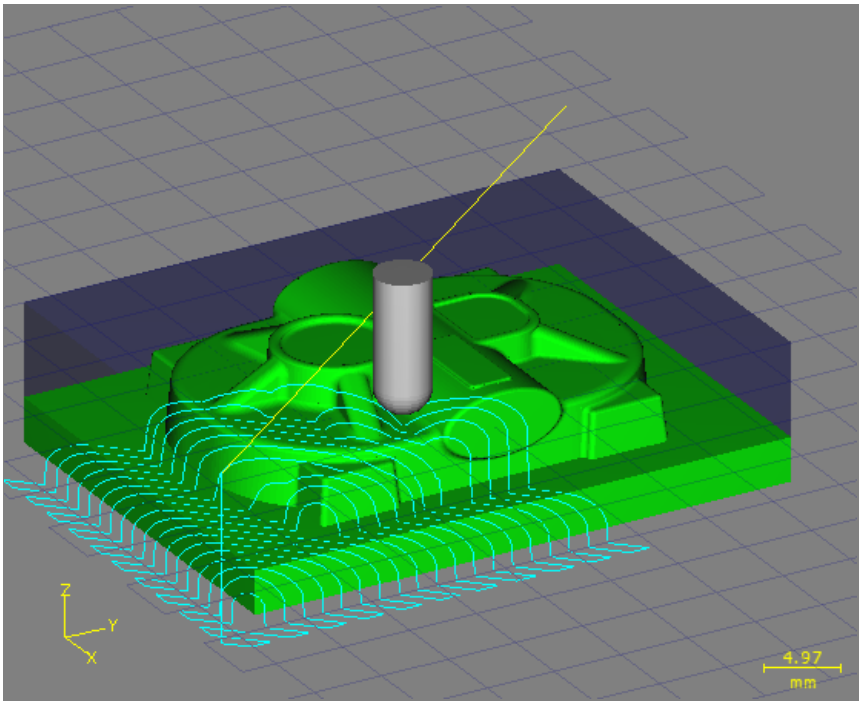
This dialog is used to define the size of the material and/or the limits of machining.
More information ....

Dialog 3 - Parallel Parameters

This dialog contains the parameters specific for the Parallel toolpath.
More information ....

Examples of Parallel toolpath



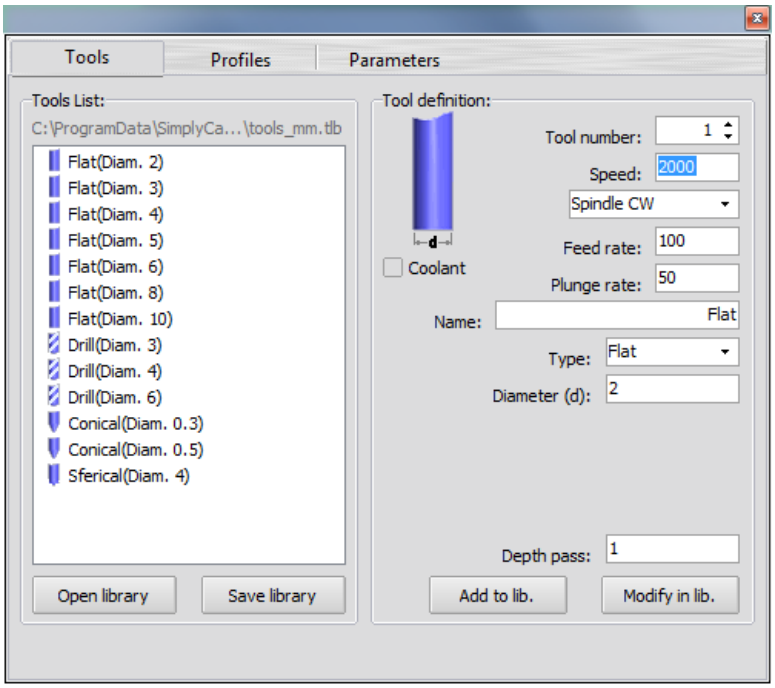


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Tools

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values â€‹â€‹that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Speed

This parameter sets the number of revolutions of the spindle.
SimplyCam normally calculates this value in revolutions / min (rpm) and produces the G-Code file the statement S (eg S3000)

Spindle: Off, CW, CCW

Sets the direction of rotation of the spindle. (Ex: M03)

Feed rate

This value controls the velocity (usually expressed in millimeters / minute or inches / minute) feed to the axes X and Y.

Plunge rate

This value controls the velocity of movement of tool penetration. The Plunge rate is only applies to tool Z movement in the negative direction (usually expressed in mm / min or inches / min).

Coolant

Turn on / Turn off the coolant.

Name

This field allows you to assign an alphanumeric string to identify the tool.

Type

This option allows you to choose the type of tool proposed by SimplyCam.

Diameter

This value sets the diameter of the SimplyCam used to calculate the toolpaths and tool compensation.

Depth pass

This value is not currently used in the calculation of the processes. Use it as a reminder.

Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

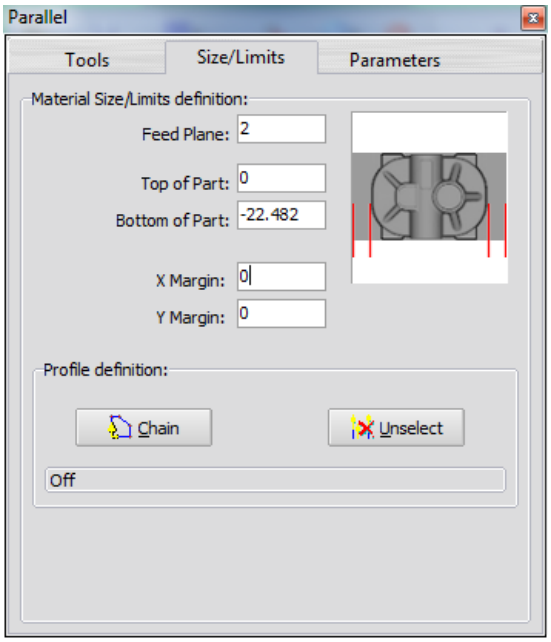
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Size/Limits

This dialog is used to define the size of the material and/or the limits of machining.



Feed Plane

The parameter set the Z coordinate at which the tool move at rapid traverse (G0).
From this position, the tool move in Z to enter into the workpiece, with feedrate (G1).
The "Feed Plane" parameter also set the Z coordinate at which the tool retract, after the processing, or before performing the move rapidly between the various profiles of the working or between the different operations (absolute).

Top Part

The parameter sets the top Z of the part. It is used as the upper limit for roughing.

Bottom Part

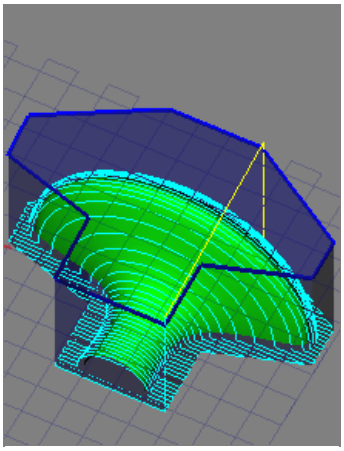
The parameter sets the lower Z of the lower part. It is used as the lower limit or supporting plane of the tool in parallel passes.

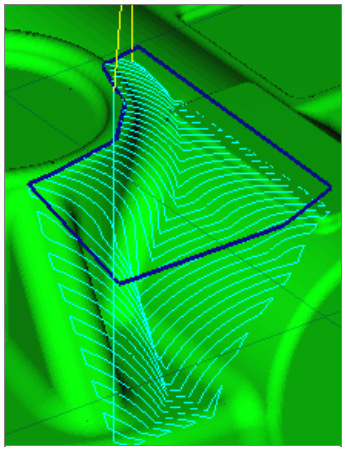
Margine X / Y

These parameters extend or restrict the stock and consequently the machining area in the X and Y directions. Enter a positive value to extend or a negative value to restrict.

Section definition Profile

This section is used to define a strock of irregular shape or to contain the machining.





Chain

Clicking on an entity, all entities connected to the respective endpoints, are concatenated to form the profile (chain) that contains the machining.

Unselect

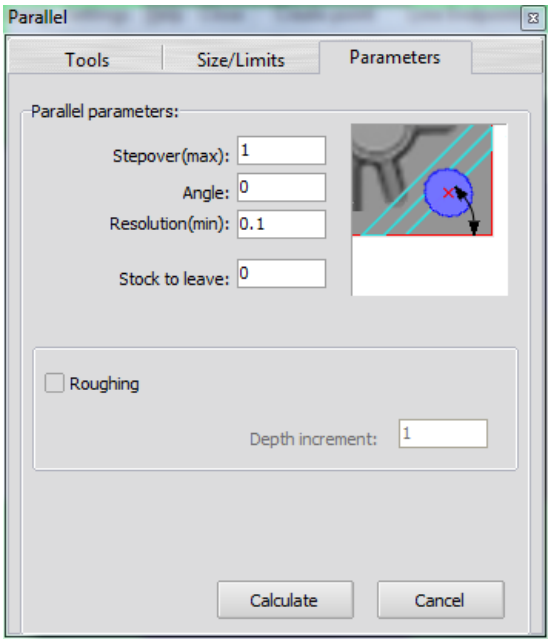
Il profilo di contenimento è rimosso dalla selezione.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Parallel Parameters

This dialog contains the parameters specific for the Parallel toolpath.



