

IGEMS Release 10

CAD/CAM/NEST

Reference manual

1	General functions	6
1.1	System menu	6
1.2	Ribbon menu	6
1.3	Quick access toolbar	6
1.4	Shortcuts	7
1.5	Command line area	7
1.6	Multiple drawings.....	7
1.7	Ortho mode.....	8
1.8	Zoom and Pan	8
1.9	Coordinate input.....	8
1.10	Distance input.....	9
1.11	Object snap	10
1.12	Select objects.....	11
1.13	Auto base point.....	13
2	Program menu	14
2.1	New.....	14
2.2	Templates	14
2.3	Open.....	15
2.4	Import	15
2.5	Recover.....	16
2.6	Recent files	16
2.7	Save	16
2.8	Save as	17
2.9	Save as template	17
2.10	Dump image	17
2.11	Export	18
2.12	Close	18
2.13	Close all.....	18
2.14	Plot	18
2.15	Limits	19
2.16	Preferences.....	20
2.17	Password	23
2.18	Drawing units.....	23
2.19	Point style.....	23
2.20	Dimension style.....	24
2.21	About	24
2.22	Exit.....	25
3	System menu.....	26
3.1	Undo.....	26
3.2	Redo.....	26
3.3	Purge.....	26
3.4	Status	27
3.5	User data folder.....	27
3.6	Shared folder.....	28
3.7	Service center.....	28
3.8	Support.....	28
4	CAD commands.....	30
4.1	Line	30
4.2	Circle	30
4.3	Rectangle.....	30
4.4	Point.....	31
4.5	Arc.....	31
4.6	Polyline	32
4.7	Spline	32
4.8	Ellipse (Shift+E).....	32
4.9	Solid	33
4.10	Hatch	33
4.11	Parametric shape	34
4.12	N-gon.....	35
4.13	Obround.....	36
4.14	Swell	36
4.15	Puzzle	36

4.16	Boundary poly	37
4.17	Offset	37
4.18	Fillet	37
4.19	Trim	38
4.20	Zero fillet	38
4.21	Chamfer.....	39
4.22	Lengthen	39
4.23	Extend	39
4.24	Boolean	39
4.25	Polyline editor	40
4.26	Break.....	40
4.27	Erase	41
4.28	Join	41
4.29	Layer	41
4.30	Explode	42
4.31	Create block.....	42
4.32	Insert block.....	43
4.33	Group	43
4.34	Save block	43
4.35	Define attributes	44
4.36	Move	45
4.37	Rotate	45
4.38	Copy.....	45
4.39	Mirror	45
4.40	Scale	46
4.41	Quick move.....	46
4.42	Stretch.....	46
4.43	Skew	47
4.44	Array	47
4.45	Zoom extents	48
4.46	Pan.....	49
4.47	Zoom.....	49
4.48	Zoom window	49
4.49	Color	49
4.50	Line type.....	50
4.51	Line weight	51
4.52	Quick select	51
4.53	Send to back.....	52
4.54	Bring to front	52
4.55	Regenerate	52
4.56	Info	52
4.57	Distance	53
4.58	Id.....	54
5	Dimension	55
5.2	Linear	56
5.3	Diameter	56
5.4	Text.....	56
5.5	Rotated dimension	57
5.6	Radius.....	57
5.7	Angular dimension	57
5.8	Leader	57
5.9	Dimension style	58
6	CAM-Tools	59
6.1	Clean up	59
6.2	Edgefix.....	59
6.3	Replace.....	59
6.4	Analyze	60
6.5	Boundary scan.....	60
6.6	Boundary trim	60
6.7	Curvefit	61
6.8	Poly joint.....	61
6.9	Vectorize	61

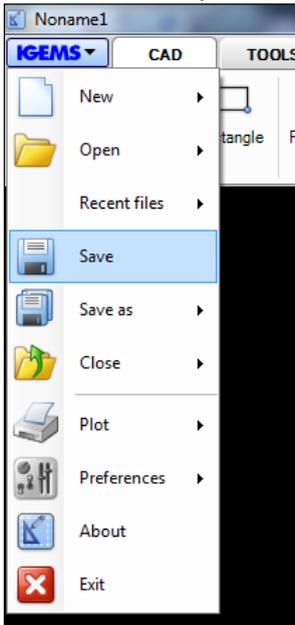
6.10	Pocketing	62
6.11	Convex hull	62
7	SignMaker	63
7.1	Font tracer	63
7.2	Image	64
7.3	Image tracer	65
8	Tile maker	66
8.1	Tile	66
8.2	Tile cut.....	66
8.3	Inlay	68
9	Data Exchange.....	73
9.1	Import	73
9.2	NC reader.....	73
10	CAM	75
10.1	Select setup	75
10.2	Machine	75
10.3	Material	76
10.4	Part create.....	76
10.5	Quality.....	79
10.6	Tool setup.....	80
10.7	Properties	82
10.8	Part text	82
10.9	Browser	83
10.10	Join	84
10.11	Split.....	84
10.12	View in 3D	84
10.13	Contour	85
10.14	Quick.....	93
10.15	Marking	95
10.16	Holes	95
10.17	Manual	96
10.18	Bridges	101
10.19	Common cutlines.....	101
10.20	Cutoff	105
10.21	Break toolpath.....	107
10.22	Edit toolpath	107
10.23	Erase toolpath.....	109
10.24	Micro joints	110
10.25	Chain.....	110
10.26	Disconnect.....	111
11	5X CAM	112
11.1	Define bevel.....	112
11.2	Cut bevel	118
12	CAM – Material setup.....	121
12.1	Material groups	122
12.2	Material qualities	125
12.3	Material thickness.....	127
12.4	General import	133
13	CAM – Sheet commands	134
13.1	Sheet Create	134
13.2	From hole	136
13.3	Lock sheet	136
13.4	Unlock	137
13.5	Divide.....	138
13.6	Sheet export	138
13.7	Sheet open	139
13.8	History.....	139
13.9	Inventory.....	140
13.10	Migrate	141
14	CAM - Process	142
14.1	Process.....	142
14.2	Order.....	147

15	Cutting parameters and Strategy	154
15.1	Cutting parameters.....	154
15.2	Strategy.....	155
16	CAM – Machine setup	168
16.1	Table layout.....	169
16.2	Cutting tools	170
16.3	Z-axis control	172
16.4	Marking tools	176
16.5	Drilling tool	177
16.6	Speeds	178
16.7	Postprocessor.....	180
16.8	Geometry.....	182
16.9	5-Axis kinematic	186
16.10	Cutting equipment.....	188
17	Organizer	193
17.1	Register	193
17.2	Organizer.....	195
17.3	Automatic count up	198
18	Nest commands	199
18.1	Quick nest.....	199
18.2	Single nest.....	200
18.3	Rectangle nest	201
18.4	Auto nest.....	202
18.5	Nest Report.....	204
19	APPS	205
19.1	Load app	205
20	Final words	207

1 General functions

1.1 System menu

Use the IGEMS system menu to handle standard functions like "Open", "Save", "Plot" and "Settings".



Picture 1

Most of these commands are self describing. Picture 2 shows more detailed instructions for each command.

1.2 Ribbon menu

IGEMS R10 is developed with a new menu system called "Ribbons". As default, IGEMS will display the menus horizontally on the top of screen after installing.

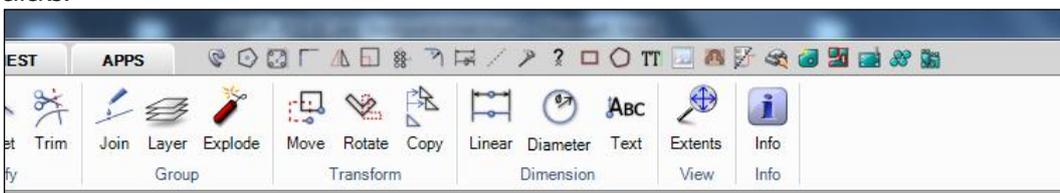


Picture 2

Menus installed by default are seen in the above picture.

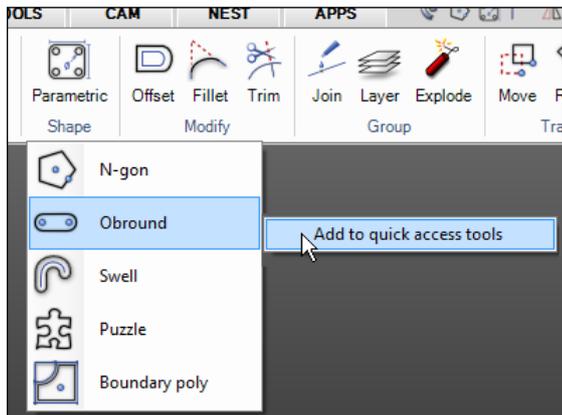
1.3 Quick access toolbar

Frequently used commands can be grouped into the "Quick access toolbar" to save time with fewer mouse clicks.



Picture 3

To add a command, right mouse click on the command and select "Add to Quick Access tools".



Picture 4

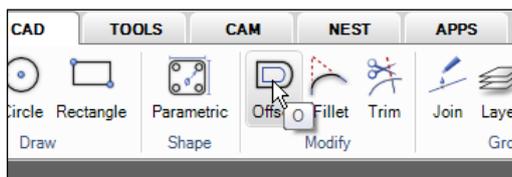
To remove the command from the Quick access toolbar, simply move the mouse pointer over the command and right mouse click to delete.



Picture 5

1.4 Shortcuts

Hold the mouse pointer over a command to see the shortcut for this command. Shortcuts are available for most commonly used commands.



Picture 6

To repeat a previously used command, simply press the spacebar or Enter key.

1.5 Command line area

On the bottom left side will be the name of the actual command.

On the bottom right side will be actual coordinates or other relevant geometry information.



Picture 7

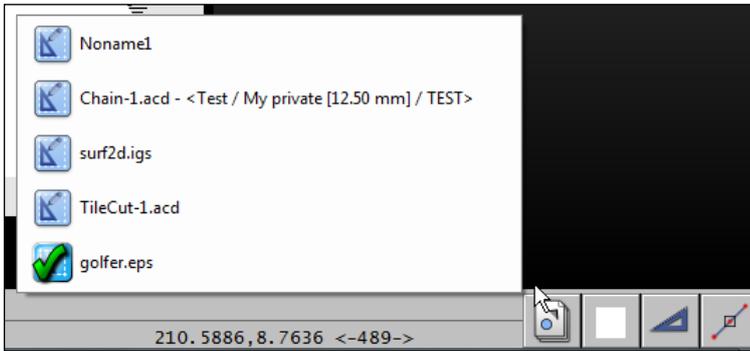
The command information is below the drawing area.

Commands that have different options have actual shortcuts inside brackets [X/B].

The sign for less and greater <Example> shows the "Default" option that can be accepted by pressing the spacebar or Enter keys. Further explanation can be found in the command description section.

1.6 Multiple drawings

Click on the "Drawing selection" button to easily switch between all drawings that are currently open.



Picture 8

1.7 Ortho mode

The drawing direction can be locked to be fixed to horizontal or vertical only or in angle steps (Ortho step).



Picture 9

Click the "Ortho" button to activate. It is also possible to activate and deactivate the mode by pressing function key F8. The Ortho settings are made in the "Preference" setup. Right mouse click to start the "Preference" command.



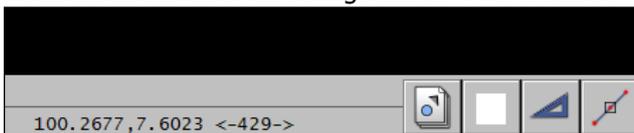
Picture 10

1.8 Zoom and Pan

A mouse with scroll wheel is needed to work properly with IGEMS.

- Zoom: Zoom in and out by scrolling the mouse wheel.
- Pan: Press and hold the mouse wheel down to pan.
- Zoom extents: Double click the scroll wheel and the drawing will zoom to full screen.

If the scroll wheel is not working in this way then check the mouse settings in the control panel. The mouse wheel button should be configured as "Mid" button.

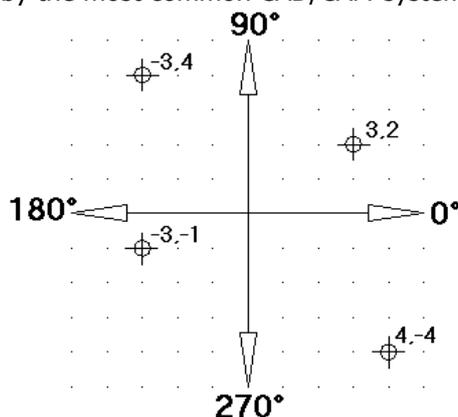


Picture 11

The value between < > indicates the width of the screen.

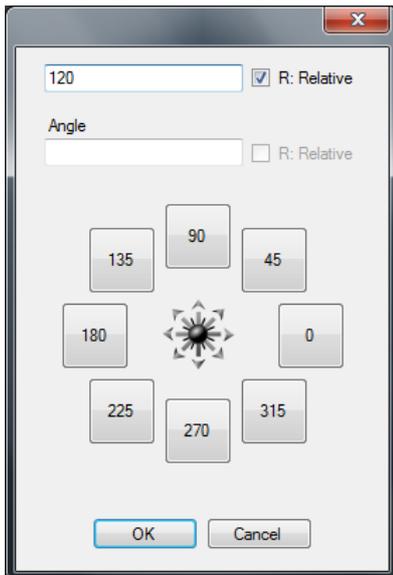
1.9 Coordinate input

The coordinate system in IGEMS supports the Cartesian coordinate system. This coordinate system is used by the most common CAD/CAM-systems.



Picture 12

When IGEMS expects a numeric value input and any digit is typed, it will show the following dialog box for input of values.



Picture 13

Absolute coordinates

Always refer to IGEMS fixed zero point. It is typed X, Y as in the following example: 110.5, 220.18

Relative coordinates

This refers to the last used point as a distance rather than a coordinate. To use relative coordinates check the "Relative" checkbox.



Picture 14

Absolute polar coordinate

Always refer to IGEMS fixed zero point.



Picture 15

Relative polar coordinate

Always refer to the last used point.



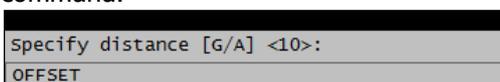
Picture 16

Direction coordinate

This is the fastest way to enter coordinates and is used for one value. This method uses the pointing direction and it is often used together with Ortho mode.

1.10 Distance input

Several commands in IGEMS ask for a numerical value that refers to a distance, for example the "Offset" command.



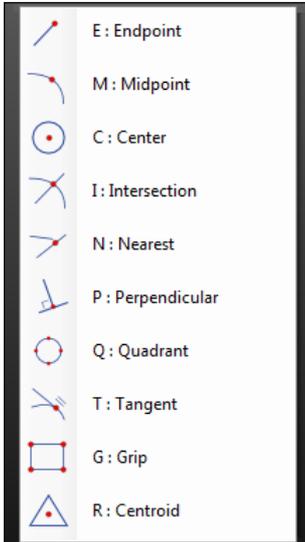
Picture 17

These options are available when a command asks for a distance:

Press the spacebar or Enter to accept the value inside <>
 Enter another value by using the numerical keyboard.
 Pick two points and the distance between the points will be the input.

1.11 Object snap

Use "Object snap" to snap exactly to different geometrical points on existing objects. The "object snap" can be used inside other commands when a point is required. The "object snap" can be selected from a menu opened with the right mouse button, when IGEMS asks for a point.



Picture 18

Available object snaps:

Snap	Shortcut	Description
Endpoint	E	Snap to endpoint of all drawn objects.
Midpoint	M	Snap to midpoint of all drawn objects.
Center	C	Snap to center of Circles, Arcs and Ellipses.
Intersection	I	Snap to Intersection between drawn objects.
Nearest	N	Snap to an object near the selected point.
Perpend	P	Snap to a perpendicular point on the picked object from the direction of the previous position.
Quadrant	Q	Snap to point on 0, 90, 180 or 270 degrees from center on Arcs, Circles or Ellipses.
Tangent	T	Snap to a Tangential position on an Arc, Circle or Ellipse.
Grip	G	Snap to Grip points on object.
Centroid	R	Snap to the center of gravity on an object.
Off	O	Turn off the Auto snap temporarily for next given point.

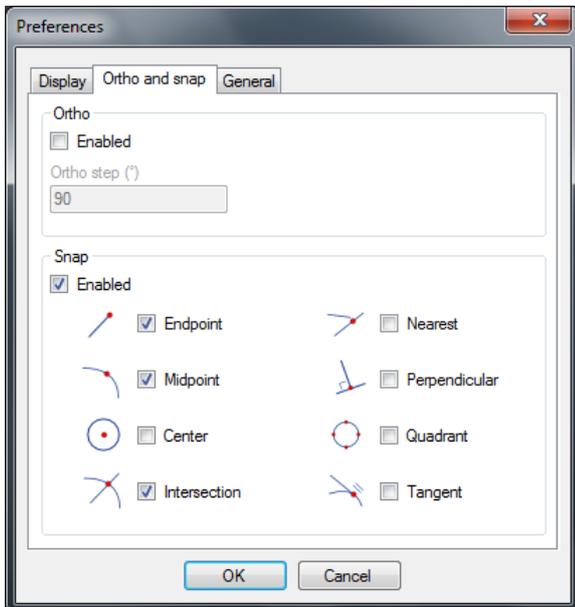
Auto snap

Use "Auto snap" to snap to selected geometrical points.
 This is activated or deactivated from the "Snap" button or by pressing F9.



Picture 19

Right mouse click to select the snap to use when the "Auto snap" function is activated.

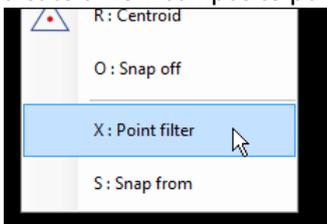


Picture 20

The same dialog box can also be started from the "Preference" command on the system menu.

Point filter

Use the "Point filter" option to retrieve individual X and Y coordinates from different points on the drawing to create a new composite point.

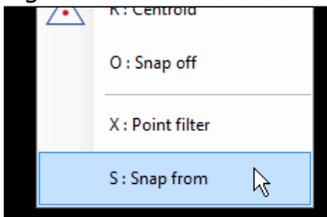


Picture 21

The "Point filter" is activated by right clicking in the context menu or by pressing "X" on the keyboard when IGEMS asks for a point.

Snap from

The "Snap from" option asks for a point that will be used as the last point. This option should be used together with relative coordinates.

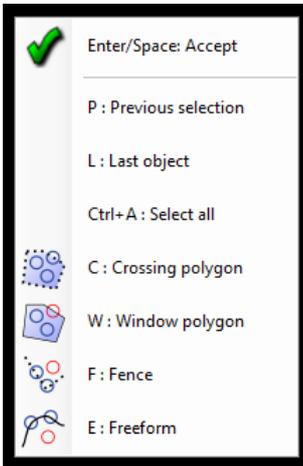


Picture 22

The "Snap from" can be activated by right clicking in the menu or by pressing "S" on the keyboard when IGEMS asks for a point.

1.12 Select objects

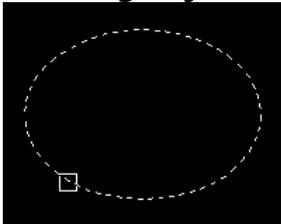
Many commands need objects as input, for example: Erase, Move and Copy. Right click to see the menu for different options. The "Select object" function in IGEMS works as follows:



Picture 23

Note! The fastest way to select different options is to use keyboard shortcuts (P, L, Ctrl-A, C, W, F and E).

Selecting Objects



Picture 24

Click on an object to select it. To unselect, hold down the SHIFT key while selecting the object.

Previous (P)

This option selects the same object from the previous time "Select object" was used.

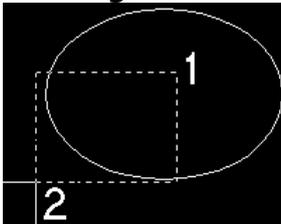
Last (L)

This option selects the last object that was drawn.

All (CTRL-A)

This option selects all objects on the drawing.

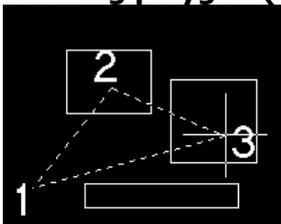
Crossing



Picture 25

This option uses two points to describe a rectangle. The second point must be to the left of the first point. If an object is inside or if the objects are crossing the rectangle then the object is selected. Hold down the SHIFT key to de-select the objects.

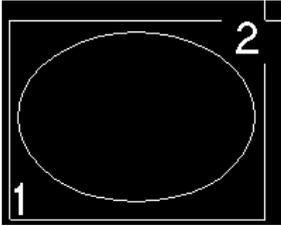
Crossing polygon (C)



Picture 26

This option uses optional number of points to describe a polygon. If an object is inside or if an object is crossing the polygon, the object will be selected. This feature does not work with the SHIFT key.

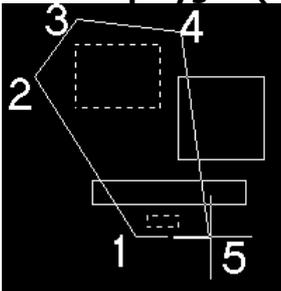
Window



Picture 27

This option uses two points to describe a rectangle. The second point must be to the right of the first point. The object must be completely inside the rectangle to be selected. Hold down the SHIFT key while using "Window" and the objects are de-selected.

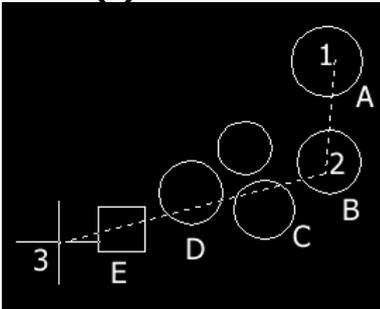
Window polygon (W)



Picture 28

This function uses optional number of points to describe a polygon. The objects must be completely inside the polygon to be selected. This function does not work with the SHIFT key.

Fence (F)

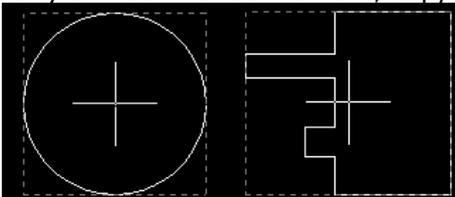


Picture 29

The "Fence" option allows a multi-segment line (polyline) to be drawn. All objects which cross the fence will be selected in the same order as the line crosses the objects. See Picture 29 - the "Fence" option is used to draw a fence line through points (1), (2) and (3). Objects (A) to (E) are then selected.

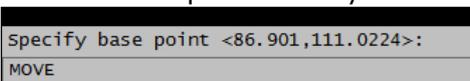
1.13 Auto base point

Many CAD commands like Move, Copy and, Scale have an automatic base point selection.



Picture 30

The "Auto base point" is always located in the center of a bounding box of the selected objects.

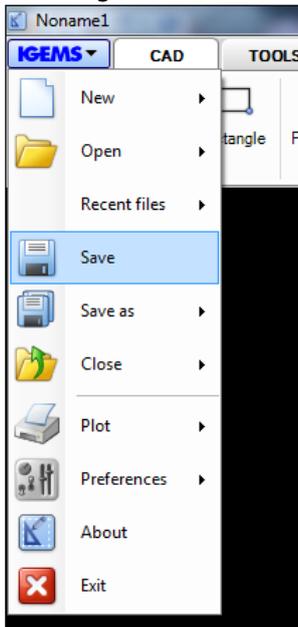


Picture 31

To use the "Auto base point" press the spacebar or Enter key.

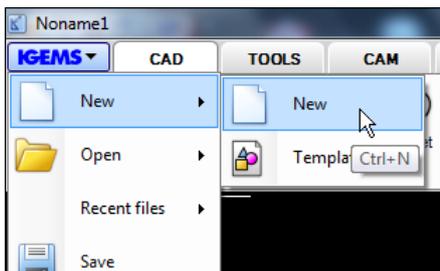
2 Program menu

The "Program" menu handles many standard features and also some IGEMS specific commands.



Picture 32

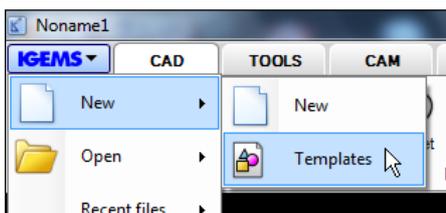
2.1 New



Picture 33

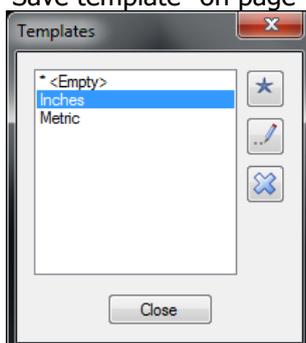
Use this command to start a new drawing.

2.2 Templates



Picture 34

A template is a normal IGEMS drawing (DIG) file. A template is used to start IGEMS with custom settings, for example, to switch between metric and inches. This option shows the following dialog box. See also "Save template" on page 17.



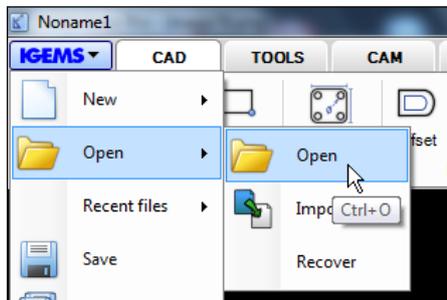
Picture 35

Use the star button to set a template to default, there is also buttons to rename or delete existing templates. The template will be used every time to start a new drawing.

? Are there other reasons to use templates?

It is a simple way to customize IGEMS. All settings like layers, colors, line types, dimension settings are saved in a template. Use a template to set these values once.

2.3 Open



Picture 36

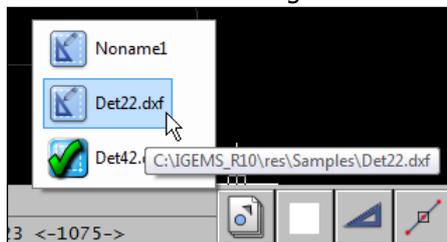
Use the "Open" command to open existing geometry files:

Type	Description
.DIG	IGEMS R10 standard drawing format
.ACD	IGEMS R3 to R9 drawing format
.DWG	AutoCAD Drawing files
.DXF	Drawing Exchange Format

Note! When "Data Exchange" module is activated 9 more file types can open. See page 73 for more information.

The command shows a windows standard open file dialog box.

IGEMS creates a drawing session for each file if multiple files are opened using CTRL or SHIFT.



Picture 37

? Is it possible to write an application that can read other kinds of files?

Yes, such applications can be made in ILISP script languages.

2.4 Import



Picture 38

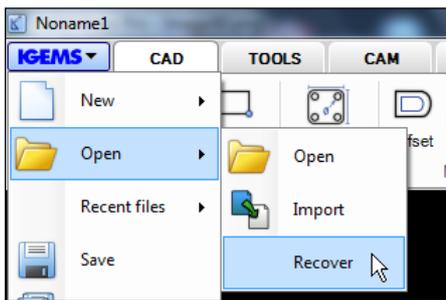
The "Import" command inserts selected drawing files into the current drawing.



Picture 39

Hold down CTRL or SHIFT while inserting the part to also insert the name of the file as text. This text can be used to identify the name when making parts. The "Import" command can use the same files as "Open".

2.5 Recover

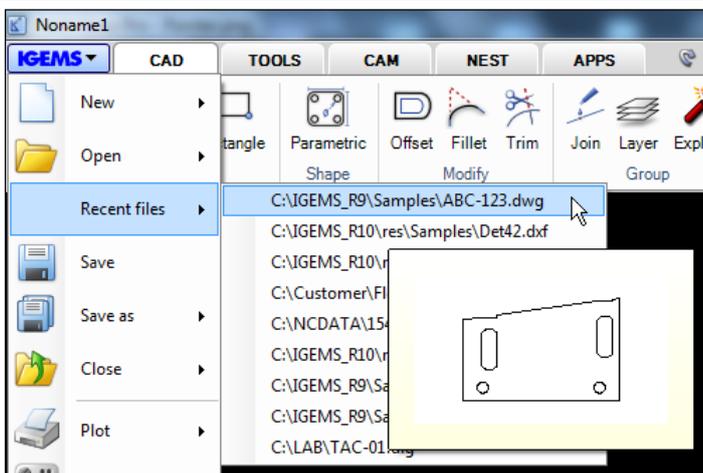


Picture 40

The "Recover" command can be used to recover drawings (DIG files) in the event of a system failure. The "Recover" opens a file dialog box with all Auto saved files. The interval (minutes) for auto save is set in the "Preferences" menu and if this value is set to zero (standard) then the "Recover" command is disabled.

Note! Auto saved files older than 24 hours will automatically be deleted from the list.

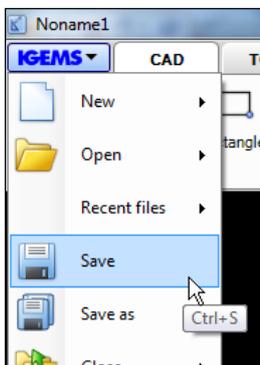
2.6 Recent files



Picture 41

This command shows the fastest way to view and find a list of recently opened files.

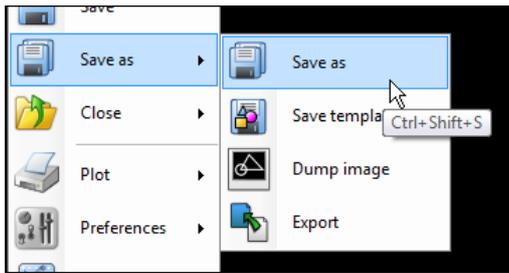
2.7 Save



Picture 42

This command saves the active drawing into an IGEMS DIG file.

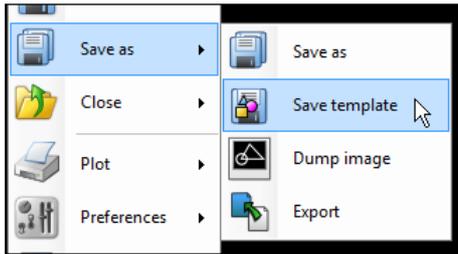
2.8 Save as



Picture 43

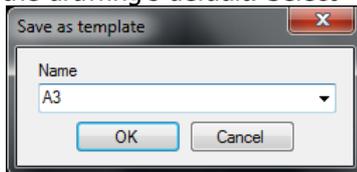
This command saves the active drawing under another name.

2.9 Save as template



Picture 44

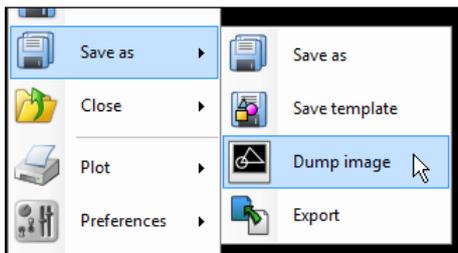
A template is a normal IGEMS drawing that can be used to start IGEMS in different predefined modes. A template's settings can be modified to create layers, limit values, modify colors or draw objects desired as the drawing's default. Select "Save as template" to have IGEMS ask for a name of the template, Picture 45.



Picture 45

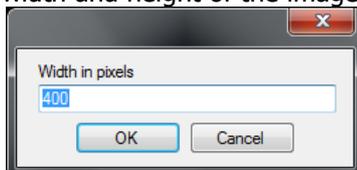
All template drawing is saved in a Shared folder that can be used by other users. See also "Templates" on page 14.

2.10 Dump image



Picture 46

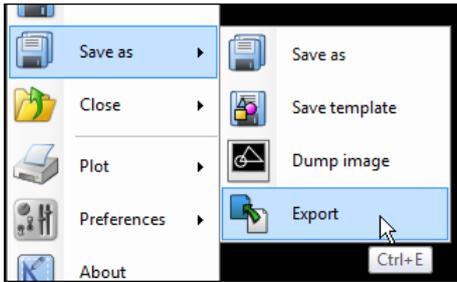
This command creates a bitmap image out of the geometry. Select the geometry to be included and enter width and height of the image before saving the file. The values are defined in pixels.



Picture 47

Supported file formats are PNG, JPG, BMP and EMF files.

2.11 Export



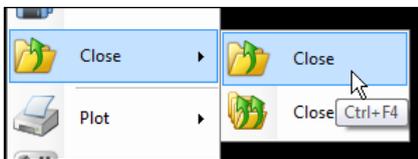
Picture 48

This command exports selected objects or the whole drawing to another format, such as a DIG, DXG, WMF or EMF file.



Picture 49

2.12 Close



Picture 50

"Close" will close the active drawing. A prompt will ask to leave the file unsaved or saved before closing if the file was modified since the last "Save".

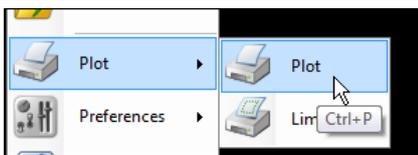
2.13 Close all



Picture 51

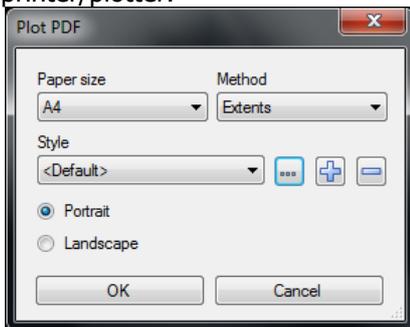
This command will close all open files. A prompt will ask to leave the file unsaved or saved before closing if the file was modified since the last "Save".

2.14 Plot



Picture 52

The "Plot" command is used to print out drawings from IGEMS to a PDF file to use to print out to any type of printer/plotter.



Picture 53

Plotting in IGEMS creates a PDF file as an intermediate step before sending it to the printer/plotter.

Paper size

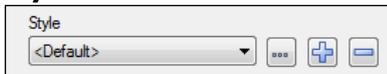
Select the paper size used for the plotting.

Method

This describes the method that will be used in the plotting area.

- Paper size: Select the proper paper size to use for the plotting.
- Method: Set the proper area to plot.
- Plot Style: Customize the plotting result with line weights and color settings.

Style



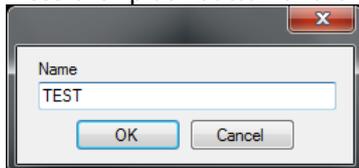
Picture 54

Select the style to use from the drop down list.

Create new or delete existing styles by using the "plus" and "minus" buttons.

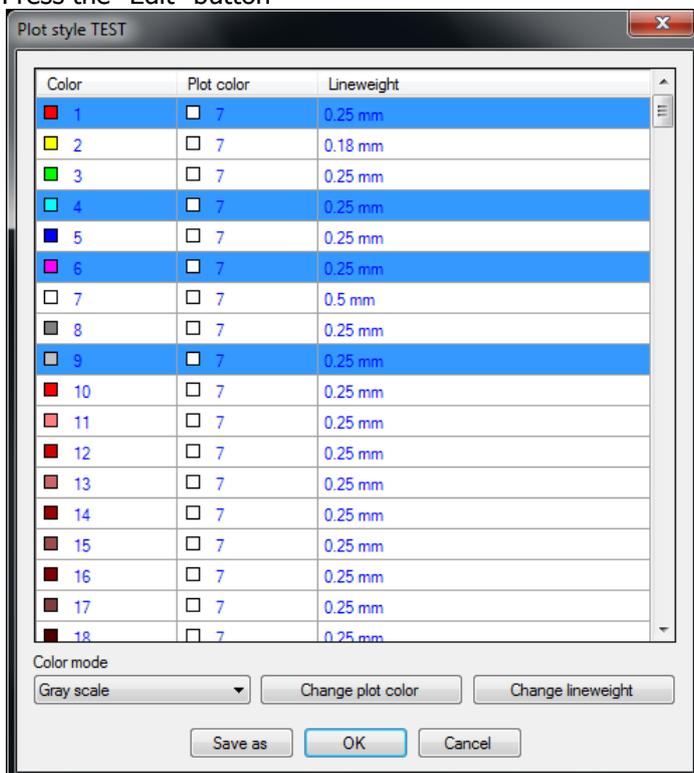
Change parameters in the selected style by the "..." button.

- ☞ Press the "plus" button and create a style called "TEST".



Picture 55

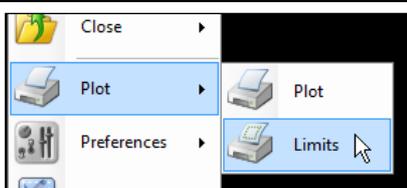
- ☞ Press the "Edit" button



Picture 56

From this dialog box, Picture 56, map the colors on the screen to different colors on the paper. Use the line weight to apply different thickness to different colors. The color mode controls how the different colors should be plotted on the paper.

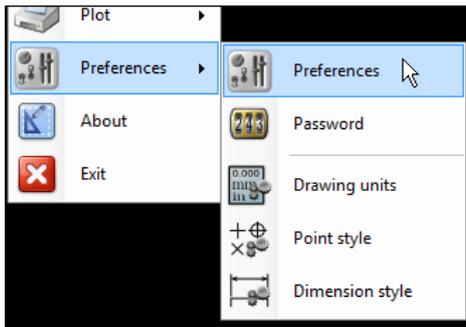
2.15 Limits



Picture 57

"Limits" predefines a rectangular plotting area by selecting 2 points diagonally on the drawing area. The limits area is saved in the drawing and is visible when the grid is activated.

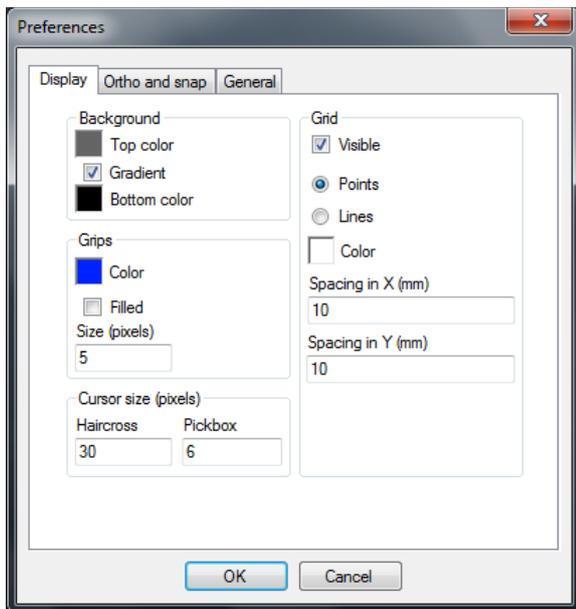
2.16 Preferences



Picture 58

The "Preferences" command includes functions to customize the drawing environment for language, units and dimensioning.

2.16.1 Display

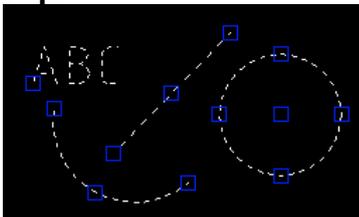


Picture 59

Background

This setting allows users to customize the preferred background color(s). Checking the gradient checkbox enables different colors to be shown in the top and bottom of the screen.

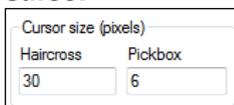
Grips



Picture 60

When no command is active and an object is selected IGEMS will show the grip points on the objects. This setting customizes the size and the color of the grips. See limits on page 18.

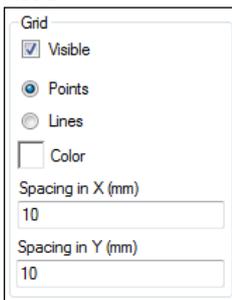
Cursor



Picture 61

This controls the size of the crosshair and the size of the selection box (Pickbox). The Pickbox is used when selecting objects and by the "Object snap".

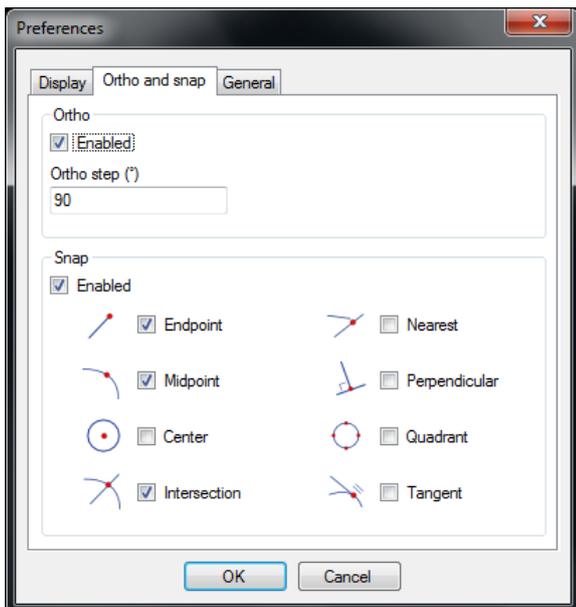
Grid



Picture 62

These settings control how the grid is shown (by points or lines), the color and the distance between the grids. The grid area is the same as the limits area.

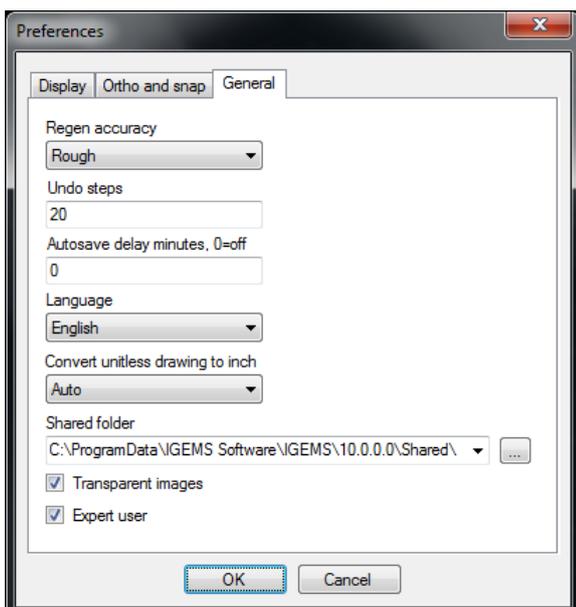
2.16.2 Ortho and snap



Picture 63

These settings control the "Ortho" and the "Auto snap". See pages 8 and 10.

2.16.3 General



Picture 64

Regen accuracy

Regen is a function that regenerates a drawing that is zoomed in/out possibly causing the geometry to not be visually perfect. This is only visual and will not affect the final result. Use the shortcut SHIFT+R to

regenerate the drawing and update the screen. If the object is too rough on the screen then set the accuracy to medium or fine.

Note! This may slow down the system.

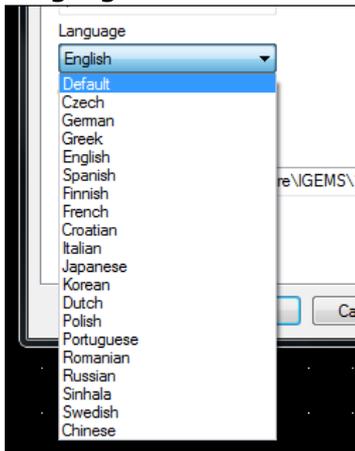
Undo steps

This is the number of Undo/Redo steps that will be possible to make. Default number is 20. Increasing the value may slow down the system on large drawings.

Auto Save

When working in an unstable environment it can be helpful for IGEMS to automatically save the drawing in a timed interval. If the value is 0 then the function is disabled. Open an auto saved drawing from the "Recover" command. See page 16.

Language



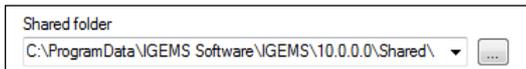
Picture 65

The language will automatically be set to the same language of the operating system but can also be changed here. If that language is currently not yet supported IGEMS will default to English.

Convert unitless drawings to inches

Some files (WMF, DXF and DWG) have no information about units. This setting controls which format these files should be converted to. If the setting is Auto then it will be converted to the same as the default drawing units.

Shared folder



Picture 66

The shared folder includes all settings that can be shared with other users in the same network (machine settings, material settings, post processor settings, etc.).

To change the location of the shared folder press the "..." button and select the folder to use.

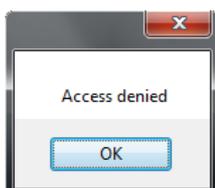
Note! If an empty folder is selected then all information in the current folder will be copied to the new folder. If a non-empty folder is selected, nothing will be copied.

Transparent images

When "Copy" and "Paste" are used to copy geometry in IGEMS, an image is created that can be inserted into other documents. This setting defines whether the background of this image should be transparent or not.

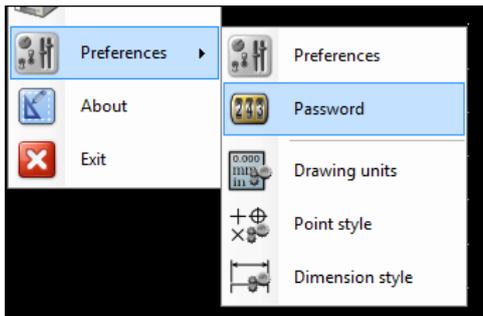
Expert user

To change machine or material settings check the "Export user" box or the following message will be shown:



Picture 67

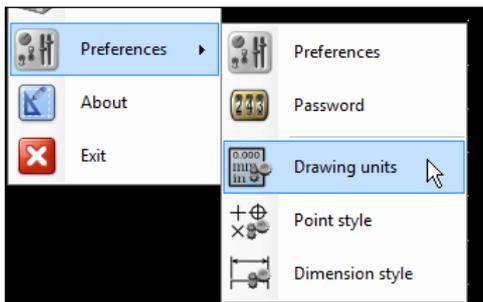
2.17 Password



Picture 68

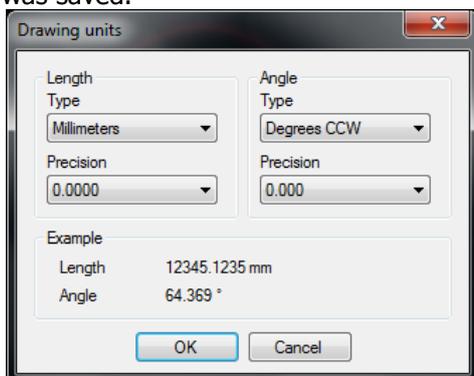
See separate "Install" documentation for more information about this command. The document can be downloaded from our web-site.