Step Over
Sets the distance between each pass in XY plane.

Angle
Sets the machining angle for the Parallel toolpath from 0 to 360 degrees in the XY plane.

Resolution
This option determines the accuracy of the toolpath. Smaller value provide more accuracy but can result in slower performance.

Stock to leave
Sets the value of material to leave on the Stl model.

Roughing
If activated, performs the roughing of the model from the upper limit(Top Part) to the lower limit(Bottom Part).

Depth Increment
Set the value of material removed for each roughing Z cut.

Calculate button
Performs the calculation process of the Pocket toolpath, using the chained profiles and the current parameters defined.

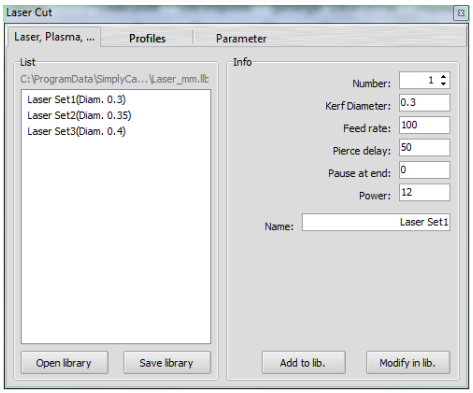
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

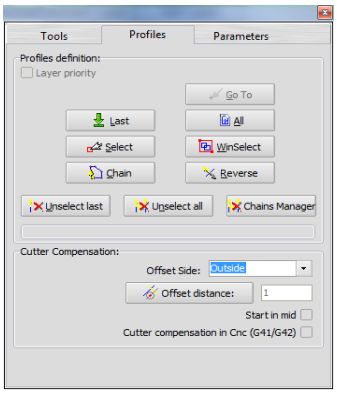
SimplyCam V3. Documentation

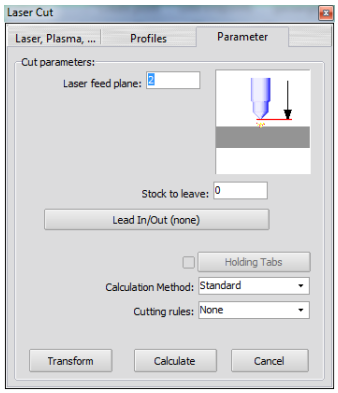
Cut Toolpath (Laser, Plasma, Waterjet)

The Cut toolpaths remove material along one or more profiles. You can select an unlimited number of profiles for each operation, but all the selected profiles, they will use the same parameters of the operation.
The selected profiles can be offsetting internally/externally (left/right for the open profiles) to compensate the kerf diameter.

Dialogues of Cut Toolpath (Laser, Plasma, Waterjet)







Dialog 1 - Laser, Plasma, Waterjet Settings

This dialog sets the technological parameters of the nozzles, heads, etc. These parameters directly affect the G-code generated by the postprocessor.
Maggiori informazioni....

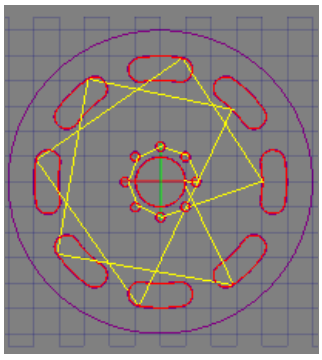
Dialog 2 - Defining Profiles

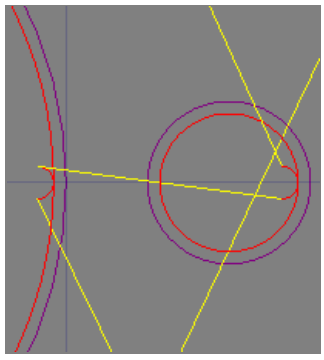
Use this dialog to define and edit the profiles for the selected operation.
This box is also used to define the tool compensation and the lead in / lead out.
More information ....

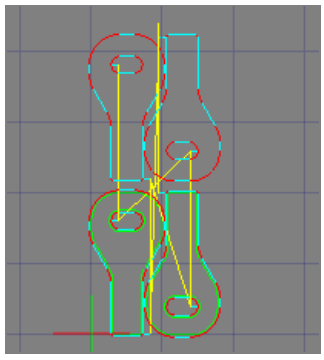
Dialog 3 - Cut Parameters

This dialog contains the parameters specific for the Cut processing of Laser, Plasma, Waterjet machine.
More information ....

Examples of Cut machining





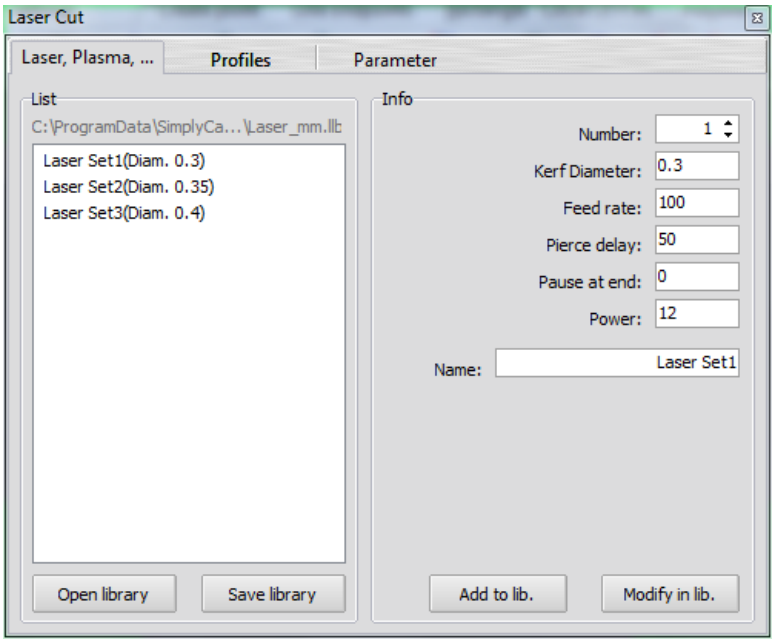


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Laser, Plasma, WaterJet Settings

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Kerf Diameter

This value sets the diameter that SimplyCam uses to calculate the toolpaths and tool compensation.
Normalmente SimplyCam calcola questo valore in giri/min (RPM) e produce nel file G-Code l'istruzione S (es: S3000)

Feed rate

This value controls the velocity (usually expressed in millimeters / minute or inches / minute) of feed to the X and Y axes.

Pierce delay

Enter the delay time here.

Pause at end

Enter the pause time here.

Power

Set the power level, if your Cnc has the power control by GCode.

Name

This field allows you to assign an alphanumeric string to identify the tool.

Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

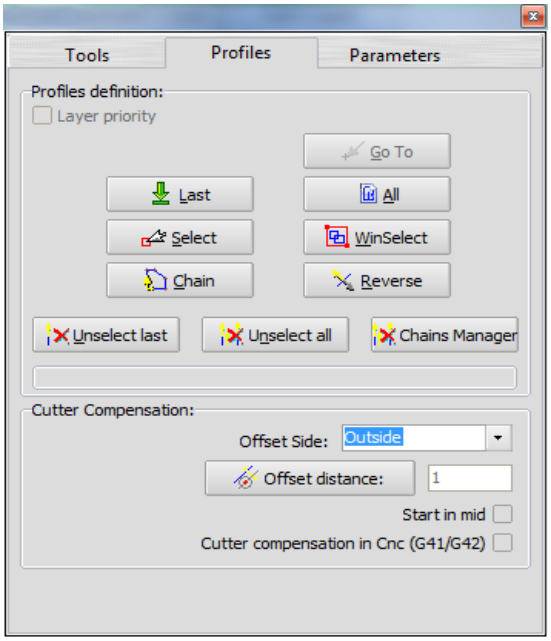
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

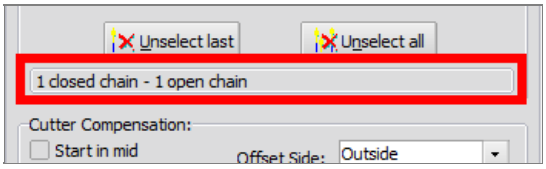
Profiles

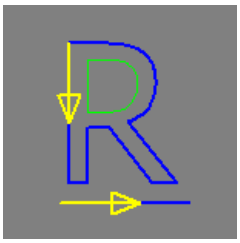
This dialog allows you to define the profiles to be machined.
It also allows you to define the tool compensation.



Profiles Definition section

Each profile defined, it is displayed on the screen with an arrow indicating the direction of the cut and the point of start.
It also increased the indicator in bottom section, showing the number and type of defined profiles.





Last
The chain defined in the last preceding operation is selected.

All
All entities on the active layers (visible) are selected and chained automatically, if possible.

Select
The selected entities, one by one, at the end are concatenated to create one or more profiles (chains).

Window Select
All entities lying inside the rectangular window are selected and concatenated to create one or more profiles (chains).

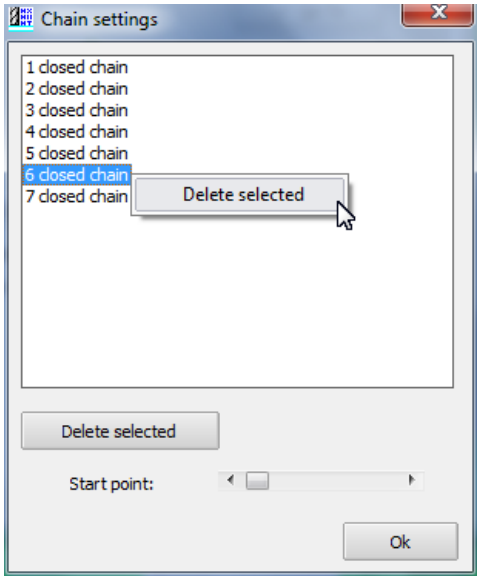
Chain
Clicking on an entity, all entities connected to the respective end points, are concatenated to form the profile (chain).

Reverse
Reverses the direction of the chains.

Unselect Last
The last defined chain is removed from the selection list.

Unselect All
All defined chains are removed from the selection list.

Chains Manager
This parameter open the dialog box containing the list of all chains with this options:



- Move the order in which they chains are performed in the machining
- Delete one or more chains
- Move the starting point each single chain.

Cutter Compensation section

Offset Side

This parameter determines the offset direction, ie the direction of the tool relative to the chained profile.
This direction is displayed on the screen with a small arrow, perpendicular to the cutting direction, which indicates the offset direction.

- Off:
No compensation, the tool is placed directly on the chained profiles.

- Outside
The tool is placed externally respect to the concatenated profile. This rule applies only to closed contours, while the open profiles this rule is not valid; normally the tool is placed to the right of the open profile.

- Inside
The tool is placed internally respect to the concatenated profile. This rule applies only to closed contours, while the open profiles this rule is not valid; normally the tool is placed to the left of the open profile.

Offset distance

This value corresponds to the tool radius. Pressing the button refers to the dialog of selection tool.

Start in mid

Setting this option, the starting point of the toolpat is moved to the middle of the first entity in the chain.
Useful to avoid having the attack on an inner corner of the path.

Cutter compensation in Cnc (G41/G42)

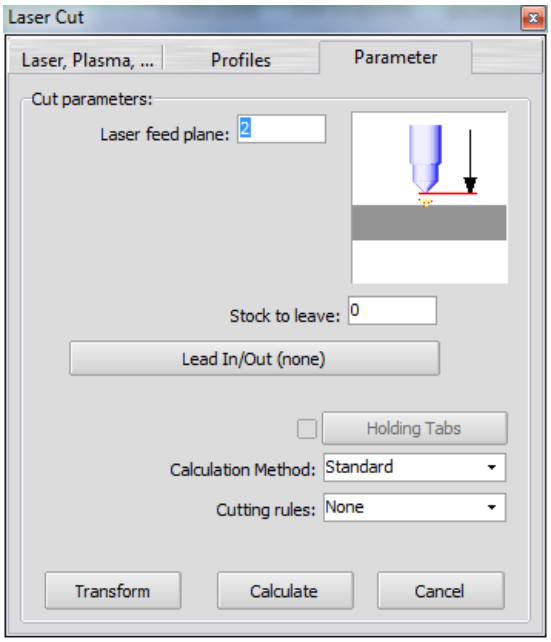
Setting this option, SimplyCam inserts at the beginning of profile G41 (left compensation in control) or G42 (right compensation in control), and at the end of the profile G40 (compensation off)

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Cut Parameters

This dialog contains the parameters specific for the Cut toolpath.

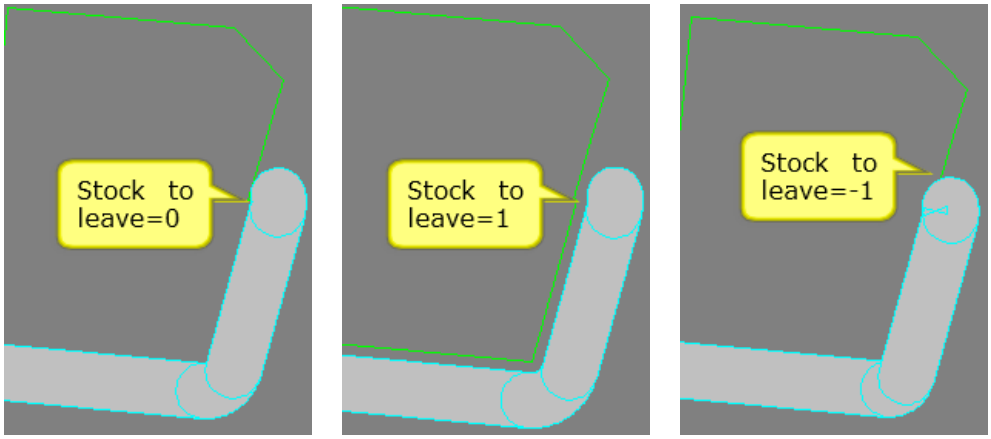


Feed Plane

The parameter sets the Z coordinate at which the tool moves at rapid traverse (G0).

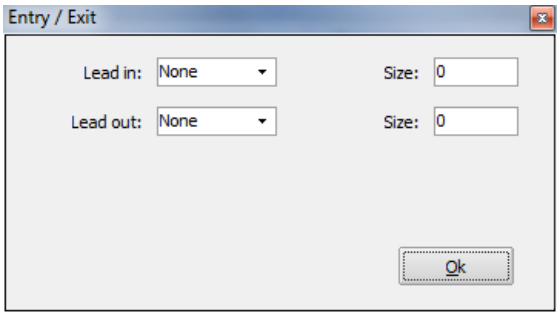
Stock to leave

Sets the value of material to leave (or remove) on the profile; example, to perform a subsequent finish pass with another tool.
A positive value leaves the material.
A negative value removes material.



Lead In / Lead Out

This parameters controlling how the tool approaches the part at the start of each pass and how the tool retracts from the part at the end of each pass in the toolpath.
More information ....



Tab

This parameter allows, in case the machining removes all the material around it, to set the supports in the chained geometry, to avoid the detachment of the part.
More information ....

Method of calculation:

Defines the algorithm used in the calculation of contour machining. There are three modes:

- Standard:

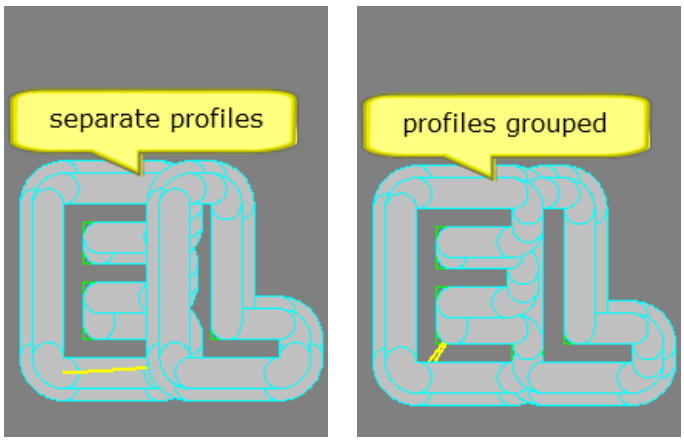
Each profile is calculated individually.
Fast calculating with complex contours by a lot of information.
Are allowed Tab
Is allowed the starting point.
Use the machining tolerance for the calculations.
Only one profile at a time is controlled and maintained intact.

- Group profiles:

All profiles defined for the machining, are grouped as a single object before the calculation.
The adjacent profiles defined in the selection, are controlled and maintained intact.
Fast calculating with complex contours by a lot of information.
Use the machining tolerance for the calculations.
Are still not allowed Tab
We currently do not use the starting point defined in the chain.

- Pairwise Algorithm:

Very precise toolpath generation, does not use the approximation of the machining tolerance.
Each profile is calculated individually.
Are allowed Tab
Is allowed the starting point.
Slower in the calculation with complex contours by a lot of information.
Fails with lots of small entities, overlapping or self-intersecting.
Only one profile at a time is controlled and maintained intact.



Cutting Rules

This parameter determines the type of optimization in the execution of the cuts, when multiple profiles are defined.

- None

No optimization is applied to the toolpath.

- Shortest path:

With this option, SimplyCam creates the shortest path between the various profiles.

- All inside first:

SimplyCam performs before cuts inside profiles and then cuts outlines. Useful when the whole piece, composed of more than one profile, is cut and would lack external support.

- Layer priority:

SimplyCam performs the cuts, in the increasing order, with which the geometric profiles are stored on graphic layers. First will be cut profiles lying on Level 0, then the profiles lying on level 1, etc.. etc..