2.18 Drawing units



Picture 69

The drawing units are saved in the drawing. IGEMS will open a file in the same units used when the drawing was saved.



Picture 70

IGEMS stores all numerical values as high precision (0.0000). These settings only affect what is displayed to the user on screen. Use this dialog box to specify special units for dimensions and other outputs.

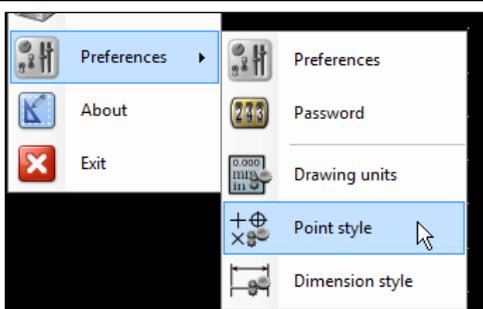
Length units

Select if the units should be in Millimeter or Inches and set how many decimals to show.

Angle

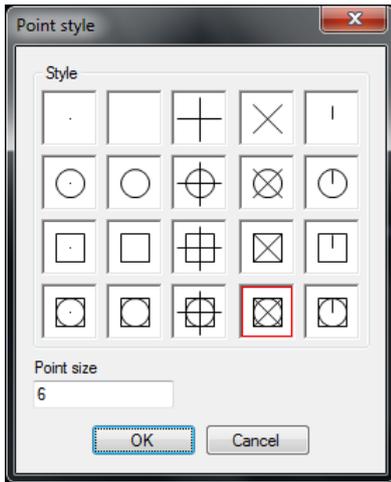
Select if the angles should be in Degrees or Radians and set how many decimals to show.

2.19 Point style



Picture 71

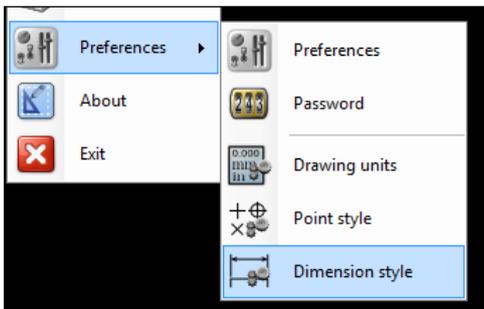
"Point style" controls how points will appear on the screen.



Picture 72

Select the type of point and the size; a larger value means a larger point.

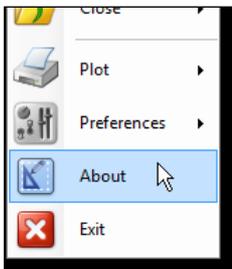
2.20 Dimension style



Picture 73

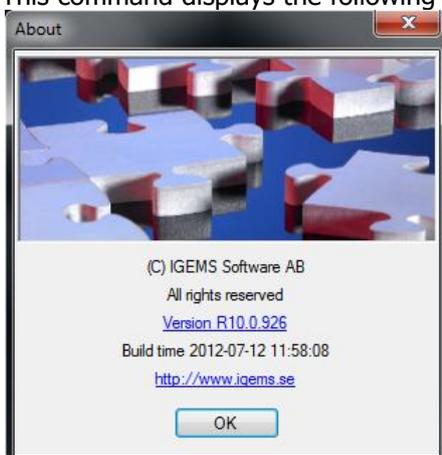
Read more about this in Section 5.

2.21 About



Picture 74

This command displays the following information.



Picture 75

Version

This is the actual version that installed. Click on the version number to connect to the internet to see if a newer version is available. This shows a revision history and what will be included in the next update.

NEXT RELEASE

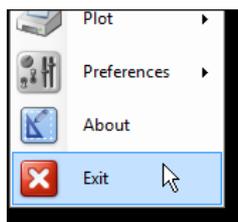
- Now possible to set quality for different edges on bevel cutting. For example X and K type can have different quality on top and bottom edges. The selected quality is visible in EditPart.
- DXF Files which had comment group codes (999) in the header section now works.
- Ruled bevel parts are now supported for all nesting methods.
- Visual enhancements in 'Load App' command window
- Fixed/directional bevel cutting , wrong direction when piercing corrected.

R10.0.912 120627

- Circle remembers radius or diameter mode when IGEMS is restarted.
- This document is now properly published.
- Better handling if report folder does not exist.

Picture 76

2.22 Exit



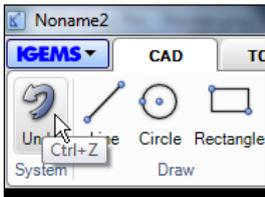
Picture 77

Use this command to close IGEMS and all open drawings.

3 System menu

The "System menu" includes commands for maintenance and file management.

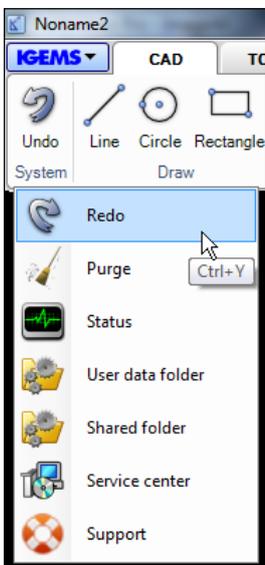
3.1 Undo



Picture 78

When working with IGEMS press the "Undo" button to correct mistakes and remove changes made with the last command. Up to 20 commands can be undone. Use backspace as a keyboard shortcut instead of CTRL-Z.

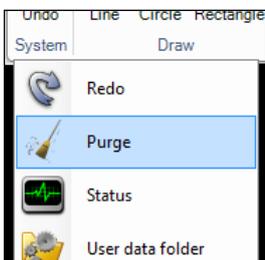
3.2 Redo



Picture 79

Use "Redo" if too many steps were "undone". Redo can only be used directly after the Undo has been used.

3.3 Purge

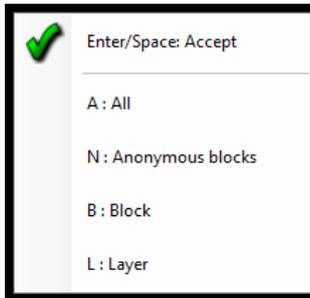


Picture 80

IGEMS will not remove unused layers or blocks when saving a drawing. To fully remove unused information use "Purge". The following information will be written on the command line:

```
select mode [A/N/B/L] <Anonymous blocks>:
PURGE
```

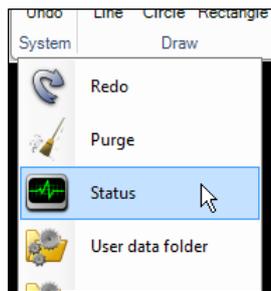
Right click on this command line to get more information.



Picture 81

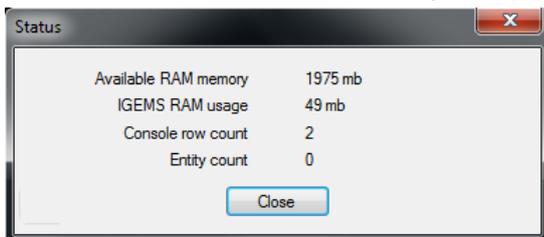
Select "All" to remove Anonymous blocks, Blocks and Layers.

3.4 Status



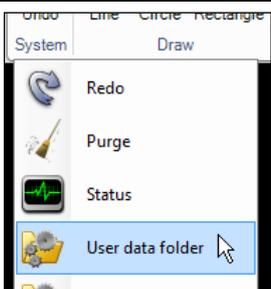
Picture 82

The "Status" command shows current use of RAM memory, the number of rows that have been used on the command line and the number of objects on the drawing.



Picture 83

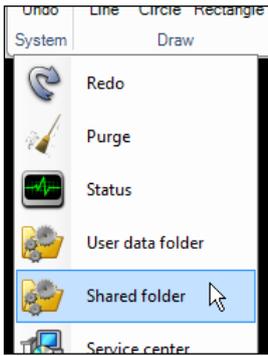
3.5 User data folder



Picture 84

This is a shortcut to the user data path. The folder has information about default values and other settings. This shortcut will probably be removed in coming versions.

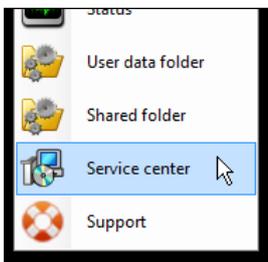
3.6 Shared folder



Picture 85

The Shared folder includes all user settings and customizations such as materials, machine settings, postprocessors, lead in/outs and more. It is recommended to make a backup copy of the shared folder from time to time in case of a computer breakdown.

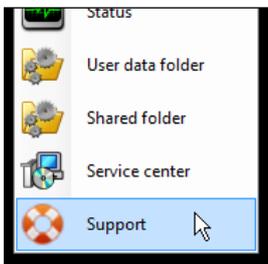
3.7 Service center



Picture 86

The "Service center" opens to a website to download the latest version, drivers, etc. "Service center" can only be used if the computer is connected to the internet and there's a valid license of IGEMS.

3.8 Support



Picture 87

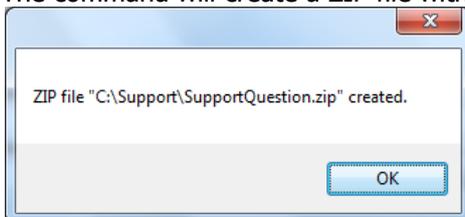
Use the "Support" command if a question or problem needs to be reported to IGEMS support team.

The 'Support' dialog box is titled 'Support' and has a close button (X) in the top right corner. It contains the following elements:

- Subject:** A text input field.
- Description:** A large text area for detailed information.
- Company:** A text input field.
- Contact:** A text input field.
- Telephone:** A text input field.
- View system info:** A button located below the telephone field.
- Files:** A section containing a 'Browse' button and a large empty area for file selection.
- Buttons:** 'Run', 'Save', and 'Close' buttons are located at the bottom of the dialog.

Picture 88

Fill in information about a problem, using the "Browse" button to include any necessary files. Press "Save". The command will create a ZIP file with all needed files and other information.



Picture 89

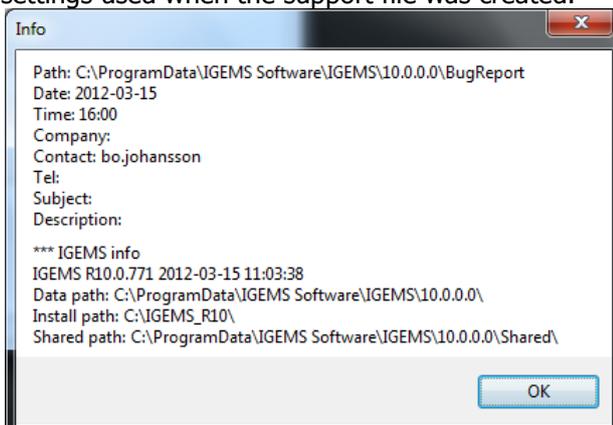
Send the ZIP file to a local IGEMS reseller or to IGEMS AB.

3.8.1 Run support files



Picture 90

Use the "Run" button to load support files. IGEMS starts up in the same mode with the same files and settings used when the support file was created.

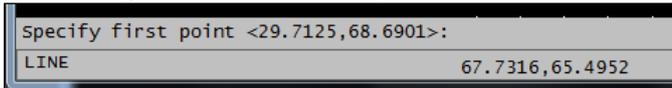


Picture 91

Some general information will show before the drawing is loaded.

4 CAD commands

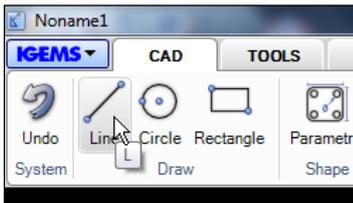
When using the CAD commands, be sure to read the information on the command line.



Picture 92

Points can be given by entering coordinates using the keyboard, by clicking on the screen or by object snap. Distance can be given by values or by clicking two points. See Section 1 to see how to enter this information.

4.1 Line

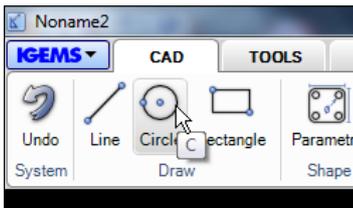


Picture 93

Shortcut "L"

"Line" command asks for a starting point and then the next point. The command is cancelled by using the spacebar, Enter or ESC keys. Backspace can be used to undo the last line segment.

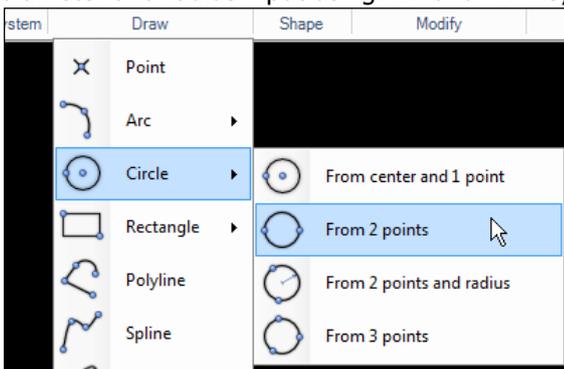
4.2 Circle



Picture 94

Shortcut "C"

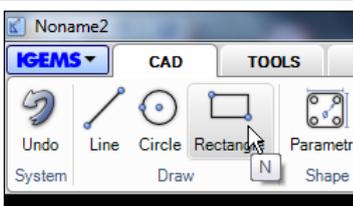
The normal "Circle" command requires a center point and a radius or a diameter as input. Toggle between diameter and radius input using "D" and "R" keys on the keyboard.



Picture 95

The "Draw" menu shows three other ways to create a circle.

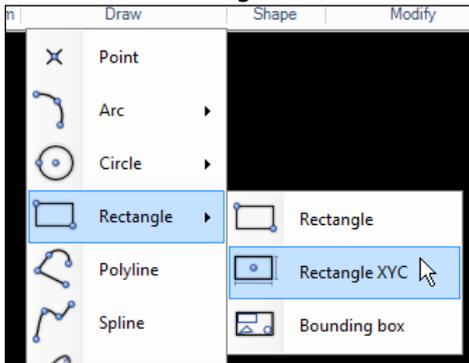
4.3 Rectangle



Picture 96

Shortcut "N"

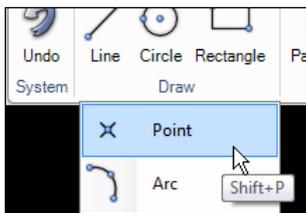
The normal "Rectangle" command creates a rectangular polyline from 2 opposite points.



Picture 97

The "Draw" menu shows two other ways to create a rectangle.

4.4 Point

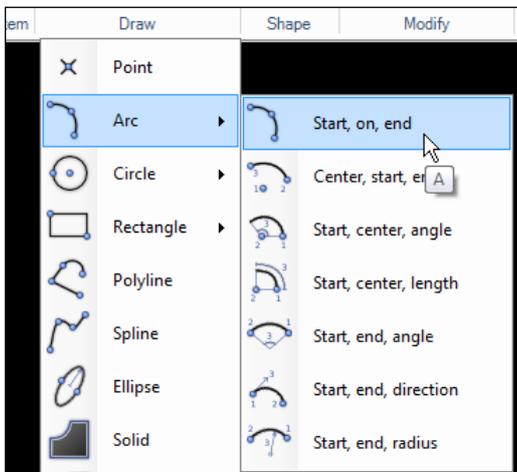


Picture 98

Shortcut "SHIFT+P"

"Point" command asks for input of point positions. To cancel the command, press the spacebar, Enter or ESC. See page 23 for more information how to change the point style.

4.5 Arc



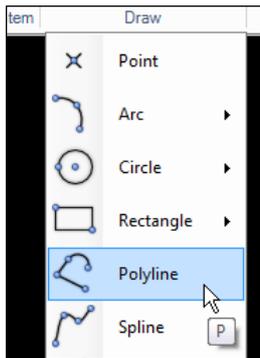
Picture 99

Shortcut "A"

"Arc" entities can be described in many ways. The default Arc mode needs input for a 3 point arc. Six other methods can be used.

Other "ARC" options can be found by selecting the "Arc" command and then click on the arrow to activate the pull down menu.

4.6 Polyline

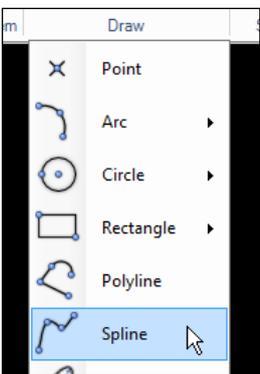


Picture 100

Shortcut "P"

A polyline is a chain of lines and arcs that are joined together into one object. When drawing a polyline toggle between "Line" (type "L") and "Arc" mode (type "A") or SHIFT+C to close the object.

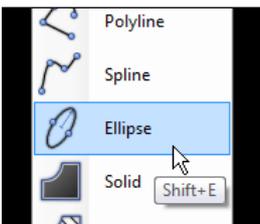
4.7 Spline



Picture 101

The "Spline" command makes a curve that goes through a set of points. Tangent condition for start and end point can be selected at the end of the command. If no tangent condition applies, then press the spacebar to execute the command.

4.8 Ellipse (Shift+E)



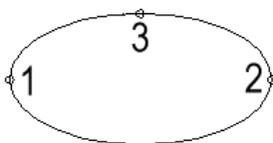
Picture 102

Shortcut "SHIFT+E"

The "Ellipse" command creates an elliptic shape by defining major and minor axis length/radius.

Default

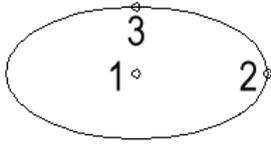
The default command will ask for two points on the major axis length 1 and 2 and the minor axis radius 3, Picture 103 below.



Picture 103

Center mode

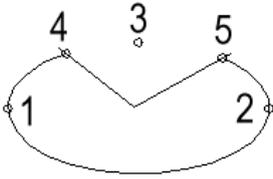
Select mode "S: Center" and input a major axis distance starting from center point of ellipse 1, the radius of the major axis 2 and the radius of minor axis 3.



Picture 104

Arc mode

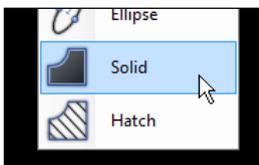
The "Arc" mode creates an open ellipse when defining a start angle and end angle. Select the "Ellipse" command and select "A: Arc" before defining the ellipse and the angles.



Picture 105

- ? My CNC-controller can't handle ellipses, how can I convert the object to arcs or lines?
You don't have to. It is done automatically by the postprocessor when you make the CNC-file.

4.9 Solid



Picture 106

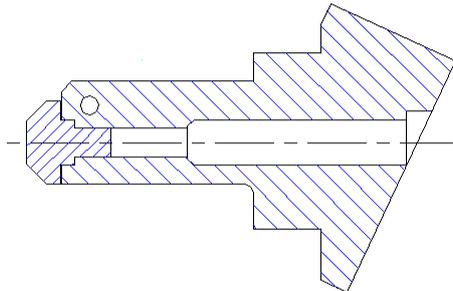
Use the "Solid" command to fill closed geometries with a selected color.

4.10 Hatch



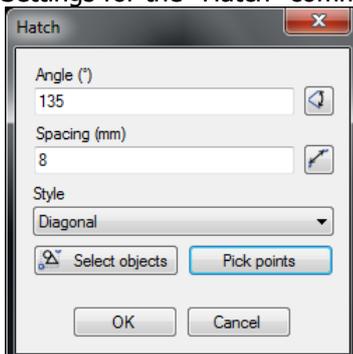
Picture 107

Use "Hatch" command to create a hatch pattern, for example to help illustrate sectioning.



Picture 108

Settings for the "Hatch" command:



Picture 109

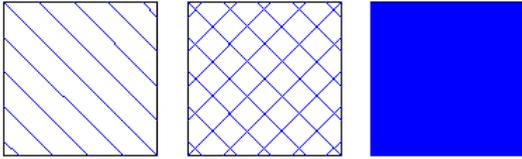
Angle

This is the angle of the Hatch lines.

Spacing

This is the distance between the hatch lines.

Style



The style can be Diagonal, Cross or Solid.

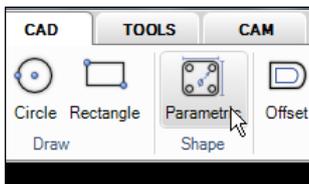
Select objects

Select a closed object and the Hatch will be inside the objects.

Pick points

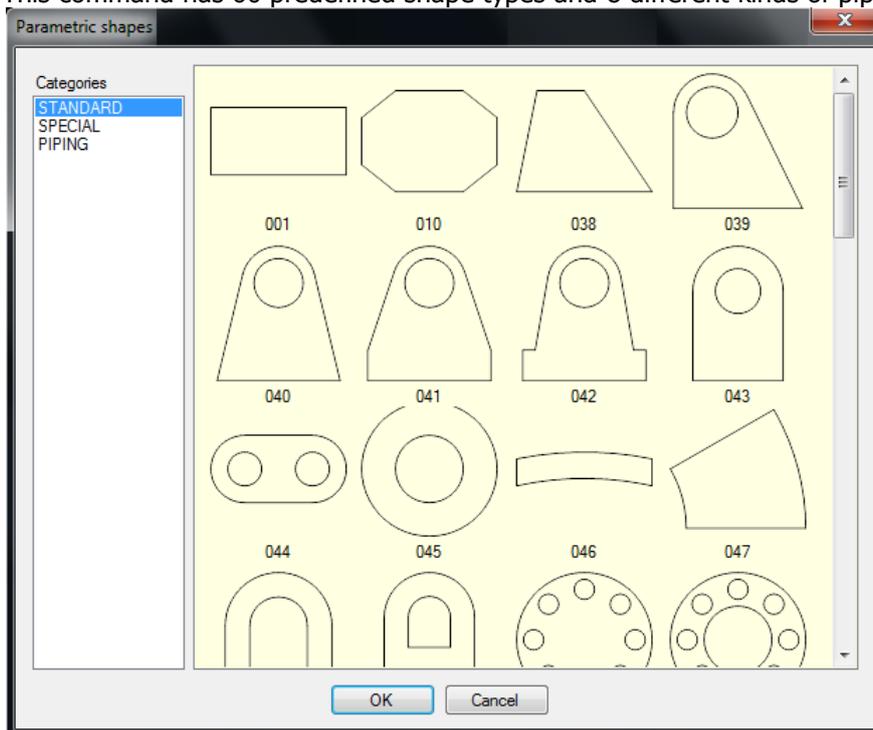
Pick a point in that area that should be hatched.

4.11 Parametric shape



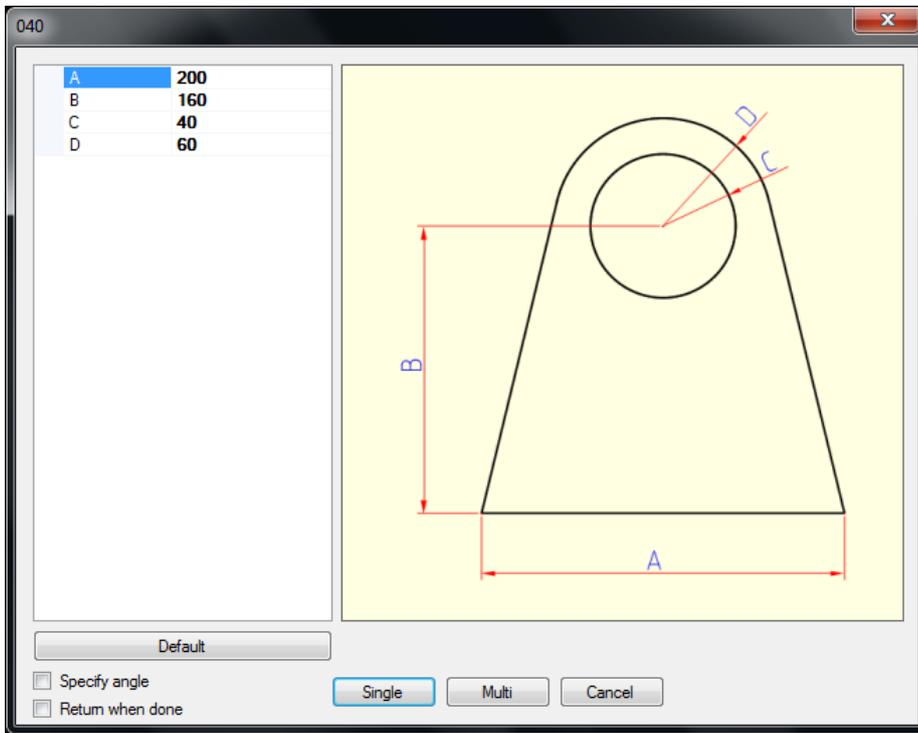
Picture 110

This command has 60 predefined shape types and 8 different kinds of pipe unfolding shapes.



Picture 111

Parametric shapes can be modified by simply opening the selected shape and entering the requested dimensions before inserting the shape into the drawing.



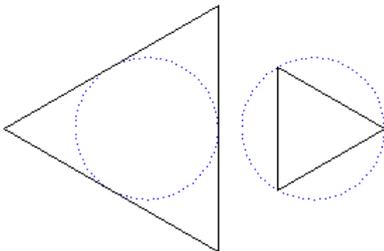
Picture 112

4.12 N-gon



Picture 113

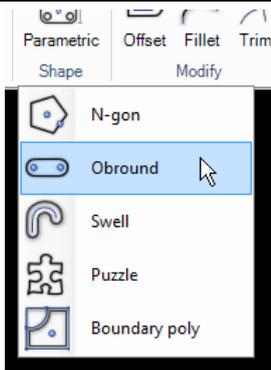
Use this command to create symmetrical polygons (triangles, squares, hexagons, etc.). Input the number of sides followed by the center position on the drawing.



Picture 114

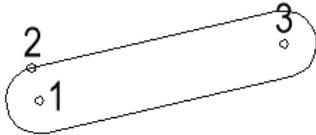
Press the "O" key to control if the N-gon should be inscribed or circumscribed.

4.13 Obround



Picture 115

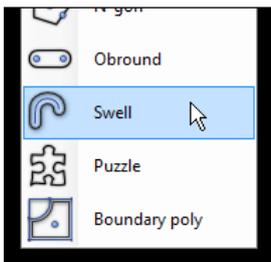
Specify starting point, radius and a distance (length) to create an obround profile.



Picture 116

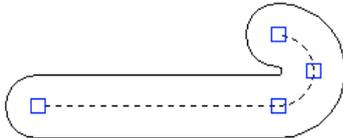
IGEMS will then automatically create the outline contour.

4.14 Swell



Picture 117

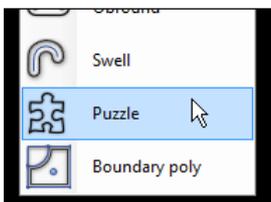
The "Swell" command makes a closed polyline around the selected object.



Picture 118

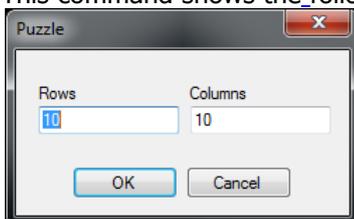
The command asks for center objects and a radius.

4.15 Puzzle



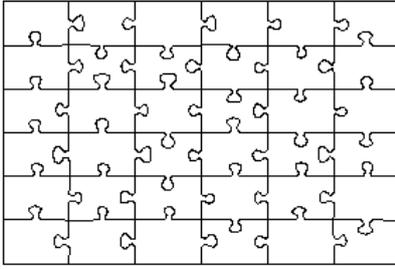
Picture 119

This command shows the following dialog box.



Picture 120

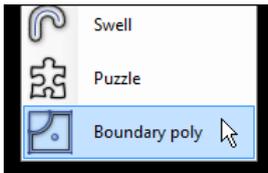
Enter two opposite points in a rectangle that describe the outside dimension. Press "OK". The command will create a random puzzle, like Picture 121.



Picture 121

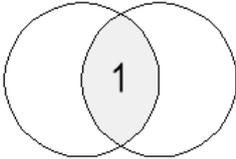
- ? Can the geometry be used to generate a toolpath?
Yes, this can be done with the "Quick" command.

4.16 Boundary poly



Picture 122

"Boundary poly" command is used to extract enclosed contours out of existing geometry. Activate the command and select the contour to extract.



Picture 123

The command will generate a polyline around the darker area when position 1 is selected, like in Picture 123.

4.17 Offset

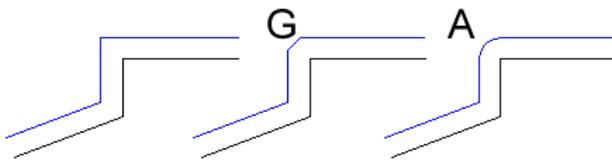


Picture 124

Shortcut "O"

This command creates an offset contour from the selected object with a specified distance. It asks for a distance, object and what side to create the new object.

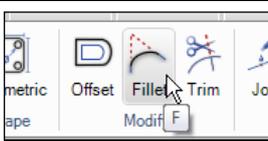
Select "G" for chamfering or "A" to make a fillet on outside corners.



Picture 125

Offset with default, Chamfer and Arc option.

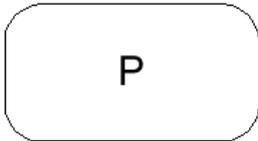
4.18 Fillet



Picture 126

Shortcut "F"

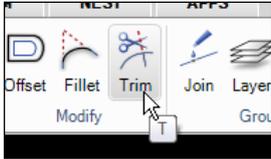
This command creates a tangential arc between two objects after having entered the size of the fillet radius.



Picture 127

To fillet all corners on a polyline with one click, first enter the fillet radius and then select "P: Polyline" option and select the objects.

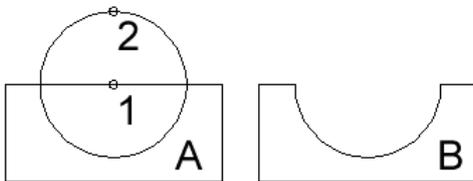
4.19 Trim



Picture 128

Shortcut "T"

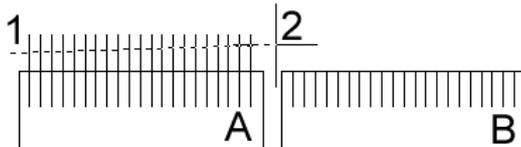
The "Trim" command deletes the portion of an object that intersects with other objects. It is necessary to have at least 2 objects to be able to trim.



Picture 129

In Picture 129, clicking on the objects at position 1 and 2 gives the result shown in B.

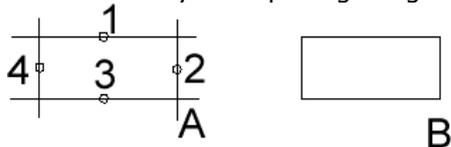
Use the "Fence" option to select many objects at the same time



Picture 130

SHIFT Trim

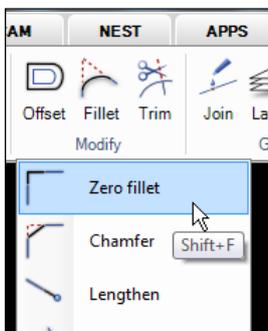
Use the SHIFT key while picking the geometry to invert the trimming to the opposite side.



Picture 131

The picking points indicate the portion of the object that should be not trimmed.

4.20 Zero fillet

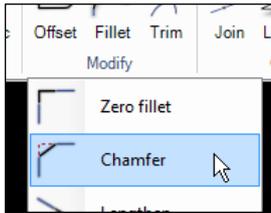


Picture 132

Shortcut "SHIFT+F"

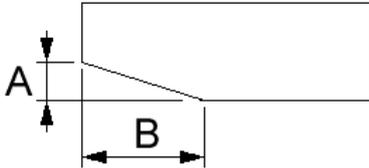
This command will extend or trim 2 segments to intersect with a zero radius.

4.21 Chamfer



Picture 133

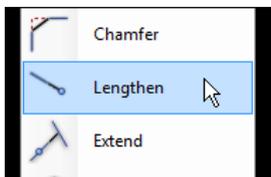
The "Chamfer" command extends or trims 2 segments to intersect and create a chamfer corner.



Define the chamfer length on the first segment and then the chamfer length on second segment before accepting.

A polyline is chamfered by holding down the SHIFT button when selecting object.

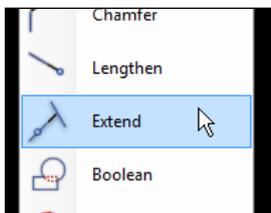
4.22 Lengthen



Picture 134

This command dynamically changes length on the selected object. Select near the end of the side to extend and drag the object to desired length.

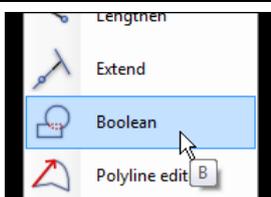
4.23 Extend



Picture 135

"Extend" an object to the next intersecting object.

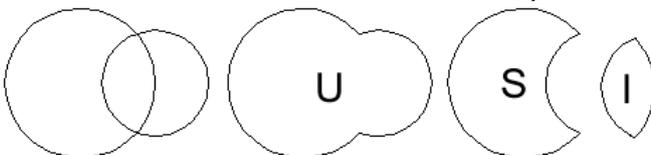
4.24 Boolean



Picture 136

Shortcut "B"

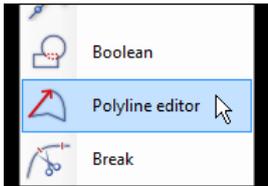
This command can make Boolean functionality on closed objects.



Picture 137

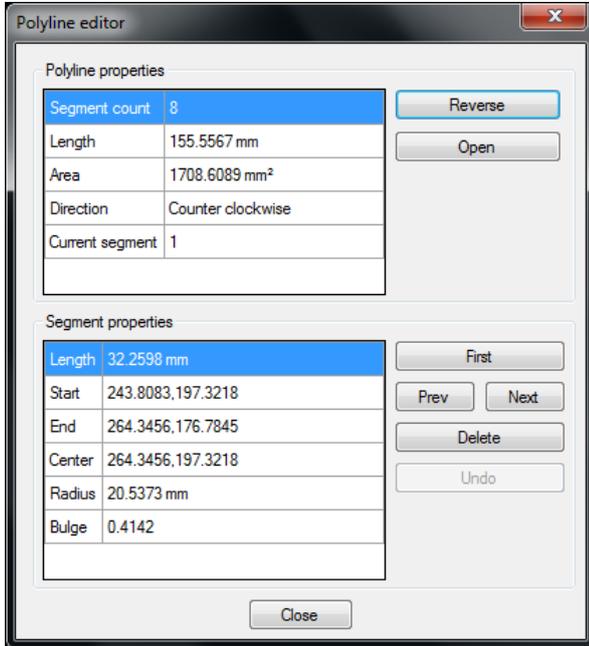
The command is set to "Union" mode by default which means that two objects will be merged together into one. The "Intersect" option creates a polyline of the area shared by the two objects. The "Subtract" option removes the overlapping area between the two geometries.

4.25 Polyline editor



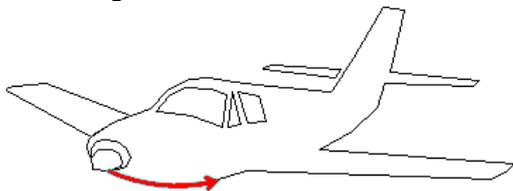
Picture 138

This command is used to check the information in a polyline and the following dialog box is shown:



Picture 139

This dialog box shows information about the complete polyline and the different segments in the polyline.



Picture 140

The actual segment is displayed in red, Picture 140, with an arrow indicating the direction of the object.

Reverse

This will reverse the direction of the polyline.

Open/Close

This option is only visible if the polyline is closed or if this option is changed to closed.

First

This will show the first object in the polyline

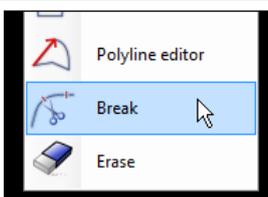
Previous/Next

This shows information about the next or previous segment.

Delete

This will delete the actual segment.

4.26 Break

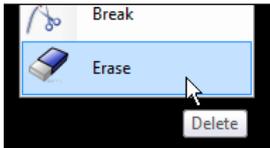


Picture 141

This command breaks objects up into several portions. The command asks for an object and a break point and splits the object at the selected point.

4.27 Erase

Shortcut "DEL"

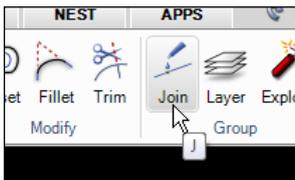


Picture 142

This command deletes the selected objects from the drawing.

Note! Use the "Undo" to restore objects that have been mistakenly deleted. See page 27.

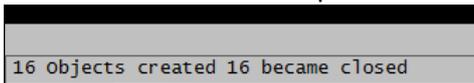
4.28 Join



Picture 143

Shortcut "J"

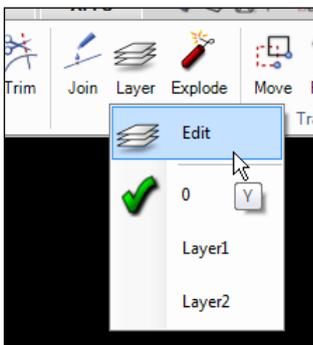
This command joins multiple objects into a single polyline. The command first asks for the objects then for a tolerance. The start or endpoints of the objects must be closer than this tolerance to be joined together.



Picture 144

The command line area has information about the result of the command.

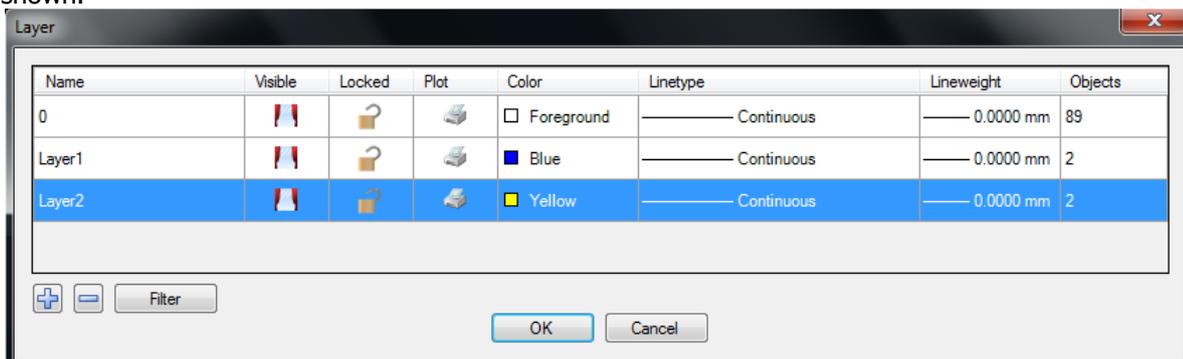
4.29 Layer



Picture 145

Shortcut "Y"

Some imported file types include layer information. The several layers on the drawing will be in the list. Click on one of the layers to activate. Press the "Edit" option or the shortcut and the following dialog box will be shown.



Picture 146

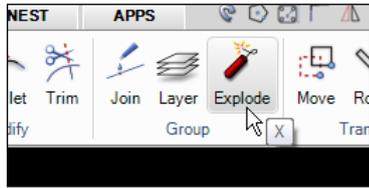
The "Layer" dialog box is used to create or modify existing layers with default line type and line weights. Layers can be "locked" to prevent modifying layer geometry or to block from being printed.

Layer for dimension

The "Dimension" style defines a layer to automatically use for all dimension objects.

4.30 Explode

Shortcut "X"

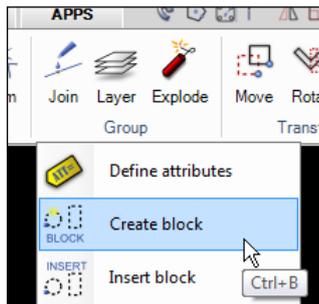


Picture 147

This command explodes a block, polyline or other composite object into separate objects. The opposite command to "Explode" is the "Join" command (see page 41).

4.31 Create block

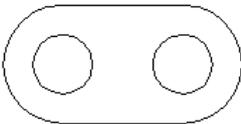
Shortcut "CTRL+B"



Picture 148

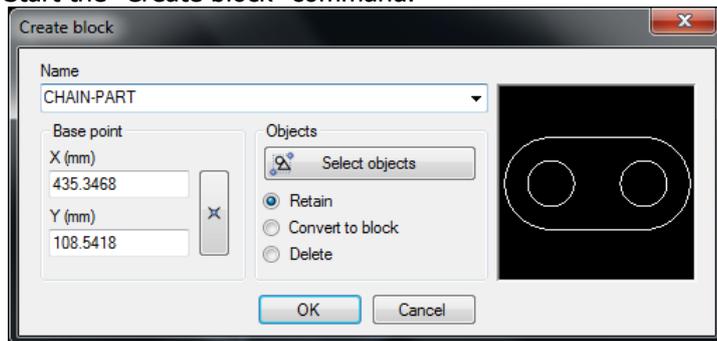
Use this command to define a set of objects that can be reused by inserting the block into the drawing.

- ☞ Create a figure, like Picture 149.



Picture 149

- ☞ Start the "Create block" command.



Picture 150

Name

This is the name of the block.

Base point

This is the base point when later inserting the block into the drawing.

Objects

Press this button to select the figure.

Retain

This leaves the figure intact as it is. The block has been saved in the block table.

Convert to block

This converts the figure to an insert and saves the information in the block table.

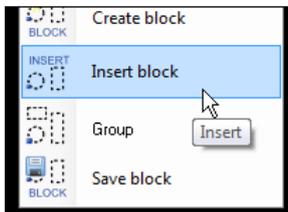
Delete

This removes the figure from the drawing and saves the block in the block table.

? Can the block be inserted into other drawings?

No, it must be defined in the drawing or in the template drawing. See pages 14 and 17. Then it can be used on all drawings that use that template.

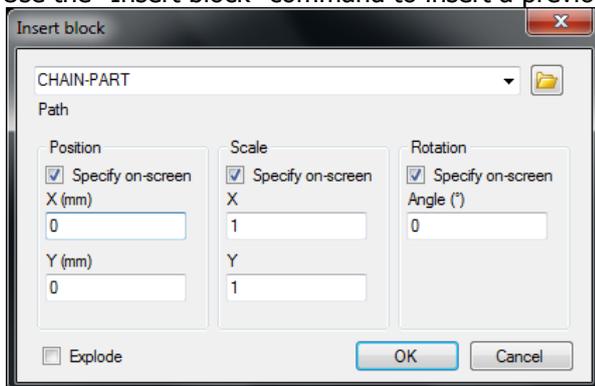
4.32 Insert block



Picture 151

Shortcut "INSERT"

Use the "Insert block" command to insert a previously defined block. The following dialog box is shown.

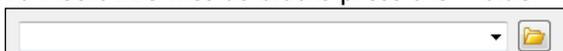


Picture 152

Select the named block to be inserted. It is then possible to position, scale or rotate the block while inserting it into the drawing.

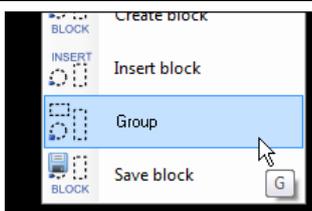
Insert DIG files

To insert DIG files as blocks press the "Folder" button in the dialog box and select the file to be inserted.



Picture 153

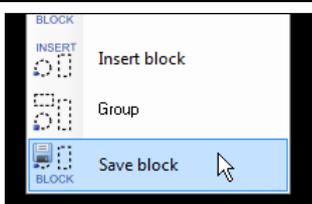
4.33 Group



Picture 154

The "Group" command asks for objects. All selected objects will be a member of that group and can now be easily handled together. The "Explode" command breaks up the group into individual segments.

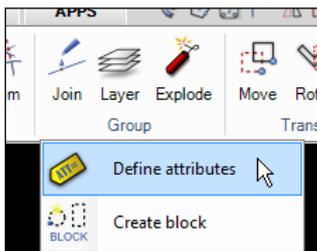
4.34 Save block



Picture 155

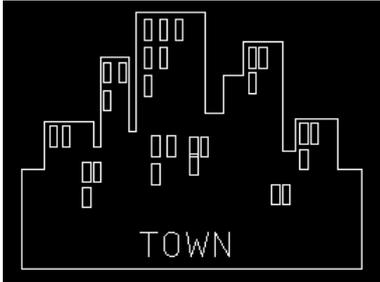
This command works the same way as the "Create block" command but it stores the block as a DIG file. It asks for the objects and a base point.

4.35 Define attributes

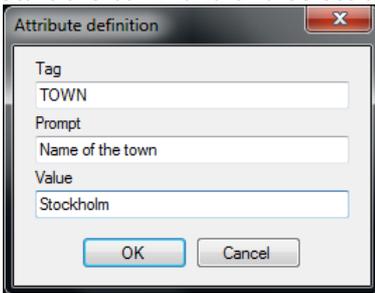


Picture 156

Create a block and include a text string that can be modified when later inserting the block in the drawing.



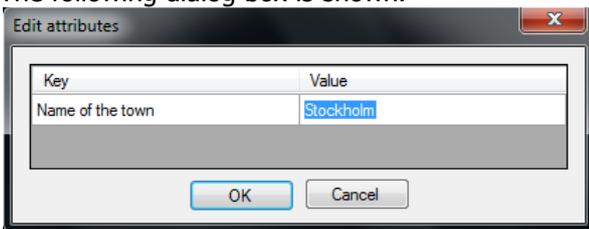
- ☞ Create a figure with a text "TOWN"
- ☞ Start the command and select the text.



Enter a prompt and a default value.

- ☞ Save the figure as a block
- ☞ Insert the block.

The following dialog box is shown.



Picture 157

- ☞ Change the value to "Gothenburg".
- When the block is inserted it will appear as follows:



Picture 158

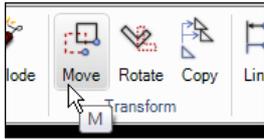
- ? What can these attributes be used for?
For example to fill in information in a title block.

? Can it be inserted in other drawings?

If you use "Create block" then it must be defined in the drawing or in the template drawing, see pages 14 and 17. Then it can be used on all drawings that use that template. If you use the "Save block" (save it on disk) option then you can use it on all drawings.

4.36 Move

Shortcut "M"

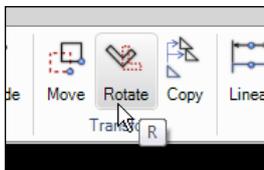


Picture 159

This command asks for an object to move, then a base point and finally a new position. The selected object will be moved to the new position.

4.37 Rotate

Shortcut "R"

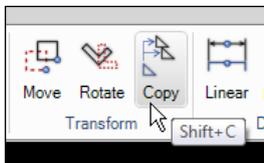


Picture 160

This command rotates an object around a specified point. Use the "Reference" option instead of entering the new rotation angle. After the command is completed there will be a request to "Keep original" or not.

4.38 Copy

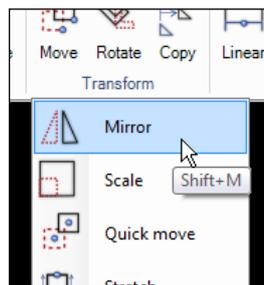
Shortcut "SHIFT+C"



Picture 161

This command asks for an object to copy, then a base point and finally a new position. The command must be interrupted by the ESC-key.

4.39 Mirror

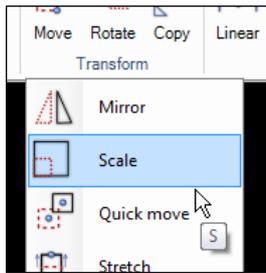


Picture 162

Shortcut "SHIFT+M"

This command asks for an object and then a mirror line. After the command is completed there will be a request to "Keep original" or not.

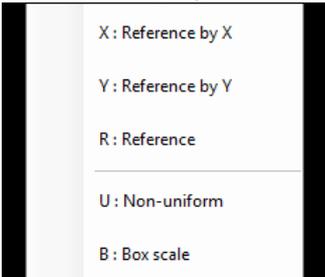
4.40 Scale



Picture 163

Shortcut "S"

This command asks for an object to scale, then a base point and a scale factor. Instead of entering the "Scale" factor, right mouse click to see the following options:



Picture 164

Reference by X

This option automatically calculates the scale factor. Enter the outermost size in X.

Reference by Y

This option automatically calculates the scale factor. Enter the outermost size in Y.

Reference

Asks for a reference length and then a new length. All objects will be scaled by reference to this length.

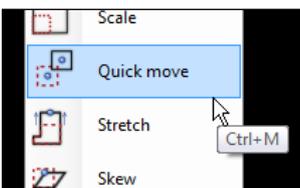
Non-uniform

Define a scale factor in X and in Y. An input of 2, 3 will make the object two times the original size in X and three times in Y.

Box scale

This option lets different entries for the outermost size in X and Y. In this option it is best to have the zero point at the lower left corner of the selected objects.

4.41 Quick move

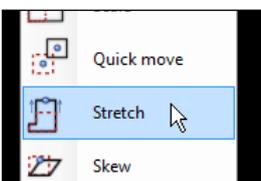


Picture 165

Shortcut "CTRL-M"

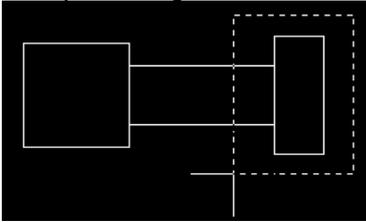
"Quick move" automatically selects an object by moving the cursor near the object. Unlike the "Move" command, there is no need to select or input a base point to move the object. The object will highlight a light blue color when the mouse cursor gets close to the object.

4.42 Stretch



Picture 166

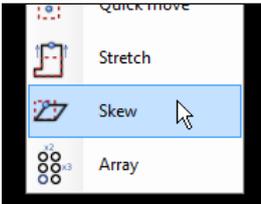
This command changes part of an object's length while keeping other parts the same. The command is mostly used together with the "Crossing" option in the select object.



Picture 167

Using Picture 167 as an example, objects completely inside will be moved. Objects that intersect with the crossing rectangle are changed. Objects that are completely outside will remain unchanged.

4.43 Skew



Picture 168

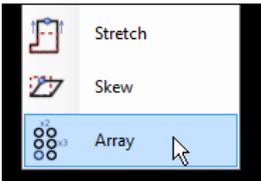
This command asks for objects to be skewed, a base point to skew from and the skew angle in Y and X.



Picture 169

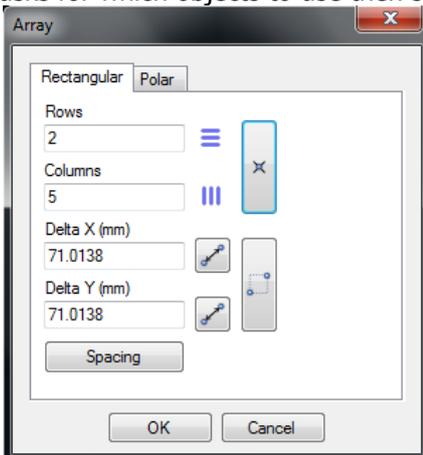
Picture 169 shows an example where Y is skewed 30 degrees (60 degrees from X).

4.44 Array



Picture 170

This command is a simple form of nesting that copies objects in a rectangular or polar array. The command asks for which objects to use then shows the following dialog box:



Picture 171

4.44.1 Rectangular

This option copies the selected objects in rows and columns.

Delta

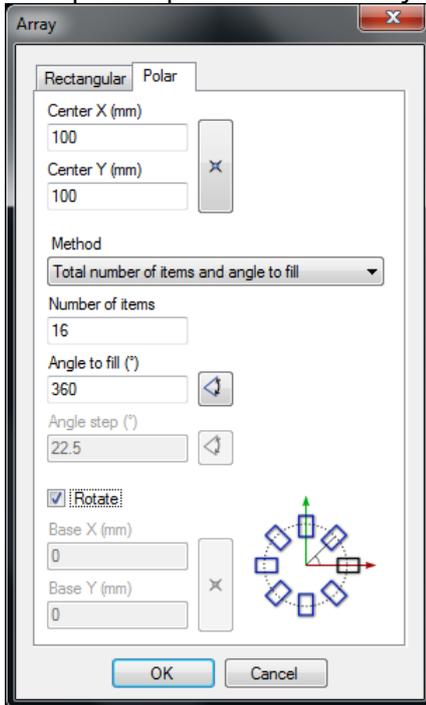
"Delta X" and "Delta Y" are the distances between the rows and columns

Spacing

To have a space between each line and column press the "Space" button and enter the space. The "Delta" between the lines and columns is calculated from the dimension of the selected objects.

4.44.2 Polar

This option copies the selected objects in a polar layout.



Picture 172

Center

Enter the center of the array. Use the "Pick" button to pick a point.

Method

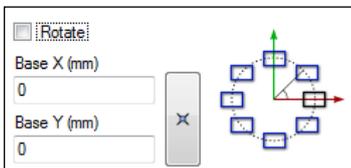
There are several methods to describe the angles between each copy.

Rotate



Picture 173

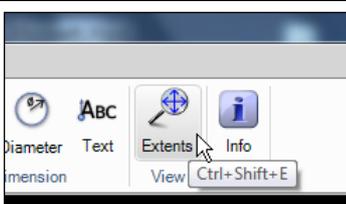
If the selected objects are rotated around the center then the command does not need any more information.



Picture 174

If the objects are copied without rotation then the command needs a base point for the objects.

4.45 Zoom extents



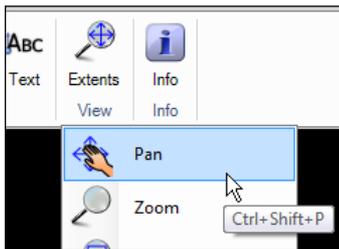
Picture 175

Shortcut "CTRL+SHIFT+E"

Click this button for IGEMS to zoom out so all objects are shown on the screen.

Note! Zoom extents by double clicking the mouse wheel if preferred.

4.46 Pan



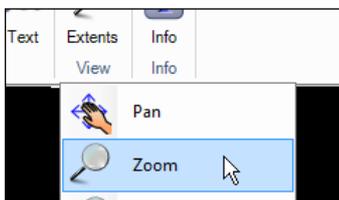
Picture 176

Shortcut "CTRL+SHIFT+P"

The "Pan" command is used to reposition the drawing area in X and Y by selecting a base point and a second point for new position.

Note! Hold down the mouse wheel while moving the mouse to pan.

4.47 Zoom

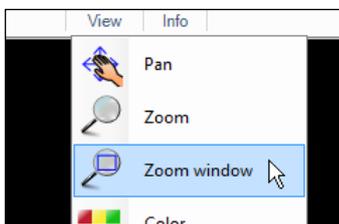


Picture 177

Start the command and then pick a point to zoom in. To zoom out hold down the CTRL key while selecting.

Note! Roll the mouse wheel to zoom out.

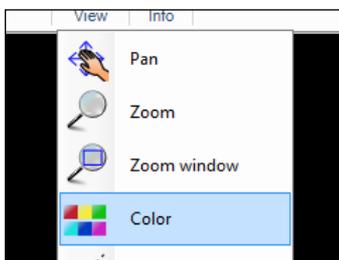
4.48 Zoom window



Picture 178

This command needs two points to describe a rectangle. IGEMS will zoom up that area.

4.49 Color



Picture 179

The currently active color is shown at the lower left corner on the screen. The "Color" command is also started by clicking on this button.



Picture 180

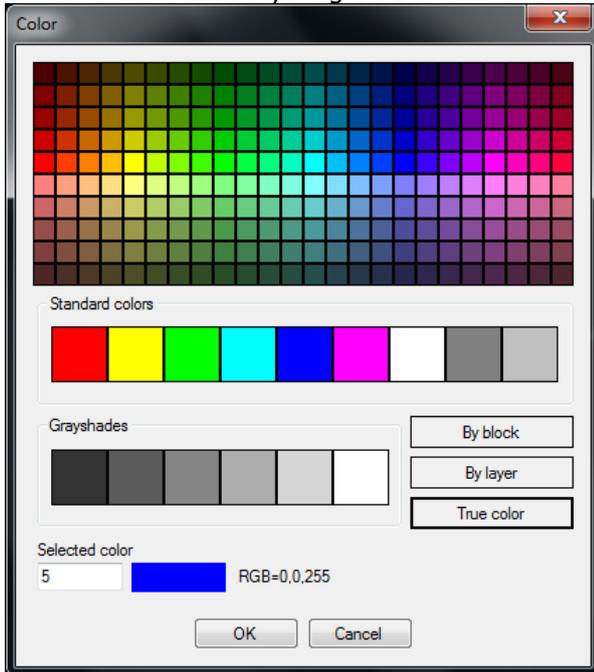
Use different color to visually separate information on the screen. To make paper drawings on a printer then use different plot line weights that are bound to different colors. There are two major ways of using colors:

Layer controlled

Bind different colors to different layers so changing layers also changes color. If a template is set up predefined layers then color doesn't need to change. The "Layer" command is described on page 41.

Different color on same layer

Use this command if layering does not matter but items need to be separated on the screen.



Picture 181

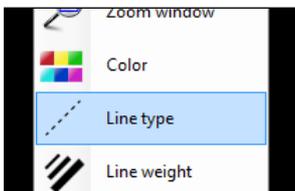
By block

Objects with color "By block" have the same color as the insert.

By layer

Objects with color "By layer" have the same color as the insert.

4.50 Line type



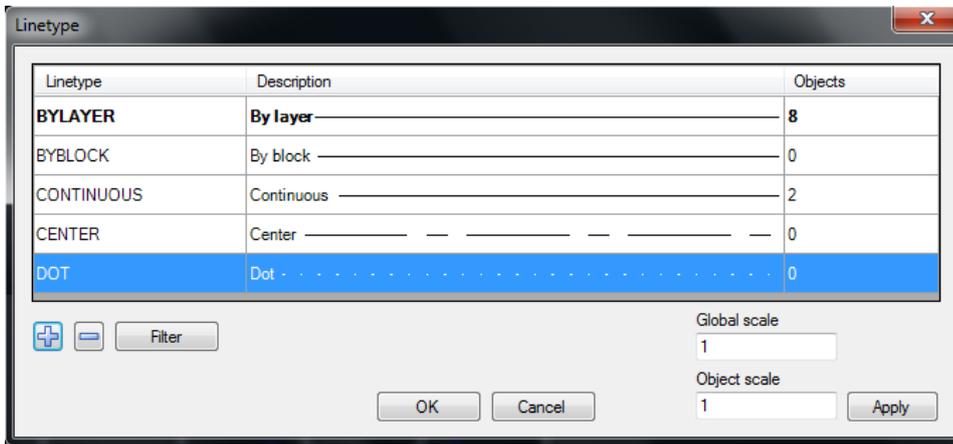
Picture 182

Layer controlled

Bind different line types to different layers. The linetype will then be changed when you change layer. If a template is configured with predefined layers and line types, line type will never have to be changed manually. The "Layer" command is described on page 41.

Different line types on same layer

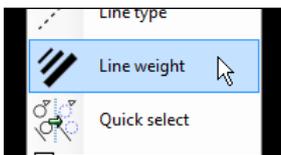
Use this command if layering does not matter but items need to be separated on the screen.



Picture 183

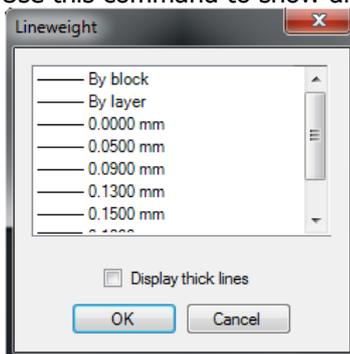
Add or remove line types from the selection list by using the "plus" and "minus" buttons. Scales can be set globally for all lines or local to modify the style on a specific object. Press "OK" button to select line type.

4.51 Line weight



Picture 184

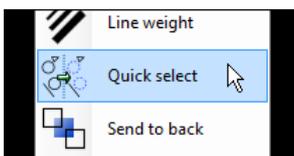
Use this command to show different line weight on the CAD screen.



Picture 185

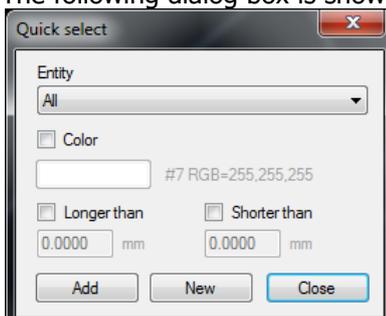
The "Display thick lines" checkbox must be activated to see the different weights on the screen. The "By block" and "By layer" works in the same ways as for color and line type.

4.52 Quick select



Picture 186

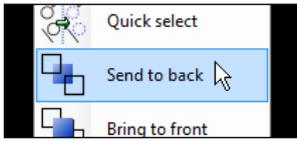
Use this command to sort out objects by object type, color or by segment available on the active drawing. The following dialog box is shown:



Picture 187

Select the filter for object types, color or length and use the "Add" button to select the objects. The "New" button starts the selection again from the beginning. The selected parts remain selected when the "Close" button is chosen. Now other commands can be used on the selected objects.

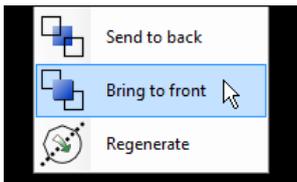
4.53 Send to back



Picture 188

Use this command to control which object should be visible when two objects are on top of each other. This command is often used for pictures or similar objects on the screen.

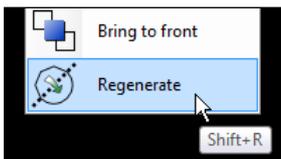
4.54 Bring to front



Picture 189

This command works opposite to "Send to back" command. Select object and pick the command to place it on top of other objects.

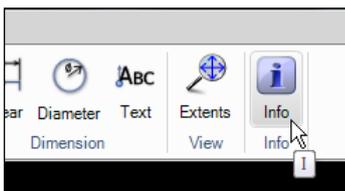
4.55 Regenerate



Picture 190

This command "regenerates" the entire drawing and updates the screen coordinates for all objects. "Regenerate" is often done automatically when zooming in or out.

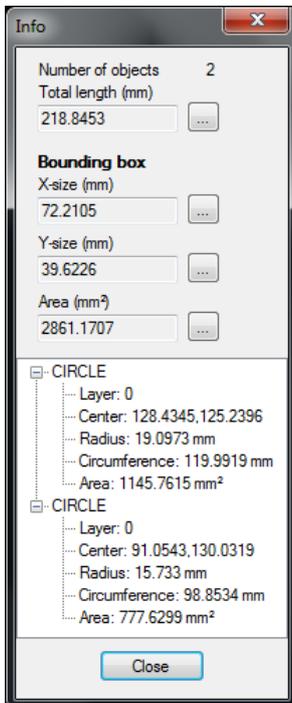
4.56 Info



Picture 191

The "Info" command is the fastest way to get information from objects on the drawing.

 Draw two circles. Start the "Info" command and select the circles.



Picture 192

This command shows the following information:

Information about the selection

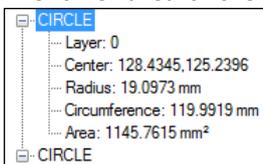
The upper area of the box provides information about how many objects are selected and their total length. The bounding box gives information about the area and the size needed for the objects.

? Can this information be used for calculating cutting costs?

Yes it can, but you have more exact tools if you make a complete cost calculation, which needs more time. This "Info" command is very quick.

Object information

The lower area of the box provides information about each object in the selection.



Picture 193

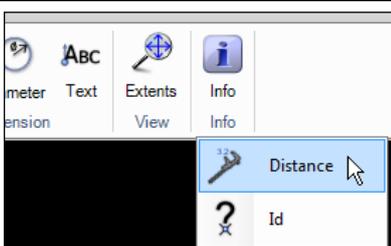
Select one object in the list to highlight that object on the screen.



Picture 194

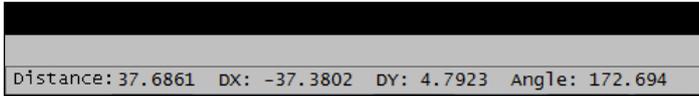
The green rectangle shows the bounding box, the yellow object is the selected object on the screen.

4.57 Distance



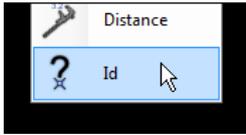
Picture 195

This command needs two points and will display the information on the command line area, Picture 196.



Picture 196

4.58 Id



Picture 197

"ID" shows the absolute coordinate in X and Y for the selected position. "Absolute" means the coordinate position in relation to IGEMS fixed zero point.

5 Dimension

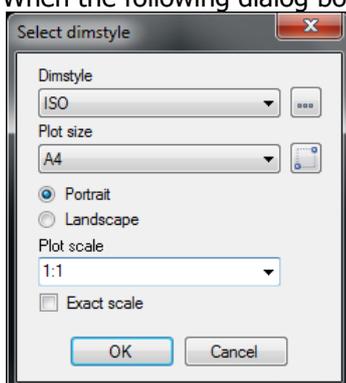
IGEMS R10 includes a basic "Dimension" command. Follow the steps below to make proper paper drawings where the dimension objects are correctly sized and the drawings are in an exact scale:

5.1.1 Step 1: Draw geometry

Draw the geometry to print out in natural size. Be sure to include the outermost geometry. Remove any unneeded objects that may change the extents of the drawing.

5.1.2 Step 2: Make dimensions

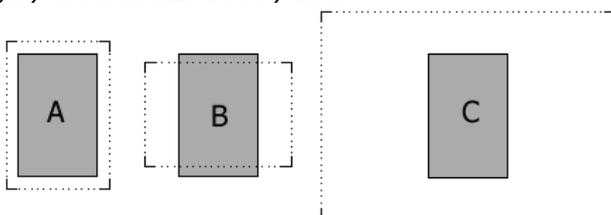
When the following dialog box shows add the dimensions:



Picture 198

- ☞ Set the plot size and the paper orientation. IGEMS will now suggest the best plot scale for the drawing.
- ☞ Press OK

Read this section if interested in how dimension scaling is done automatically. The command will compare the plot size and the orientation of the paper with the extents of the drawing and will suggest a plot scale for the drawing. In the examples below the paper size is described with dotted lines, the extents are the gray areas in size 160 by 250.



Picture 199

In example A the drawing should print out in Portrait on an A4. The area fits well inside the paper and IGEMS will suggest plot scale 1:1 (Drawing units: plot units). The dimension scale will be set to 1.0 and text height can be 3.5 arrow length 3mm and so on.

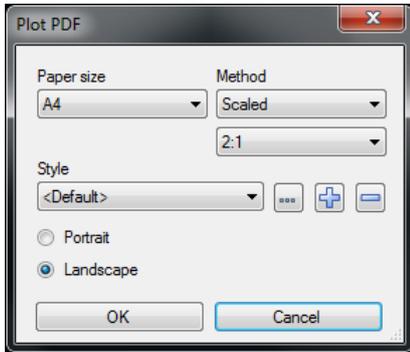
In example B the same drawing should print out in Landscape mode on an A4. Since the area on the paper does not fit when the paper is oriented like this IGEMS recommends example C.

In example C the plot scale will be 2:1 and all dimensions are scaled to double size (dimension scale will be set to 2.0). Text height will be set to 7.0 and arrows to 6 mm and so on.

When the drawing is plotted, the size will be reduced to 50% and the geometry and the dimension object will be in correct size.

5.1.3 Step 3 Plot

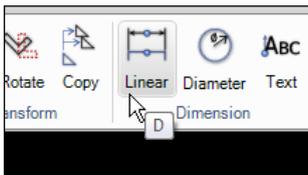
To plot in correct scale use the "Scaled" method.



Picture 200

Set the Paper size, plot scale and orientation to the same as in step 2. Press "OK" to place a scaled paper rectangle on the screen. The drawing and the dimension will automatically be in correct size. Read more about the "Plot" command on page 18.

5.2 Linear

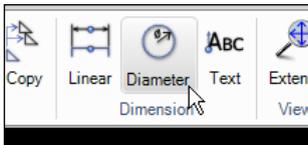


Picture 201

Shortcut "D"

The linear dimension is used for horizontal, vertical or tilted dimensions. To enter the dimension either select an object or press the spacebar and then enter two points.

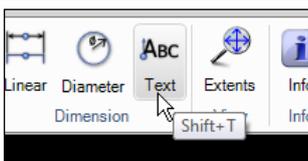
5.3 Diameter



Picture 202

This command is used to add dimensions on circles and arc objects. Select a point on the arc or circle and place the dimension.

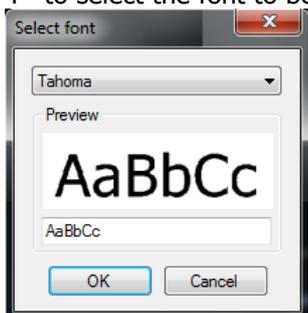
5.4 Text



Picture 203

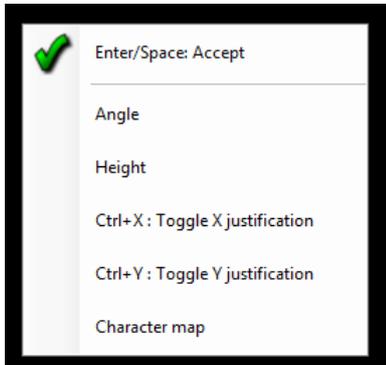
Shortcut "SHIFT+F"

Use this command to insert text into the drawing area. Instead of selecting the start point of the text press "F" to select the font to be used.



Picture 204

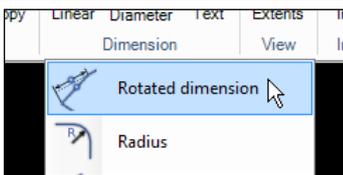
Select font style, insertion base point, height and type the text. Right mouse click while writing the text to control the angle, height and justification.



Picture 205

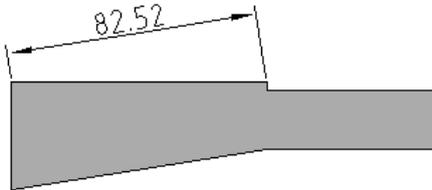
It is also possible to show a character map of all characters in actual font.

5.5 Rotated dimension



Picture 206

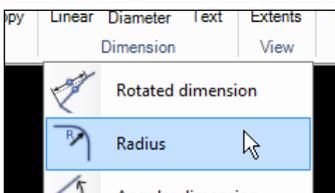
Use this command to place a dimension line that is not aligned to the measurement.



Picture 207

Select the angle of the dimension line and then select the object or two points.

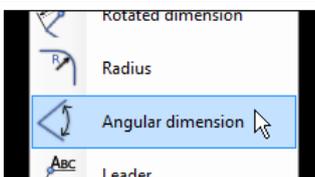
5.6 Radius



Picture 208

This command creates a dimension showing the Radius for an arc or circle.

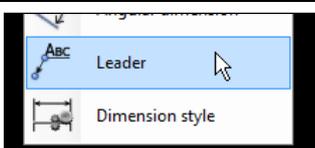
5.7 Angular dimension



Picture 209

This command creates an angle dimensioning between 2 linear segments. Move the cursor before placing the dimension to select which quadrant to use.

5.8 Leader



Picture 210

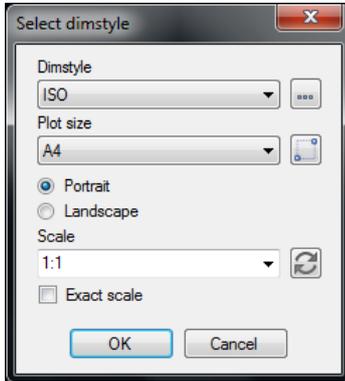
This command will add a leader with information text to an existing object. Select the command and enter the text. Place the text on the drawing area or press the spacebar or Enter to select an object.

5.9 Dimension style



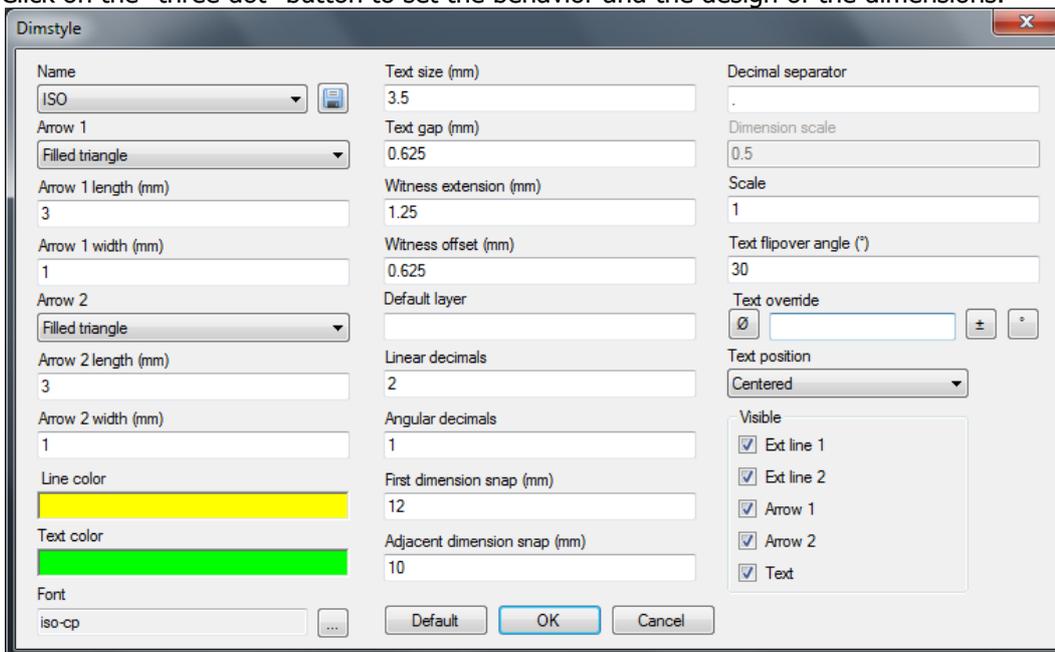
Picture 211

Click to show the dimension style dialog box. Read more on this command on page 55.



Picture 212

Click on the "three dot" button to set the behavior and the design of the dimensions.



Picture 213

6 CAM-Tools

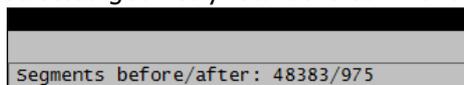
The "CAM-Tools" module includes commands for analyzing, preparing and optimizing the geometry before starting to make a toolpath. To use these commands there must be a license for the "CAM-Tools" module.

6.1 Clean up



Picture 214

This command removes double objects, overlapping geometry and closes gaps that may occur in the selected geometry. Start the command and enter a tolerance to fit a new curve and select objects.



Picture 215

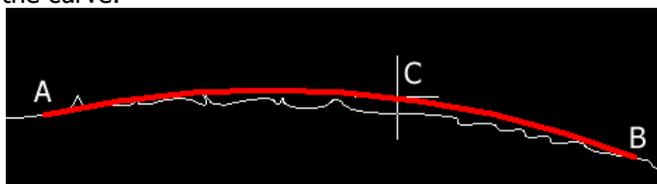
The command line area shows a summary of the operation.

6.2 Edgefix



Picture 216

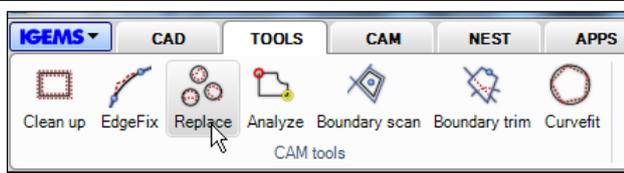
This command replaces a portion of a polyline with an arc or line to increase the quality and smoothness of the curve.



Picture 217

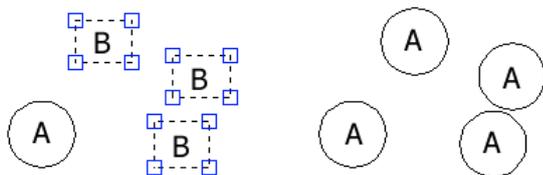
Picture 217 shows a start point A and an end point B. Place the cursor between points and press the spacebar to select position C for the new curve which will replace the old curve with an arc or a line.

6.3 Replace



Picture 218

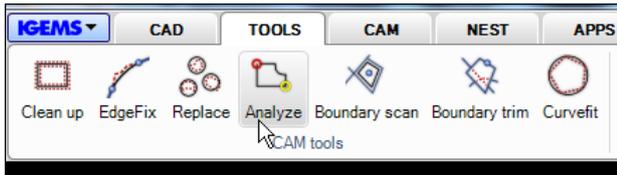
This command replaces objects with other objects.



Picture 219

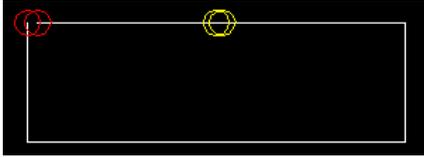
The command asks for a source object (A) and then target objects (B). All target objects will be replaced by the source object. The position is calculated by the center of gravity position.

6.4 Analyze



Picture 220

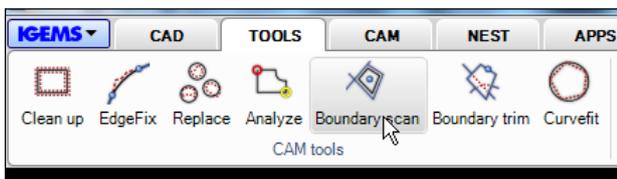
This command asks for objects and creates red circles on gaps between objects and yellow circles if the objects are overlapped.



Picture 221

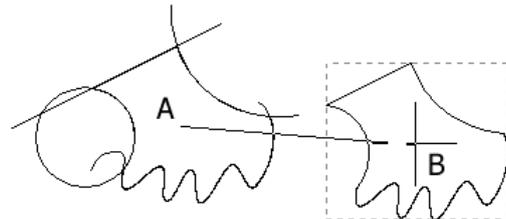
The circles are joined in a group. Select one circle and then press delete.

6.5 Boundary scan



Picture 222

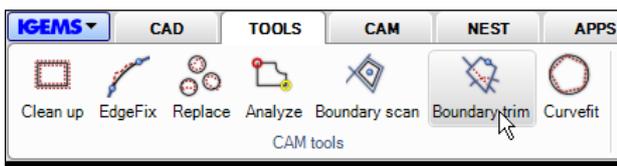
This command extracts closed polylines from enclosed areas.



Picture 223

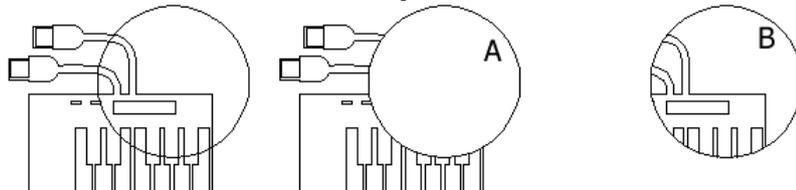
Specify a tolerance for automatically closing gaps. Select the objects and click inside the area that describes the wanted geometry (A). Press spacebar or Enter and move the geometry (B). Select a color by using shortcut "C" while picking the point.

6.6 Boundary trim



Picture 224

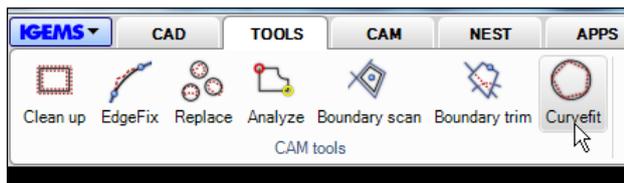
This command trims or deletes objects that are inside or outside a closed geometry.



Picture 225

Start the command and select a point on the circle (A). Everything inside the circle is deleted. Objects that are intersecting with the circle are trimmed. Hold down the SHIFT key while selecting the circle to make the opposite result.

6.7 Curvfit



Picture 226

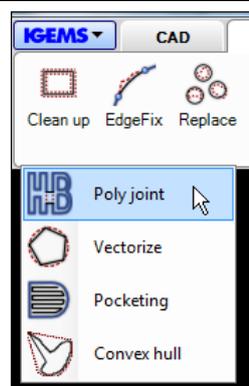
This command converts a polyline consisting of many short objects into a smooth curve with lines and arcs based on a given tolerance. The feature is useful to eliminate short line segments that may cause the machine not to operate as it should. It also may reduce file size.



Picture 227

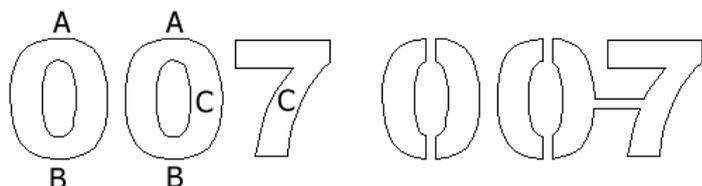
The command line area shows a summary of the operation.

6.8 Poly joint



Picture 228

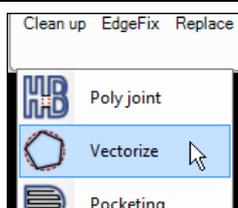
This command is used to join areas together or to split areas. The function must be applied on closed polylines.



Picture 229

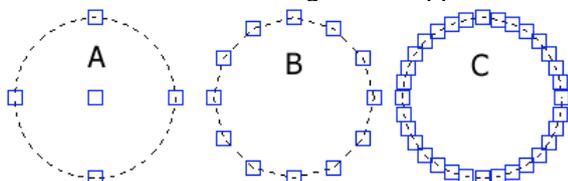
The command asks for a width and then for points. Picture 229 is an example: Select a point outside (A) to outside (B) and the geometry is spit up in two geometries. Select a point inside geometry (C) to another inside geometry (C) and the command will join the two geometries.

6.9 Vectorize



Picture 230

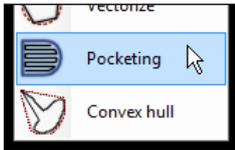
The "Vectorize" command gives an opposite result as the "Curvfit" command.



Picture 231

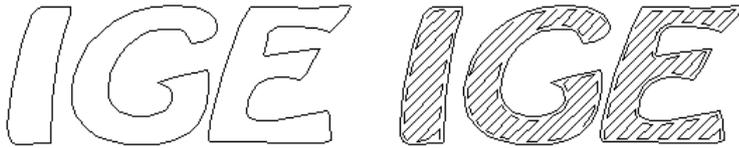
Select the command and enter a tolerance. The tolerance decides how much the new curve will deviate from the original curve (A). A small tolerance (C) will create more and shorter line segments and will stay closer to the original curve compared to entering a large tolerance (B).

6.10 Pocketing



Picture 232

"Pocketing" generates a zigzag pattern that can later be used to create a toolpath for milling.



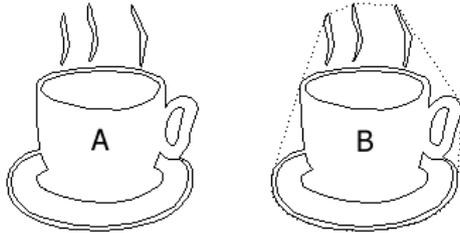
This command expects input for angle and a value for the step over before selecting the closed contour to apply the pocket pattern.

6.11 Convex hull



Picture 233

"Convex hull" creates an enclosing polygon around selected objects.



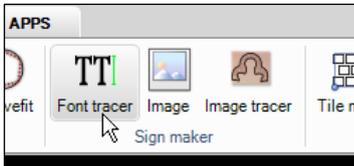
Picture 234

"Convex hull" has only convex geometry - no concave.

7 SignMaker

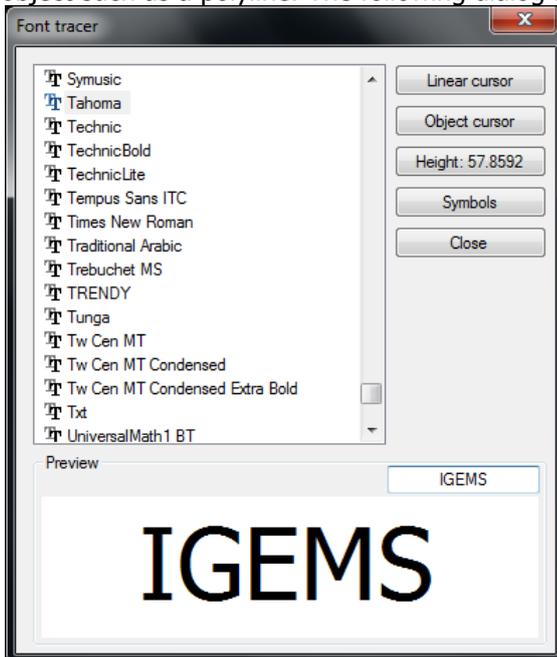
This module includes commands for making signs. For example: Converting images to CAD vectors, converting true type fonts, etc. There must be a license for the "SignMaker" module to use these commands.

7.1 Font tracer



Picture 235

This command converts true type fonts into vector graphics so text can be linear follow an existing curved object such as a polyline. The following dialog box is shown:



Picture 236

Select the font

The preview area will show example text of differing fonts.



Picture 237

Linear cursor

Click on this button to set the start point for the text.

Object cursor

Use this option to select an object to follow.

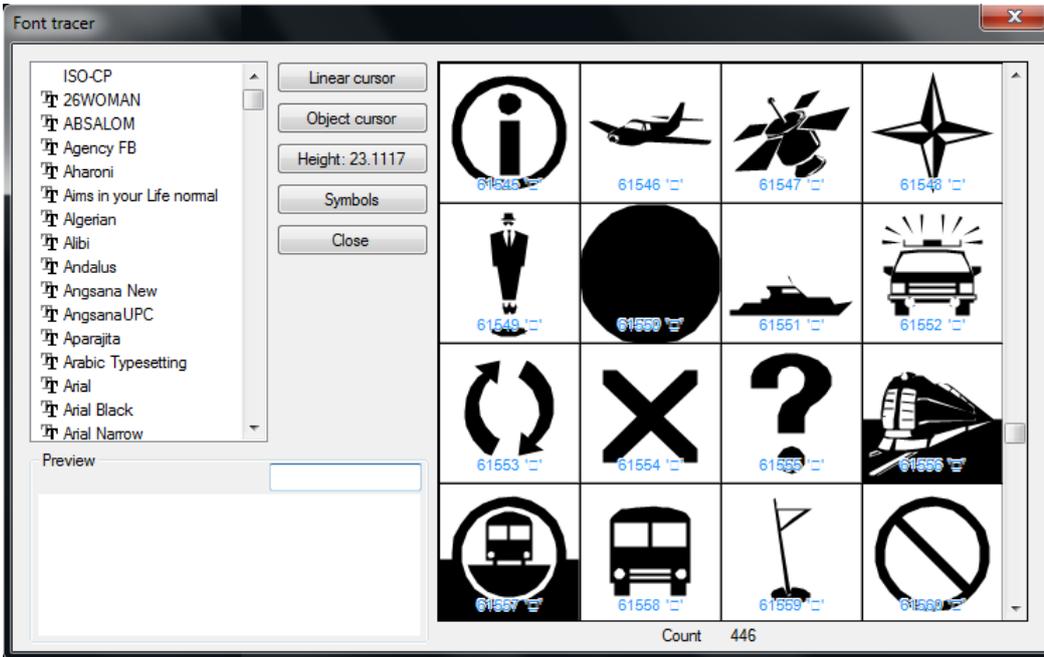


Picture 238

Select the start point and the cursor will follow the geometry as the text is typed.

Symbols

Use the "Symbols" button to expand the dialog box:



Picture 239

Double click on the sign to insert it onto the drawing.

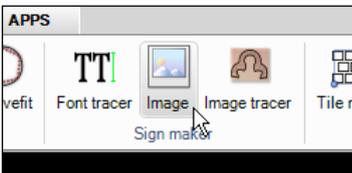
7.1.1 Explode

Use the "Explode" command (described on page 42) to convert the text to CAD vectors.



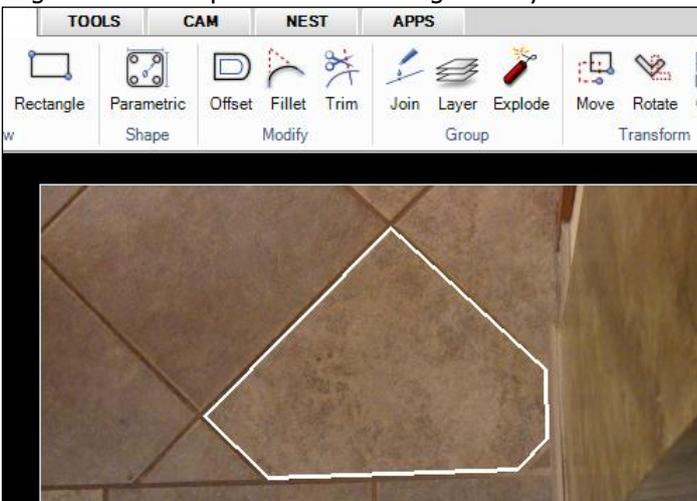
Picture 240

7.2 Image



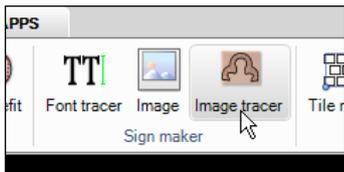
Picture 241

Use the "Image" command to insert images into the IGEMS drawing. The image can be scaled to correct size and used as a background. A common method is to draw an object on the image. These objects can be used to generate a toolpath that cuts the geometry in correct size.