Transform

This option allows you to arrange multiple parts on the same workpiece.

Calculate button

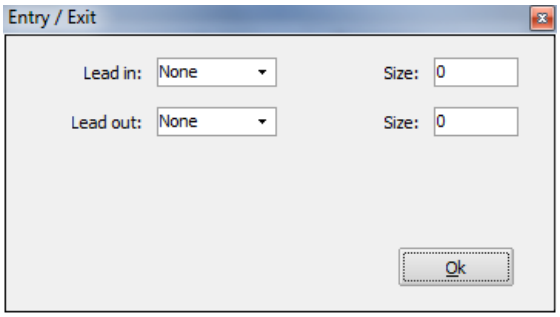
Performs the calculation process of the toolpath, using the profiles chained and the current parameters defined.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Lead In / Lead Out

This dialogs controlling how the tool approaches and retracts to the part at the start and at the end of profile.



Lead In / Size

This option adds a line or arc the beginning of each rough cut and / or finish cut of the profile.

- Line:
A perpendicular line is added to the start of the pass.

- Arc:
An tangent arc is added to the start of the pass.

- Size
If the lead in is a line, the value refers to the length, however, if the lead in is an arc, the value refers to the radius.

Lead out / Size

This option adds a line or an arc at the end of each rough cut and / or finish cut of the profile.

- Line:
A perpendicular line is added at the end of pass.

- Arc:
An tangent arc is added at the end of the pass.

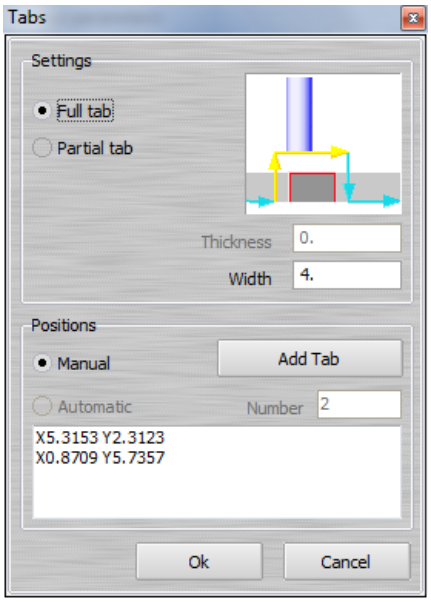
- Size
If the lead out is a line, the value refers to the length, however, if the lead out is an arc, the value refers to the radius.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Tabs

This dialog allows you to insert a 'holding tab' on the part at a location of your choosing.



Tab Full / Partial

Select Full for tabs that equal the full thickness of the material.
Select Partial for tabs less than the full thickness of the material. For partial tabs, enter the Tab thickness.

Width

Enter the tab width.

Tab Manual / Automatic

Select Manual, then choose the Position button to place each tab on the part.
Select Automatic, then enter the number of tabs. SimplyCam distributes the tabs evenly around the part.

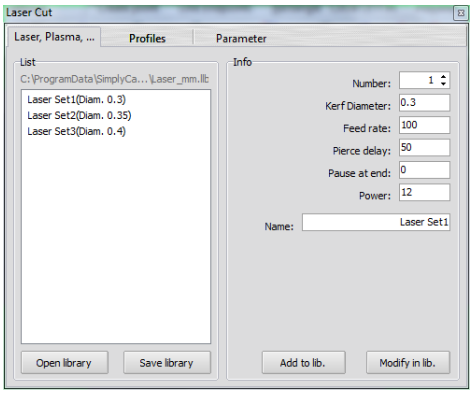
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

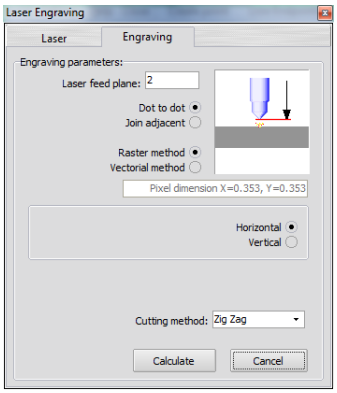
SimplyCam V3. Documentation

Laser Engraving Toolpath

This option, performs theÂ machining Laser Engraving on raster image.

Dialoghi della lavorazione Incisione





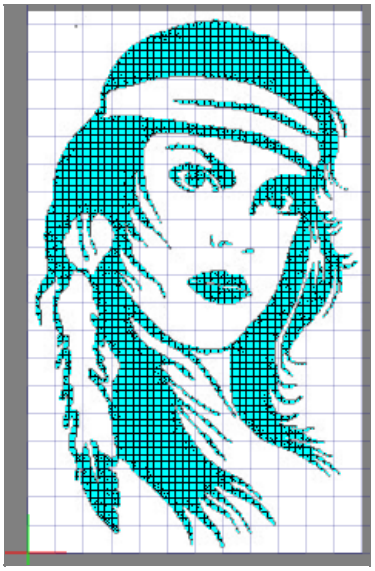
Dialog 1 - Laser Settings

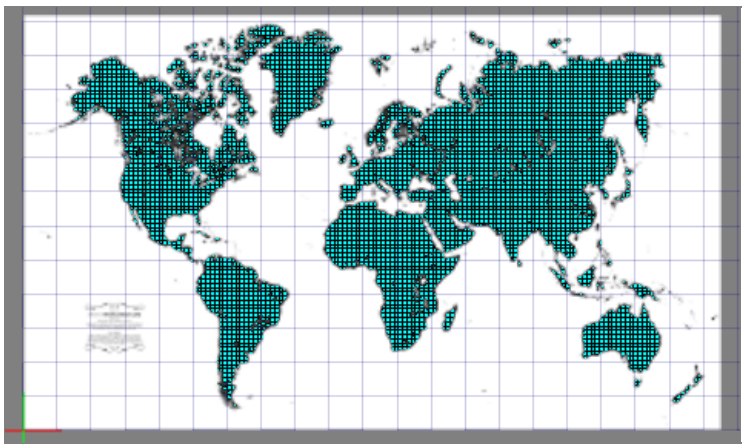
This dialog sets the technological parameters of the Laser. These parameters directly affect the G-code generated by the postprocessor.
More information ....

Dialog 2 - Engraving Parameters

Questo dialogo contiene i parametri specifici per la lavorazione Laser Incisione.
More information ....

Examples of machining Engraving



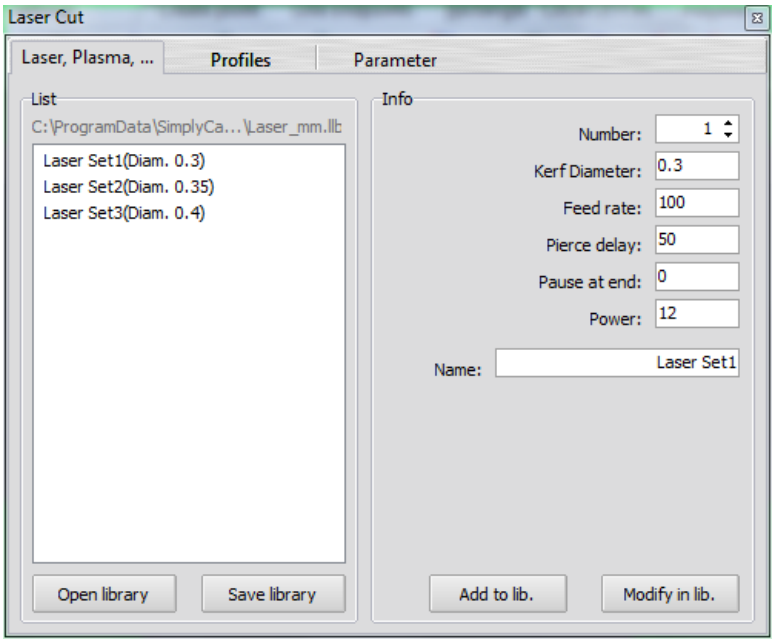


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Laser, Plasma, WaterJet Settings

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Kerf Diameter

This value sets the diameter that SimplyCam uses to calculate the toolpaths and tool compensation.
Normalmente SimplyCam calcola questo valore in giri/min (RPM) e produce nel file G-Code l'istruzione S (es: S3000)

Feed rate

This value controls the velocity (usually expressed in millimeters / minute or inches / minute) of feed to the X and Y axes.

Pierce delay

Enter the delay time here.

Pause at end

Enter the pause time here.

Power

Set the power level, if your Cnc has the power control by GCode.

Name

This field allows you to assign an alphanumeric string to identify the tool.

Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

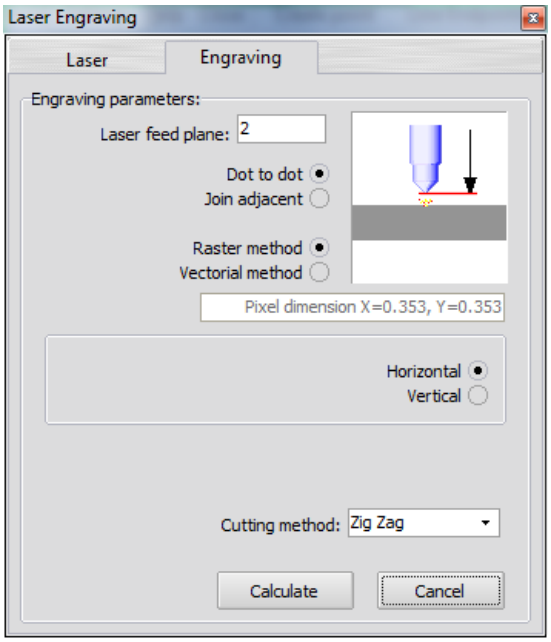
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Laser Engraving Parameters

This dialog contains the parameters specific for the Laser Engraving.



Feed Plane

The parameter sets the Z coordinate at which the tool moves at rapid traverse (G0).

Dot to Dot / Join adiacent:

This gives the choice of producing individual dots or lines joining adjacent dots.

Raster / Vectorial method

Defines the algorithm used in the calculation of engraving machining.

- Raster

Each point of the image is scanned.
Is allowed only the working direction horizontal or vertical.

- Vectorial

To scan the image is used an along step and an across step.
Each working direction is allowed.

Cutting method

This parameter allows you to choose between Zig-Zag or One Way working.

Calculate button

Performs the calculation process of the toolpath, using the current parameters defined.

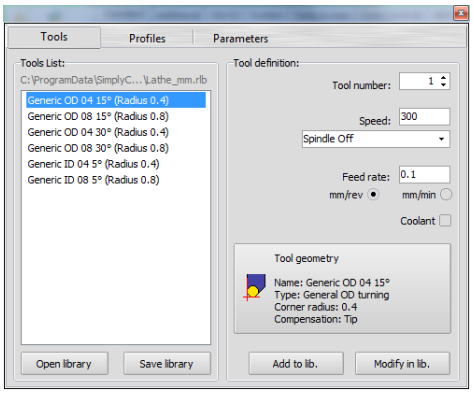
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

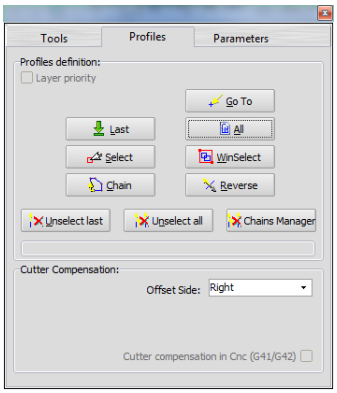
SimplyCam V3. Documentation

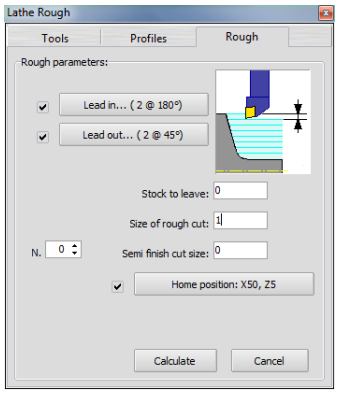
Lathe Rough Toolpath

Roughing toolpaths coarsely cut the part geometry in preparation of a finish toolpath

Dialogs of Lathe Rough Toolpath







Dialog 1 - Tool Definition

This dialog sets the technological parameters of the tool. These parameters directly affect the G-code generated by the postprocessor.
More information....

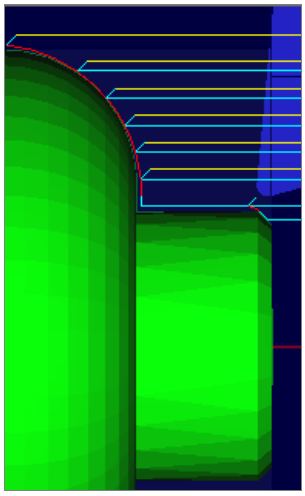
Dialog 2 - Defining Profiles

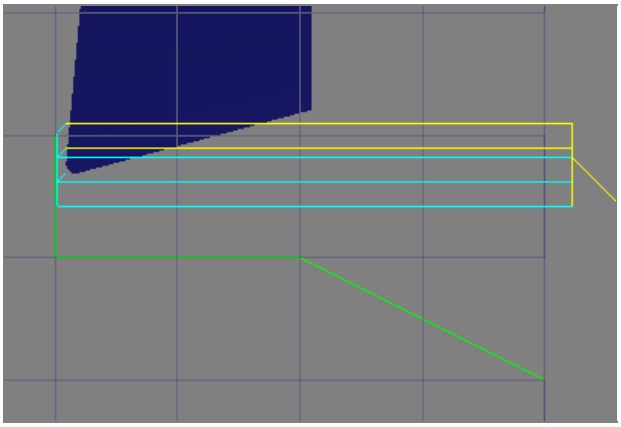
Use this dialog to define and edit the profiles for the selected operation.
The tool compensation is handled automatically.
More information....

Dialog 3 - Contour Parameters

This dialog contains the parameters specific for the Contour machining.
More information....

Examples of Lathe Rough toolpath



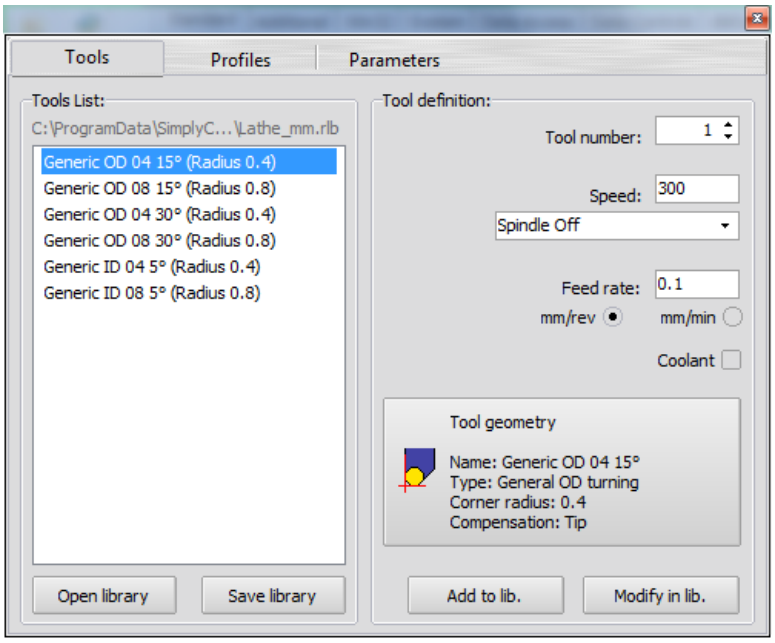


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Lathe Tools

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values â€‹â€‹that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

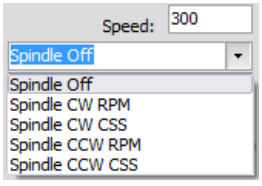
This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Speed

This parameter sets the spindle speed.
SimplyCam generates in the G-Code file the statement S (example: S3000)

Spindle:

Sets the direction of rotation of the spindle. (Ex: M03) and code of the type of cutting speed (Ex: G96)



- Spindle Off
Spindle turned off.

- Spindle CW RPM
The spindle rotates clockwise RPM (Revolutions Per Minute).

- Spindle CW CSS
The spindle rotates clockwise with Constant Surface Speed.

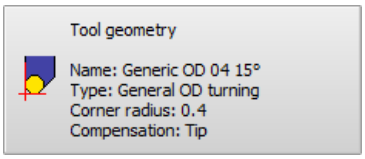
- Spindle CCW RPM
The spindle rotates counterclockwise RPM (Revolutions Per Minute).

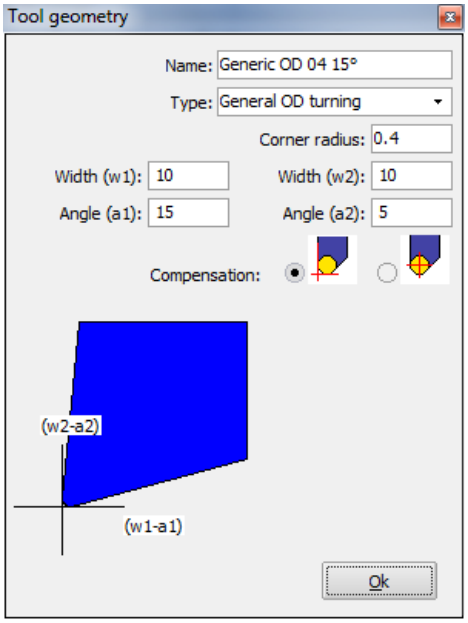
- Spindle CCW CSS
The spindle rotates counterclockwise with Constant Surface Speed.

Feed rate
This value controls the velocity (expressed in mm(inch)/revolution or in mm(inch)/minute) feed to the Cnc axes.

Coolant
Turn On / Turn Off the coolant.

Tool Geometry
This button opens the dialog box, which lets you change the geometric parameters of the tool.