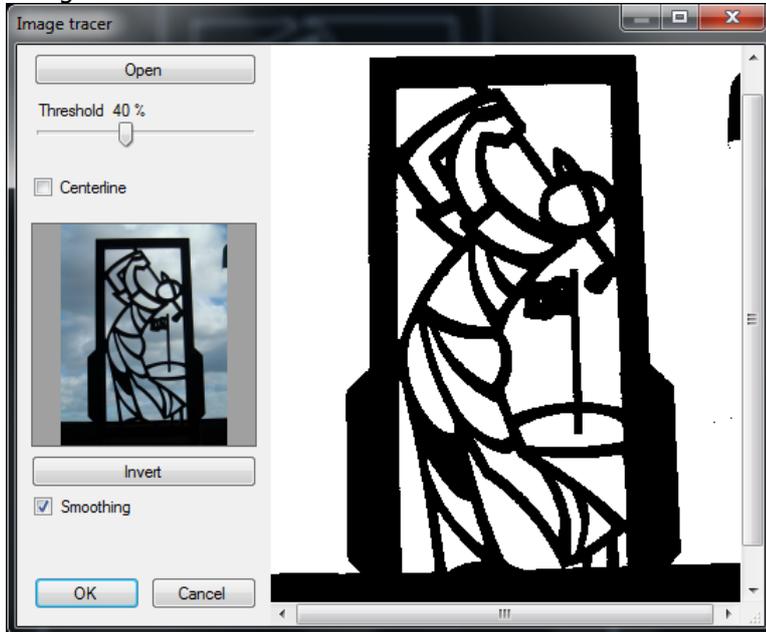
Picture 242

7.3 Image tracer



Picture 243

"Image tracer" reads and converts a raster file into lines and arc segments.



Picture 244

Open

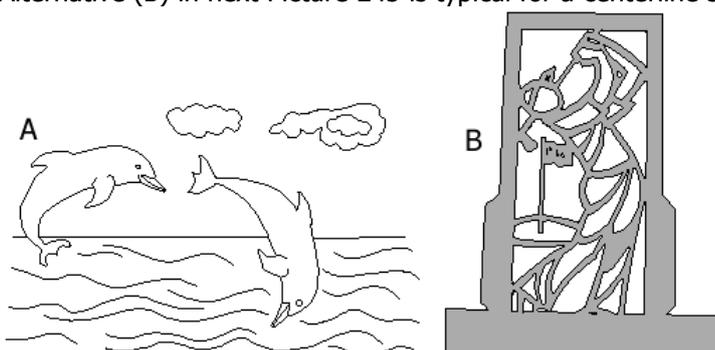
Use this button to select the file and the image will show on the right side of the dialog box. Supported files are PNG, JPG and BMP.

Threshold

Use this setting to control the brightness of the image.

Centerline

This option should be used on scanned drawings when the geometry is drawn by pen or something similar. Alternative (B) in next Picture 245 is typical for a centerline scanning.



Picture 245

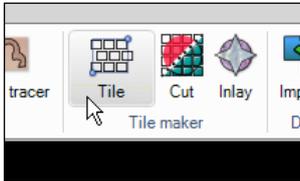
Alternative B shows an outline scanned example. This option will draw objects around black areas.

Note! The Image tracer does not run any automatic "Curvefit". For smoother geometry use the "Curvefit" command afterwards. See page 61.

8 Tile maker

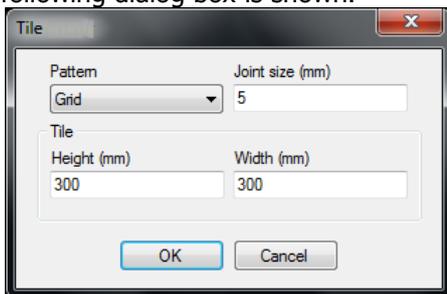
The "Tile maker" module includes a set of commands especially designed for tile cutting. There must be a license for the "Tile maker" module to use these commands.

8.1 Tile



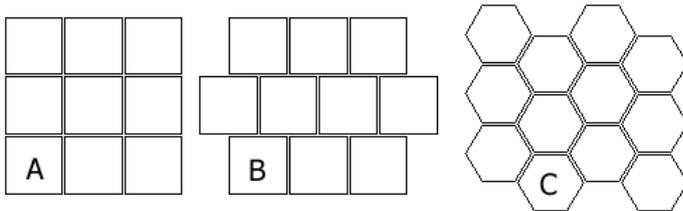
Picture 246

The "Tile" command creates a grid of rectangle or hexagon shapes with a specified part distance. The following dialog box is shown.



Picture 247

Pattern



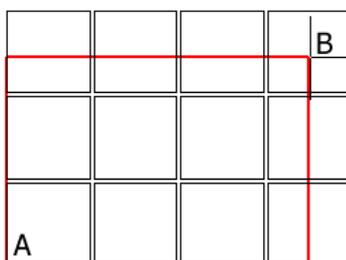
Picture 248

The "Tile" command supports the pattern type: Grid (A), Jagged Grid (B) and Hexagons (C).

Dimensions

The "Height" and "Width" is the X and Y sizes of the tiles and the "Joint size" is the distance between the tiles.

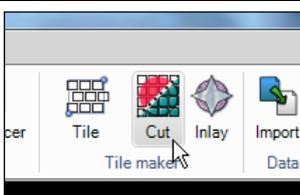
Number of tiles



Picture 249

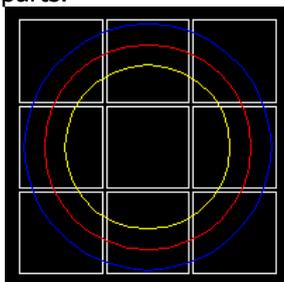
Press "OK" and IGEMS asks for two corner points (A) and (B) that describe the area for the tiles.

8.2 Tile cut



Picture 250

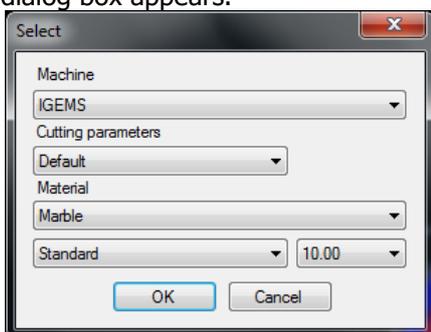
The "Tile cut" command extracts patterns from a drawing and automatically nests and cuts the extracted parts.



Picture 251

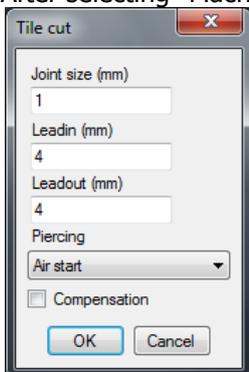
- ☞ Prepare a drawing like the above Picture 251.
- Make 9 tiles with the "Tile" command.
- Make three circles: blue, red and yellow.
- Start the "Cut" command.

This command needs information from the machine and material. If no machine is selected the following dialog box appears.



Picture 252

After selecting "Machine" and "Material" the dialog box for "Tile cut" command is shown:



Picture 253

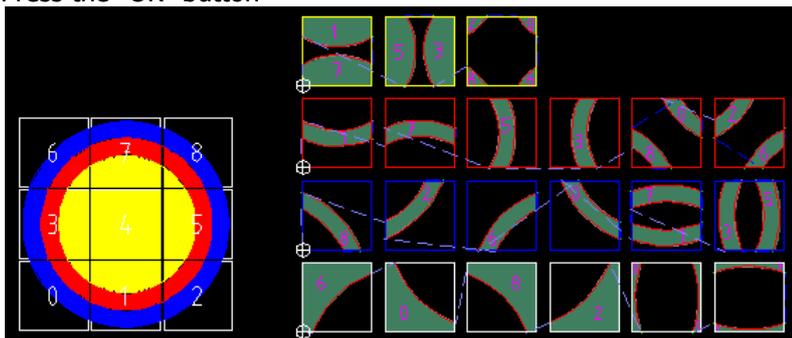
Joint size

This is the distance between the tiles.

Cutting information

See the "2D-CAM" module for more information about leadin, leadout and piercing type. When the compensation checkbox is activated the toolpath will be compensated for the tool radius.

- ☞ Press the "OK" button

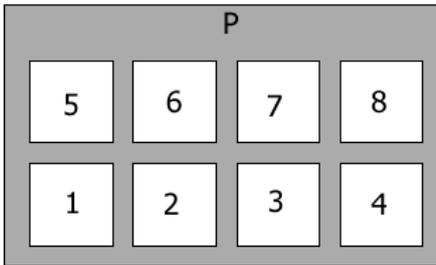


Picture 254

IGEMS will now nest all tiles by color, all toolpaths are added and the job will be ready to post.

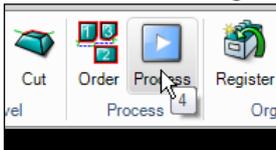
8.2.1 Tile fixture

The "Tile fixture" is simple fixture to locate the tiles at correct positions on the cutting machine. The fixture can be made of optional material. The rectangles in the fixture should have the same size as the tiles to be cut.



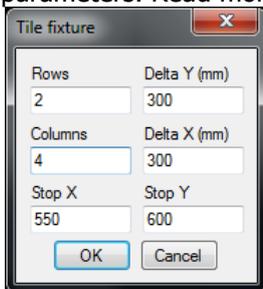
Picture 255

When ready to cut tiles, place them in the fixture. The machine will cut tile 1,2,3,4,5,6,7 and 8 then the machine will go to stop position (P) and stop. It is now possible to unload and load the next 8 tiles, then start the machine again.



Picture 256

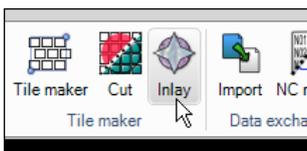
When the postprocessing is started then another dialog box appears that will handle all tile fixture parameters. Read more about the post processing in Section 14.



Picture 257

Input the positions of the tiles on the fixture. When tiles are cut the zero point is located on the lower left corner on tile rectangle number one.

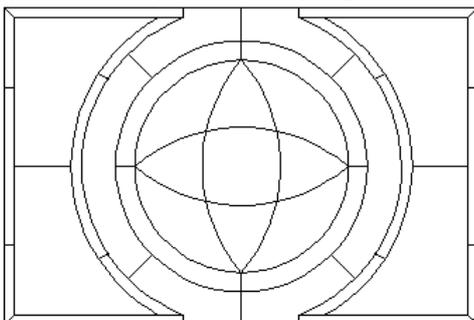
8.3 Inlay



Picture 258

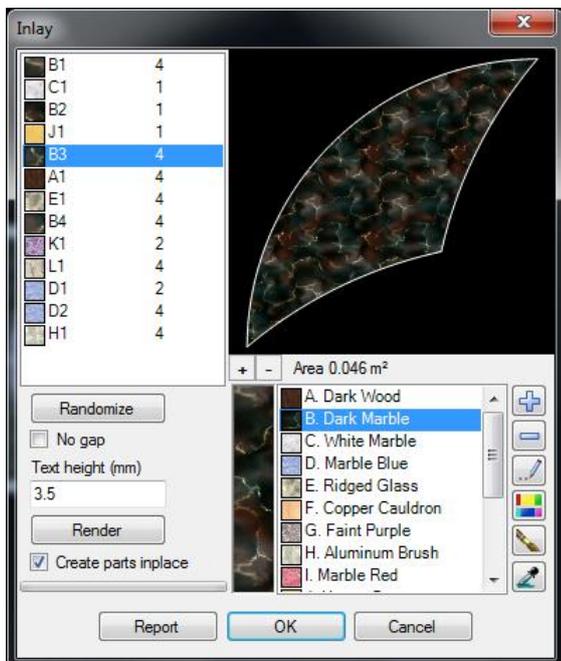
The "Inlay" command is an advanced tool that allows designing of tile floors or other tile decorations by adding textures or colors before extracting the parts.

- Open the sample file "Inlay.dig" located in the Sample directory.



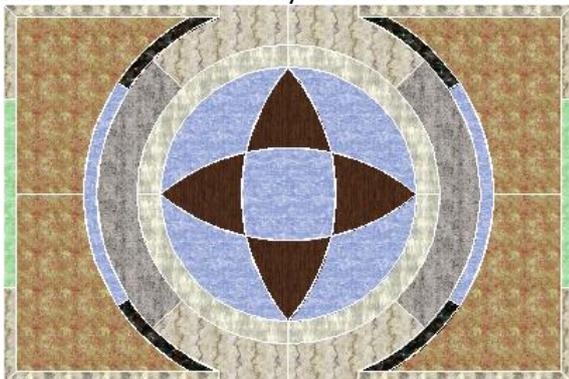
Picture 259

- Start the "Inlay" command.



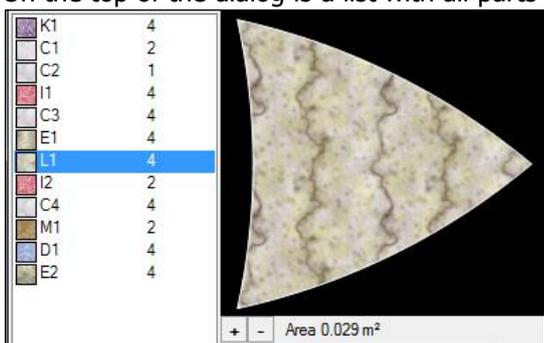
Picture 260

IGEMS will now randomly add textures on different areas.



Picture 261

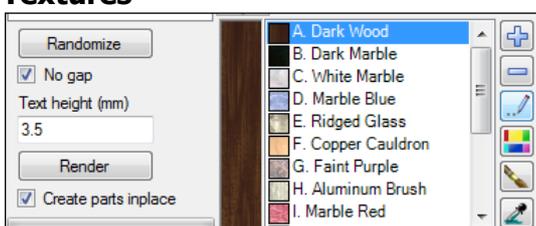
On the top of the dialog is a list with all parts in the Inlay. Selected parts are highlighted on the drawing.



Picture 262

The parts are named as follows: The first character stands for the texture material. Number one indicates the first part in this texture and number 2 the second and so on. The last number is the quantity of that part. Press the "plus" or "minus" signs to change the scale of the texture.

Textures



Picture 263

The "Randomize" button randomly changes the texture on all parts. Select a material on the texture list to preview it on the left.



Picture 264

Load or delete textures in the list with the "plus" and "minus" sign. To add new texture files add to the "IGEMS_R10/Res/Texture" folder.



Picture 265

The "edit" button can be used to change the description of the texture. It will not change the texture itself.



Picture 266

It is also possible to add solid colors to the texture list. Press the button and select the color to add.



Picture 267

This button temporarily closes the dialog box. Hold down the CTRL key to choose different areas on the drawing to have that selected texture. Press the spacebar or Enter to go back to the dialog box.

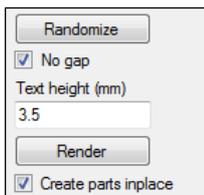
Select a texture and use the button to attach textures to the drawing.



Picture 268

This button temporarily closes the dialog box. Use this feature to select a part and to see the actual texture on the drawing.

8.3.1 Gaps

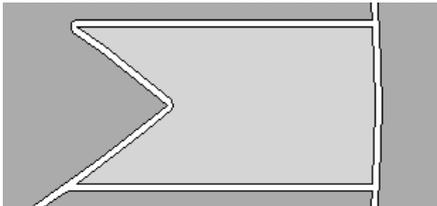


Picture 269

Use the checkbox "No gap" to control cutting with or without gaps.

With gaps

To cut with gaps cut the parts with no tool radius compensation (no kerf). This means that all external measurements will be smaller and all internal measurements larger.

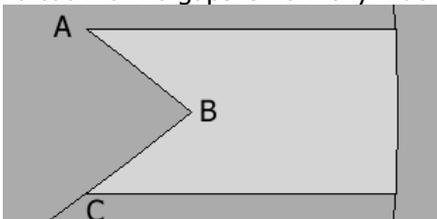


Picture 270

The size of the gap will be the same as the diameter of the jet.

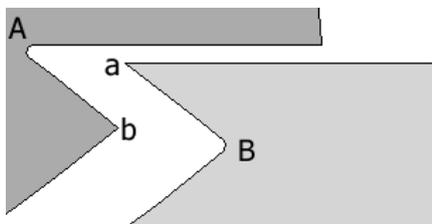
No gaps

To cut with no gaps is normally much more difficult. Cut all parts with tool radius compensation.



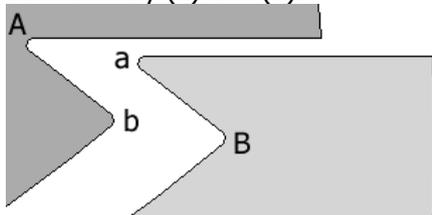
Picture 271

Since the jet cannot make sharp inside corners manual work at positions (A) and (B) in Picture 271 or at (a) and (b) in Picture 272 are required. Without doing this job the parts will not have proper fit.



Picture 272

With "No gaps" activated then all outside corners that cause a fitting problem will be adjusted by IGEMS automatically (a) and (b).

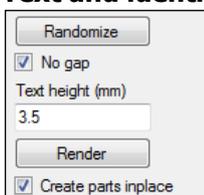


Picture 273

The radius (A) and (a) and the radius (B) and (b) now have the same size and the parts will fit together.

- ? What setting do I use if I want to modify the inside corners manually (A) and (B)?
In that case you should cut with gap and with tool radius compensation (with kerf).

Text and identification



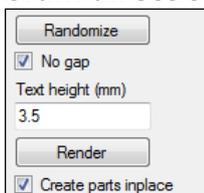
Picture 274

If the text height is larger than 0 then the command will automatically place a text with the name on each part. This can be used as identification of the parts.

Render

The "Render" button creates a more photorealistic image of the layout.

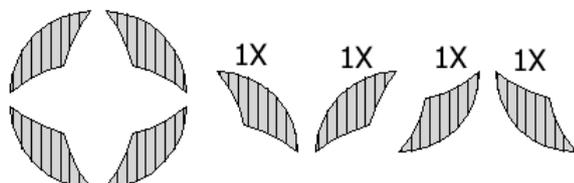
Grain direction



Picture 275

This is controlled by the checkbox "Create parts inplace"

Grain direction

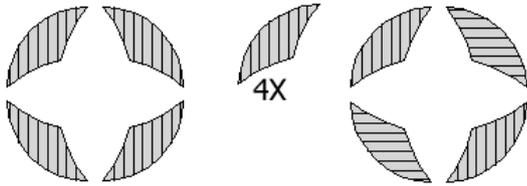


Picture 276

In the above example, Picture 276, the command will create 4 different parts with different rotation. When the parts are mounted the grain direction will be the same for all parts

No grain direction

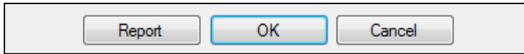
For material with solid colors the grain direction may not be important.



Picture 277

In the above example, Picture 277, the command will create one part with the quantity flag set to 4. When the parts are mounted the grain direction will be in different directions.

8.3.2 Report



Picture 278

Click the "Report" button to generate a complete report of the inlay.

8.3.3 Generate inlay parts

Press "OK" to generate all parts from the drawing.



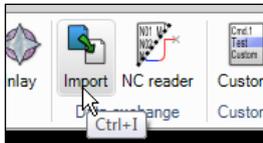
Picture 279

The parts are lined up by material. The final step is to use the normal "Nesting" and "CAM" commands to create the CNC file.

9 Data Exchange

The "Data Exchange" module includes commands that allow the import of extended file formats and analyzing and conversion of existing CNC files. There must be a license for the "Data Exchange" module to use these commands.

9.1 Import



Picture 280

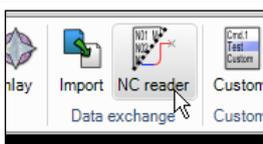
The "Import" command the user to import previously saved drawings and place them in the current drawing. The following file formats can be imported:

Type	Description	IGEMS	Data Exchange
.DIG	IGEMS R10 standard drawing format	Yes	
.ACD	IGEMS R3 to R9 drawing format	Yes	
.DWG	AutoCAD Drawing files	Yes	
.DXF	Drawing Exchange Format	Yes	
.WMF	Windows Meta Files		Required
.PS	Postscript (level 1 and some objects in level 2)		Required
.EPS	Encapsulated Post Script (level 1 and some objects in level 2)		Required
.GEN	Tribon Generic Files.		Required
.GEO	TOPS CAD/CAM system from Trumpf.		Required
.IGS	IGES Files (a subset of 2D object)		Required
.MEC	Files from Lantek		Required
.ORD	CNC-Files from Flow		Required
.TAG	Files from Taglio		Required

Note!

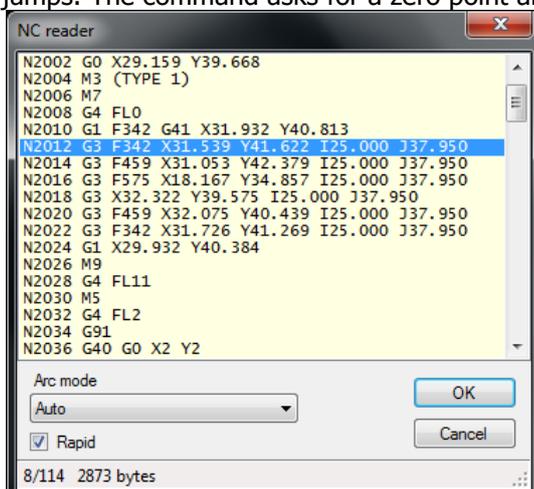
The imported parts can include a file name just as when using the import menu described on page 15.

9.2 NC reader



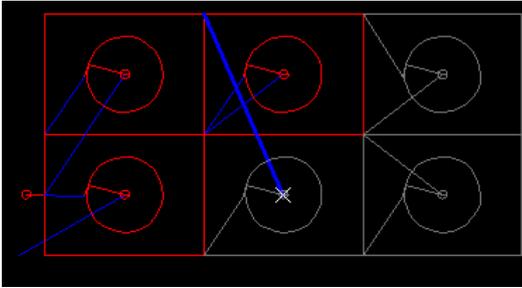
Picture 281

This command imports CNC files and creates lines and arcs according to coordinates in the CNC file. Only the most common format is supported in this command and it will not understand sub-program calls or jumps. The command asks for a zero point and then the name of the CNC-file.



Picture 282

The whole geometry of the file is displayed in gray on the drawing area.



Picture 283

Select a line in the dialog box and the gray object will be replaced by a red or blue object up to that position. The actual movement is highlighted. Click on an object in the drawing area and this line is selected in the dialog box. The NC-reader is a perfect tool to analyze an existing CNC-file.

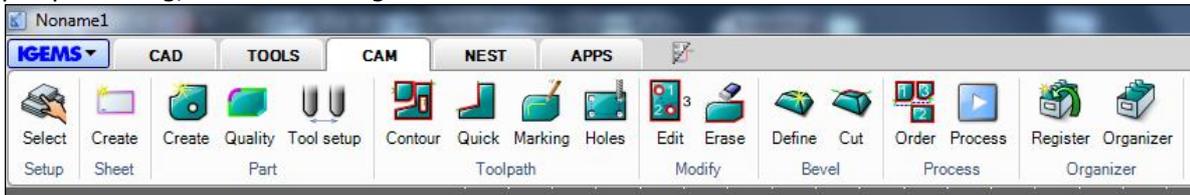


Picture 284

If the geometry looks strange, change the settings of the arc mode. The checkbox "Rapid" determines if the blue rapid lines are displayed or not. The bottom line is a summary of the selected line / total number of lines and the file size.

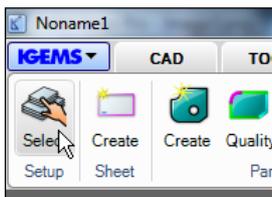
10 CAM

This chapter contains information about the most common "CAM" commands. It includes all the tools required to setup the machine and material database, prepare CNC files for the cutting process, postprocessing, calculate cutting times and much more.



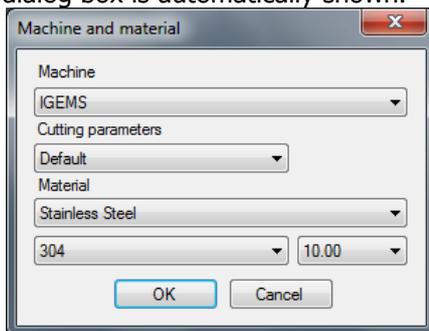
Picture 285

10.1 Select setup



Picture 286

When any "CAM" command is started that needs information from the "Machine" or the "Material" then this dialog box is automatically shown.



Picture 287

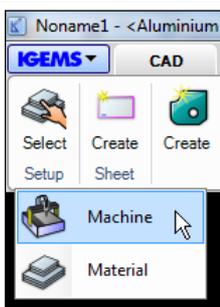
To select this information at the beginning of the process gives IGEMS possibilities to suggest the best default values. Select another machine or material at any time.



Picture 288

The actual filename and selections are shown at IGEMS title row at the top of the program and at the Drawing File Selector at the lower right corner of the screen.

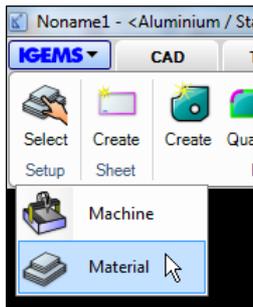
10.2 Machine



Picture 289

This command is used to set up the actual Machine equipment. The command is covered completely in Section 16.

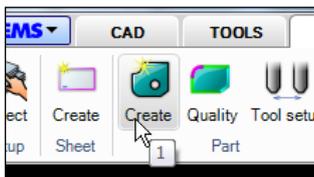
10.3 Material



Picture 290

This command handles all cutting parameters that are related to the material. Detailed information about this is found in Section 12.

10.4 Part create

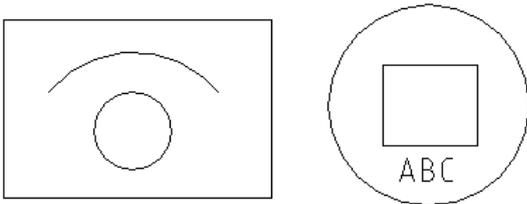


Picture 291

Shortcut "1"

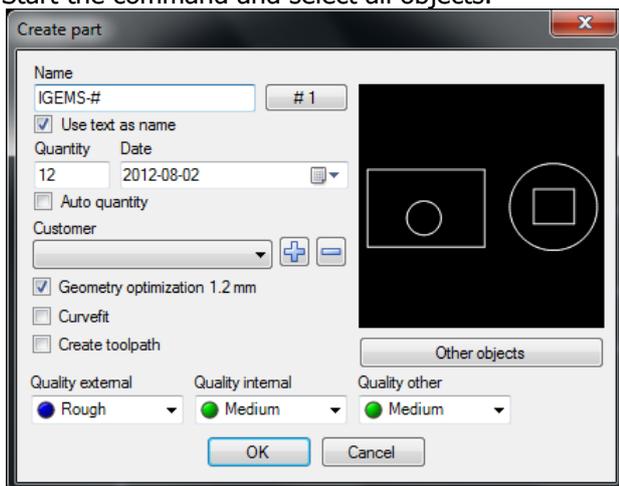
The "Part create" command creates parts from existing geometry on the screen. The parts are used to extract toolpath and are also used by the "Nesting" commands.

- Make geometries like below. The rectangle should be about 300-400 mm in X. Add the text "ABC" inside the second part.



Picture 292

- Start the command and select all objects.



Picture 293

10.4.1 Name

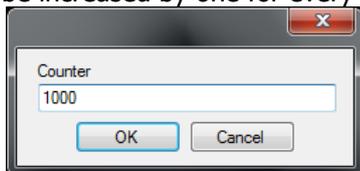


Picture 294

The name of the part is used by the postprocessor and the organizer and doesn't have to be unique. There are two ways of giving parts automatic names:

By increasing number

If the name includes a "#" sign then this sign will be replaced by the counter on the button. The counter will be increased by one for every new part created.



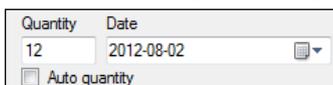
Picture 295

To change counter series press the button and set a new start value. If the name is "IGEMS-#" and the counter is 1000 then the parts will be named: IGEMS-1000, IGEMS-1001, IGEMS-1002 and so on.

By text information inside the geometry

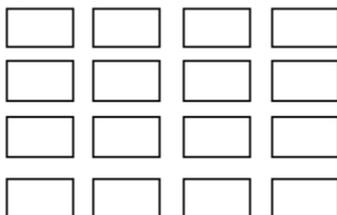
If the checkbox "Use text as name" is activated then the actual text will be the part name. See the "Import" command on page to read more about how to generate names from imported files.

10.4.2 Quantity



Picture 296

Enter the number of parts to produce. The information is used by the "Nesting" and the "Organizer". When importing a file with several identical geometries there are two choices:

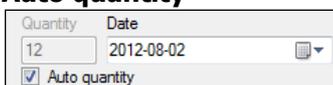


Picture 297

By quantity

Set the quantity to 1 and generate 16 identical parts with different names.

Auto quantity



Picture 298

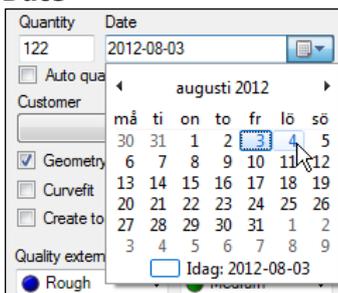
By activating the checkbox the command will count all identical parts and handle the quantity information automatically. This means that only one part with a quantity of 16 will be created when using this option. All other identical parts are deleted.

- ? Must the part be oriented in the same angle?
No, the part can be rotated in optional directions.

10.4.3 Production information

The production information is saved on the part. Change production information using the "Part properties" command. The information can be used by the "Organizer", "Applications" and the "Postprocessor".

Date



Picture 299

The date information is used for production date or delivery date. The information is useful if using the "Organizer" module.

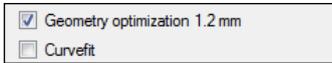
Customer



Picture 300

Select an existing customer name in the list. Use the "plus" and "minus" button to add new and delete existing names.

10.4.4 Geometry



Picture 301

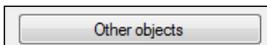
Geometry optimization <value>

It is recommended that the "Geometry optimization" is always activated. The optimization guarantees that the toolpath created on the part is possible to cut with a tool diameter smaller than the optimization value. The value can be changed in the geometry section in the machine setting. See Max tool diameter on page 182.

Curvefit

If this checkbox is activated then "Create" is performing an automatic "Curvefit" operation on the selected geometry. More information on the "Curvefit" command is on page 61.

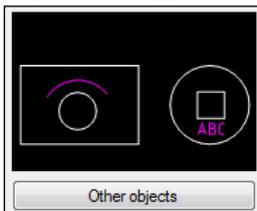
10.4.5 Other objects



Picture 302

Click the "Other objects" button to select optional objects to include in the part.

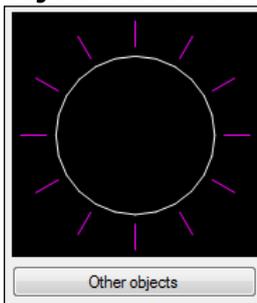
Press the button and select the arc and the text.



Picture 303

The selected objects are now included in the parts and can be used by other functions in the "2D-CAM" module.

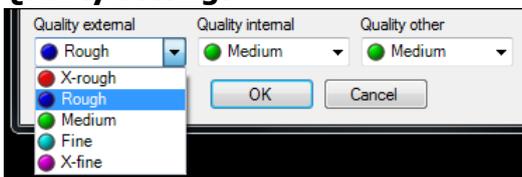
Object located outside external geometry



Picture 304

To add "Other objects" that are located outside an external geometry, like in Picture 304 above, then only one part can be created at a time.

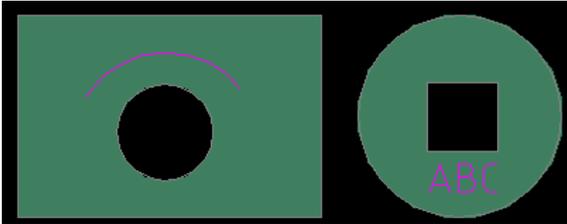
Quality settings



Picture 305

The default attaches different cutting quality to external, internal and other objects.

Press "OK"



Picture 306

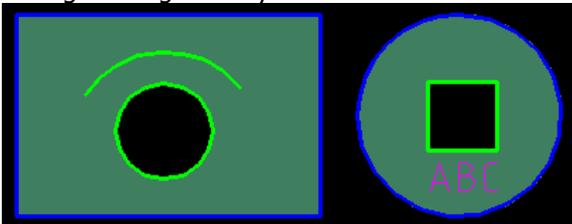
The gray color indicates that the parts are correctly described. Other commands will know what is inside and outside.

10.5 Quality



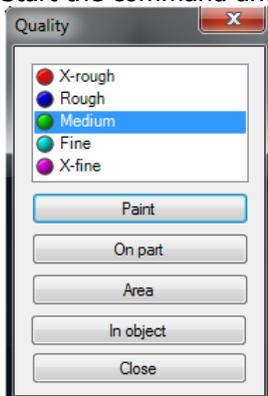
Picture 307

This command allows the user to change cutting quality on a part. When the command is started, the quality setting on all geometry is shown.



Picture 308

Start the command and select the rectangular part.

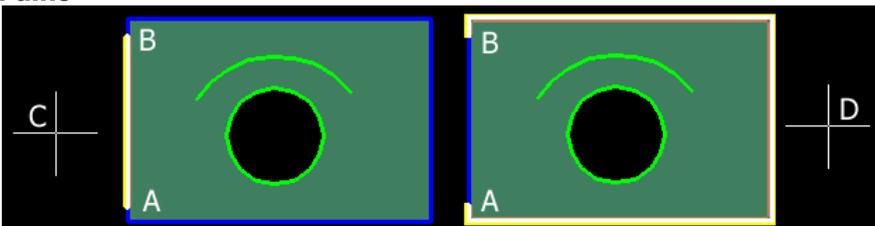


Picture 309

There are 5 different quality levels for abrasive waterjet cutting. The difference in cutting speed can be up to 500% or more between the roughest and the finest quality. Therefore it is important to never use a finer quality than necessary. The "X-rough" cut should only be used for separation of material and the "X-fine" is only used when extreme finish is required.

Select a quality.

Paint

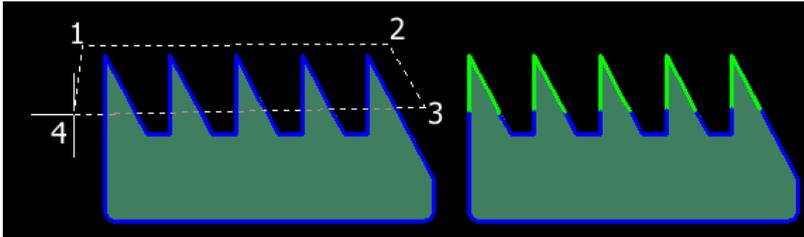


Picture 310

Use the "Paint" option to add the quality on optional positions of the part. Press the "Paint" button and set the start point (A) and end point (B) for the paint. The third point (C) or (D) controls what portion of the geometry should be painted.

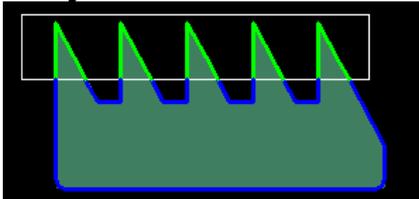
On part

When using this option the whole part gets the selected quality.

Area

Picture 311

The "Area" option defines a polygon by clicking on optional points. Everything inside the polygon will have the selected quality.

In object

Picture 312

This is a similar option as the "Area". Use a closed object to describe the area instead of defining a polygon.

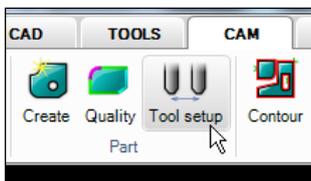
10.5.1 Relations between parts

When there are several copies of the same part then it is sufficient to modify the quality on just one part. All sister parts will automatically get the same quality.

References

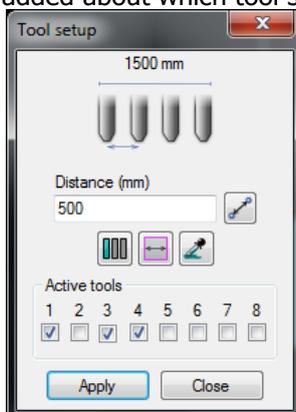
The "Create" command sets the base quality. See page 76.

Modify the cutting parameters for different qualities in the material. See page 131.

10.6 Tool setup

Picture 313

Use this command for Machines equipped with more than one cutting tool when information needs to be added about which tool should be used and the distance between each tool.



Picture 314

10.6.1 Active tool

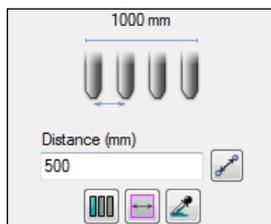
Use the checkboxes to control what tools should be active. Above tool number 1, 3 and 4 is activated. This should result in following:



Picture 315

Tool number 1 is always the master tool.

10.6.2 Distance



Picture 316

The distance should be given as distance between each tool. The total distance between the two outermost tools are given as information in the top of the dialog.

Pick



Picture 317

Click on this button to pick the distance between each tool.

Minimize



Picture 318

This option puts the tools as close together as possible. Select the part and enter the distance between the parts. The distance will be calculated by the size information on the part.

Equal on sheet



Picture 319

Click on the sheet. The distance will be calculated so all tools get equal distance on the sheet.

Pipette

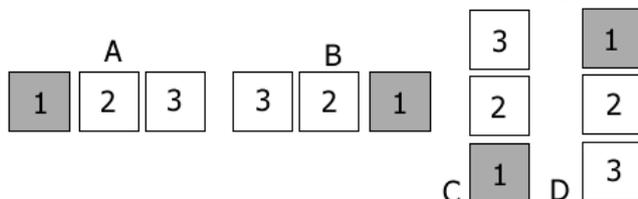


Picture 320

Select a part. All settings in the dialog will be the same as the selected part.

10.6.3 Layout

IGEMS supports different layouts on the tool alignments.



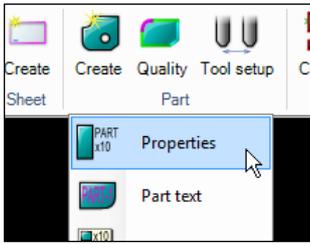
Picture 321

The number of tools and layout is set in the "Machine" setup. All layouts are compatible on all machines. This means that a completed job done for a machine with three tools can also be run on a machine with one tool or on a machine with a different layout.

References

Configure machine layout on the "Machine" setup. See page 169.
The "Auto nesting" command makes an own tool configuration.

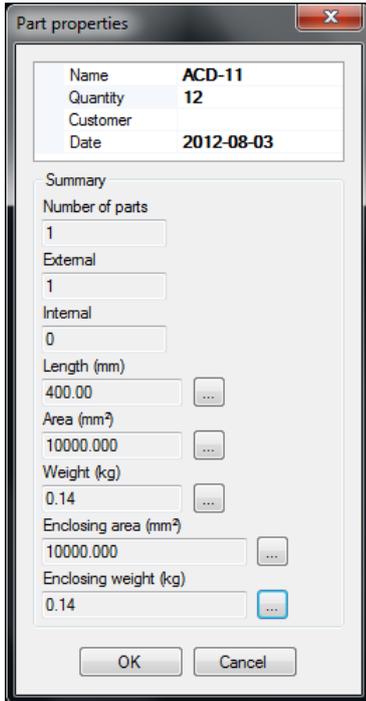
10.7 Properties



Picture 322

To change the property of the parts afterwards use the "Properties" command.

Start the command and select one or several parts.



Picture 323

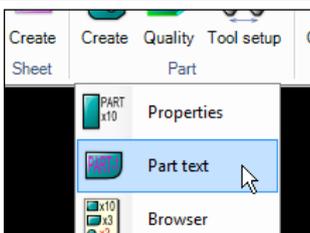
The command shows a lot of information for the part; however it is only possible to change the information below:



Picture 324

Multi selection of parts is possible with the properties command. If multiple parts are chosen at once, the quantity will not be able to be changed.

10.8 Part text



Picture 325

The "Part text" command is used to display the name of the part graphically.

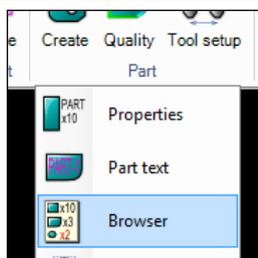
Start the command and select parts and enter a text height.



Picture 326

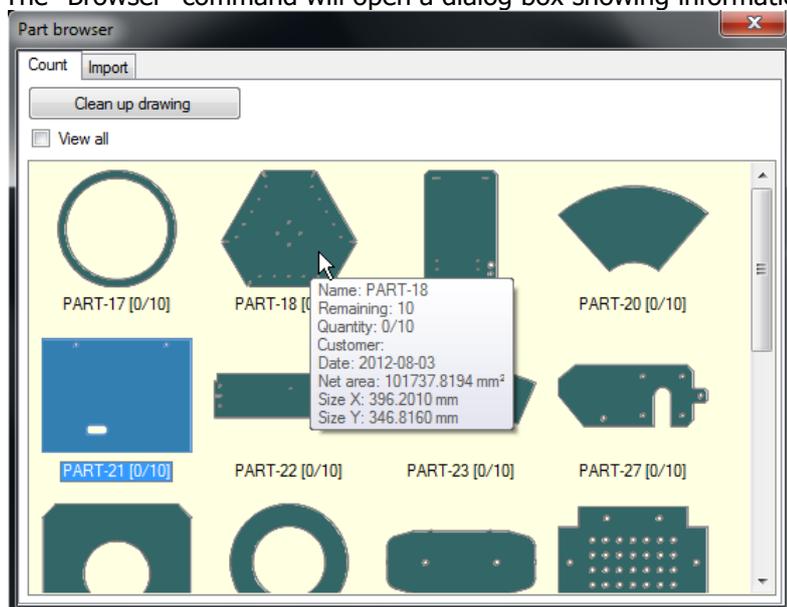
The name or an index number can be written on the parts.

10.9 Browser



Picture 327

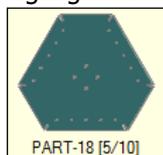
The "Browser" command will open a dialog box showing information about all existing parts on the drawing.



Picture 328

10.9.1 Information

Hold the mouse pointer over a part to see information about the part in a hint window. The selected part is highlighted on the screen.



Picture 329

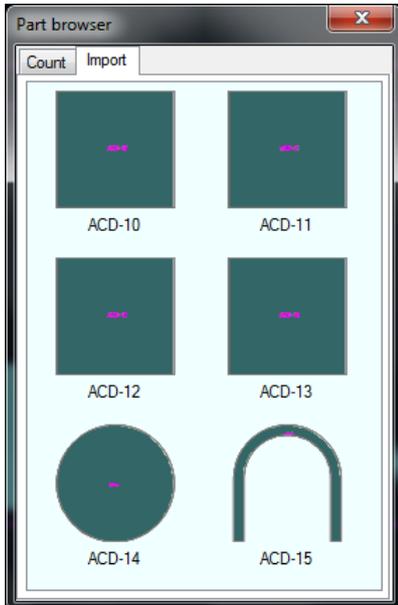
In Picture 329, the name of this part is "PART-18". The quantity is set to 10 and 5 of these parts are located on sheets.

10.9.2 Nesting

Double click on a part to use the "Quick nest" command.

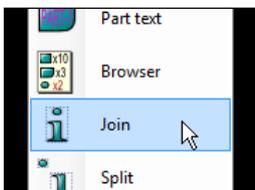
10.9.3 Import

If there are multiple drawings using the same material in the IGEMS session, then they can be imported to active drawing.



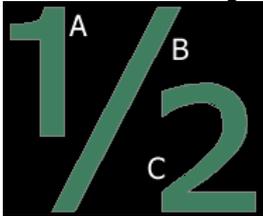
Picture 330

10.10 Join



Picture 331

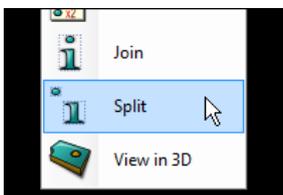
This command merges several parts into one part.



Picture 332

The command will ask for the master part (A) and then for parts that should be joined to this part (B) and (C). The result will be one part with three external geometries with name (A).

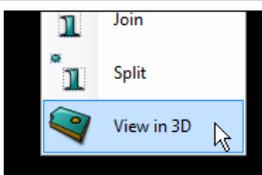
10.11 Split



Picture 333

"Split" is used to separate a part that has previously been joined together.

10.12 View in 3D

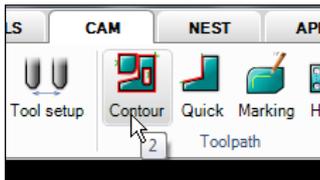


Picture 334

This command can view the part in 3D. The viewer has no other purpose then to visually display the 2D part in 3D. The different colors show the cutting qualities.

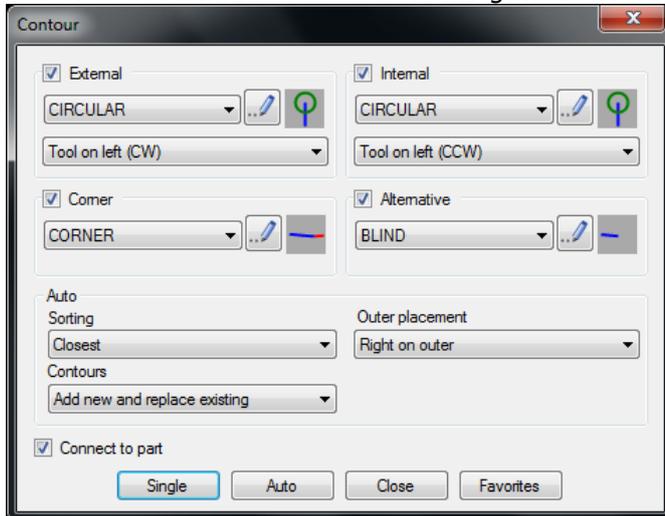
10.13 Contour

Shortcut "2"



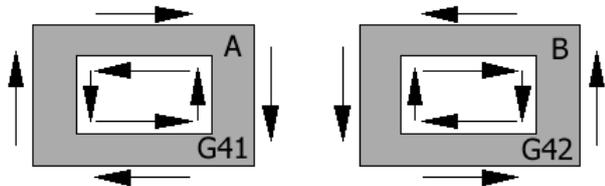
Picture 335

This command can be used on all closed geometries that should be cut all the way around.