Name
This field allows you to assign an alphanumeric string to identify the tool.

Type
This option allows you to choose the type of tool proposed by SimplyCam.

- General OD turning
Generic tool for external turning.

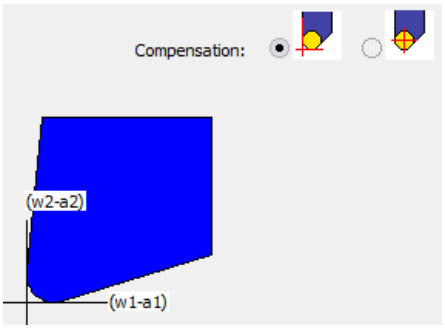
- General ID turning
Generic tool for internal turning.

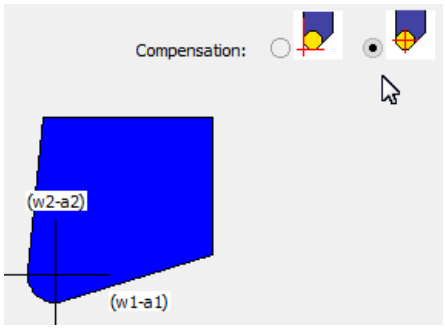
Corner radius
This value sets the radius of the SimplyCam used to calculate the toolpaths and tool compensation.

Width(W1) / Angle(A1)
These values define the geometric shape of the tool and are used in the simulation to view the tool.

Width(W2) / Angle(A2)
These values define the geometric shape of the tool and are used in the simulation to view the tool.

Compensation
Select the type of compensation in the same way as the tool is set in the CNC.





Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

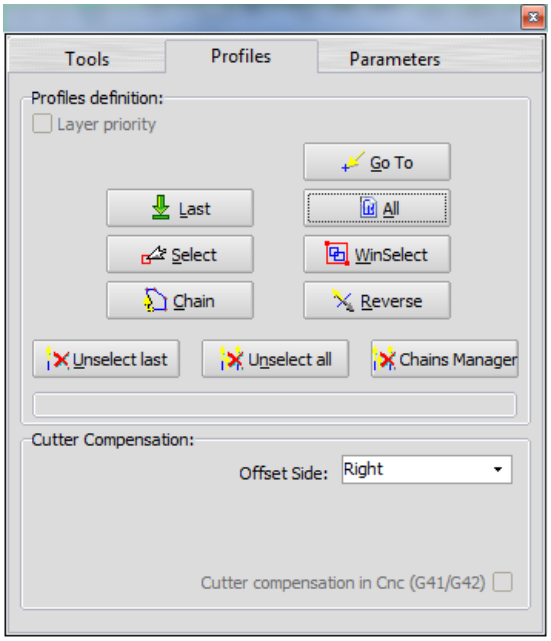
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

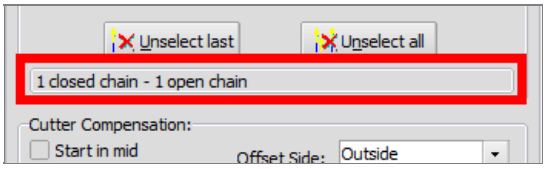
Profiles

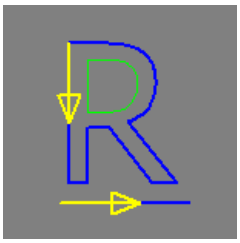
This dialog allows you to define the profiles on which to perform the contouring machining.
It also allows you to define the tool compensation.



Profiles Definition section

Each profile defined, it is displayed on the screen with an arrow indicating the direction of the cut and the point of start.
It also increased the indicator in bottom section, showing the number and type of defined profiles.





Go To
This option moves the tool to the defined position. Multiple points can be set.

Last
The chain defined in the last preceding operation is selected.

All
All entities on the active layers (visible) are selected and chained automatically, if possible.

Select
The selected entities, one by one, at the end are concatenated to create one or more profiles (chains).

Window Select
All entities lying inside the rectangular window are selected and concatenated to create one or more profiles (chains).

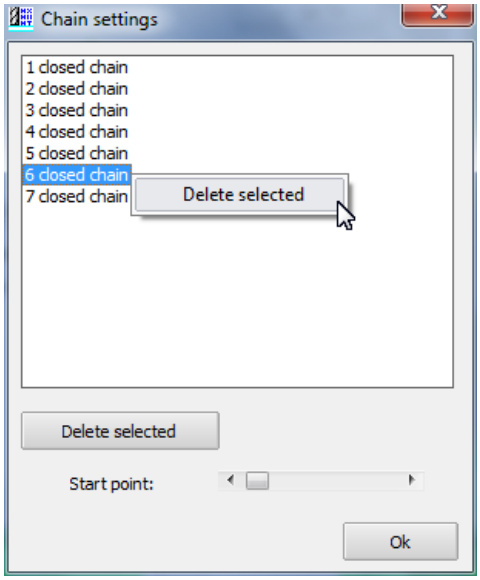
Chain
Clicking on an entity, all entities connected to the respective end points, are concatenated to form the profile (chain).

Reverse
Reverses the direction of the chains.

Unselect Last
The last defined chain is removed from the selection list.

Unselect All
All defined chains are removed from the selection list.

Chains Manager
This parameter open the dialog box containing the list of all chains with this options:



- Move the order in which they chains are performed in the machining
- Delete one or more chains
- Move the starting point each single chain.

Cutter Compensation section

Offset Side

This parameter determines the offset direction, ie the direction of the tool relative to the chained profile.
This direction is displayed on the screen with a small arrow, perpendicular to the cutting direction, which indicates the offset direction.

In the Lathe Rough toolpath, the direction is handled automatically.

- Off:
No compensation, the tool is placed directly on the chained profiles.

- Right:
The tool is placed to the right of the chained profile.

- Left:
The tool is placed to the Left of the chained profile.

Cutter compensation in Cnc (G41/G42)

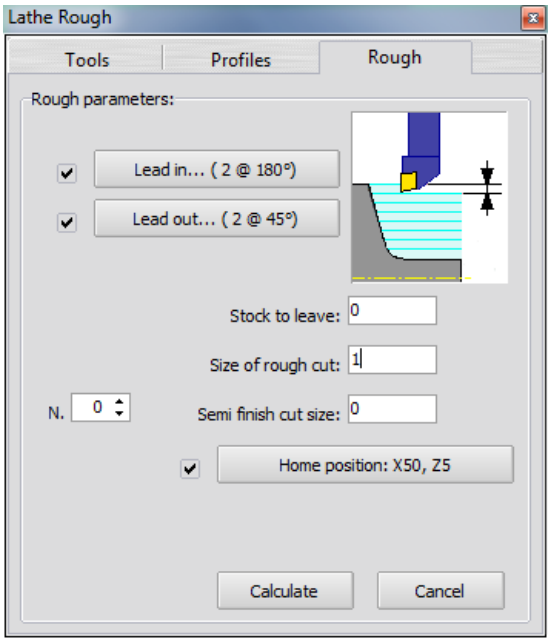
Setting this option, SimplyCam inserts at the beginning of profile G41 (left compensation in control) or G42 (right compensation in control), and at the end of the profile G40 (compensation off)

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Lathe Rough Parameters

This dialog contains the parameters specific for the Lathe Rough toolpath.



Lead in

This parameter controlling how the tool approaches to the part at the start of cut.

More information....

Lead out

This parameter controlling how the tool retracts to the part at the end of cut.

More information....

Stock to leave

Sets the value of material to leave (or remove) on the profile; example, to perform a subsequent finish pass with another tool.
A positive value leaves the material.
A negative value removes material.

Rough cuts size

Set the removal value for each roughing cut.

Number / Semi Finish cut size

Set the number of semi-finishing cuts and the removal value for each semi-finishing cut.

Home Position

The "Home Position" it allows you to determine the X and Z coordinates of where the turret moves at the end of operation or for tool changes. On many Cnc is already automatically defined as a macro (Ex: G28 or M6) and don't need you to define it.

Calculate button

Performs the calculation process of the toolpath, using the chained profiles and the current parameters defined.

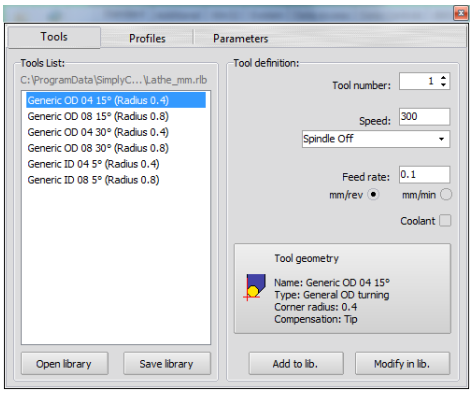
Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

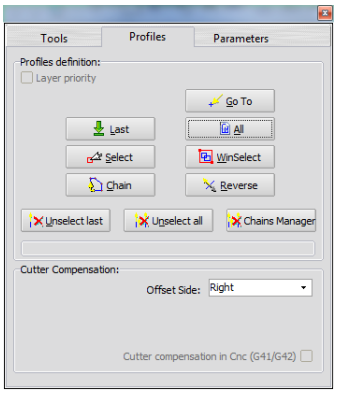
SimplyCam V3. Documentation

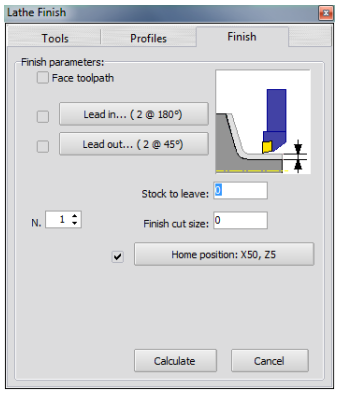
Lathe Finish Toolpath

The Finish toolpath, removes material along a selected profile. On the profile can be applied the tool tip radius compensation.

Dialogs of Lathe Finish Toolpath







Dialog 1 - Tool Definition

This dialog sets the technological parameters of the tool. These parameters directly affect the G-code generated by the postprocessor.
More information....

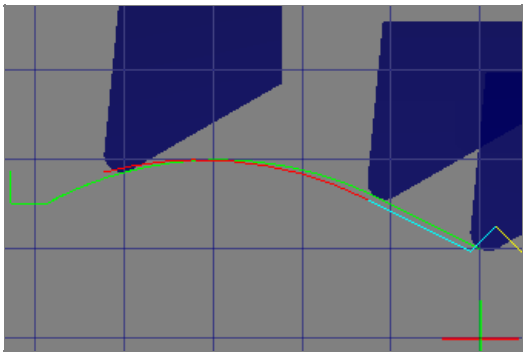
Dialog 2 - Defining Profiles

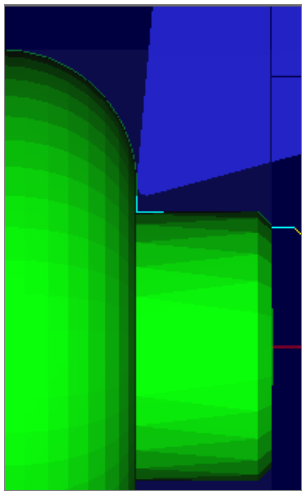
Use this dialog to define and edit the profiles for the selected operation.
This dialogue is also used to define the tool compensation.
More information....

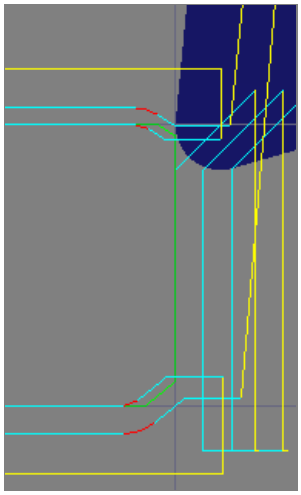
Dialog 3 - Contour Parameters

This dialog contains the parameters specific for the Lathe Finish toolpath.
More information....

Examples of Lathe Finish toolpath





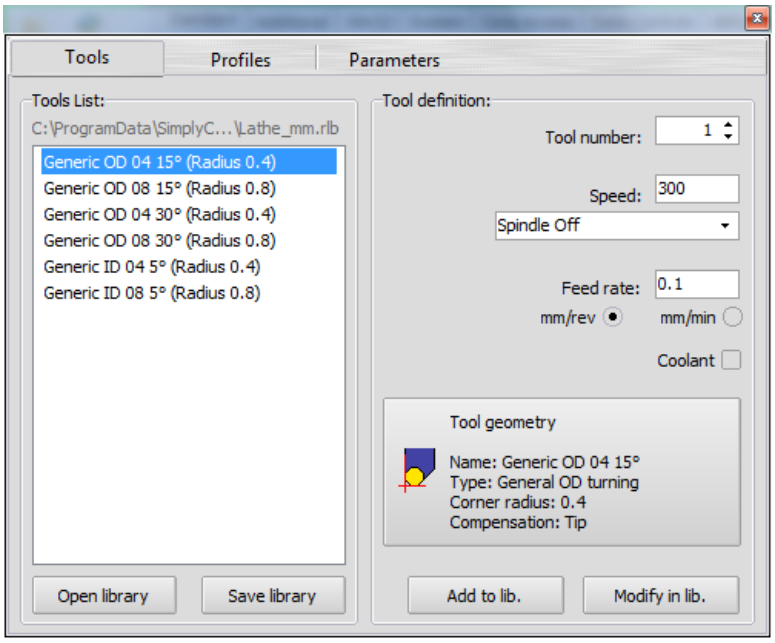


Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Lathe Tools

This dialog allows you to define the technological parameters of the tool that will be used in toolpath.



Tools List section

This dialog box list all tools in the current tool library.
By clicking on a tool in this list, the associated parameters with it are displayed in the right section.

Delete (Right-click on the tool)

The Delete menu item removes the selected tool from the current list. You must use the "Save Library" button to update the external file.

Drag-and-drop tool

Click, drag to another location and release to sort the list to your liking. You must use the "Save Library" button to update the external file.

Open Library

The button "Load Library" allows you to select a new tools library.

Save library

The "Save the Library" button allows you to save the current list of tools, in an external file that can have the same name or different name.

Tool Info section

These are the current values â€‹â€‹that will be used to create the machining. These parameters directly affect the G-code generated by the postprocessor.

Tool Number

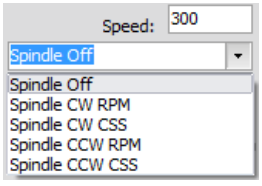
This value set the number of the tool used in the processing.
More operations, with different tool numbers, produce the G-Code file, instructions for tool change (eg T3 M06).

Speed

This parameter sets the spindle speed.
SimplyCam generates in the G-Code file the statement S (example: S3000)

Spindle:

Sets the direction of rotation of the spindle. (Ex: M03) and code of the type of cutting speed (Ex: G96)



- Spindle Off
Spindle turned off.

- Spindle CW RPM
The spindle rotates clockwise RPM (Revolutions Per Minute).

- Spindle CW CSS
The spindle rotates clockwise with Constant Surface Speed.

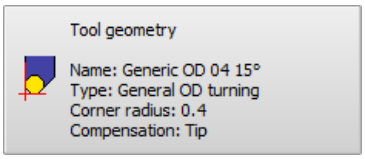
- Spindle CCW RPM
The spindle rotates counterclockwise RPM (Revolutions Per Minute).

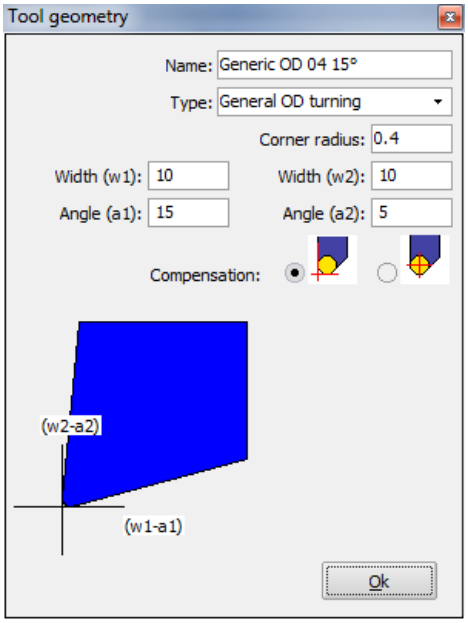
- Spindle CCW CSS
The spindle rotates counterclockwise with Constant Surface Speed.

Feed rate
This value controls the velocity (expressed in mm(inch)/revolution or in mm(inch)/minute) feed to the Cnc axes.

Coolant
Turn On / Turn Off the coolant.

Tool Geometry
This button opens the dialog box, which lets you change the geometric parameters of the tool.





Name
This field allows you to assign an alphanumeric string to identify the tool.

Type
This option allows you to choose the type of tool proposed by SimplyCam.

- General OD turning
Generic tool for external turning.

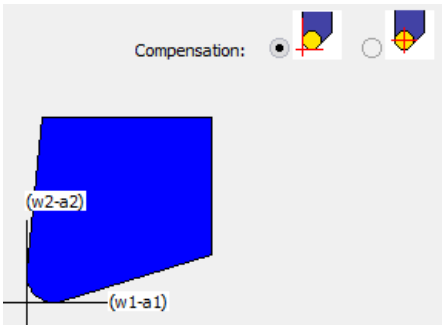
- General ID turning
Generic tool for internal turning.

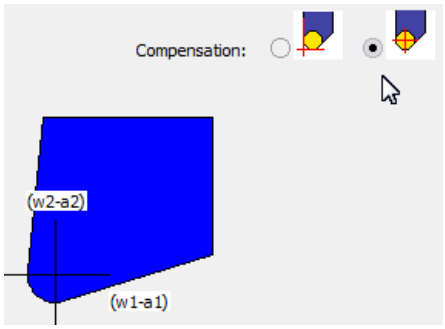
Corner radius
This value sets the radius of the SimplyCam used to calculate the toolpaths and tool compensation.