Picture 336

10.13.1 Tool compensation (kerf)

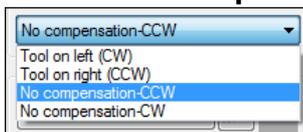


Picture 337

Picture 337 above shows Alternative (A) with the Tool on left option. This will move the jet on the left side of the geometry when looking in the direction of the movement. This is the same as clockwise (CW) for external geometry and counter clockwise (CCW) for internal geometries. Alternative B shows the Tool on right option. This option makes the movement in opposite directions.

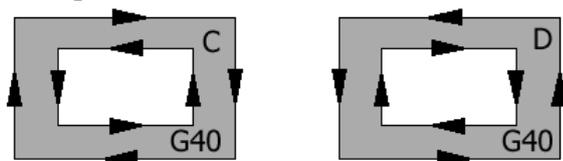
In a waterjet machine the result will be the same on G41 and G42 but it is more common to use the "G41" option.

Without tool compensation



Picture 338

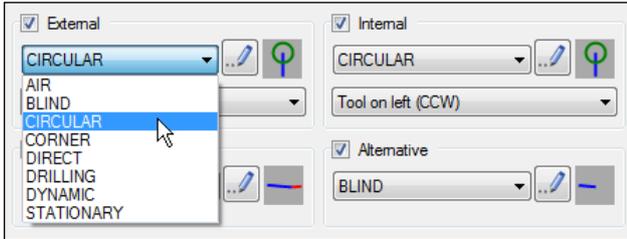
When cutting without tool compensation: the external geometry will be too small and the internal geometry too large.



Picture 339

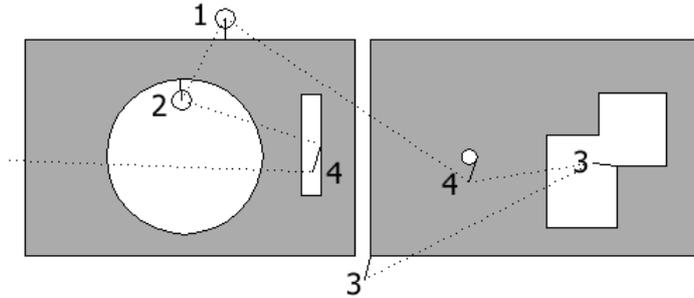
10.13.2 Lead selection

It is possible to predefine leads and to give them optional names, creating a library of leads that can be selected in the lead lists.



Picture 340

IGEMS can be set to use different leads for external and internal geometries. It is also possible to use a special lead if the lead is placed in a sharp corner. If these leads cannot be used, then an alternative lead can be used.



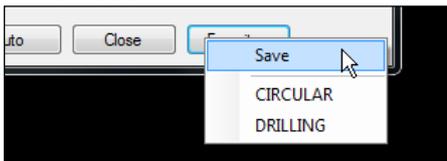
Picture 341

Lead 1 is an external lead, lead 2 is an internal lead, lead 3 is a corner lead and lead 4 is an alternative lead.

? Lead 4 seems to damage the material?

No, this is a blind lead and is not used for cutting. Read more in "Lead definitions" on page 90.

Favorites



Picture 342

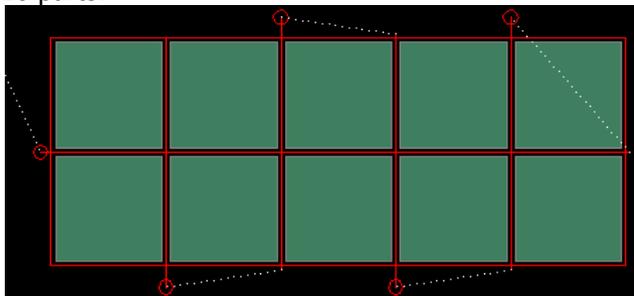
Use the "Favorites" button to save and restore sets of lead definitions.

10.13.3 Connect to part



Picture 343

This option is also available in other commands. When the toolpath is connected then it is bound to the part. When the part is moved, then the toolpath should be moved. A toolpath that handles more than one part cannot be connected, for example "Common outline". In the example below the same toolpaths are used for 10 parts.



Picture 344

By deactivating this checkbox a toolpath is created that is disconnected from the parts.

? What can a disconnected toolpath be used for?

You can then create several toolpath on the same part in order to assign different tools on the same parts.

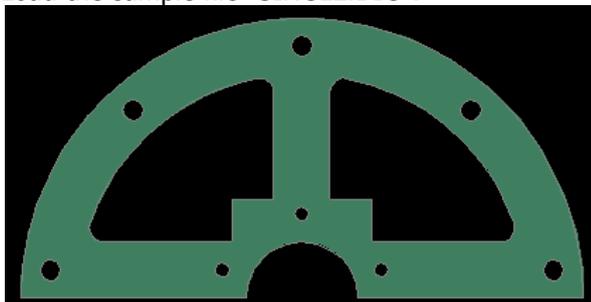


Picture 345

Only the internal cut is in the example above. The contours on the left side (A) are connected to the part, and the contours on the right side (B) are not. It is now possible to assign two different tools to these two geometries.

10.13.4 Single mode

🔗 Load the sample file "SINGLE.DIG".



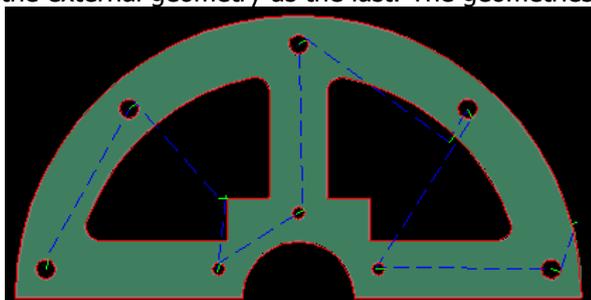
Picture 346

Use the "Single mode" to manually attach leads to geometries.



Picture 347

🔗 Attach the leads on desired positions by clicking the "Single" button. Start with internal geometries and take the external geometry as the last. The geometries on the part will be cut in the order inputted.



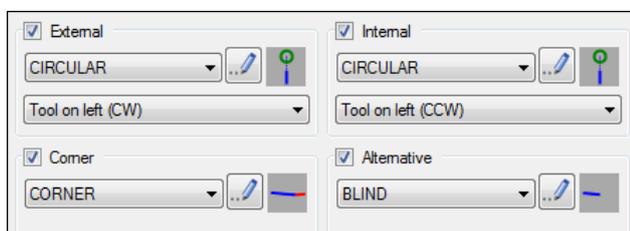
Picture 348

Change position of leads

Change the position of the lead by re-clicking on the same contour.

? Can I change the cutting order?
Not in the "Single" mode. This is done in "Auto" mode and using "Edit" toolpath.

10.13.5 Auto mode

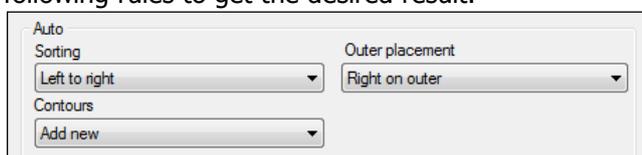


Picture 349

Activate or deactivate the checkboxes "External", "Internal", "Corner" and "Alternative" to control which contours the leads are attached to.

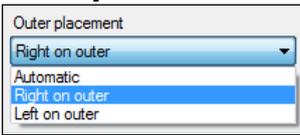
Rules

With the "Auto mode" leads can be set automatically on one or many parts at the same time. Use the following rules to get the desired result.



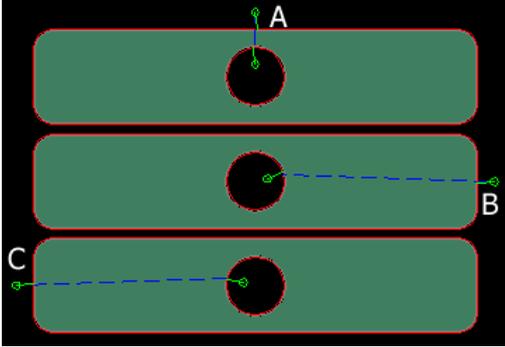
Picture 350

Outer placement



Picture 351

This setting controls where the lead should be placed on external geometry.

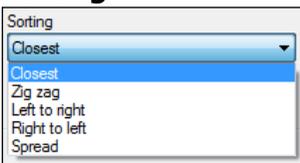


Picture 352

The "Automatic" option (A) will try to put the external lead as close as possible to the last internal lead. If the external geometry has one or several sharp outside corners then the lead will be placed on the closest corner.

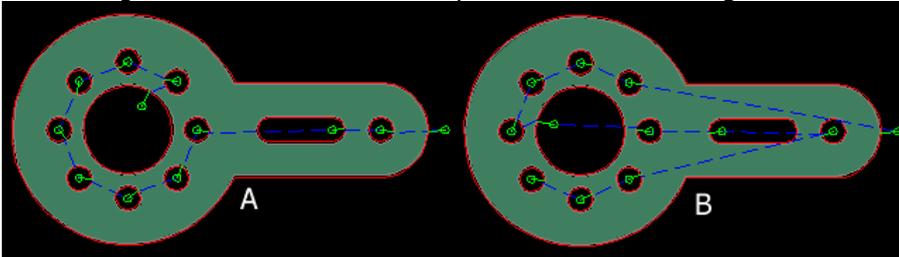
The "Right on outer" (B) will try to put the external lead on the farthest right position as possible. The "Left on outer" option will put the external lead on the farthest left side of the part (C).

Sorting



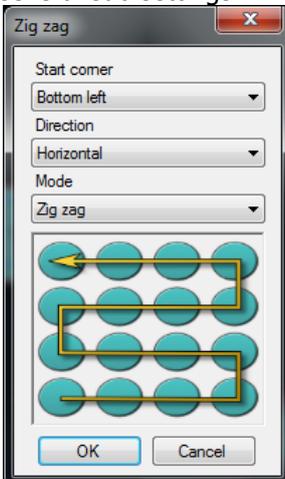
Picture 353

This setting controls where the lead is placed on the internal geometries.



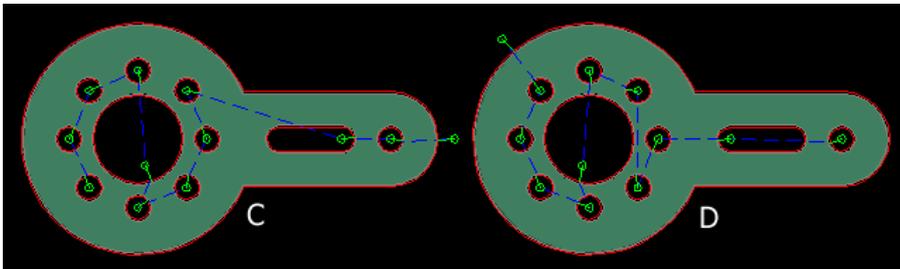
Picture 354

Alternative (A) is the option "Closest". The algorithm starts from the external position and looks for the closest internal geometry until all geometries are found. Alternative (B) is the "Zig zag" option and has several sub settings.



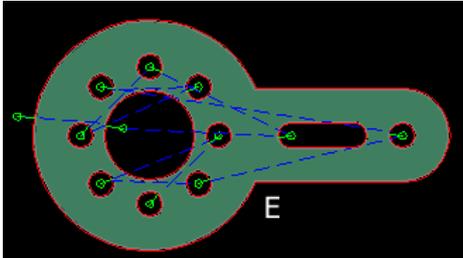
Picture 355

The user selects "Start corner", "Direction" and "Mode". Picture 356 shows the different options.



Picture 356

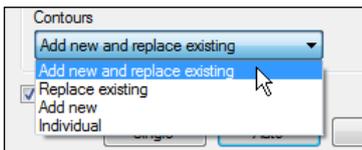
The (C) shows the "Right to left" option and (D) the "Left to right" option with outer placement set to "automatic".



Picture 357

The (E) shows the "Spread" options that can be used on thermal cutting, so the heat is spread out as much as possible.

10.13.6 Contours



Picture 358

This settings controls which contours the leads are attached to.

Add new and replace existing

This method adds new and replaces all existing leads on the selected parts. Do not use this option if some leads are placed with other commands that shouldn't change.

Replace existing

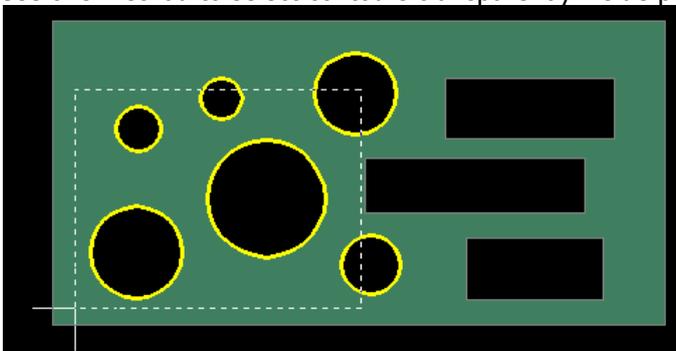
In parts with no leads, this choice will not produce anything since it replaces existing leads with the defined leads. This method uses the same lead positions as the already existing leads.

Add new

This method adds leads on contours that have no lead. Use this method when leads were placed by other commands.

Individual

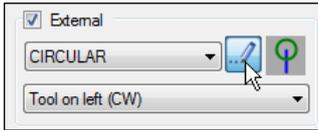
Use this method to select contours transparently inside parts.



Picture 359

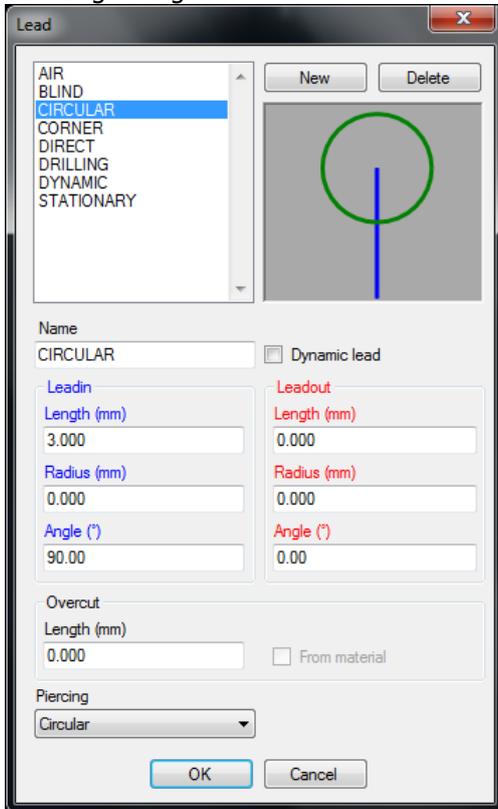
This method adds new and replaces existing leads.

10.13.7 Lead definitions



Picture 360

Click on one of the "edit" buttons to change the settings of the piercing methods and lead geometries. The following dialog box is shown:



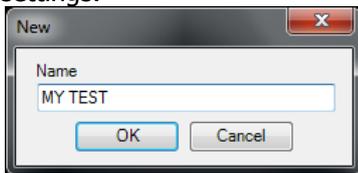
Picture 361

Create new or delete existing definitions



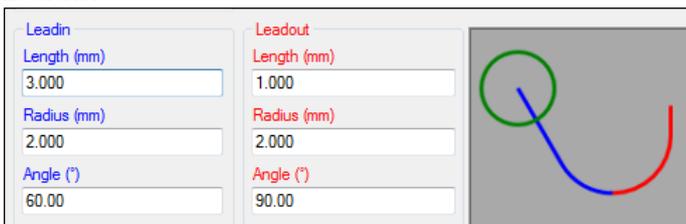
Picture 362

Click on these buttons to manage the list of lead definitions. Delete leads that are never used to make as short a list of leads as short as possible. Press the "New" button to create a new definition based on current settings.



Picture 363

Leadin



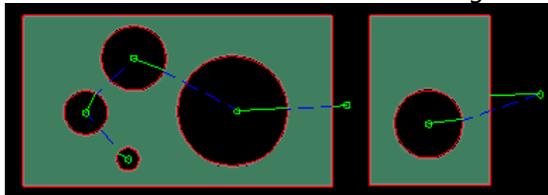
The dynamic picture located in the dialog box shows the result of different parameters. When using tool radius compensation (kerf with G41 or G42) in the machine it is important that the linear part of the leadin is longer than the tool radius.

Dynamic leads

Name CIRCULAR <input checked="" type="checkbox"/> Dynamic lead	
Leadin Length (mm) [8.000,20.000]	Leadout Length (mm) 0.000

Picture 364

When the Dynamic lead is activated the linear length of the leadin is variable. In Picture 364 the length is between 8 to 20 millimeters. This setting can be used with as long a lead as possible.



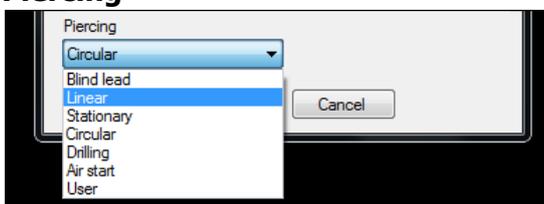
Picture 365

In Picture 365 the lead has different lengths depending on where it is placed.

? In which case is dynamic leadin usable?

When cutting in material that could break during the cutting. This method moves the piercing point out from the geometry. It is mostly used in AWJ cutting of stone or glass.

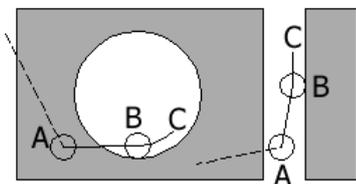
Piercing



Picture 366

A piercing type must be attached to every lead definition. The piercing controls how the jet cuts through the material. The following methods are available:

Blind lead



Picture 367

With this option the Jet On is delayed until after the leadin position is reached. This method should never have any lead geometry with a radius. Movements are as follows: Rapid to point (A); Activate the kerf (G41/G42) with a linear movement to point (B); At this point the jet is opened and without any delays the movement continues to cut the geometry; After some distance (C) the jet cuts through the material.

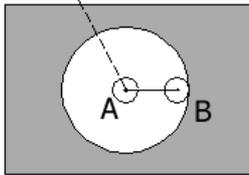
In AWJ cutting this piercing method can be used on more or less all material. The benefit is that it does not require any leadin that can damage parts and material.

Length (mm) 4.000
Radius (mm) 0.000
Angle (°) 5.00

Picture 368

Never use a radius on this kind of piercing. The angle should be between 0 and 5 degrees. There is no benefit of this piercing type if an angle around 90 is used. It is important to use an overcut (see page 92) that is longer than the distance between (B) and (C).

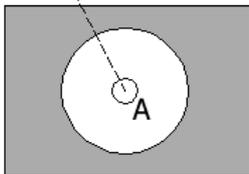
Linear piercing



Picture 369

With this method the jet is opened at point (A) and then continues to point (B) without any delay. If the distance between point (A) and (B) is so short that the jet cannot cut through the material then the jet goes back and forth between (A) and (B) as many times as needed. The kerf (G41/G42) is activated with the last movement between (A) to (B). The linear piercing is the fastest way to cut through the material with a lead. The length of the linear piercing can be set individually on each material and thickness. Read more about the linear piercing on the material settings on page 129.

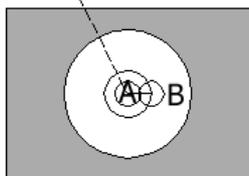
Stationary piercing



Picture 370

The jet goes to point (A) with rapid transport. At this point the jet is open and remains there for a time period. Since this method takes the longest time to cut through the material it is recommended to not use it. It is still supported for compatibility reasons. The stationary piercing time can be set on each material (see page 128 for more information).

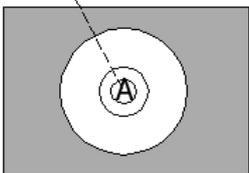
Circular piercing



Picture 371

This is not the fastest method but probably the most common. The jet goes to point (A) with rapid transport. At this point the jet is opened and then it goes to point (B). The jet now makes circular movements around point (A) until the jet has cut through the material. This is an excellent method to use on thick material. All parameters like the radius, speed of the movement and time can be set individually for each material. See page 128 for more information.

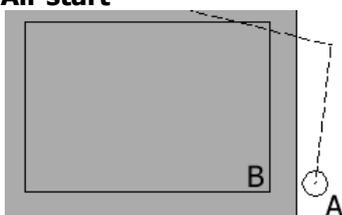
Drilling



Picture 372

Since there can be problems using traditional piercing in laminated material use this method to perform the piercing with a drill. If the machine has no drilling unit then IGEMS uses circular piercing. Set drilling parameters on the "material setup" (see page 124) and control the drilling sequences in the "Strategy" command (see page 162).

Air start



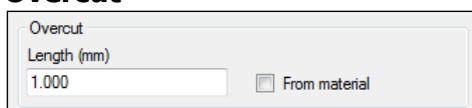
Picture 373

This method is used when a piercing is not needed. For example: Starting outside a sheet or in an existing hole. This method is automatically used for cutting when the start hole has been drilled.

User piercing

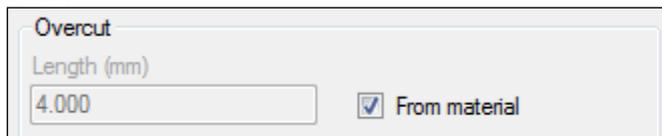
This method is specified in the postprocessor for customization of a special process. Do not use this method if the postprocessor is not developed for this method.

Overcut



Picture 374

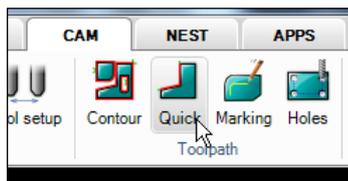
A zero value means that the cut will complete the contour. A positive value will extend the cut to cover more than a full lap. If a negative value is entered, the toolpath will stop before a full lap is completed and create a tab.



Picture 375

Activate the checkbox "From material" when using the piercing type "Blind lead". This will use the overcut value from the material (see page 129 for more information).

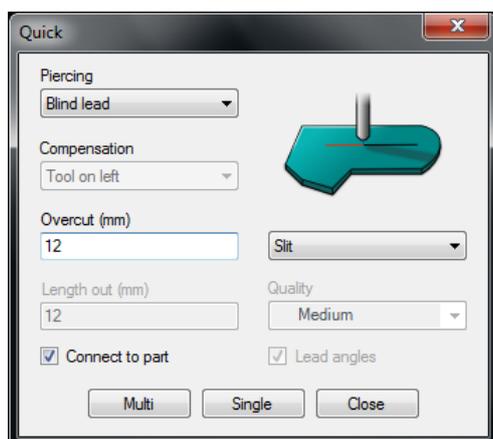
10.14 Quick



Picture 376

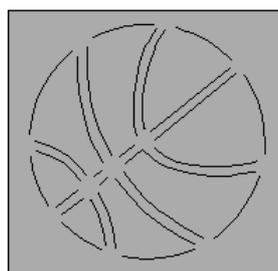
The "Quick" command can be used on all kinds of geometries but is mostly used when it is necessary to only cut contours partially. The command can be used to cut slits or contours. Slit geometry can be created by the "Other object" button in the "Create part" command (see page 78).

10.14.1 Slit



Picture 377

The "Slit" option can be used to cut open or closed objects. Since the geometry itself is used as the leadin it is only possible to use the piercing type "Blind lead". Read more on this on page 91.

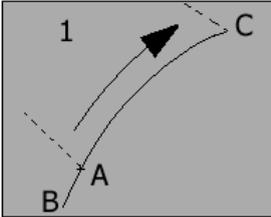


Picture 378

Picture 378 is a typical example. The arcs should be cut with the "Slit" command and the rectangle with "Normal" command.

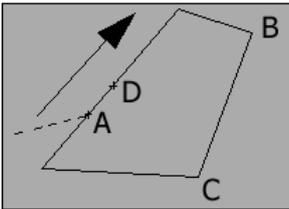
Overcut

The "Slit" option of the "Quick" command can only use the blind lead. The overcut value is the same length it takes the jet to cut through the material.



Picture 379

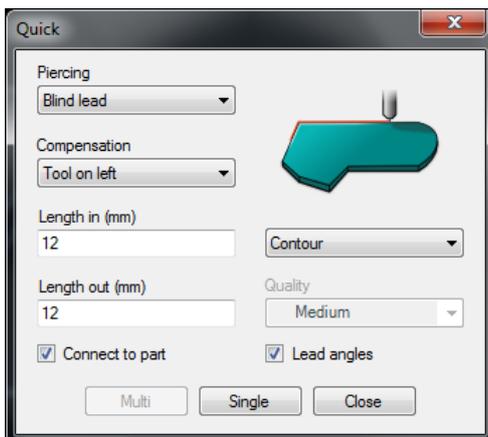
Picture 379 shows an open slit. The cutting starts at point (A) and continues to point (B). At this point the jet has cut through the material and then moves to point (C). The distances that have been cut two times (A) to (B) is the overcut distance.



Picture 380

Picture 380 shows a closed slit. The cutting starts at point (A) and continues all the way around back to (A) and then to (D). The distance between (A) and (D) is the overcut distance.

10.14.2 Contours



Picture 381

Since this option uses a lead in all piercing types can be used. See page 91 for more information about different piercings.

Connect to part



Picture 382

Page 86 has more information about connected and disconnected toolpath. Make disconnected toolpath when creating toolpath from Lines, Arcs and other CAD objects. This object will have the selected quality.

Lead angle

Only use this option together with the "Contour" option. The command also asks for the lead in and lead out angles.

Multi or Single

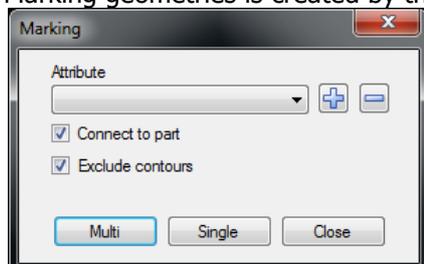
The "Multi" button is used to select multiple objects at the same time. When the "Single" button is used each object must be clicked on. The starting position will then be at the position where the cursor is placed.

10.15 Marking



Picture 383

The "Marking" command makes it possible to cut a mark in the material rather than cutting through. Marking of text and contours is made with abrasive waterjet or pure waterjet or with other marking equipment. Marking geometries is created by the "Other object" button in the "Create part" command (see page 78).



Picture 384

Attribute

The "Attribute" supports extended functionality. The postprocessors must be specially developed to support such equipment.

Connect to part

See page 87 for more on the "Create" command.

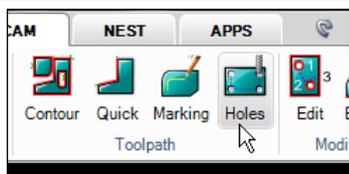
Exclude contours

When this checkbox is activated no geometries that belong to the contours can be selected.

Multi or Single

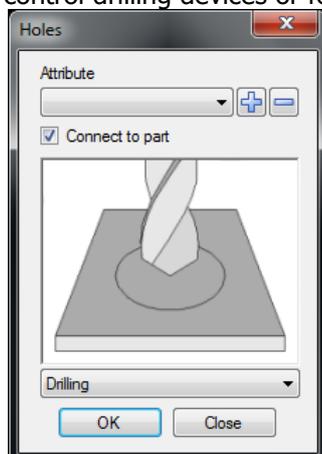
Use the "Multi" button to select multiple objects at the same time. Select the "Single" button and click on each object. The starting position will then be at the position where the cursor is placed.

10.16 Holes



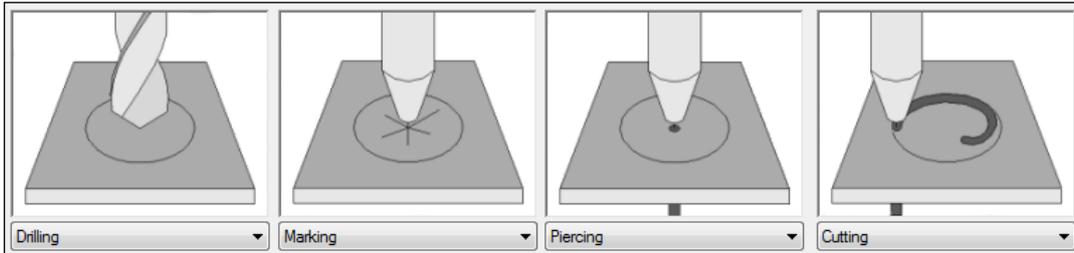
Picture 385

This command is used for different types of cutting processes on point or circular objects. It can be used to control drilling devices or for special waterjet cutting. The following dialog box is shown:



Picture 386

Select the option from the drop down menu at the bottom.



Drilling

This option makes a drilling hole at an optional position.

Marking

This option makes a marking point at an optional position.

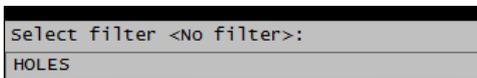
Piercing

This option makes a circular piercing hole.

Cutting

This option makes a circular hole. The holes are machined in a spiral from inside out in order to get a precision hole.

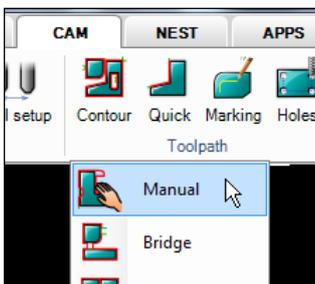
10.16.1 Filter



Picture 387

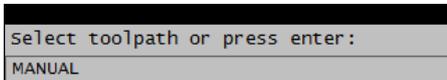
Press "OK" and IGEMS first asks for a filter object. When one of the small circles below is selected as a filter, then a window is made around the whole drawing. Only objects that are identical with the filter object will be used. Press Enter <No filter> and all selected objects will be used.

10.17 Manual



Picture 388

The "Manual" command is extremely flexible, but requires more manual input from the user. The command can mix created toolpath from IGEMS parts and normal CAD geometry such as lines and arcs. All points and information needed is entered the same way as using "CAD" commands. "Ortho" or "Snap" can be used. When the command starts there is the following:



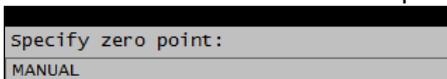
Picture 389

Add movement to a disconnected toolpath

The command only works on disconnected toolpath. To continue to work on an existing toolpath select the disconnected toolpath.

Make a new toolpath

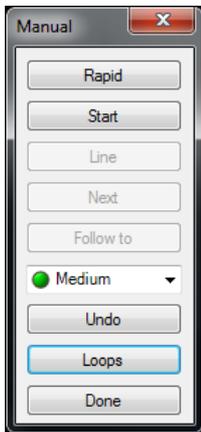
Press "Enter" to create a new toolpath. Then there is an additional question:



Picture 390

Specify a point for the zero point of the new toolpath.

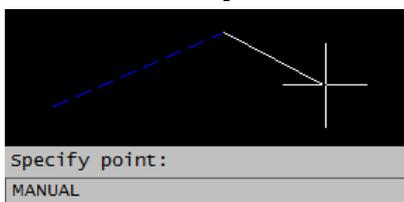
- ? What is this zero point used for?
The selected position normally has no impact on the result but it can be used by the postprocessors. Set the point as close as possible to desired toolpath.



Picture 391

Use this command to create a toolpath step by step. All movements in the machine will be in the same order as the inputted sub-commands.

10.17.1 Rapid

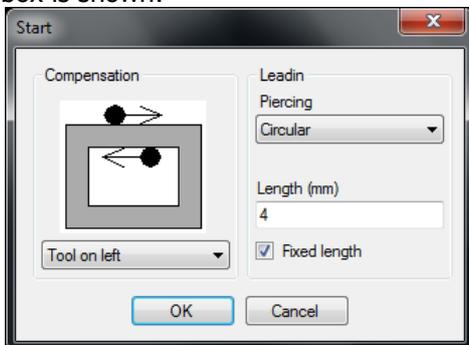


Picture 392

Press this button to add "rapid movements". The jet will always be off when the machine executes this movement. Use the same methods to enter the coordinates as for the "Line" command. Interrupt the "Rapid" command by pressing the spacebar or Enter.

10.17.2 Start

The "Start" command ends the "rapid", activates compensation and turns on the jets. The following dialog box is shown:



Picture 393

Compensation

Select one of the following tool radius compensation to use: "Tool on left", "Tool on right" or "No compensation". Read more about compensation on page 85.

Piercing

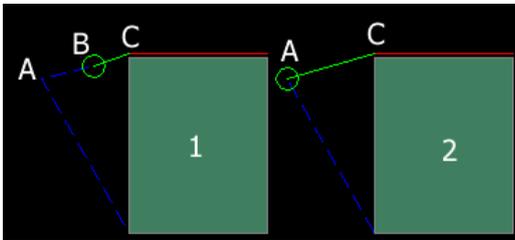
Select one of the 7 different piercing types. Read more about piercing types on page 91.

Fixed leadin length



Picture 394

When "Fixed length" box is checked the lead length is always fixed to this value.



Picture 395

The "fixed length" is used on example (1) in Picture 395 above. When at point (A) and want to start cutting at point (C) then the piercing will be at point (B). The distance from (B) to (C) is the fixed lead length. If this option is not activated (2) then the piercing point will be at the last rapid position.

Start point

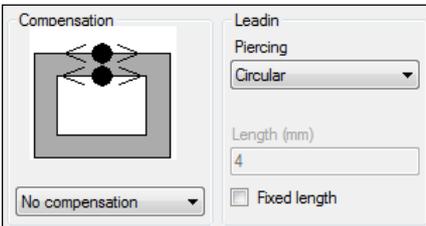
Press "OK" to pick a point on the geometry to follow. To use commands that follow other objects, put the start point on the object. One example to do this is by using "Object snap" commands.



Picture 396

Note! When using tool radius compensation the lead length must be longer than the tool radius or else there may be an error message in the CNC-controller.

Start without leadin



Picture 397

The only way to start "without lead" is when cutting without tool compensation and the "Fixed length" is not activated.

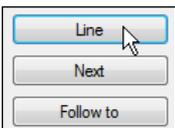


Picture 398

Press enter or the spacebar to accept current position.

Note! If piercing type is "Linear" the postprocessor will use stationary piercing.

10.17.3 Line



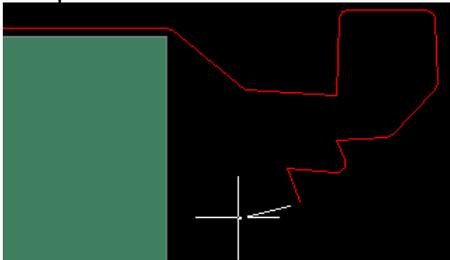
Picture 399

The "Line" option can only be used when the jet is on.



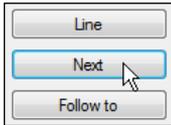
Picture 400

Enter as many lines as needed and move the jet to any position. Interrupt the "Line" command by pressing the spacebar or Enter.



Picture 401

10.17.4 Next



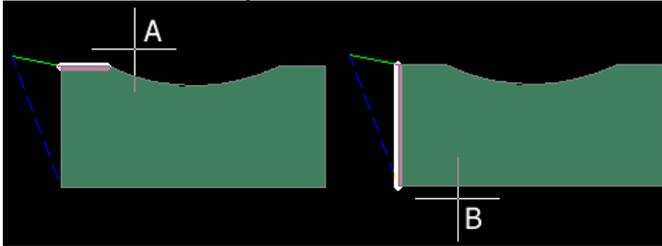
Picture 402

The "Next" option is used when existing objects are located in a chain. For every click on the "Next" button the cursor moves one step forward. To be able to use this function the cursor must be located on an object. When there is more than one object that can be used the path to use must be selected.



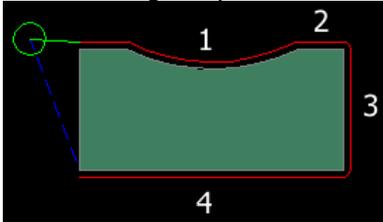
Picture 403

Click to select either path A or B.



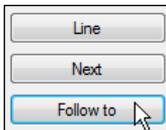
Picture 404

After deciding the path direction then use the "next" button to add one step with every click.



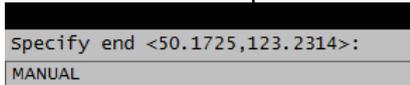
Picture 405

10.17.5 Follow to



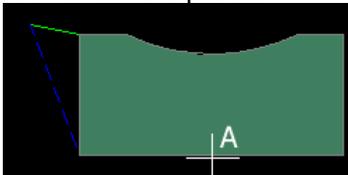
Picture 406

Use the "Follow to" option instead of clicking on the "Next" button.



Picture 407

The coordinates show the current position and if accepted the cursor goes all the way around the part and back to current position.

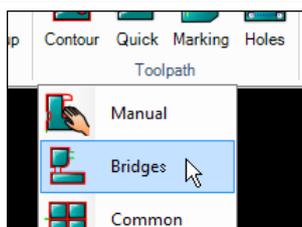


Picture 408

When a position on the contour (A) is selected then all objects up to that position will be added. Use "object snap" when selecting this point.

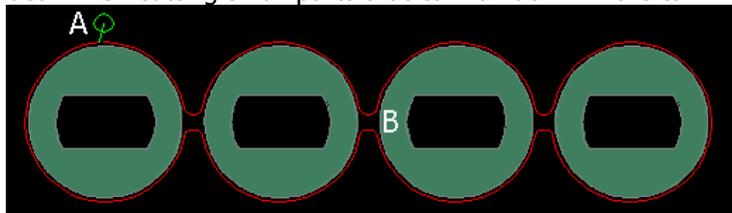
Note! If there is a different solution on how to come to that position, there may be a prompt to select path as described on the "Next" option (see page 99).

10.18 Bridges



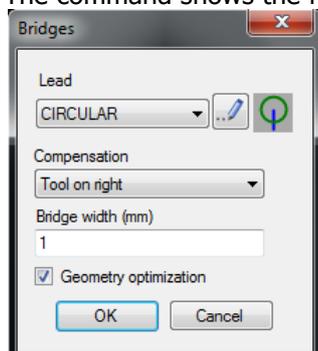
Picture 415

This command creates a disconnected toolpath and connects parts with a bridge. This kind of toolpath is ideal when cutting small parts that can fall down in the tank.



Picture 416

The command shows the following dialog box:



Picture 417

Lead

The "Lead" uses the same settings as the "Contour" command. Read about the lead definitions on page 90.

Compensation

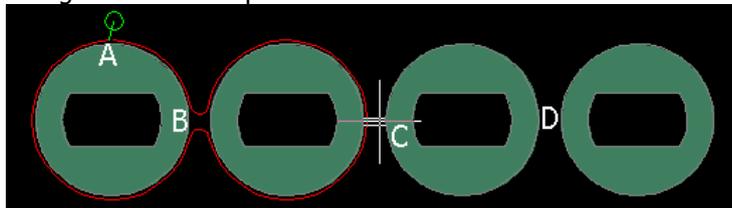
Use normal methods for tool radius compensation (kerf). Read more about the compensation on page 85.

Bridge

This value is the width of the bridge which may look wider on the screen since the toolpath shows the center of the tool.

Geometry optimization

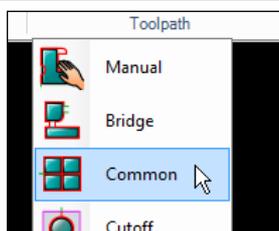
This guarantees the part is able to be cut. Read more about this on page 78.



Picture 418

Click on the position to put the lead (A) then on the position for the bridges (B), (C) and (D).

10.19 Common cutlines



Picture 419

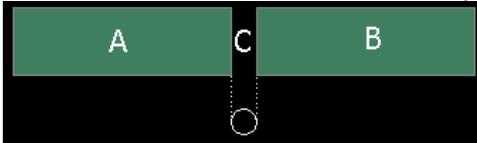
The "Common cutlines" command is used for separating parts with only one cut. This cutting method is much faster than traditional cutting. The following settings are common for all methods:

Tool radius compensation (kerf)

Common cutline generates a toolpath without kerf information. The coordinates in the CNC-file always describe the center position of the jet.

Relation between part distance and tool diameter

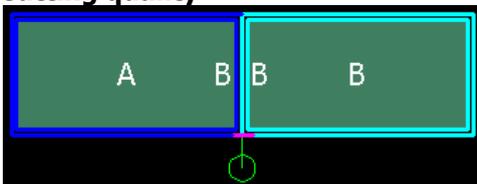
The tool center is located between the parts to be separated.



Picture 420

It is important that the distance between the parts are the same as the dimension of the jet. If the distance between the parts (C) is closer then no toolpath are generated between the parts. If the parts have a larger distance then (A) and (B) will be cut separately and the (C) portions will be cut two times. The tolerances for the distance are approximately plus/minus 0.01 to 0.05 millimeters.

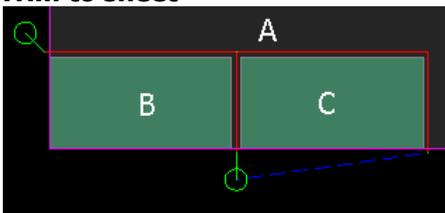
Cutting quality



Picture 421

The cutting quality is inherited from the parts. When cutting between two parts that have different cutting quality the toolpath which is common for the part will have the best quality.

Trim to sheet



Picture 422

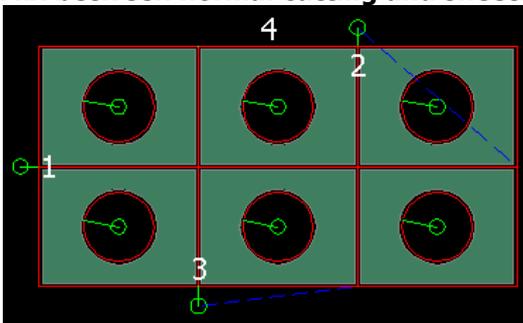
If the tolerance of a part is not critical, possibly save time by using the edge of the sheet as the edge of the part.



Picture 423

To use this, the parts must be nested close to the edge of the sheet. Check the "Trim to sheet" box.

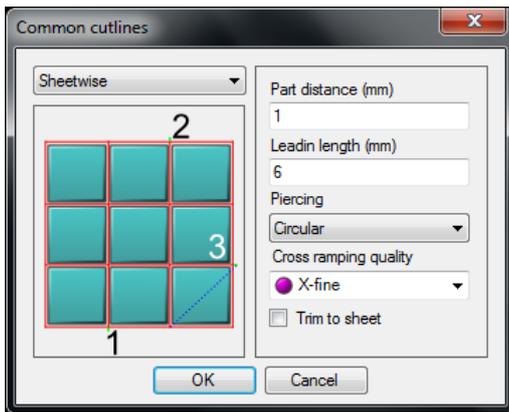
Mix between normal cutting and sheetwise



Picture 424

When internal geometries already have a toolpath the command will skip these holes. Cut holes with another command before using the common cut. In the example in Picture 424 the holes are cut with the "Contour" command.

10.19.1 Sheetwise



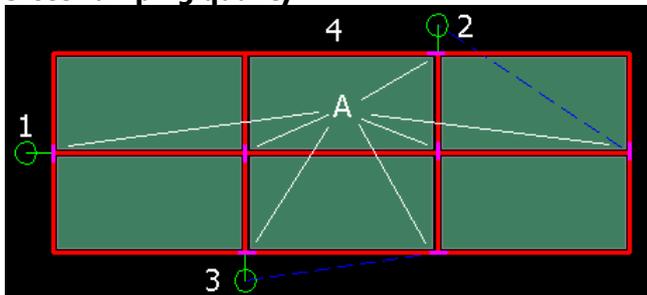
Picture 425

This method needs material free from stresses when there is no risk that the sheet will be moved during the cutting.

Lead length and piercing

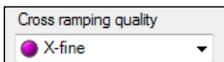
Use any length leadin and use any piercing method. Information about piercings is described on page 91.

Cross ramping quality



Picture 426

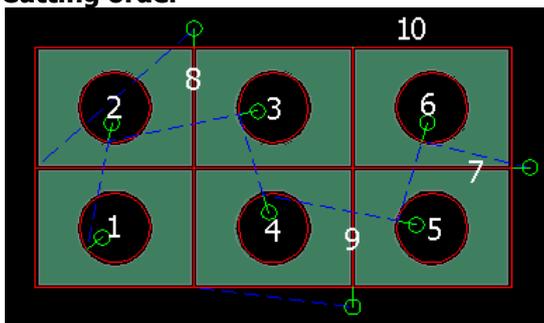
If the material is thick then the jet will make a jump when crossing a path that is already cut. There is a risk that the material will not be cut at these points.