Width(W1) / Angle(A1)
These values define the geometric shape of the tool and are used in the simulation to view the tool.

Width(W2) / Angle(A2)
These values define the geometric shape of the tool and are used in the simulation to view the tool.

Compensation
Select the type of compensation in the same way as the tool is set in the CNC.





Add to Library

By pressing this button, the current tool and its parameters is added to the tools list on the left.
You must use the "Save Library" button to update the external file.

Modify in Library

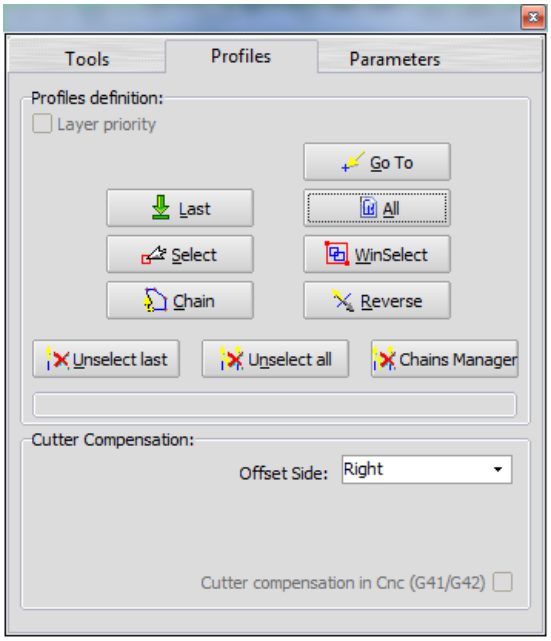
By pressing this button, the current tool and its parameters, is amended in the tool list on the left.
You must use the "Save Library" button to update the external file.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

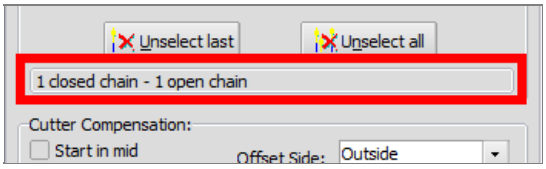
Profiles

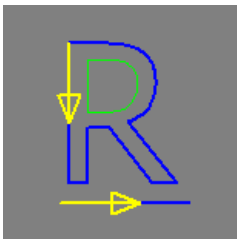
This dialog allows you to define the profiles on which to perform the contouring machining.
It also allows you to define the tool compensation.



Profiles Definition section

Each profile defined, it is displayed on the screen with an arrow indicating the direction of the cut and the point of start.
It also increased the indicator in bottom section, showing the number and type of defined profiles.





Go To
This option moves the tool to the defined position. Multiple points can be set.

Last
The chain defined in the last preceding operation is selected.

All
All entities on the active layers (visible) are selected and chained automatically, if possible.

Select
The selected entities, one by one, at the end are concatenated to create one or more profiles (chains).

Window Select
All entities lying inside the rectangular window are selected and concatenated to create one or more profiles (chains).

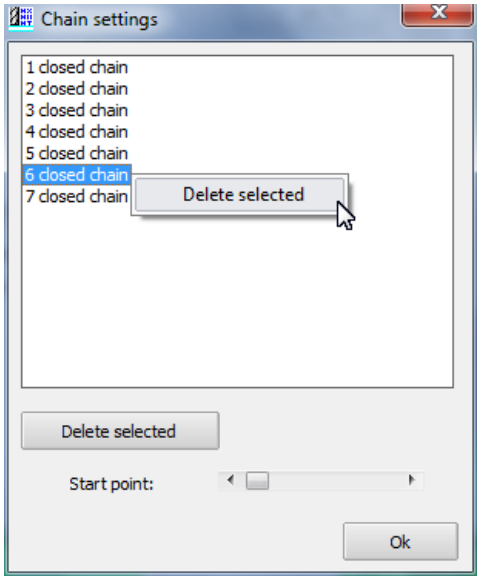
Chain
Clicking on an entity, all entities connected to the respective end points, are concatenated to form the profile (chain).

Reverse
Reverses the direction of the chains.

Unselect Last
The last defined chain is removed from the selection list.

Unselect All
All defined chains are removed from the selection list.

Chains Manager
This parameter open the dialog box containing the list of all chains with this options:



- Move the order in which they chains are performed in the machining
- Delete one or more chains
- Move the starting point each single chain.

Cutter Compensation section

Offset Side

This parameter determines the offset direction, ie the direction of the tool relative to the chained profile.
This direction is displayed on the screen with a small arrow, perpendicular to the cutting direction, which indicates the offset direction.

In the Lathe Rough toolpath, the direction is handled automatically.

- Off:
No compensation, the tool is placed directly on the chained profiles.

- Right:
The tool is placed to the right of the chained profile.

- Left:
The tool is placed to the Left of the chained profile.

Cutter compensation in Cnc (G41/G42)

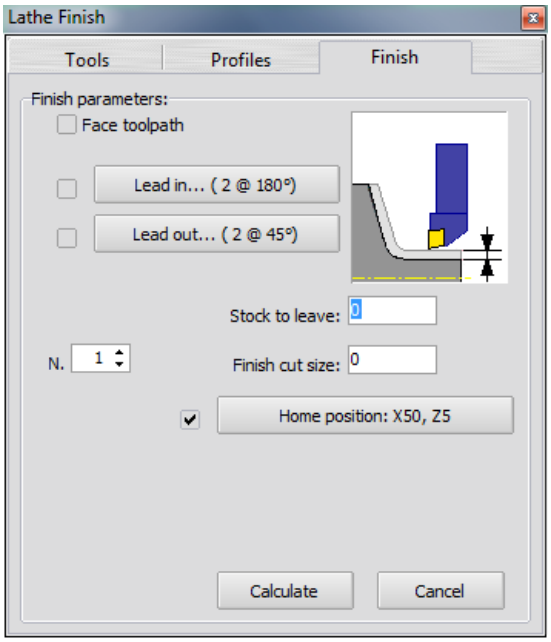
Setting this option, SimplyCam inserts at the beginning of profile G41 (left compensation in control) or G42 (right compensation in control), and at the end of the profile G40 (compensation off)

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Documentation

Parametri Tornio Finitura

This dialog contains the parameters specific for the Lathe Finish toolpath.



Face toolpath

This option allows you to perform the cut on the front face of the part.

Lead in

This parameter controlling how the tool approaches to the part at the start of cut.

More information....

Lead out

This parameter controlling how the tool retracts to the part at the end of cut.

More information....

Stock to leave

Sets the value of material to leave (or remove) on the profile; example, to perform a subsequent finish pass with another tool.
A positive value leaves the material.
A negative value removes material.

Number of finish cuts / Finish cut size

Set the number of finishing cuts and the removal value for each finishing cut.

Home Position

The "Home Position" it allows you to determine the X and Z coordinates of where the turret moves at the end of operation or for tool changes. On many Cnc is already automatically defined as a macro (Ex: G28 or M6) and don't need you to define it.

Calculate button

Performs the calculation process of the toolpath, using the chained profiles and the current parameters defined.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015

SimplyCam V3. Tutorials

Machining

page under preparation...."

Tutorial 1 - Vectorize Jpg image and create Engraving toolpath.

Tutorial 2 - Contour toolpath.

Tutorial 3 - Pocket toolpath.

Tutorial 4 - Drilling toolpath.

Tutorial 5 - PizzaLed, drilling toolpath along curves.

Tutorial 6 - Laser, Plasma WaterJet, Cut toolpath.

Tutorial 7 - Laser Engraving.

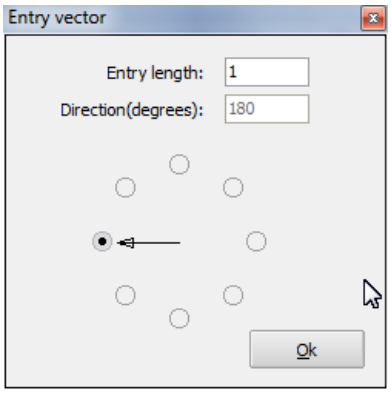
Tutorial 8 - Rough Lathe toolpath.

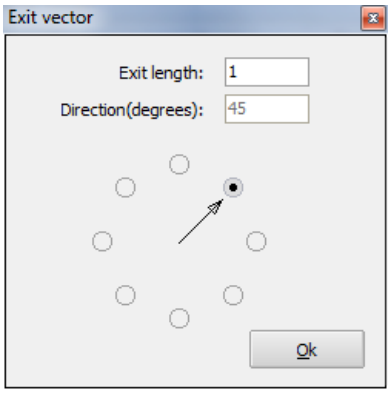
Tutorial 9 - Finish Lathe toolpath.

SimplyCam V3. Documentation

Lathe Lead in/out

These dialogs allows you to add an Entry and/or an Exit vector to the finishing and/or roughing cuts.





Length

Sets the length in degrees of the entry/exit vector.

Direction

Sets the direction in degrees of the entry/exit vector.

Copyright ©2015 MR-Soft - SimplyCam version 3.0.6 - Help file built on 12/08/2015