Picture 427

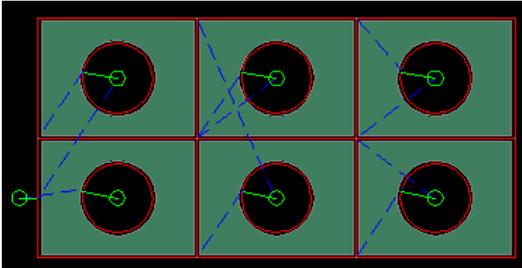
Set the cross ramping quality to a finer value and the cutting speed will slow down at these points (A).

Cutting order



Picture 428

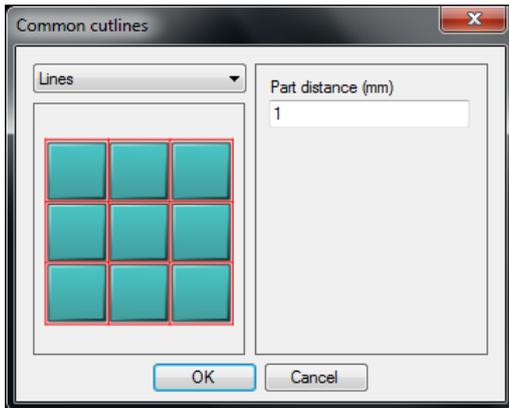
Sheetwise method cuts all holes in all parts and then cuts the lines between the parts. Then the external geometries are cut.



Picture 433

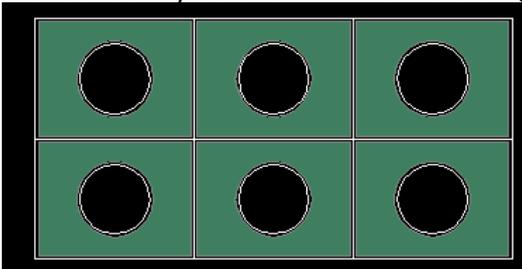
The example above shows a possible result of a mix between "normal" cutting and "common cut partwise".

10.19.3 Lines



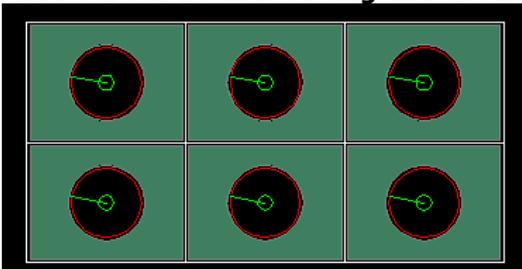
Picture 434

This method only adds CAD objects between the parts and does not add any toolpath. These objects can later be used by other commands like the "Quick" or "Manual" commands.



Picture 435

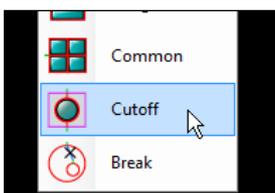
Mix between normal cutting and the Lines option



Picture 436

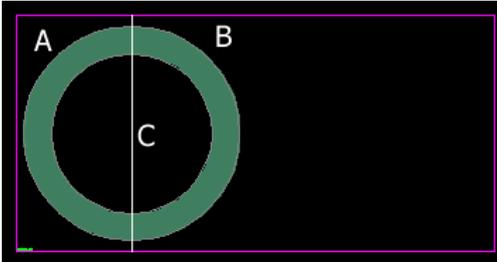
When internal geometries already have a toolpath the command will skip these holes. Cut holes with another command before using the "Line" option.

10.20 Cutoff



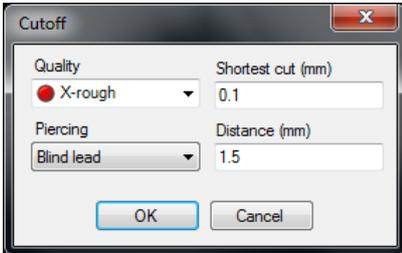
Picture 437

The "Cutoff" command is used to divide a sheet into one or several portions.



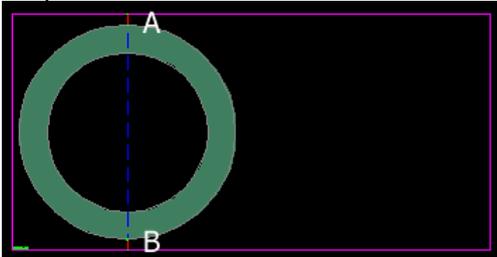
Picture 438

In Picture 438 the sheet needs to be "cutoff" into two portions (A) and (B). To do this make a line or a polyline (C) that describes the cutoff geometry. Start the command and select the cutoff object (C).



Picture 439

The command will cut off the sheet at point (A) and (B) below. It will not destroy the internal geometry of the part.



Picture 440

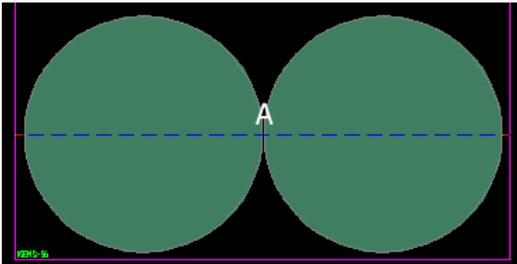
Quality

Since the cutoff is only used for separation the X-rough cutting quality may be used to make the fastest cut.

Piercing

Read more about piercings on page 91. The "Blind lead" is probably sufficient in most cases.

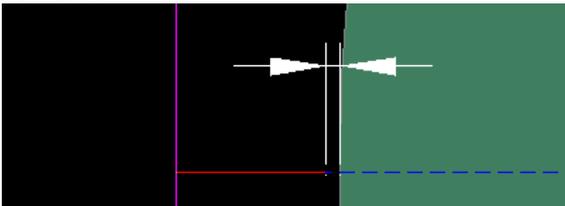
Shortest cut



Picture 441

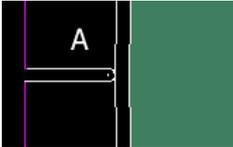
There will only be a cut made if the length of the cut is longer than this value. In Picture 441 the distance between the two parts at point (A) is too short so no cutting will be done there.

Distance



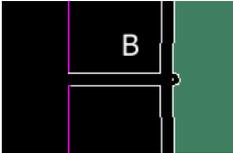
Picture 442

This value controls the distance between the parts, the center of the starts and the end of the cutoffs. Setting the distance to more than 1.5 times tool diameter leaves a small bridge between the cutting paths (A).



Picture 443

If the parts are located on the machine when the machine cuts the toolpath and the value is a smaller distance value than the tool radius the parts are destroyed (B).



Picture 444

Different methods

There are three different methods to use the "Cutoff":

Cutoff first

Do the cutoff first and then cut the parts on the sheet. Be sure the distance value is large enough to not destroy the parts. In this case run the cutoff first in the "Order" command (see page 148).

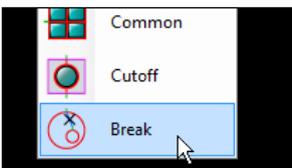
Cut part first

Cut all parts first and then the cutoff. If the parts are not unloaded from the machine be sure the distance value is large enough. In this case run the cutoff at the end in the "Order" command (see page 148).

Cut part first, unload and then cut

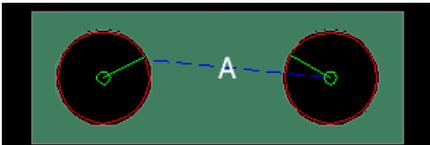
To first cut and then unload all parts set the distance value to zero. In this case there will be no bridges.

10.21 Break toolpath



Picture 445

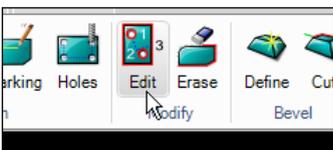
This command breaks a disconnected toolpath in two by dividing at the selected "rapid".



Picture 446

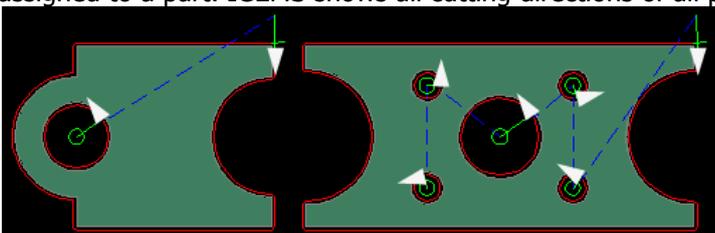
Click point (A) to get two disconnected toolpaths.

10.22 Edit toolpath



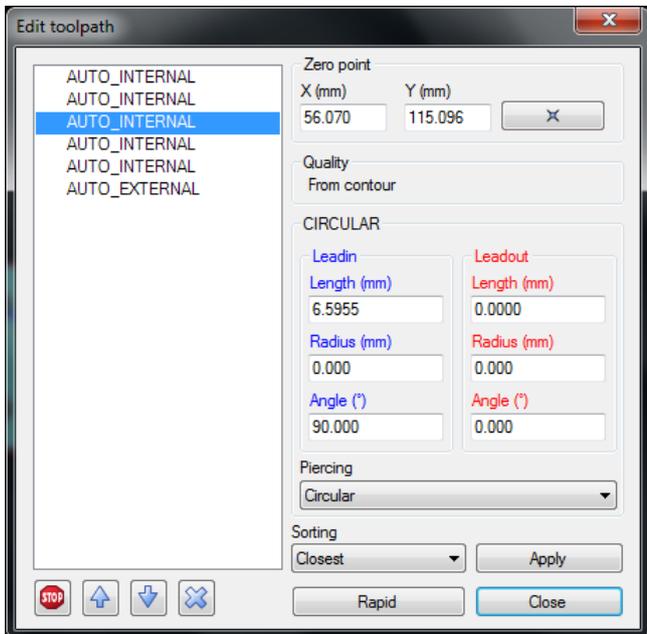
Picture 447

Use the "Edit part" command to modify, reorder and apply the "Sort" command after the toolpath has been assigned to a part. IGEMS shows all cutting directions of all parts on the drawing.



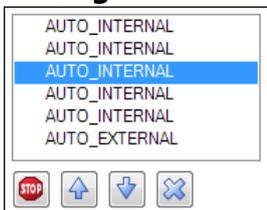
Picture 448

Select the part to edit



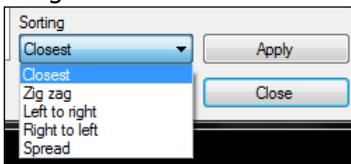
Picture 449

Cutting order



Picture 450

The internal cutting sequence is sorted manually by selecting a contour and moving it up or down in the list using the "arrow" buttons.



Picture 451

To re-order using a predefined sorting algorithm select the algorithm and press "Apply". Read more about the sorting on the "Contour" command on page 88.

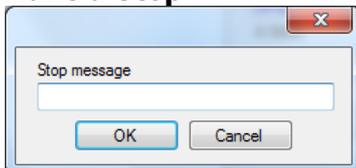
Delete a toolpath



Picture 452

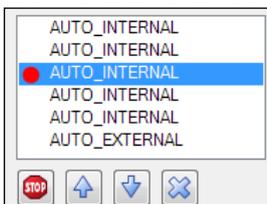
Use the "X" button to delete a toolpath from a contour. This can also be done by using the "Erase Toolpath" command (see page 109).

Make a Stop



Picture 453

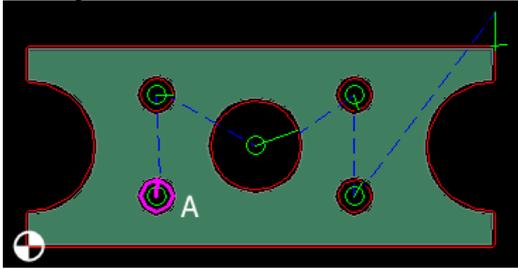
Press the "Stop" button to add a stop in the CNC-file. The stop will take place before the geometry should be cut. If the CNC-controller supports messages the message will be displayed for the operator.



Picture 454

A red circle in front of the text indicates that a stop has been assigned to the node. To remove the stop, press the "Stop" button again. The stop will be done only during the cutting (not on pre-piercing or drilling).

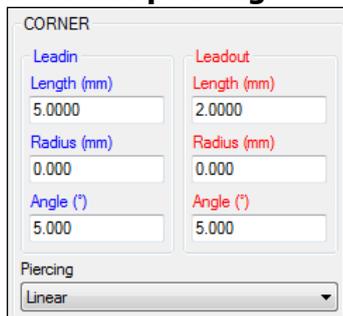
Zero point



Picture 455

As default the zero point of each part is located in the lower left corner. There is no reason to move this point, unless there are special applications that will not use this point. The selected geometry (A) is always highlighted on the drawing.

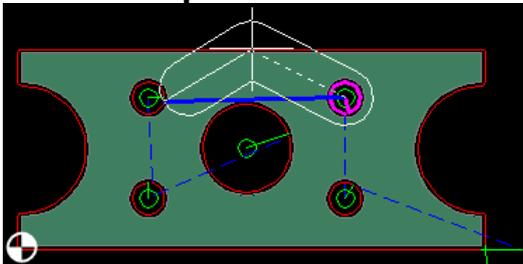
Lead and piercings



Picture 456

Change the lead and piercings the same way as in the "Contour" command. See pages 90 and 91 for more information.

Add extra rapids between contours



Picture 457

When it is necessary to add extra rapids between nodes in the part to avoid collisions, for example then click on the node to move and press the "Rapid" button. Add as many rapids as needed and press the spacebar to interrupt. To delete an extra rapid press the "rapid" again without giving any points.

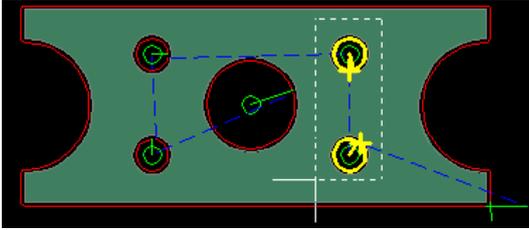
- ? What does the thick area around the rapid points mean?
This is the outside tool diameter. The value can be set in the machine setup (see page 172).

10.23 Erase toolpath



Picture 458

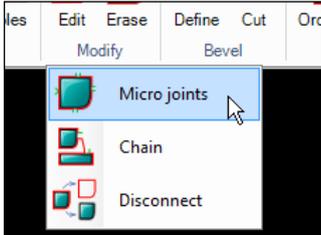
To easily delete an existing toolpath, start the command and select the leads to delete and press Enter.



Picture 459

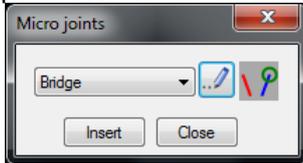
This will erase the toolpath on these two holes.

10.24 Micro joints



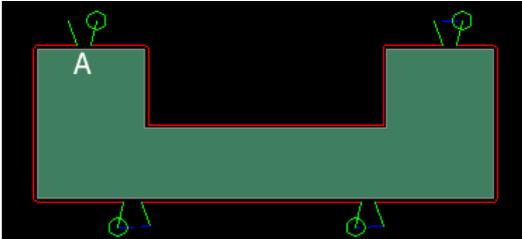
Picture 460

Use this command to add several leadin on the same geometry. The lead's design can prevent small parts from falling down into the water tank of the machine. It can also be used to fix the part in the sheet so the parts are not moved because of stresses in the material.



Picture 461

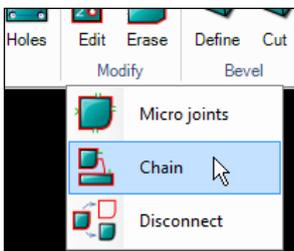
Click the "Edit" button to make lead definitions and piercing settings in the same way as on the "Contour" command described on pages 90 to 93. Click on the "Insert" button to attach the lead to the part.



Picture 462

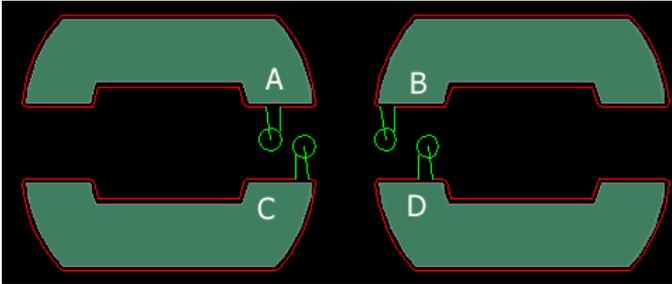
The standard lead is inserted at (A). The other three are micro joints with the same settings as (A).

10.25 Chain



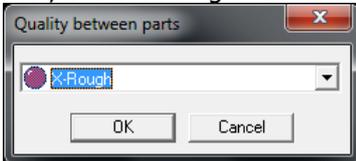
Picture 463

This command can only be used on parts that already have a toolpath. The command joins the different toolpaths together by replacing Rapid movements to cutting. The "Chain cut" command creates a toolpath with only one piercing for all selected parts. "Chain" is often use in materials which are difficult to pierce.



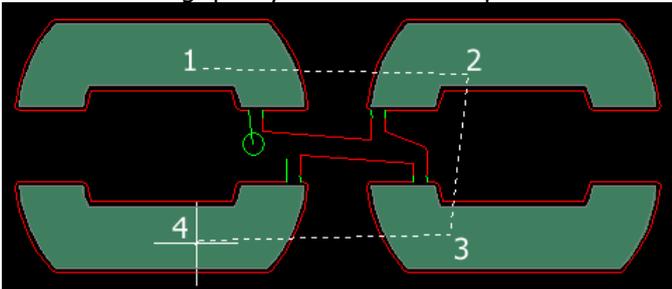
Picture 464

First, use the "Single" command and place the lead on good positions.



Picture 465

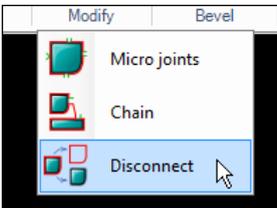
Enter the cutting quality to use for the rapids that will be converted to cutting objects.



Picture 466

IGEMS will automatically use the "Fence" option on selected parts. The parts are selected in the same order the fence line crosses the parts.

10.26 Disconnect



Picture 467

A toolpath connected to the part is simple to work with. The toolpath is attached and will move with the part. In some specific cases, it is necessary to disconnect the toolpath from the part. This command will separate the part and toolpath into individual units.

11 5X CAM

IGEMS "5X-CAM" module provides tools to prepare cutting in a 5-axis cutting machine. Strategies supported are cutting with fixed or various angles in flat material.

11.1 Define bevel



Picture 468

This command is used to setup the bevel parameters by selecting a 2D part made by the "Create" command and then applying requested bevel angles on individual contours.

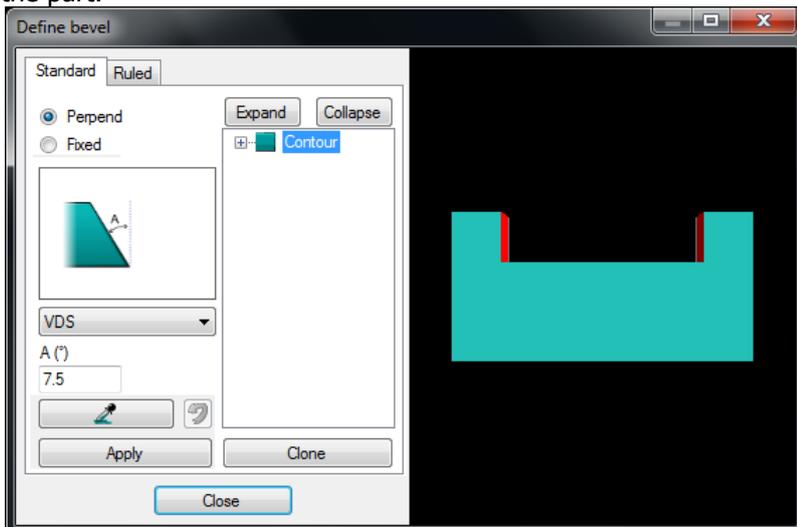


Picture 469

Select the part that should be modified with Bevel information.

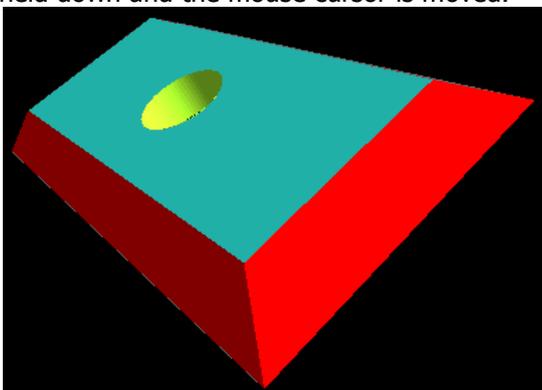
11.1.1 Standard bevel

The "Standard bevel" command works on normal parts. Add different types of bevel on different edges of the part.



Picture 470

Picture 470 is an example of looking on the Bevel parts from different views when the left mouse button is held down and the mouse cursor is moved.

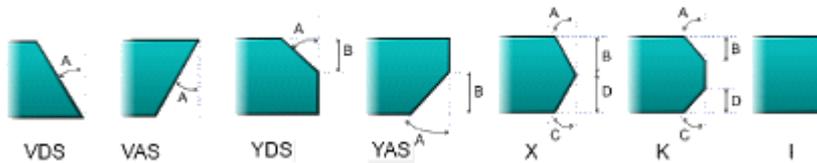


Picture 471

Hold down the mouse wheel while moving the mouse to pan and roll the mouse wheel to zoom.

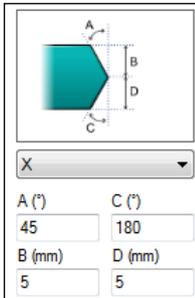
Perpendicular

This method is cuts the part with a perpendicular cut. The angle of the jet will never cut with a larger angle then defined. The following perpendicular modes are available:



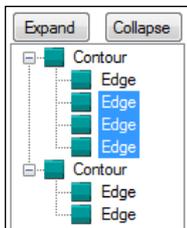
Picture 472

Select desired method and set the other values for the bevel.



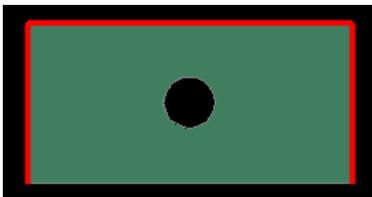
Picture 473

Select the edges on the part that the bevel setting will be attached to.



Picture 474

Select a Contour node and all edges on that node will be selected. Select a single node or select multiple nodes by using SHIFT or CTRL.



Picture 475

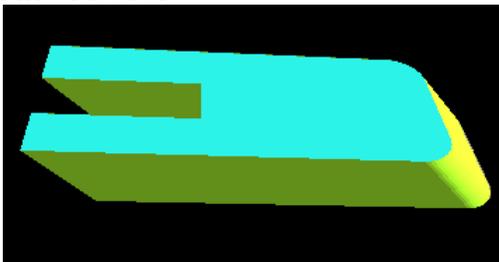
The CAD window shows graphically which nodes are selected.



Picture 476

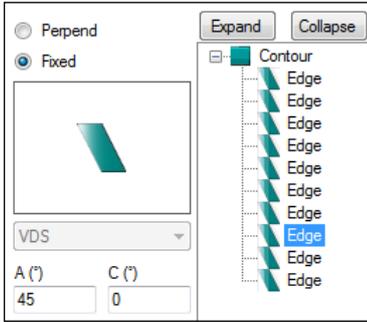
Press "Apply" to attach the bevel settings to the part.

Fixed bevel



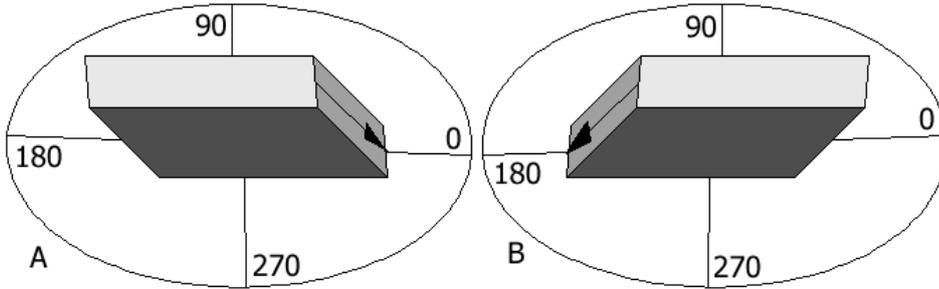
Picture 477

Use the fixed bevel to cut whole contours or edges with a fixed angle.



Picture 478

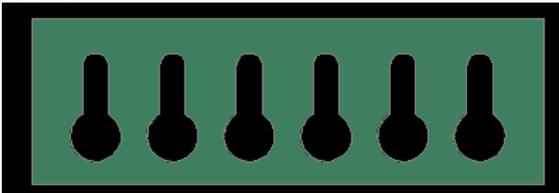
The A value is the Bevel angle and the C value is the slope down direction of the fixed angle.



Picture 479

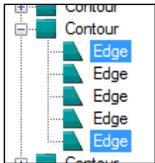
The first picture above has slope down direction 0 degree and the second slope down direction of 180 degrees.

Clone



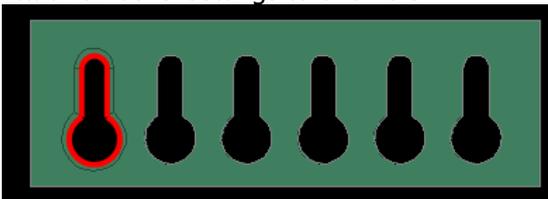
Picture 480

Use the "Clone" option to have many identical holes.



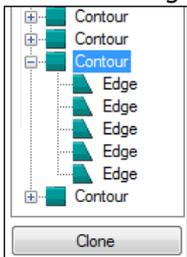
Picture 481

Attach all bevel settings to one hole.



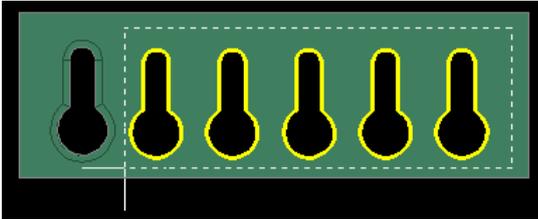
Picture 482

After activating the Contour node press the "Clone" button.



Picture 483

Select all identical holes on the part.



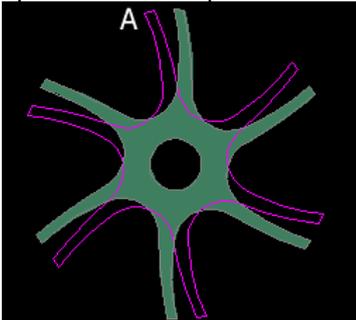
Picture 484

All six holes have now identical setting.

11.1.2 Ruled bevel

In "Ruled bevel" the bevel part is described by a top and bottom geometry. The part contours is always the top. The bottom geometry can be defined by "Other objects" in the "Create" command or by the "Uniform" option in this command.

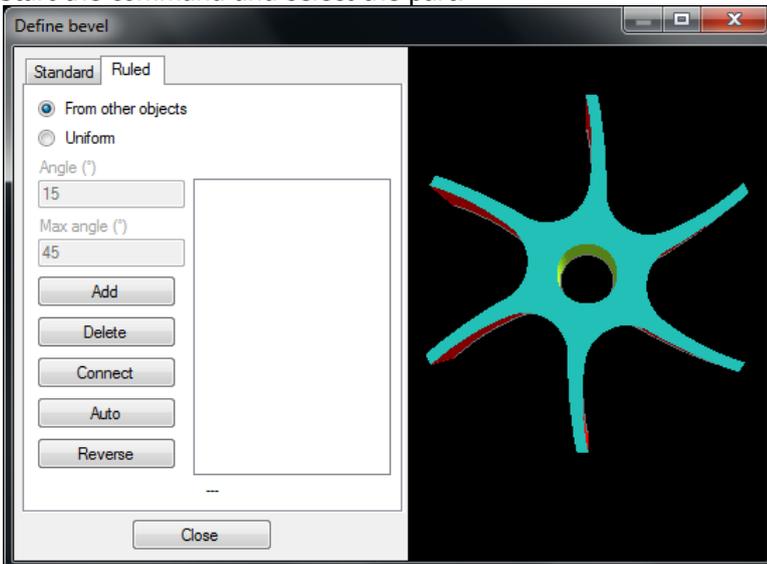
- Open the file sample file "FANWHEEL.DIG.



Picture 485

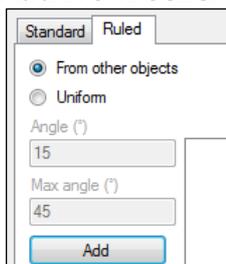
Use a material that is about 10 mm thick and create a part. Select one of the outside geometries as "Other objects"(see page 78 for more information).

- Start the command and select the part.



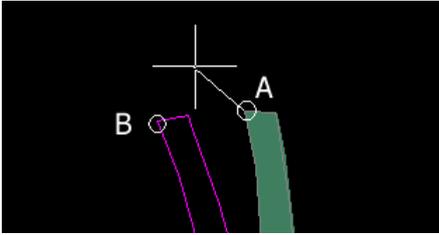
Picture 486

Add from other object



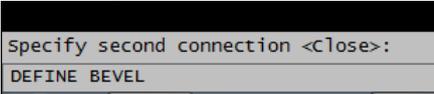
Picture 487

Select the "Add" button and select a point on the top geometry then a point on the bottom geometry.



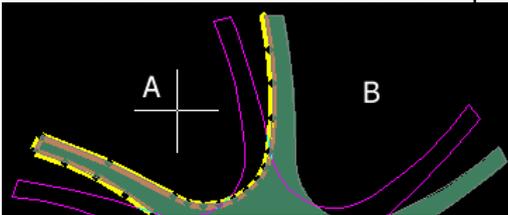
Picture 488

The selected points will create a connection line between the two geometries which is the start position for the cutting.



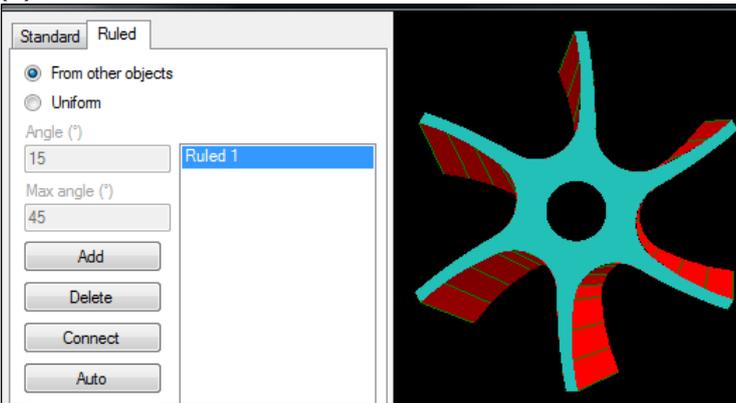
Picture 489

Define a second connection if the "ruled bevel" should not be all the way around. This will be the end of the cut and this bevel will be all around the parts. Press the spacebar or Enter to accept the "Close" option.



Picture 490

The final information is to select the direction of the cut. Click near (A) to have a CCW direction. Click near (B) to have a CW direction.



Picture 491

The bevel will now be defined as in Picture 491 above.

Add by uniform offset

This option will create the "Other objects" from the part geometry.

🔗 Open the sample file "UNIFORM.DIG"



Picture 492

Use a material thickness around 10 mm and create a part. Start the "Define" command and select the part.

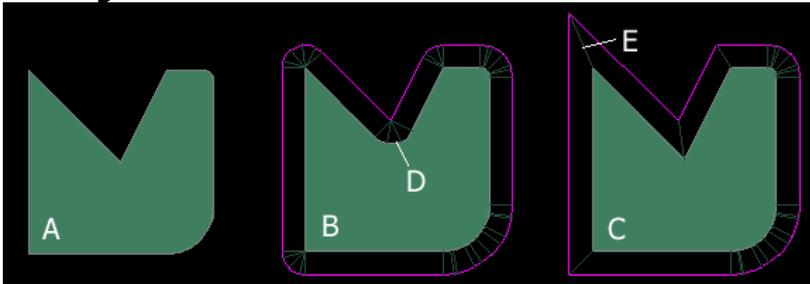


Picture 493

Angle

The offset will be calculated depending on the angle and thickness of the material. Only positive angles are supported so the outside bottom measurements will be wider.

Max angle



Picture 494

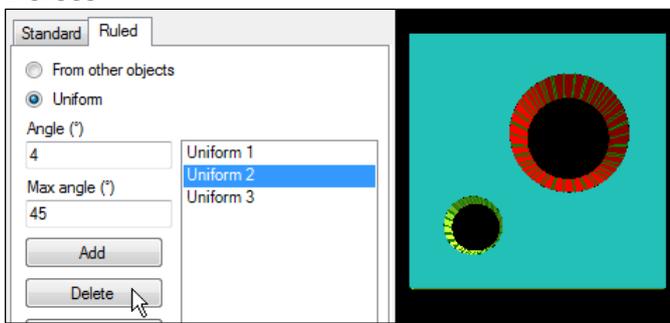
Part (A) is the part before any bevel is added.

Part (B) is the result when the "Angle" and "Max angle" are equal. The cutting angle will be the same on all edges on the part.

Note! Inside corners may be modified (D).

Part (C) has an "Angle" of 10 degrees and "Max angle" of 45 degrees so the outside corners are extended. If the edge at (E) uses a "Max angle" more than allowed the offset will be rounded.

Delete



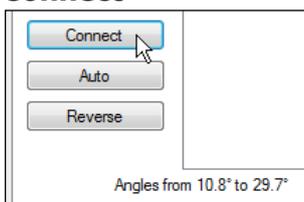
Picture 495

All ruled definitions are shown in the list, and the actual bevel is highlighted in the graphic screen.

Press the "Delete" button to delete a definition.

Note! Deleting a uniform bevel that has changed the part geometry will not restore the original contour. Close the "Define" command and use "Undo" to do that.

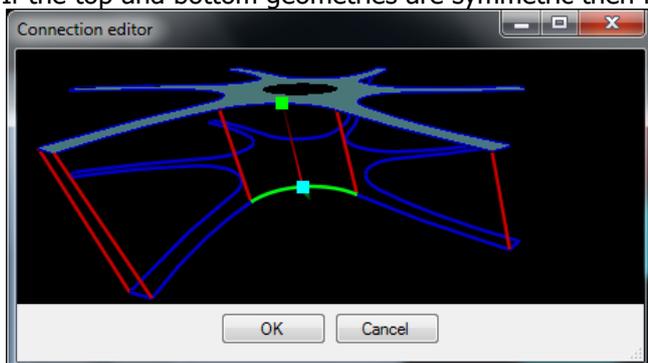
Connect



Picture 496

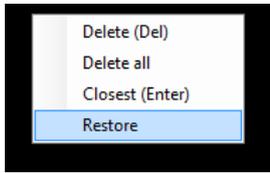
Use the "Connect" command to manually add connection lines. This feature is usable when there is an advance part where the geometry on top and bottom side is very different.

If the top and bottom geometries are symmetric then no more connection lines are needed.



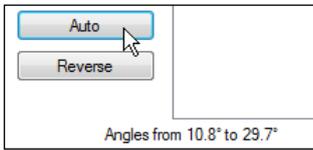
Picture 497

Picture 497 shows the window where the top and bottom geometry by lines can be connected. Hold down the right mouse button for extra functions.



Picture 498

Auto



Picture 499

The "Auto" button automatically makes connection lines on the parts. It may be helpful to use this command and then change the connection lines by using the "Connect" command.

Reverse

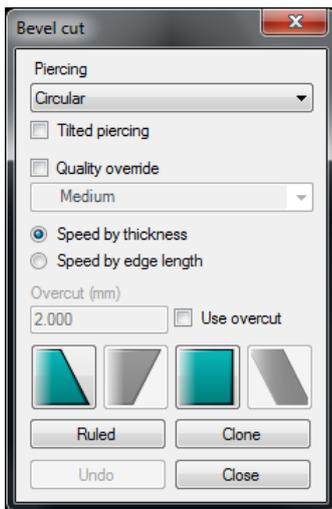
This command will probably be removed.

11.2 Cut bevel



Picture 500

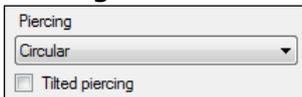
The "Cut bevel" command is used to add toolpaths to bevel parts.



Picture 501

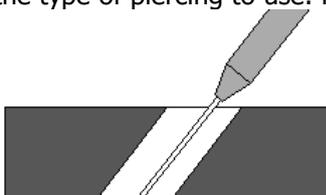
Before adding the toolpath set the following parameters:

Piercing



Picture 502

Set the type of piercing to use. Read more about different piercings on page 91.



Picture 503

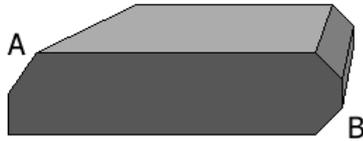
When it is necessary to make a tilted pierce, the tilting direction will be the same as the first point on the contour. Typically it is better to use a vertical pierce instead of a tilted pierce.

Quality and speed



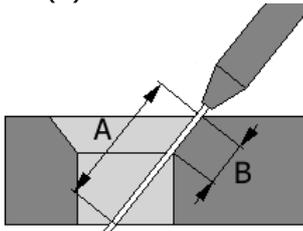
Picture 504

Normally the cutting quality is inherited from the part. Check the "Quality override" for different qualities on different cuts. This is usable for a two or three time bevel cut.



Picture 505

Side (A) shows a two time bevel and (B) a three time bevel.



Picture 506

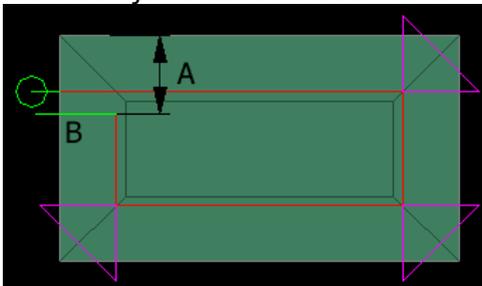
Example (A) shows the thickness of the material in the actual tilted angle. Example (B) shows the edge length. In one time cutting the result will be the same. In two or three time bevel the cutting speed can be very different.

Overcut or micro joints



Picture 507

Overcut (positive values) cannot be used when leads are placed in sharp corners. If the value is negative then micro joints can be made. This is very useful when making two or three time bevels.



Picture 508

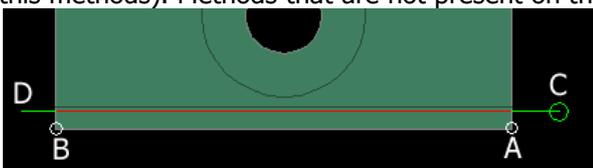
The negative overcut value will make a micro joint (A). The leadout (B) can be in a direction that will create a micro joint.

Applying toolpaths on Standard bevels



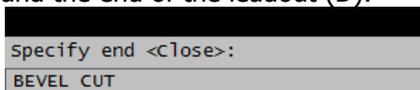
Picture 509

These four buttons show the major standard bevel cutting method (YDS, YAS, X and K is a combination of this methods). Methods that are not present on the part are disabled.



Picture 510

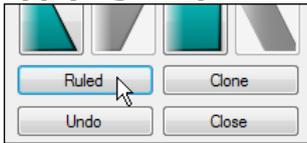
Select the start point (A) and the end point (B). The command then asks for (C) the piercing point (leadin) and the end of the leadout (D).



Picture 511

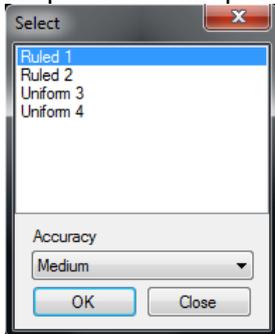
If there is the same type of bevel around the part, then accept the "Close" option by pressing spacebar or Enter.

Applying toolpaths on Ruled bevels



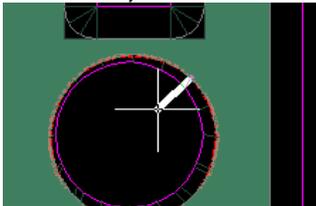
Picture 512

The ruled bevels already contain information about start and endpoint of the cut. Select specific bevel and the position of the piercing point.



Picture 513

All ruled bevels contain only lines. The "Accuracy" controls the tolerance of this vectorization. "Rough" settings are better than 0.01 mm, "Medium" is better than 0.005 mm and "Fine" is better than 0.001 mm. It is not always true that the finest accuracy will give the best cutting quality.

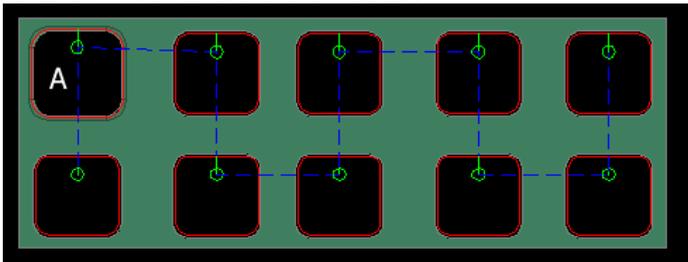


Picture 514

Press the "OK" button to set the leadin and leadout length. The leadin position will be the position for the piercing.

Clone

This feature is helpful when cutting many identical contours. The command asks for a master and then for holes on which to clone the master.



Picture 515

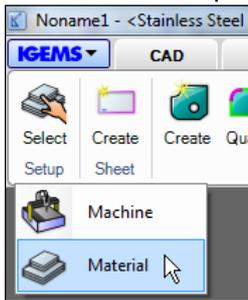
In the example above (A) is the master. The toolpath has been cloned to all other selected holes. Bevel definition must be added to all holes. Clone the toolpath to the contours which have a similar geometry and bevel definition.

11.2.1 Assign bevel cutting to a tool

All parts and toolpaths are assigned to tool one by default. If the 5-axis cutting head is located on another tool then change the default tool assignment in the "machine" settings.

12 CAM – Material setup

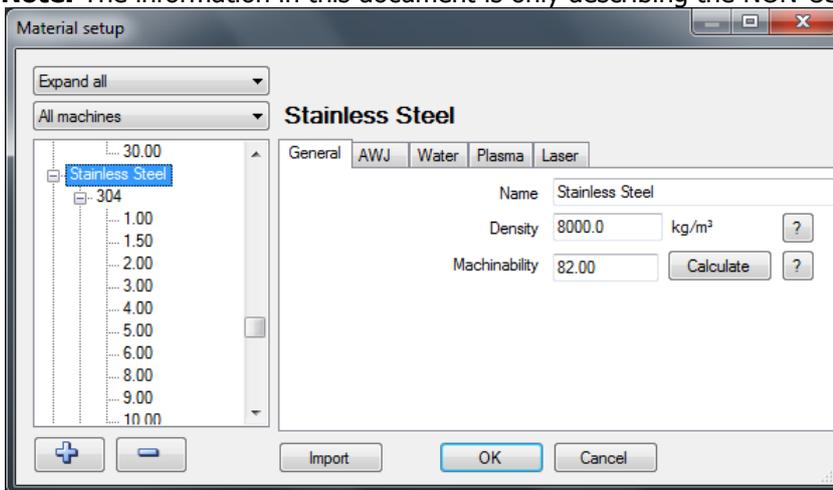
The "Material setup" is started from the "Material" command on the Setup menu.



Picture 516

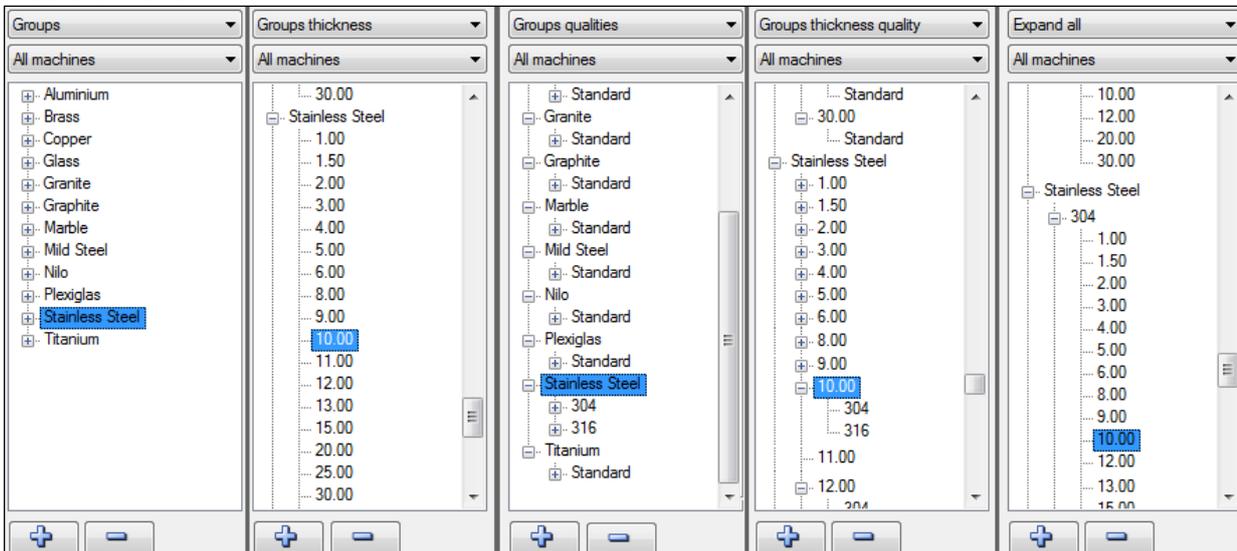
Use the "Material" command to create, delete and change material parameters.

Note! The information in this document is only describing the NON-US version of IGEMS.



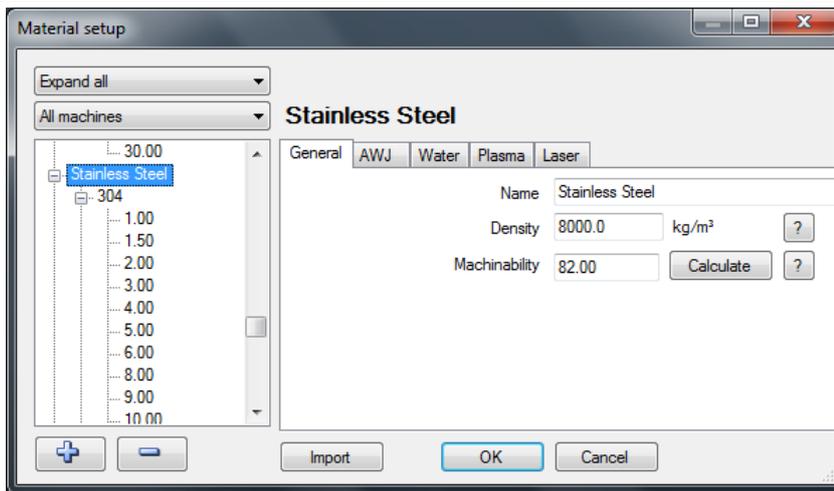
Picture 517

On the left side is a tree view. Change the options in the upper left drop down lists to look at the database in several views.



Picture 518

12.1 Material groups



Picture 519

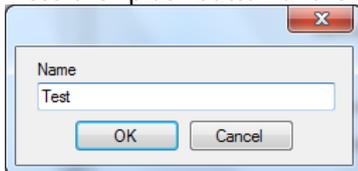
12.1.1 Add or delete groups

All material in a group has the same cutting properties. If a group is selected then add a new group or delete an existing group by clicking on the "plus" or "minus" buttons.



Picture 520

Press the "plus" button and enter the name "Test" for the new material group.

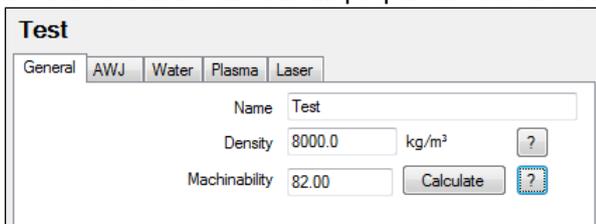


Picture 521

The new group "Test" will be an identical copy of the selected material. Delete the group and the material file will be deleted.

12.1.2 General properties

On the "General" tab edit all properties that are common for all qualities, thickness and machine types.



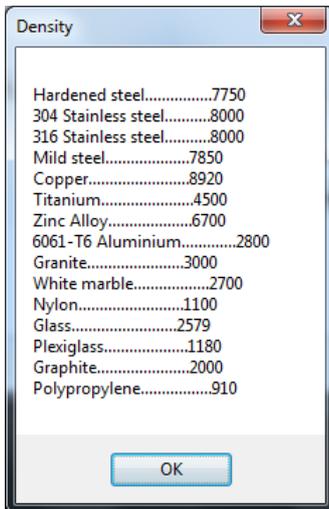
Picture 522

Name

Change the name of the group by changing this field.

Density

IGEMS needs this information to calculate weight on parts. The "question mark" button will show a list of densities for some common materials.

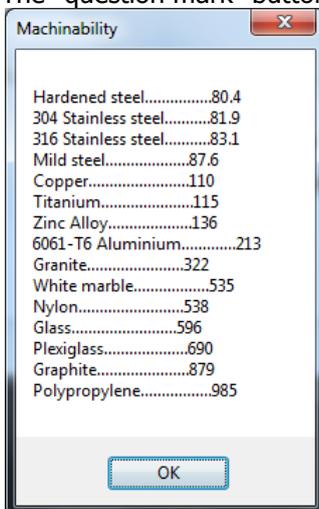


Picture 523

Machinability

This is a material property that describes how easy the material can be cut with abrasive water jet. Material with a high value is more difficult to cut than materials with a low value.

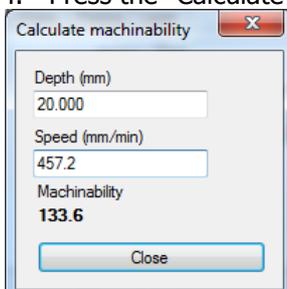
The "question mark" button will show a list of machinability values for some common materials.



Picture 524

If the Machinability is unknown then calculate the value by making a test cut and then calculate the value. Follow these steps:

1. Place the new material on the machine table.
2. Make a straight cut with a speed that is so high that the jet cannot cut thru the material.
3. Measure the depth of the cut.
4. Press the "Calculate" button.



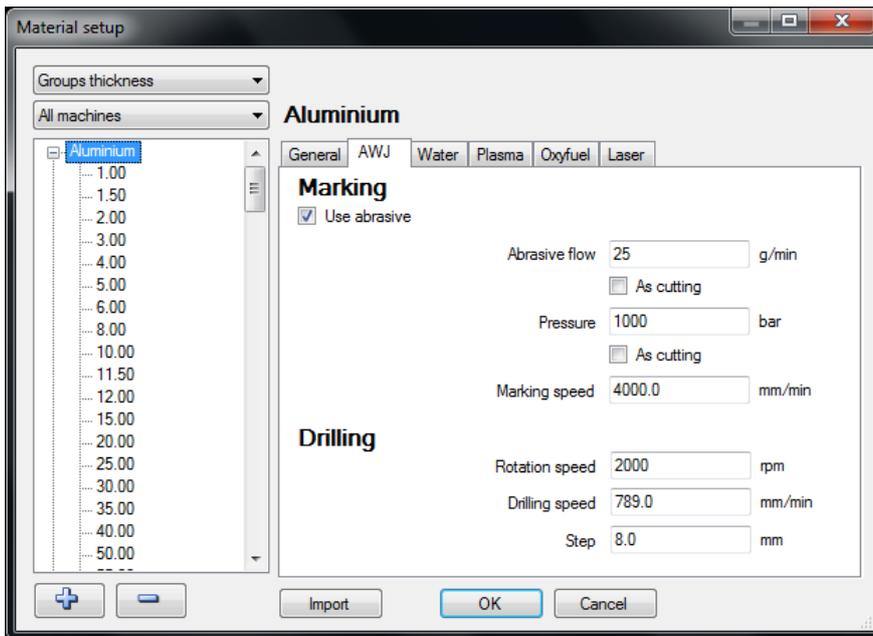
Picture 525

Enter the depth of the cut and the actual cutting speed. The calculator shows the actual machinability value.

? Is the machinability depending of other cutting parameters than depth and speed?

Yes, it uses the pressure, orifice size, size of the mixing tube and other parameters defined in the Cutting parameters.

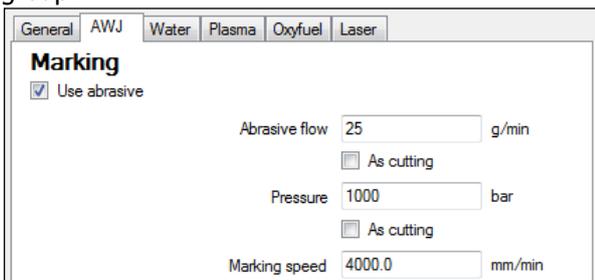
12.1.3 AWJ



Picture 526

Marking

"Marking" affects only the surface of the material and the information is identical for all thicknesses in this group.



Picture 527

Abrasive flow

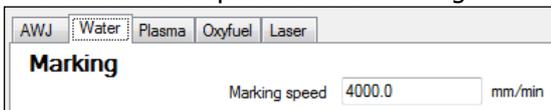
On an AWJ machine select whether or not to use abrasive. Select the amount of abrasive in gram/minute or lbs/min. With the "As cutting" option the amount of abrasive will be the same as for cutting.

Pressure

Select the pressure to use for "marking". With the "As cutting" option the pressure will be the same as for cutting.

Marking speed:

This controls the speed for the marking.

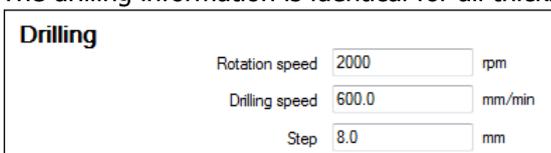


Picture 528

All machine types other than AWJ have only the Marking speed input.

Drilling

The drilling information is identical for all thicknesses.



Picture 529

Most drilling units on waterjet machines are pneumatic and so it is difficult to control the rotation and velocity. If the machine has the possibility to use these parameters then the postprocessor writes this information to the CNC-file.

Rotation speed

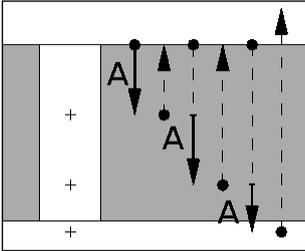
Enter the rpm/min.

Drilling speed

With a servo controlled drilling spindle this value will be used. This value affects the time calculation.

Step

These inputs make it possible to use a peck drilling cycle.



Picture 530

The value controls the depth of each step (see the A value on the picture).

Note! Not all machines support this feature.

12.1.4 Water, Plasma, Oxyfuel and Laser

General	AWJ	Water	Plasma	Oxyfuel	Laser
Marking					
		Marking speed	4000.0	mm/min	
Drilling					
		Rotation speed	2000	rpm	
		Drilling speed	1000.0	mm/min	
		Step	0.0	mm	

Picture 531

The settings for these four machine types are identical.

Marking

This value controls the marking speed.

Drilling

These values have the same meaning as for the AWJ machine.

12.2 Material qualities

The material qualities are used to handle material cost and the "Sheet" commands handle the material stock. The quality information is saved as different tags on each thickness in the material. The information is identical for all machine types.

Available for thickness			
<input checked="" type="checkbox"/> 1.000	<input checked="" type="checkbox"/> 6.000	<input checked="" type="checkbox"/> 13.000	<input checked="" type="checkbox"/> 50
<input checked="" type="checkbox"/> 1.500	<input checked="" type="checkbox"/> 8.000	<input checked="" type="checkbox"/> 15.000	<input checked="" type="checkbox"/> 55
<input checked="" type="checkbox"/> 2.000	<input checked="" type="checkbox"/> 9.000	<input checked="" type="checkbox"/> 20.000	<input checked="" type="checkbox"/> 60
<input checked="" type="checkbox"/> 3.000	<input checked="" type="checkbox"/> 10.000	<input checked="" type="checkbox"/> 25.000	<input checked="" type="checkbox"/> 80
<input checked="" type="checkbox"/> 4.000	<input type="checkbox"/> 11.000	<input checked="" type="checkbox"/> 30.000	
<input checked="" type="checkbox"/> 5.000	<input checked="" type="checkbox"/> 12.000	<input checked="" type="checkbox"/> 40.000	

Picture 532

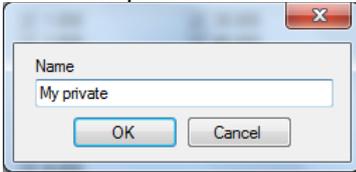
12.2.1 Add or delete qualities

If a quality is selected then add a quality or delete an existing quality by clicking on the "plus" or "minus" buttons.



Picture 533

Press the "plus" button and enter the name "My private" for the new quality.



Picture 534

The new quality "My private" will be an identical copy of the selected quality. Press the "minus" button to delete the quality.

12.2.2 Quality properties

Test / My private		
Name	My private	
Cost	6.00	/kg
Group	Test	
Density	8000 kg/m ³	
Machinability	82.0	
Thickness		
<input checked="" type="checkbox"/> 1.000	<input checked="" type="checkbox"/> 9.000	<input checked="" type="checkbox"/> 30.000
<input checked="" type="checkbox"/> 1.500	<input checked="" type="checkbox"/> 10.000	<input checked="" type="checkbox"/> 40.000
<input checked="" type="checkbox"/> 2.000	<input type="checkbox"/> 11.000	<input checked="" type="checkbox"/> 50.000
<input checked="" type="checkbox"/> 3.000	<input checked="" type="checkbox"/> 12.000	<input checked="" type="checkbox"/> 55.000
<input checked="" type="checkbox"/> 4.000	<input checked="" type="checkbox"/> 13.000	<input checked="" type="checkbox"/> 60.000
<input checked="" type="checkbox"/> 5.000	<input checked="" type="checkbox"/> 15.000	<input checked="" type="checkbox"/> 80.000
<input checked="" type="checkbox"/> 6.000	<input checked="" type="checkbox"/> 20.000	
<input checked="" type="checkbox"/> 8.000	<input checked="" type="checkbox"/> 25.000	

Picture 535

Name

Use this field to change the name of the quality.

Cost

This value is used in the cost calculations.

Thickness

Thickness		
<input checked="" type="checkbox"/> 1.000	<input checked="" type="checkbox"/> 9.000	<input checked="" type="checkbox"/> 30.000
<input checked="" type="checkbox"/> 1.500	<input checked="" type="checkbox"/> 10.000	<input checked="" type="checkbox"/> 40.000
<input checked="" type="checkbox"/> 2.000	<input type="checkbox"/> 11.000	<input checked="" type="checkbox"/> 50.000
<input checked="" type="checkbox"/> 3.000	<input checked="" type="checkbox"/> 12.000	<input checked="" type="checkbox"/> 55.000
<input checked="" type="checkbox"/> 4.000	<input checked="" type="checkbox"/> 13.000	<input checked="" type="checkbox"/> 60.000
<input checked="" type="checkbox"/> 5.000	<input checked="" type="checkbox"/> 15.000	<input checked="" type="checkbox"/> 80.000
<input checked="" type="checkbox"/> 6.000	<input checked="" type="checkbox"/> 20.000	
<input checked="" type="checkbox"/> 8.000	<input checked="" type="checkbox"/> 25.000	

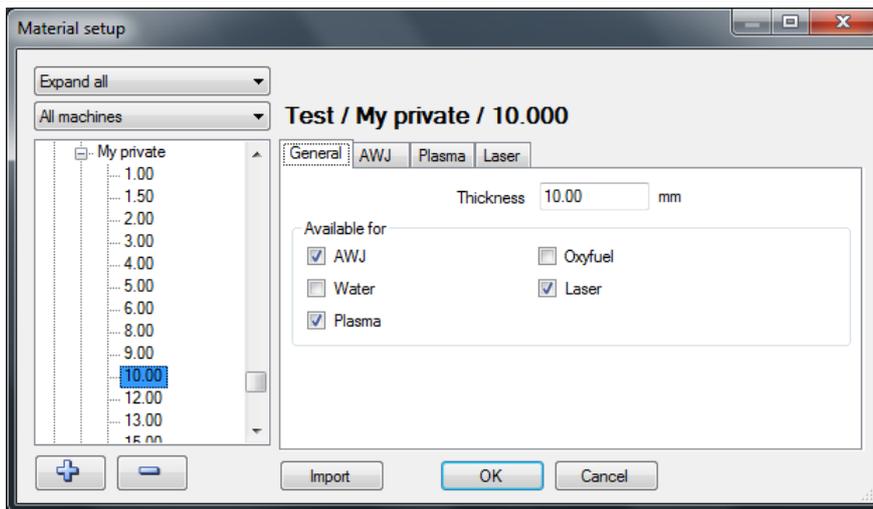
Picture 536

Check the thickness that should be available for the quality.

? Why can't I change the density and machinability?

If your material has different weight or machinability, then you should add a new group instead.

12.3 Material thickness



Picture 537

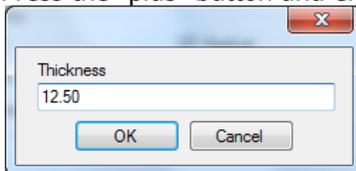
12.3.1 Add or delete thickness

If a thickness is selected in the list then add a new thickness or delete an existing thickness by clicking on the "plus" or "minus" buttons.



Picture 538

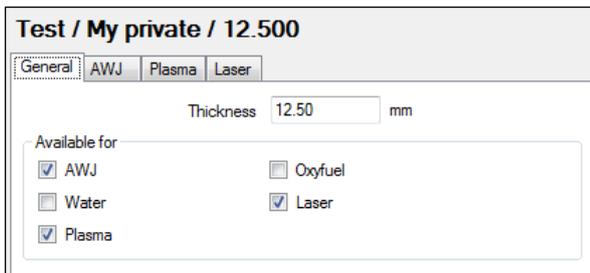
Press the "plus" button and enter the thickness "12.50".



Picture 539

Press the "minus" button to delete.

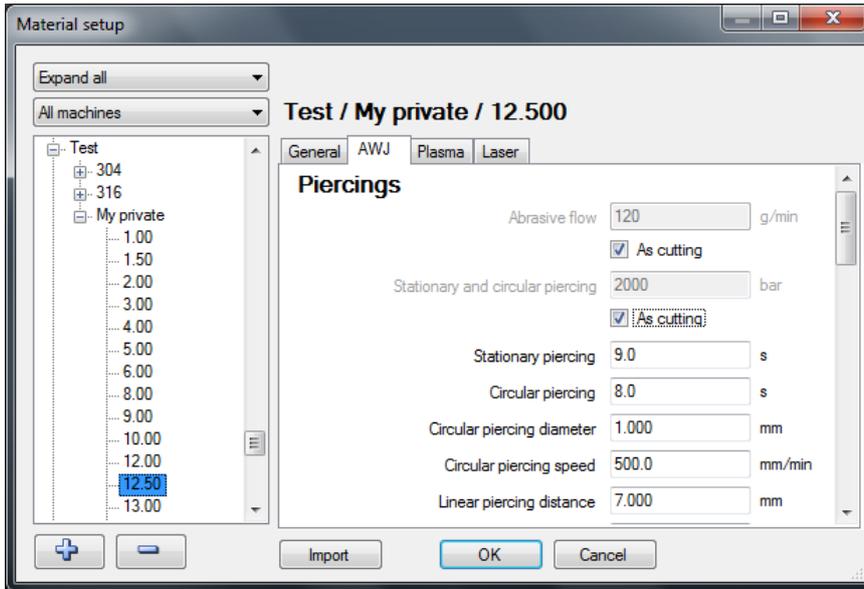
12.3.2 General



Picture 540

On the general tab change the thickness and control what machine types can be used for this specific material.

12.3.3 AWJ



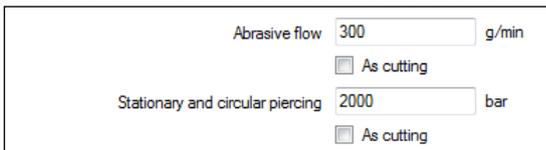
Picture 541

Piercings

IGEMS supports different methods to pierce through material. Read more about piercings on page 91.

Individual piercing parameters

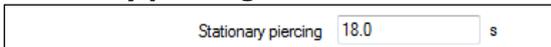
To use a different amount of abrasive or a different pressure for the piercing enter values here:



Picture 542

Check the two boxes "As cutting" to not use any special values. When cutting brittle material, piercing at a lower pressure is often needed.

Stationary piercing



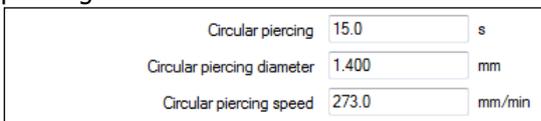
Picture 543

The jet will stay in the same position during the entire pierce.

The value is the length of time in seconds that the jet will be on. This piercing method works best on very thin material.

Circular piercing

The jet goes around a small circle during the entire pierce. This is one of the most common methods of piercing on an AWJ machine and works well on thin and thick material.



Picture 544

Circular piercing

This controls the piercing time it takes for the jet to cut through the material.

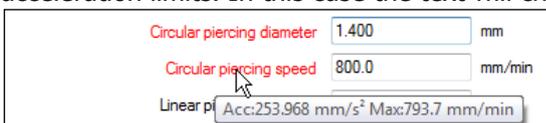
Circular piercing diameter

This is the movement diameter. The diameter in the material will be this value plus the diameter of the jet.

Circular piercing speed

This value is in mm/min. The faster the speed the more number of revolutions is used.

Note! If the diameter is too small, or the speed too high, then the centrifugal acceleration may exceed the acceleration limits. In this case the text will change to red.



Picture 545

Hold the pointer on the text to see a tooltip with information about the actual accelerations.
The maximum acceleration is set in the machine setup.

Linear piercing

Linear piercing distance	<input type="text" value="8.000"/>	mm
Linear piercing speed	<input type="text" value="273.0"/>	mm/min

Picture 546

The jet makes a linear movement during the entire piercing. If the distance is longer than the length of the lead then the movement will go back and forth until the specified distance is reached.

Linear piercing distance

This is the length needed for the piercing to be able to cut through the material.

Linear piercing speed

This is the speed that is used for the piercing.

Overcut distance

This value is used as the default overcut distance when the cutting starts directly on the geometry (Lind lead).

Overcut distance	<input type="text" value="7.000"/>	mm
------------------	------------------------------------	----

Picture 547

This value controls the length of the overlapped cut.

Dynamic leads

Dynamic leads are used to have the piercing position as far away as possible from the geometry. It is a nice feature when cutting material that often breaks on the bottom side during the piercing.

Dynamic lead min length	<input type="text" value="3.000"/>	mm
Dynamic lead max length	<input type="text" value="10.000"/>	mm

Picture 548

Dynamic lead min length

This should be the shortest acceptable lead length for this material.

Dynamic lead max length

Set this value to the maximum required lead length. While safer to have a longer lead this takes time to cut.

Cutting

Cutting					
Fixed speed	<input type="text" value="1000.0"/> mm/min				
Abrasive flow	<input type="text" value="350"/> g/min				
	<input type="checkbox"/> By machine				
Pressure	<input type="text" value="3800"/> bar				
	<input type="checkbox"/> By machine				
TAC for Rough	<input type="text" value="1.40"/> °				
TAC for X-fine	<input type="text" value="-1.00"/> °				
	...				
	X-rough	Rough	Medium	Fine	X-fine
High speed	913.1	655.9	411.5	295.6	228.7
Low speed	314.9	273.0	214.5	175.8	148.3
Steps	<input type="text" value="3"/>	<input type="text" value="3"/>	<input type="text" value="3"/>	<input type="text" value="3"/>	<input type="text" value="3"/>
Acc dist	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>
Dec dist	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>	<input type="text" value="3.000"/>
High fac	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>
Low fac	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>	<input type="text" value="1.00"/>
	<input type="checkbox"/> Automatic				
	<input type="button" value="Default"/> <input type="button" value="Import"/>				
Arc fac	<input type="text" value="1.00"/>				

Picture 549

The information in this area controls the thickness depending information for the cutting process.

Fixed speed

This value is used when "Fixed speed" is selected in the Machine setup. When using fixed speed there is no possibility to use qualities or speed ramping. The value is only used on an AWJ machine without using the "AWJ" module.

Fixed speed	1000.0	mm/min
-------------	--------	--------

Picture 550

Individual cutting parameters

It may be an advantage for certain materials to use other cutting parameters than the machine uses as standard. If that case, set up the individual cutting parameters here.

Abrasive flow	350	g/min
	<input type="checkbox"/>	By machine
Pressure	3800	bar
	<input type="checkbox"/>	By machine

Picture 551

Abrasive flow

This controls the amount of abrasive. To use the standard amount of abrasive check the box "By machine".

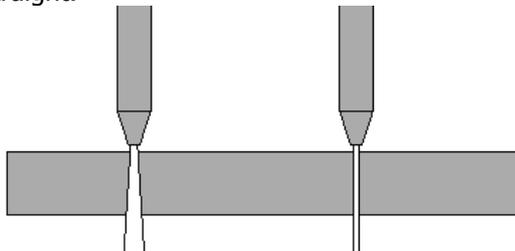
Pressure

This value controls the pressure. To use the standard pressure for cutting check the box "By machine".

Note! If there is no machine equipment to automatically change the abrasive amount or the pump pressure then check these boxes, allowing fewer changes for the machine operator.

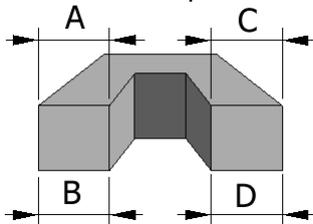
Taper Angle Control

Depending on material properties and cutting parameters the cut through the material is not always 100% straight.



Picture 552

By using the Taper Angle Control feature manually adjust the inclination of the nozzle. IGEMS does not use any mathematic formula to calculate the taper. Make a test cut to adjust the TAC parameters. Cut a small test part without TAC using cutting quality Rough and X-Fine.



Picture 553

Measure the Rough side A and B. Measure the X-Fine side C and D. Start the "TAC Calculator" by clicking on the button.

TAC Calculator	
Top measure (mm)	10.00
Bottom measure (mm)	10.50
Thickness (mm)	10.00
Angle	-1.4°
Close	

Picture 554

The TAC Calculator calculates the angles. If the measurements are smaller on the bottom side on an outside geometry then the angles are negative else positive.

TAC for Rough	-1.00	°
TAC for X-fine	-0.50	°

Picture 555

TAC for Rough

Enter the angle for the "rough" measurement.

TAC for X-Fine

Enter the angle for the "X-fine" measurements.

Note! To use the "TAC" option a standard CNC machine is needed with at least 4 or 5-axes controlled by the CNC and the nozzle can be tilted in different directions.

Automatic speed optimizing

The automatic speed optimizing adjusts the cutting speed depending on cutting parameters, material properties and geometry to be cut. Straight lines and large arcs are cut with a higher speed than sharp corners or small arcs. The automatic speed optimizing uses another algorithm than earlier versions of IGEMS. The speeds are therefore not the same as before, but still divided into five qualities.

	X-rough	Rough	Medium	Fine	X-fine
High speed	791.3	568.4	356.6	256.1	198.2
Low speed	272.9	236.6	185.9	152.3	128.5
Steps	5	4	3	2	2
Acc dist	5.000	4.000	3.000	2.000	1.000
Dec dist	5.000	4.000	3.000	2.000	1.000
High fac	1.00	1.00	1.00	1.00	1.00
Low fac	1.00	1.00	1.00	1.00	1.00
<input checked="" type="checkbox"/> Automatic		Default		Import	
Arc fac	1.00				

Picture 556

At IGEMS R&D Center, we are constantly updating and fine-tuning cutting parameters. Updating IGEMS may also update cutting speed parameters. The automatic speed optimizing is activated when the checkbox "Automatic" is checked.

Manual speed optimizing

The manual speed optimizing uses the same formulas as IGEMS R8 and R9. Use this option to fine-tune parameters. The values are not changed in an update of IGEMS.

	X-rough	Rough	Medium	Fine	X-fine
High speed	913.1	655.9	411.5	295.6	228.7
Low speed	314.9	273.0	214.5	175.8	148.3
Steps	3	3	3	3	3
Acc dist	3.000	3.000	3.000	3.000	3.000
Dec dist	3.000	3.000	3.000	3.000	3.000
High fac	1.00	1.00	1.00	1.00	1.00
Low fac	1.00	1.00	1.00	1.00	1.00
<input type="checkbox"/> Automatic		Default		Import	
Arc fac	1.00				

Picture 557

The following parameters can be adjusted:

High speed

This speed is used by straight lines and arc radius larger than the thickness of the material.

Low speed

The low speed is dependent on the geometry. The sharper the corner or the smaller radius size of an arc the lower speed will be used. The speed will never be lower than this value.

Steps

This value is only used if when using the ramping method "Speed interpolation by CNC-file" (defined in the machine setup).

Acceleration distance

This is the total length of all speed increasing steps when the speed changes from low to high speed.

Deceleration distance

This is the total length of all speed decreasing steps when the speed changes from high to low speed.

High speed factor

The speed is calculated by known parameters such as: Machinability, Pressure, Orifice size, Abrasive quality, Amount of abrasive, Nozzle size, Cutting quality and the thickness of the material. If the actual high speed is

according to this algorithm then the height factor is 1.00. If the high speed is changed it will also change the high speed factor.

Low speed factor

This factor does the same as "High speed factor" but for the "Low speed" values.

Arc factor



Arc fac 1.00

Picture 558

This value controls the cutting speed in the arcs. If this value is set to a higher value then the small arcs will use a higher speed. Normal value is 1.00

Default

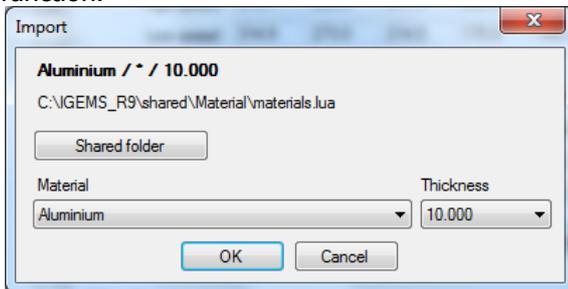


Default Import

Press this button to reset the values to default. This will change all values on this material according to the algorithm.

Import

The database in R10 is different from previous versions. To use old values in IGEMS R10 use the import function.



Import

Aluminium / * / 10.000

C:\IGEMS_R9\shared\Material\materials.lua

Shared folder

Material Aluminium Thickness 10.000

OK Cancel

Picture 559

Shared folder

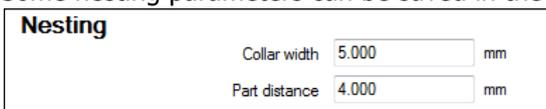
When using this command for the first time, select the location of the shared folder. IGEMS R10 will then remember this location.

Material and Thickness

Be sure to use the same material and thicknesses as in IGEMS R10. Repeat these steps for every material imported to IGEMS R10.

12.3.4 Nesting

Some nesting parameters can be saved in the material database.



Nesting

Collar width 5.000 mm

Part distance 4.000 mm

Picture 560

Collar width

This value sets the default value used when creating new sheets.

Part distance

This value sets the default value for the part distance. The value is used in the "Nesting" commands.

12.3.5 Various



Various

Technology

Options

User

Picture 561

The "Technology", "Options" and "User" settings are read by the postprocessor for handling different equipment or functions in the postprocessor.

Plasma, Oxyfuel and Laser settings

IGEMS is developed for Abrasive Water Jet Cutting, but also supports other machine types such as Plasma, Oxyfuel and Laser machines. These machines have similar inputs.

12.4 General import

If there is an older version of IGEMS then import the complete material database.

13 CAM – Sheet commands

13.1 Sheet Create

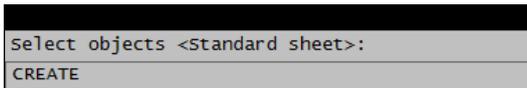


Picture 562

This command creates and inserts sheets onto the drawing area.

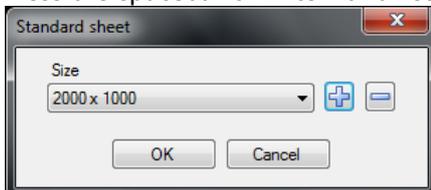
13.1.1 Standard sheets

When the command is started there is the following question:



Picture 563

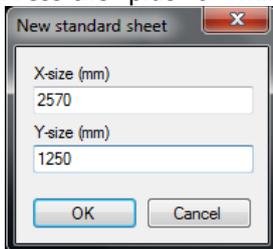
Press the spacebar or Enter for a list of standard sheets.



Picture 564

Modify list of standard sheet

Press the "plus" or "minus" buttons to modify the list of standard sheets.

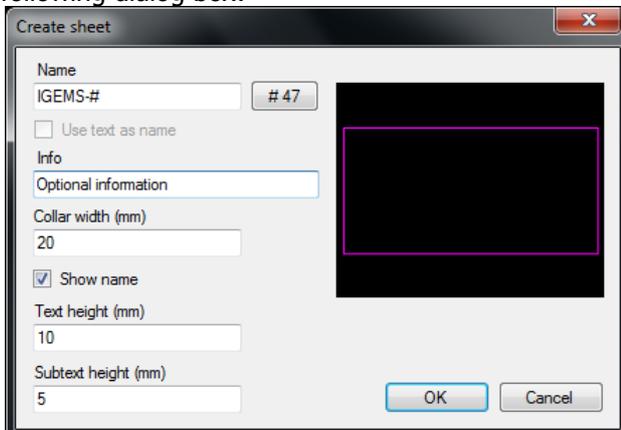


Picture 565

Press the "plus" button to add a new sheet size.

Sheet properties

Select a size in the drop down list and press "OK". Add additional information about the sheet in the following dialog box:



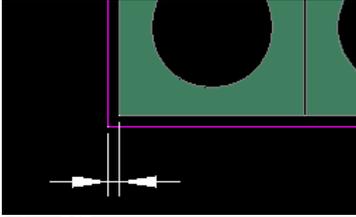
Picture 566

Name

To track all sheets it is important to use a unique name for the sheet. The "#" sign uses an automatic counter in the same way as on "Part create" (see page 77 for more information).

Info

This information is used by the postprocessors and in the reports.

Collar width

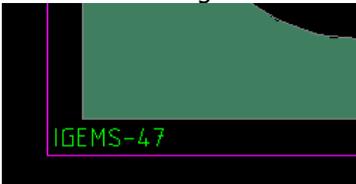
Picture 567

This value controls the distance between the edge of the parts and the edge of the sheets when using "Nesting" commands.

Show name

Picture 568

With these settings control height and the name of the sheet is visible on the lower left corner on the sheet.



Picture 569

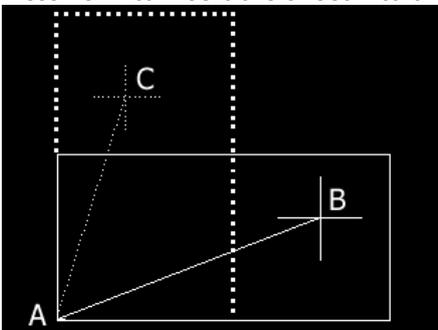
Subtext

Picture 570

This setting controls the height of the text placed on the sheet when using the "Lock sheet" command.

Insert sheet

Press "OK" to insert the sheet into the drawing.

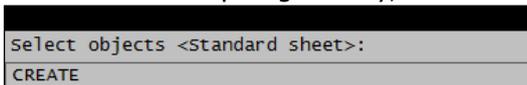


Picture 571

Pick the position for the lower left corner (A) then the orientation of the sheet (B) or (C).

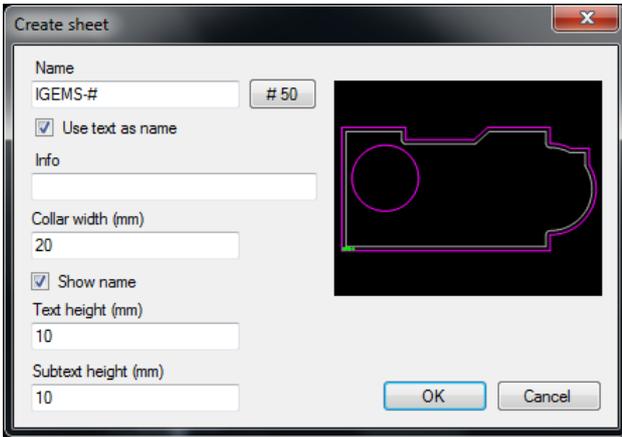
13.1.2 Irregular sheets

If a sheet has complex geometry, be sure that the geometries are closed before starting the command.



Picture 572

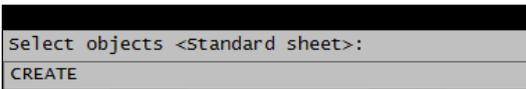
Select the objects that describe the sheet.



Picture 573

The sheet can contain holes and it is also possible to use the "Text as names" function. See page 77 for more information.

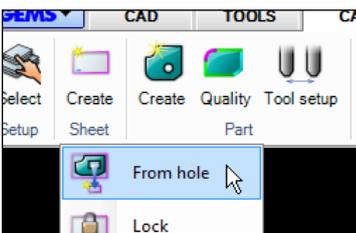
13.1.3 Edit sheet properties



Picture 574

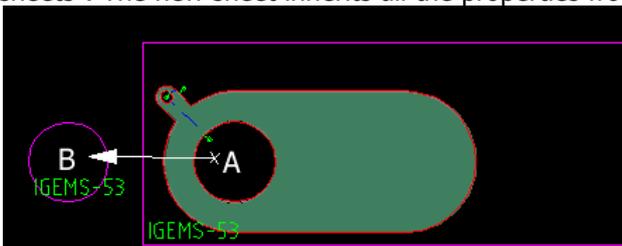
To change any parameters on an existing sheet use the same command to change the parameters. Select the sheet instead of geometry. It is also possible to double click on the sheet without starting the command.

13.2 From hole



Picture 575

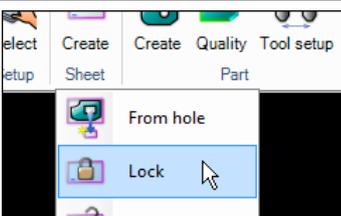
In parts with large internal contours, use this command to keep the internal areas and store as "rest sheets". The new sheet inherits all the properties from the original sheet.



Picture 576

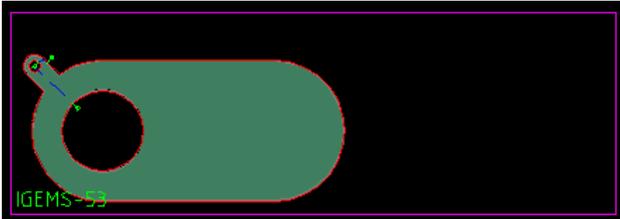
Start the command and click inside a hole in the part (A). This creates a new sheet with same geometry as the hole. Move the mouse and the sheet to a position (B). The new sheet (B) has the same properties as its parent.

13.3 Lock sheet



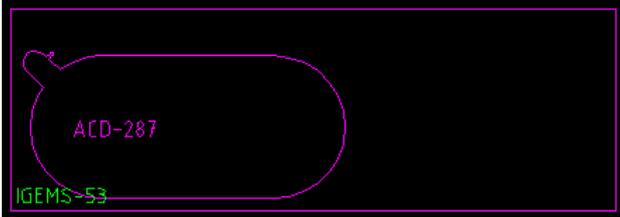
Picture 577

Use this function to convert parts located on sheets to holes.



Picture 578

The sheet now looks as follows:



Picture 579

The external contour and the toolpath creates a hole in the sheet. The text located in the hole is the name of the part. This text height is controlled by the "Subtext height" in the "Sheet create command". See page 135 for more information.

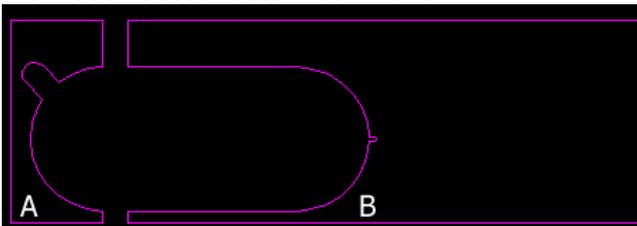


Picture 580

The exact geometry of the part and the piercing geometry (A) are shown on the sheet.

Automatic divide

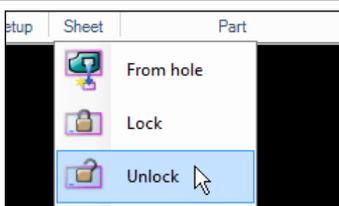
If a cutoff crosses the sheet, then the sheet automatically will be divided into two. See page 105 for more information about "Cutoff" command.



Picture 581

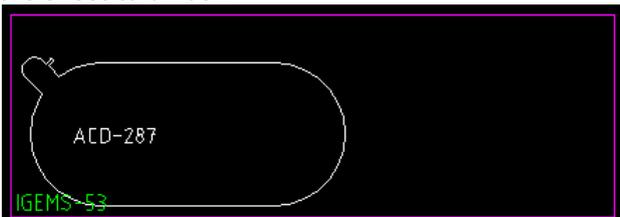
Both sheets (A) and (B) have the same properties.

13.4 Unlock



Picture 582

To modify the position of the part on the sheet, use the "Unlock" command. Start the command and select the sheet to unlock.



Picture 583

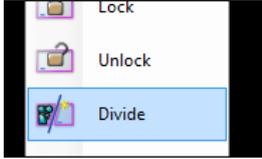
Now move the geometry on the sheet. When the geometry is in the correct position, lock the sheet again.

- ? Why is it important to move the geometry afterwards?
It is only important if there is some space left on the sheet and you plan to use the sheet again.

? Why can the hole be in the wrong position?

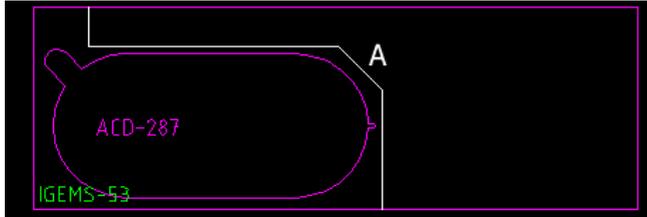
For example: If the sheet was not aligned correctly on the machine or the zero point was wrong. Use "rotate" or "move" to correct this.

13.5 Divide



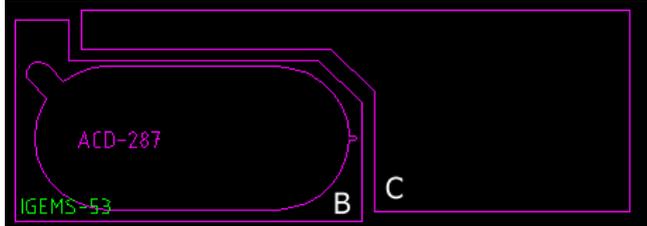
Picture 584

The "Lock sheet" command automatically divides a sheet into several parts. To divide a sheet manually, use the "Divide" command.



Picture 585

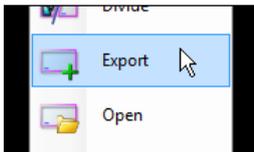
Make a polyline that describes how to divide the sheet.



Picture 586

The command creates two sheets (B) and (C) with the same properties. The "Divide" command also works with toolpaths from the "Cutoff" command. See page 105.

13.6 Sheet export

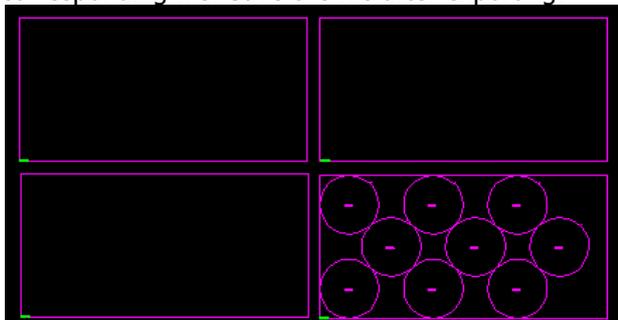


Picture 587

IGEMS has an excellent way to handle the library of sheets and rest sheets.

For every material there is a corresponding DIG file that contains the sheets. If the actual material is: "Stainless Steel" quality "304" and thickness 10.00, then the corresponding "DIG" file is "Stainless Steel_304_10.00 mm.dig".

Use this command to export Sheets, rest sheets, parts or locked sheets from the drawing into the corresponding file. Save the file after exporting.

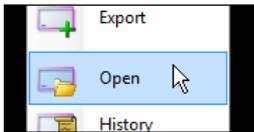


Picture 588

If the corresponding file does not exist then IGEMS will create a file. All sheet files are normal IGEMS drawing files located in the Sheet directory in the Shared folder. See Section 3.6 for more information about the shared folder). The file will contain all rest sheets.

- ? Should I lock the sheet before I use the "Export" command?
Do this before or after.
- ? Can I make the nesting and toolpath in the Sheet file?
Yes, handle this file as any other file.

13.7 Sheet open



Picture 589

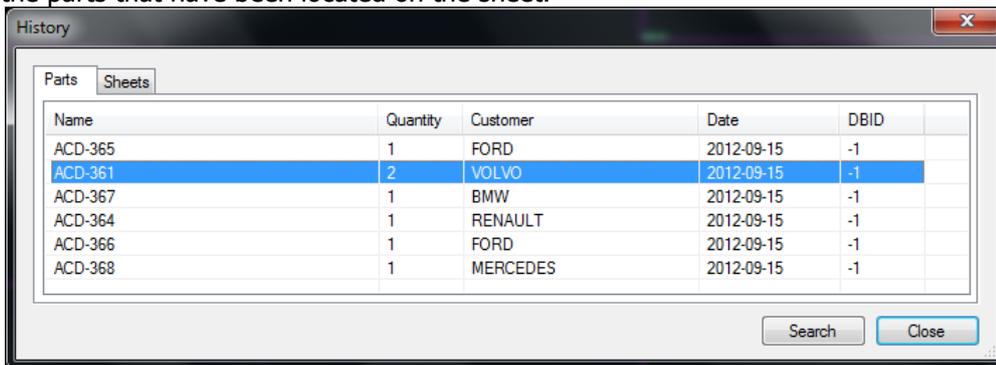
This command opens the corresponding file but will not export any information. See Section 13.6. Use this command to look in the rest sheet library.

13.8 History



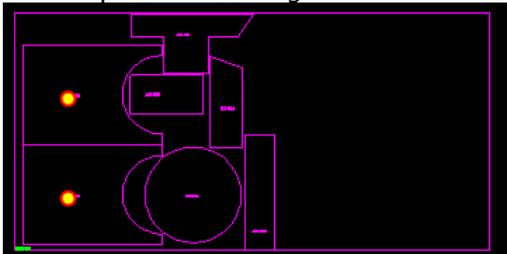
Picture 590

This command asks for the selection of one or several sheets. It then shows information about sheets and the parts that have been located on the sheet.



Picture 591

Select a part in the dialog box and the location will be highlighted on the sheet.

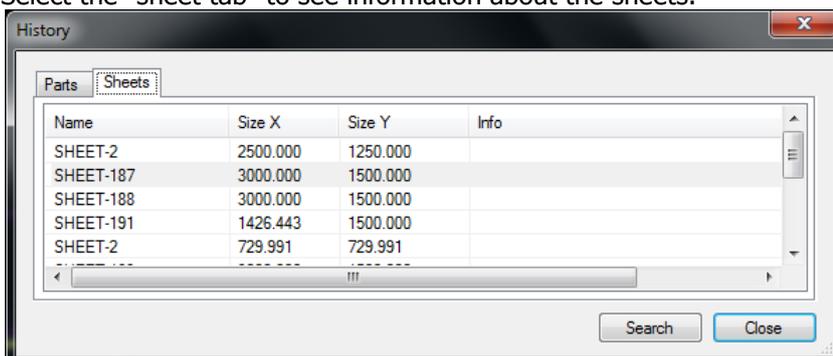


Picture 592

If the parts were registered in the "Organizer" module then the DBID flag has a positive number. Double click on that line and the "Organizer" shows more detailed information about the part.

Sheets

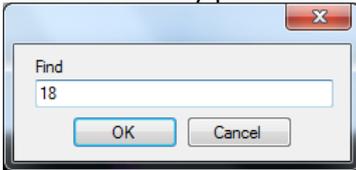
Select the "sheet tab" to see information about the sheets.



Picture 593

Search

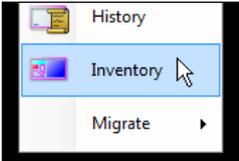
If there are many parts and sheets use the Search function to find the information.



Picture 594

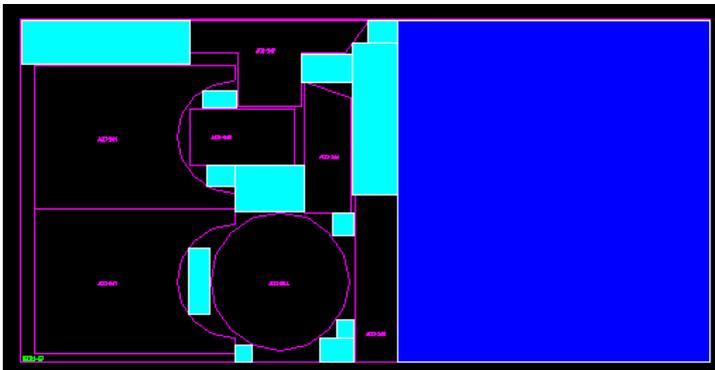
Use the "Search" function to make a search on all.

13.9 Inventory



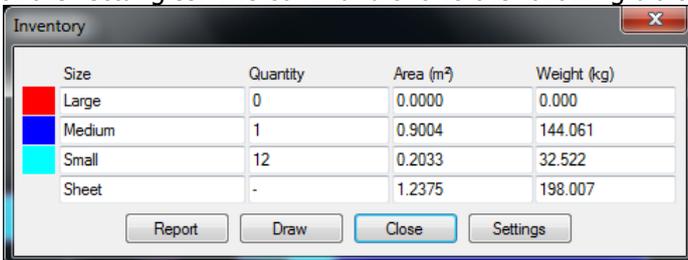
Picture 595

"Inventory" locates rectangular areas on a locked sheet and gives information on area and weight of the rest of the material based on the size. Different colors show the categorized size of each rectangle.



Picture 596

Picture 596 above shows the screen after using "Inventory". Areas are different colors depending on the size of the rectangles. The command shows the following dialog box:

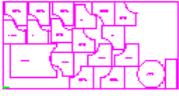
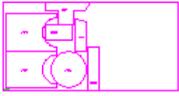


Picture 597

The following options are available in the "Inventory" command:

Reports

This option creates a full report on selected sheets.

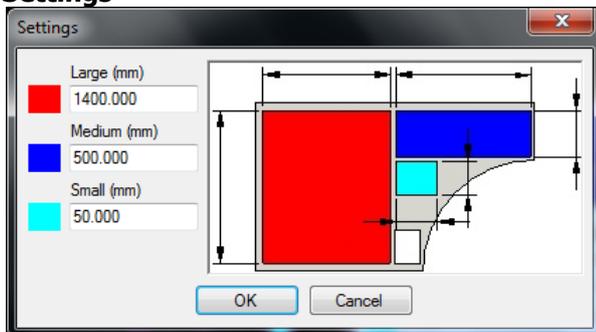
Sheet inventory			
Material	Stainless Steel/304 20.00 mm mm		
Large	1400.0 mm		
Medium	500.0 mm		
Small	50.0 mm		
		Area m ²	Weight kg
	SHEET-1 2000.0x1001.3		
	Large	0.000	0.000
	Medium	0.000	0.000
	Small	0.642	102.787
	Sheet	0.797	127.569
	IGEMS-57 2000.0x1001.0		
	Large	0.000	0.000
	Medium	0.900	144.061
	Small	0.203	32.522
	Sheet	1.238	198.007
Total	Large	0.000	0.000
	Medium	0.900	144.061
	Small	0.846	135.309
	Sheet	2.035	325.576

Picture 598

Draw

This command creates rectangles around the different colors.

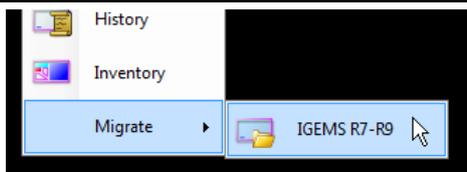
Settings



Picture 599

The "Settings" option defines the categorized size of the areas used in the Inventory.

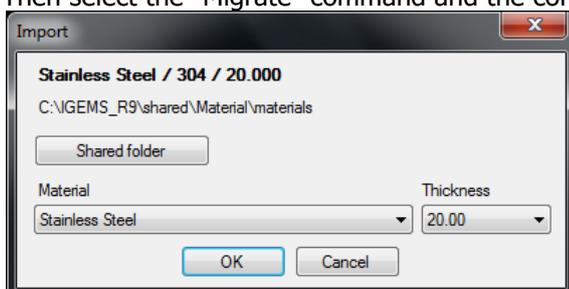
13.10 Migrate



Picture 600

Use the "Migrate" command to migrate sheets/rest sheets previously stored in the "Organizer" from older IGEMS versions into IGEMS R10. Before using this command run the "Open" command in the sheet menu and select the name of material and thickness.

Then select the "Migrate" command and the correct "Shared" folder to import from.



Picture 601

After the migration is completed, the previously stored sheets are imported into the IGEMS R10 sheet files.

14 CAM - Process

This chapter contains detailed information about the commands and possibilities for postprocessing.

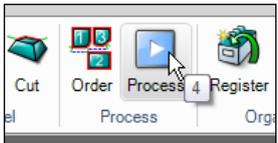


Picture 602

Prepare something to process before starting the postprocessing.

- ☞ Create a closed geometry to be cut.
- ☞ Create the part.
- ☞ Add a toolpath using the "Auto" command.

14.1 Process



Picture 603

The postprocessor system in IGEMS R10 is built around our internal LISP language called ILISP. The postprocessor is used for creating CNC-files, estimate cutting times, making reports and simulations.

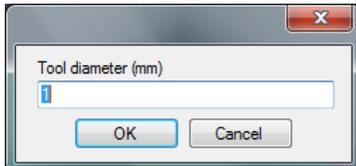
- ☞ Press the "Process" button
- The "Process" command needs a cut order to continue.



Picture 604

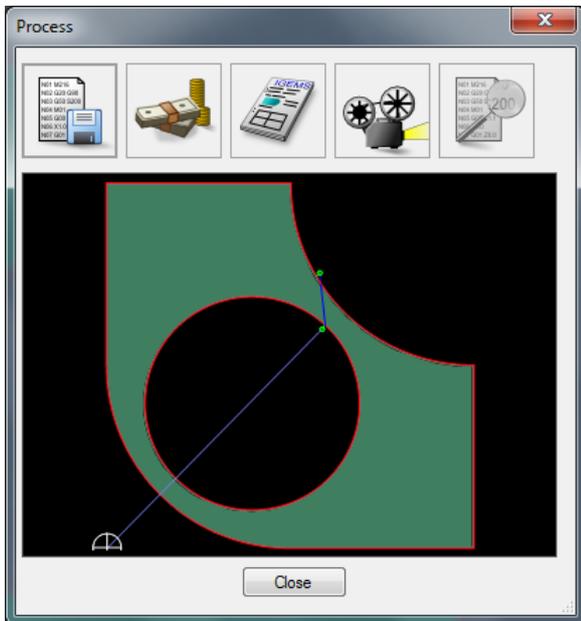
- ☞ Select the parts to cut.
- This creates a cut order with the default settings from the "Order" command.

The tool radius compensation (kerf) can be calculated in IGEMS or at the CNC-controller. If the kerf is calculated in IGEMS then there is the following dialog box about the Tool diameter to use:



Picture 605

IGEMS automatically calculates the toolpath to obtain the correct measurements of the job. If the kerf needs to be calculated in the CNC-controller there will not be this question.



Picture 606

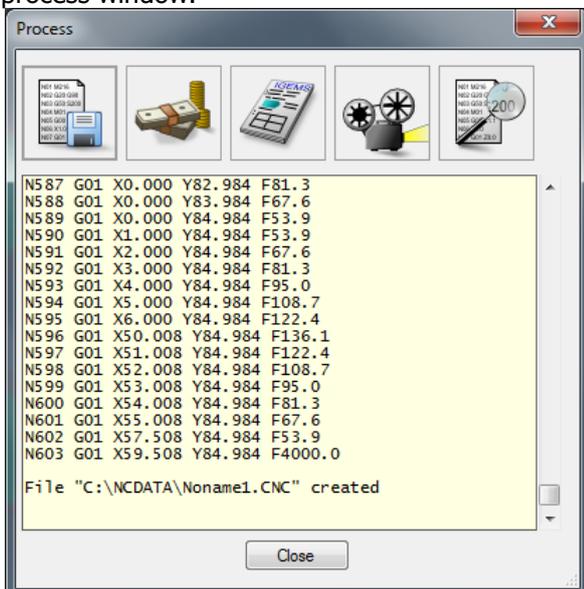
The postprocessor window shows the selected parts.
The process window has the following choices:

14.1.1 Create CNC-file



Picture 607

- Press the "Process" button.
The postprocessor ask for the name of the CNC-file. When the file has been created it is shown in the process window.



Picture 608

- ? How can I change to another postprocessor?
The postprocessor is controlled by the machine setup. To make a file for another machine then change to another machine in the "Select" command.
- ? Can I modify or make my own postprocessor?
Yes, modify or make new postprocessors or make applications. Contact IGEMS Software for more information.

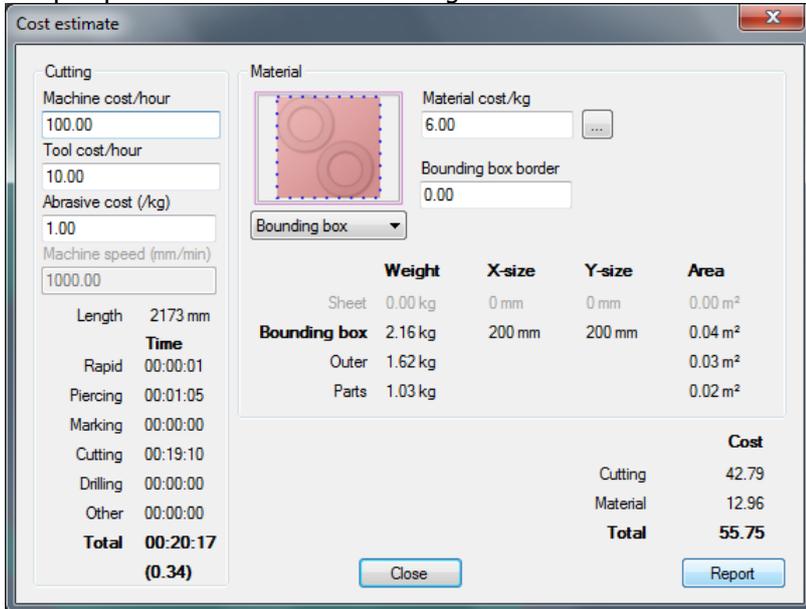
14.1.2 Cost estimate



The "Cost estimate" calculates a cutting time, the cost for the cutting and the cost for the material.

Press the "Cost estimate" button

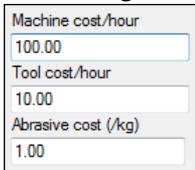
The postprocessor shows the following:



Picture 609

Cutting cost

The cutting cost for an AWJ machine depends on the following cost parameters:



Picture 610

Machine cost/hour

This cost is the general cost for the machine equipment

Tool cost/hour

To use a machine with several cutting heads is more expensive than using one head. On the other hand it cut double when using two heads. To make it possible to calculate this IGEMS needs to know the cost for each head. The hourly cost should be the sum of the following:

Water + Orifice + Mixing tube + Other costs related to the head

Abrasive cost

This cost should include cost for deposit or recycling. The cost for the grid can also be added to the abrasive cost.

Material cost

The material cost is loaded from the quality level in the material database.

Material

Material cost/kg
6.00

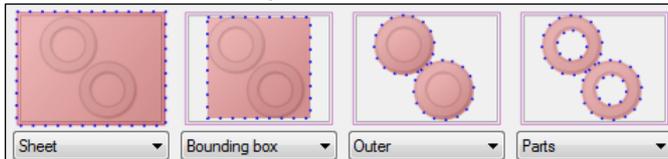
Bounding box border
0.00

Bounding box

	Weight	X-size	Y-size	Area
Sheet	0.00 kg	0 mm	0 mm	0.00 m ²
Bounding box	2.16 kg	200 mm	200 mm	0.04 m²
Outer	1.62 kg			0.03 m ²
Parts	1.03 kg			0.02 m ²

Picture 611

There are different ways to calculate the area for the material cost:



Picture 612

Sheet

This option uses the whole sheet. The option is only available if the cut order is placed on a sheet.

Bounding box

This option uses the area of a bounding box around the parts. When using this option add an extra border around the area.

Outer

This option calculates the area of all parts. It will not reduce the area of the holes in the parts.

Parts

This option calculates the net area of all parts.

Material cost/kg
6.00

Bounding box border
0.00

Picture 613

Material cost/kg

Click on the "dot" button to change the units to square meters.

Bounding box border

This option is only available when the "Bounding box" option is selected.

	Cost
Cutting	42.79
Material	12.96
Total	55.75

Close Report

Picture 614

The bottom of the window shows the total cost of the job.



Press the "Report" button

This shows a report that can be printed out and used as base for a quotation.

? Are the values calculated here the price or the cost?

That is up to you.

? Can the report be customized?

Yes, Chapter X will contain more information about the reports.

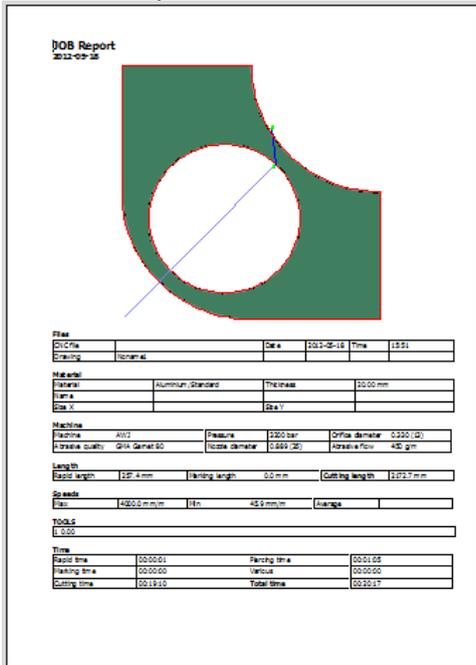
14.1.3 Job report



Picture 615

Use this command to make a job report.

Press the "Report" button



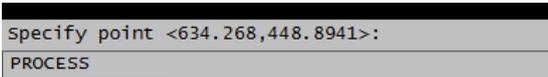
Picture 616 Chapter X will have more information about the job reports.

14.1.4 Simulation

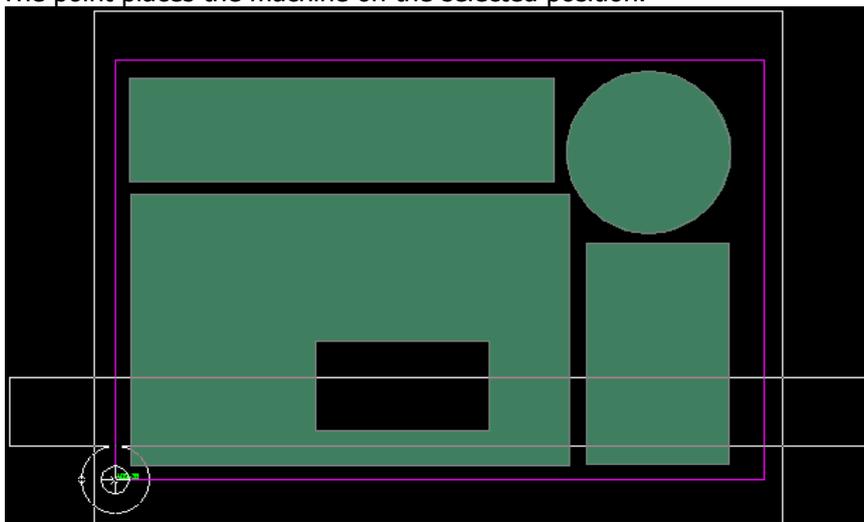


The "Simulation" command visualizes the motions as they will come out in the machine. The simulation will show a simple layout of the machine with cutting heads, drilling and marking units.

Press the "Simulate" command



Picture 618 The point places the machine on the selected position.

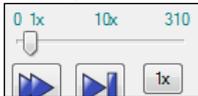


Picture 619 Use these different buttons to play, stop and go forwards or backwards one movement or one geometry at a time.



Picture 620

The right slider controls the simulation speed in play mode.



Picture 621

The "1x" button sets the simulation speed to 1. The simulation then takes the same time as the cutting time.

14.1.5 View code

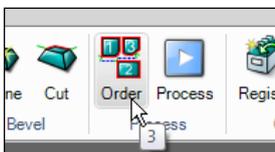


Picture 622

This command opens the output file by using Notepad.

Note! Create a CNC file before pressing this button.

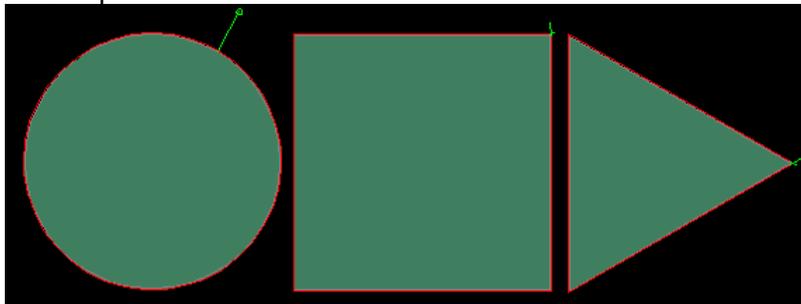
14.2 Order



Picture 623

If cutting only one part or if the cutting order between parts is not important, then skip this command and use the "Process" command directly. Prepare parts before starting.

- ☞ Start a new drawing and create three parts in the form of a disk, square and a triangle.
- ☞ Add toolpath.



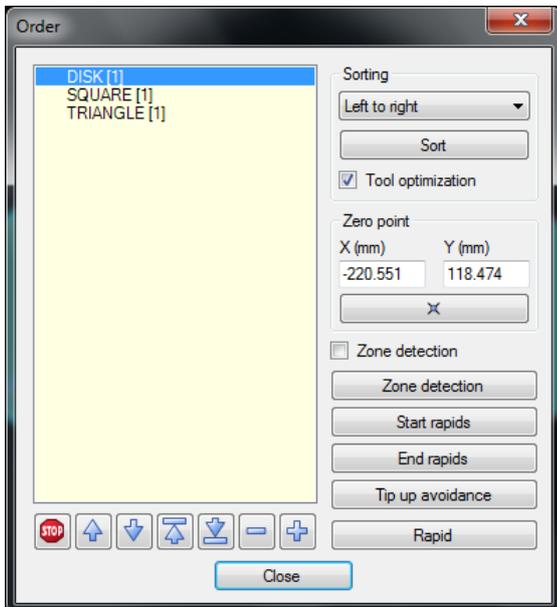
Picture 624

- ☞ Start the "Order" command.
- The following is shown:



Picture 625

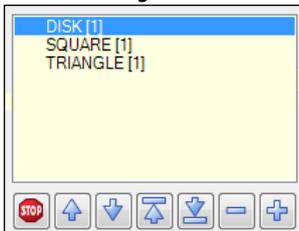
- ☞ Select the parts. The following window is shown:



Picture 626

? What does the number in parentheses after the part name mean?
This is the tool number that will be used to cut the part.

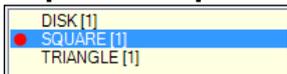
Manual adjustment of the cut order



Picture 627

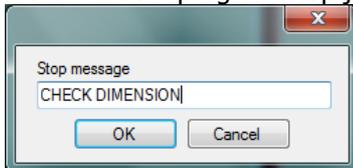
The list shows the cutting order of the parts. The DISK will be cut first and the TRIANGLE will be the last part cut. To change the cutting order use the "arrow" buttons to move the parts up or down. To remove or add parts to the cut order use the "plus" or "minus" buttons.

Stop between parts



Picture 628

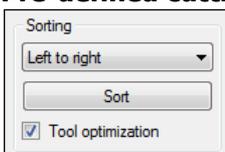
Use the "Stop" button to add a program stop in front of the parts. If the machine supports "stops" then there will be a program stop just before this part is cut.



Picture 629

If the machine supports the feature to give a message to the machine operator, then the message can be a helpful hint.

Pre-defined cutting orders

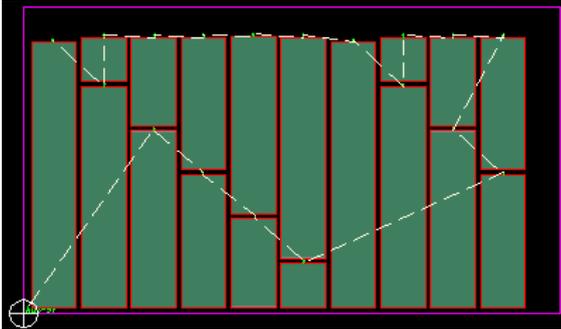


Picture 630

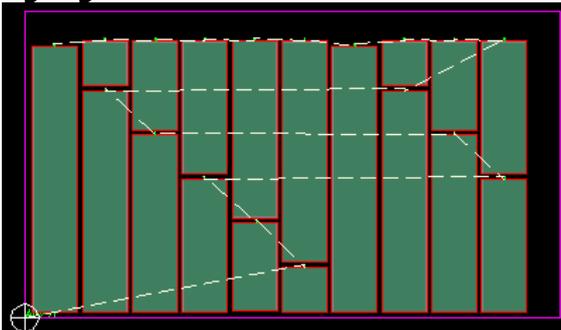
When cutting many parts use pre-defined methods to set up the cut order. The result of the "strategies" depends on the quantity and geometry of the parts and sheets. Select a strategy from the drop down list and press the "Sort" button to sort the cut order in the following ways:

Closest

This method looks from the start point to the next closest point and then to the next closest point again and so on. It optimizes the order to avoid long movements at the end.

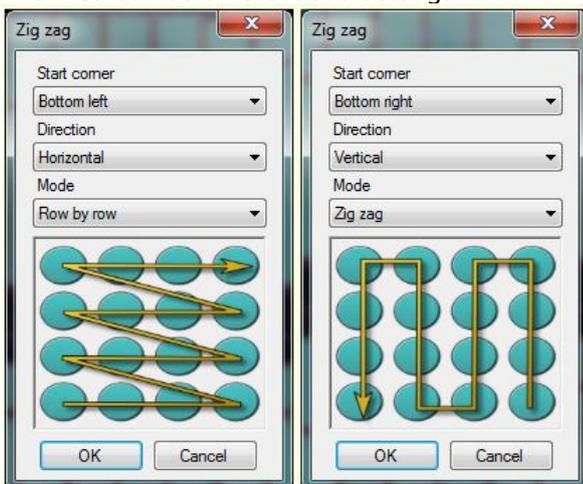


Picture 631

Zig zag

Picture 632

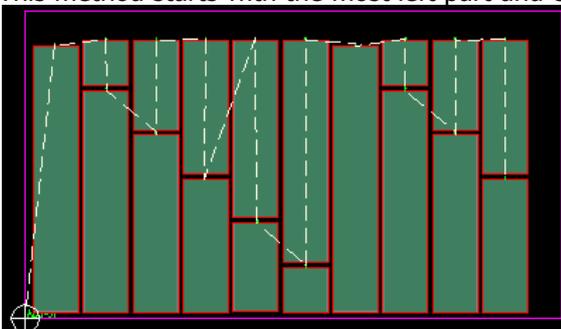
This method has different sub-settings:



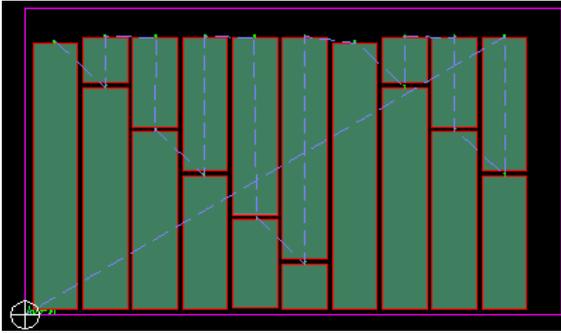
Picture 633

Left to right

This method starts with the most left part and ends with the most right parts.

**Right to Left**

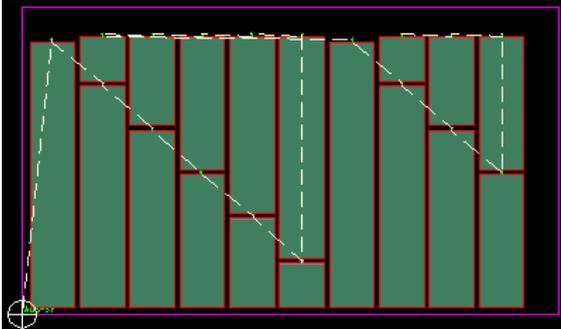
This method starts with the most right part and ends with the most left part.



Picture 634

As placed

This method takes the parts in the order they have been placed on the sheet.



Picture 635

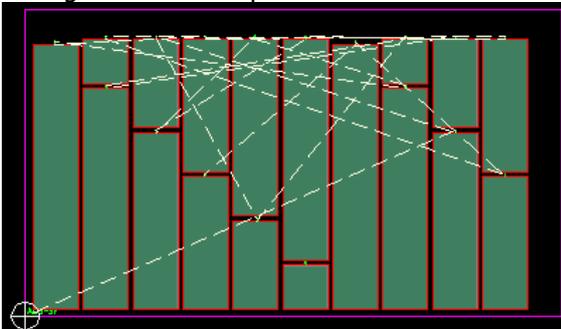
This method is mostly used together with other applications.

Manual

Use this method to select the parts in the order they should be cut.

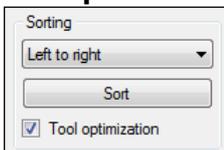
Spread

This method takes the parts randomly and not close to each other. The method is mostly used by Plasma cutting machines to spread the heat out over the whole sheet.



Picture 636

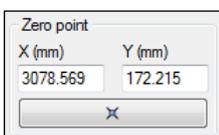
Tool optimization



Picture 637

This example uses only one tool. If a machine is equipped with several tools then this feature is useful since it keeps parts together that use the same tools resulting in less tool changes. Read more about cutting with multiple tools in CHAPTER X.

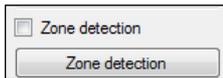
14.2.1 Zero point



Picture 638

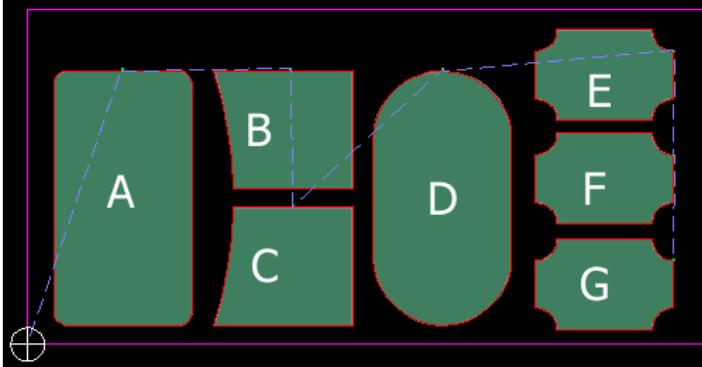
The program has a default location of the zero point that is setup in the "Strategy". Click on this button or enter coordinates to change the zero point to an optional point.

14.2.2 Zone detection



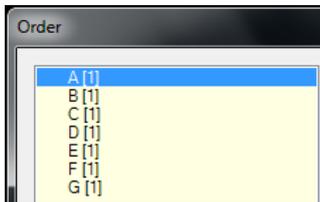
Picture 639

This feature divides the material into working zones with the equal number of tools the machine is equipped with. If zone detection is not activated all parts are cut with tool number 1.



Picture 640

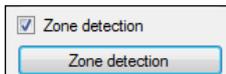
The list will look like this:



Picture 641

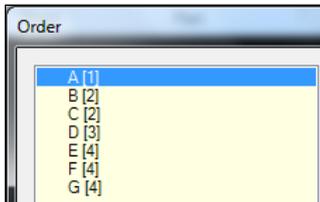
The number inside the parentheses shows the tool number to be used.

Activate the checkbox and then press the button to apply. Tool 1 primarily works in zone 1 and tool 4 in zone 4 as illustrated in the below pictures:



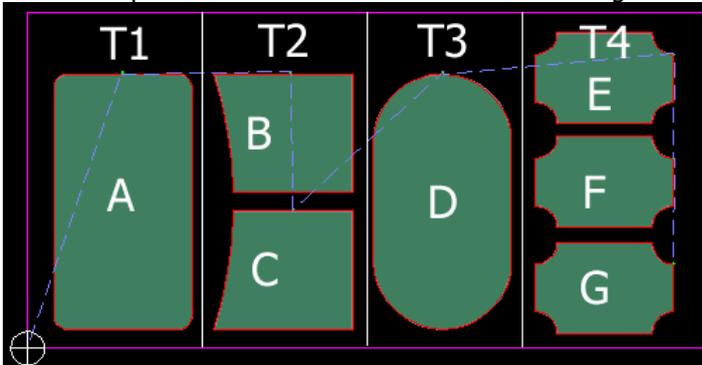
Picture 642

The list will look like this:



Picture 643

The above picture shows how tools 1-2-3-4 are assigned with the zone detection feature.



Picture 644

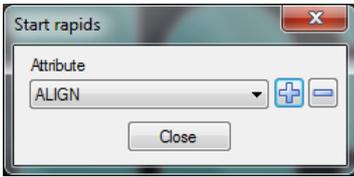
14.2.3 Start rapids

"Start rapids" defines extra rapids from the zero point to the first contour. This feature supports special sheet alignment tools.



Picture 645

Add different "Attributes" for the postprocessor to support different options.



Picture 646

Note! The postprocessor must be specially designed to support this kind of feature.

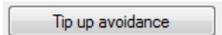
14.2.4 End rapids

When it is necessary to make a final movement of the tool away from the material, click on the "End rapids" button.



Picture 647

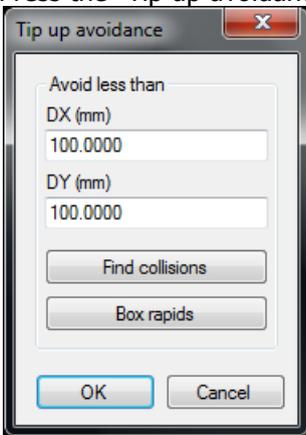
14.2.5 Tip up avoidance



Picture 648

Use this button to detect and avoid a collision and redefine a rapid transportation to avoid crossing parts that are smaller than a specified size in X and Y.

☞ Press the "Tip up avoidance" button

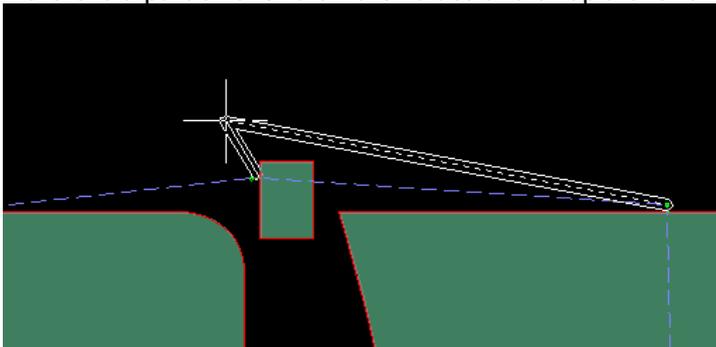


Picture 649

Define the smallest parts to avoid by entering the values for DX and DY.

☞ Click on the "Find collisions" button

If there is a part smaller than the values and a rapid over this part, then define a way around this part.



Picture 650

Click the "Find collisions" button to redefine a rapid toolpath around parts that are smaller from what is defined in the DX, DY.

Box rapid

This method automatically generates extra rapids to avoid moving over existing parts. For best results the cut order should be sorted from left to right and the internal cut order on the part should also be sorted in left to right.

? When should this be used?

It is not possible on some older machines to lift the nozzle between each cut. In that case it is important to move around parts to avoid a crash. On modern machines it is not necessary to use this feature.

14.2.6 Rapids

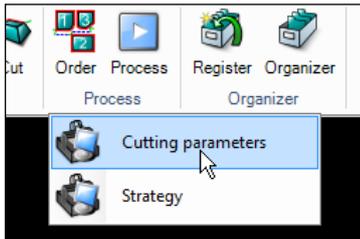


Picture 651

Click this button to add extra rapids to the selected part on the list.

- ? When should this be used?
It makes it possible to move around clamps and other things.

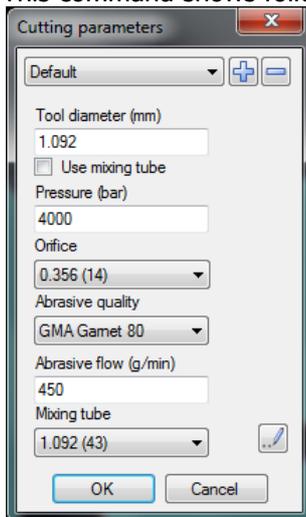
15 Cutting parameters and Strategy



Picture 652

15.1 Cutting parameters

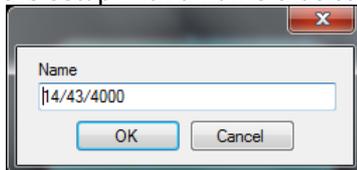
This command shows following dialog box:



Picture 653

Plus button

Press the "plus" button to create a new parameter setup. IGEMS asks for a name for the new setup. Save the setup with a name that corresponds to the size of orifice and mixing tube.

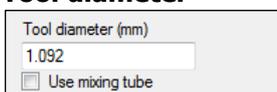


Picture 654

Minus button

This button will delete actual parameter setup.

Tool diameter



Picture 655

This is the actual "tool diameter" used as default when IGEMS calculates the tool radius compensation. It is also default value for the part distance when using commands for common cutline. If the checkbox "Use mixing tube" is active then the tool diameter automatically is the same as the diameter of the mixing tube.

AWJ settings

Picture 656

The "Pressure" value should be the pressure the pump is set to (the pressure loss is handled by IGEMS). All these settings are used in the mathematical formula to calculate the cutting speed.

Click the "Lamp" button to have some hints of good combinations of Nozzle/Orifice/Abrasive.

Nozzle	Orifice	Abrasive flow
0.179 [7]	0.508 [20]	100-250g/min [0.220-0.551 lbs]
0.204 [8]	0.508 [20]	125-300g/min [0.276-0.661 lbs]
0.229 [9]	0.762 [30]	150-400g/min [0.331-0.882 lbs]
0.254 [10]	0.762 [30]	200-450g/min [0.441-0.992 lbs]
0.279 [11]	0.762 [30]	250-475g/min [0.551-1.047 lbs]
0.304 [12]	1.016 [40]	300-500g/min [0.661-1.102 lbs]
0.330 [13]	1.016 [40]	400-600g/min [0.882-1.323 lbs]
0.356 [14]	1.093 [43]	450-650g/min [0.992-1.433 lbs]
0.381 [15]	1.270 [50]	500-750g/min [1.102-1.653 lbs]
0.406 [16]	1.270 [50]	550-800g/min [1.213-1.764 lbs]

Picture 657

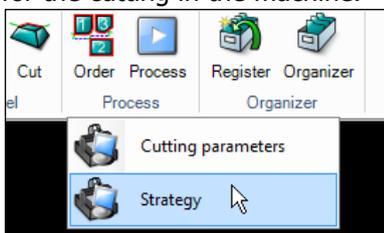
? Where do I select the parameters I want to use?

When you select machine and material in the "select" command then you also select cutting parameters.

Picture 658

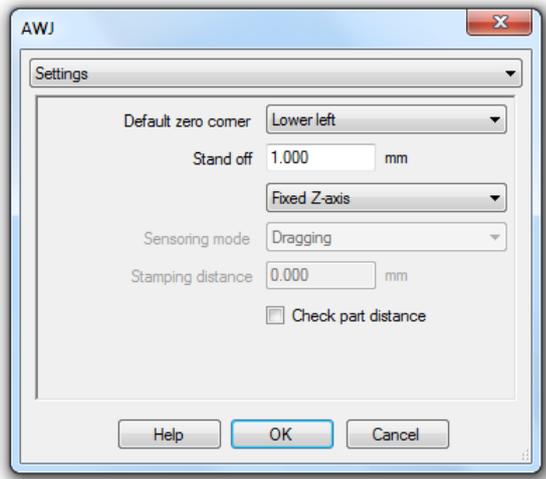
15.2 Strategy

The "Strategy" command affects how the postprocessor generates the CNC-file, the strategy and workflow for the cutting in the machine.



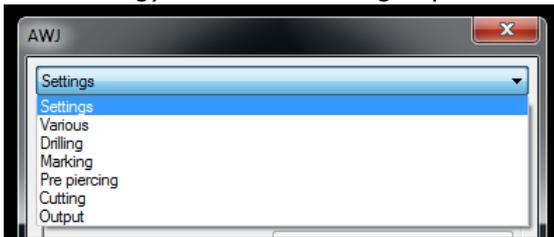
Picture 659

Press the "Strategy" button.



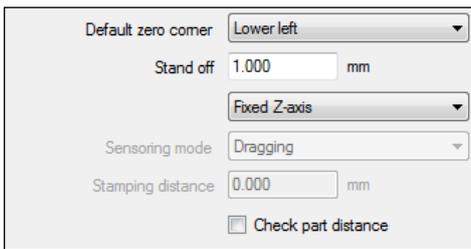
Picture 660

The "Strategy" is divided into 7 groups. Select the group in the drop down list located at the top:



Picture 661

15.2.1 Settings



Picture 662

Default zero corner

When a cut order is created then the program zero point is automatically set to the default corner. Lower left, Lower right, Upper left and Upper right.



Picture 663

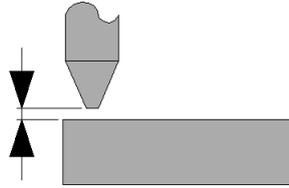
If there is a sheet it will be located at the corner of the sheet; if there is no sheet it will be located at the corner of a bounding including all parts.

? Can I change the zero point to an optional point?

Yes, please see the "Order" command.

Stand off

This value or feature is not implemented as standard on all postprocessors.



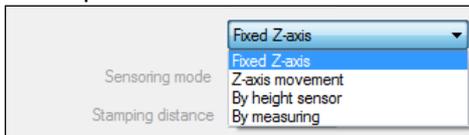
Picture 664

The value is used to control the distance between the end of the nozzle to the material.

Z-axis mode

In the machine setup select up to 4 different Z-axis modes. Only the options available for the machine are in this list.

The implementation of these modes can be different depending of the machine supplier.



Picture 665

Fixed Z-axis

This method is used to not have any Z-axis movement at all.

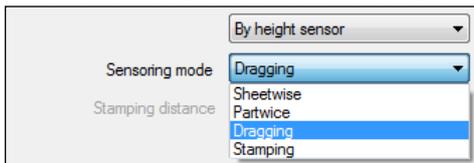
CNC-controlled

Use this method to move the Z-axis up and down using coordinates.

By height sensor

Use this method to use the height sensor.

When this mode is activated choose one of four different "Sensing modes":



Picture 666

Sheetwise

The height sensor measures the height once in the beginning of the process and not again.

Partwise

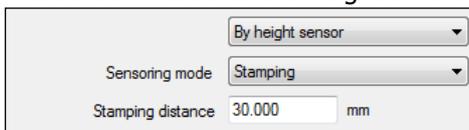
The sensor measures the height once when starting to cut on a new part.

Dragging

The sensor measures the height constantly and the sensor ring is dragged on the material.

Stamping

The sensor measures the height on a time period or a cutting distance.



Picture 667

The time period is calculated from the "Stamping distance", which is the time it takes to cut this distance in medium quality.

By measuring

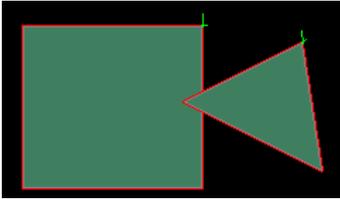
Use this method when there is a separate height measuring system. This is mostly used on 5-axis machines.

Check part distance



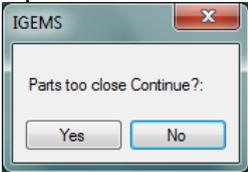
Picture 668

This feature checks the distance between all parts before starting the process.



Picture 669

If parts interfere with each other or are closer than tool distance there is the following message:



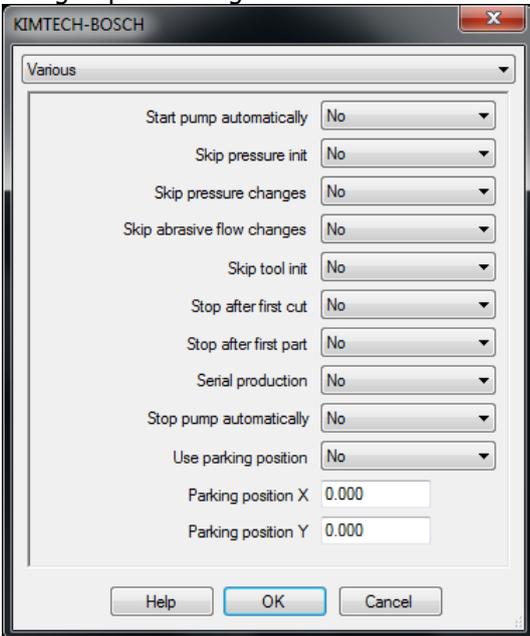
Picture 670

Modify the position of the part to avoid cutting damaged parts.

- ? Is there any reason to not having this feature always activated?
If you have extremely large jobs then this check can take some time.

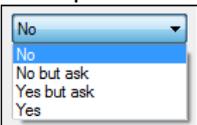
15.2.2 Various

This group of setting handles various functions in the postprocessor.



Picture 671

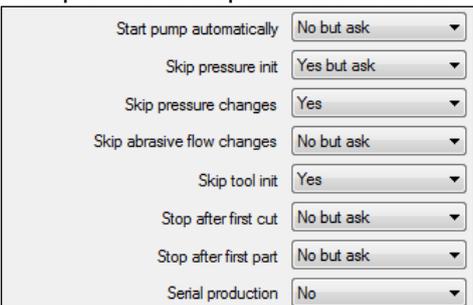
All drop down menus handle 4 different options:



Picture 672

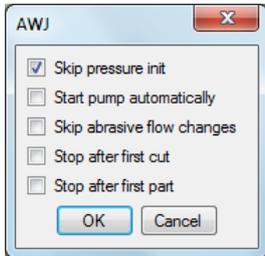
The choices "No" and "Yes" activate different options in the postprocessor. If "No but ask" or "Yes but ask" is used then there is an extra question during the postprocessing.

Example: If the drop downs are set as follows:



Picture 673

When the postprocessing starts the following dialog box is shown:



Picture 674

All options that were set to "No" or "Yes" are not visible.

All options that were set to "Yes but ask" are shown and checked.

All options that were set to "No but ask" are shown but not checked.

Start pump automatically

This option is only enabled if the machine can be turned on/off from the CNC-file.

Skip pressure init

There is always a pressure initialization if the start pressure is not the same as the cutting pressure. Set this value to "Yes" to always have a pressure initialization.

Skip pressure changes

If the job contains different pressures, then turn of the changes by setting the value to "Yes".

Skip abrasive flow changes

If the job contains different amount of abrasive then turn off this by setting the value to "Yes".

Skip tool init

If the job starts with several tools or not with T1 then there is always a tool distance initialization. To not have a tool distance initialization when starting with T1 only then set this option to "Yes".

Stop after first cut

This option provides a stop after the first geometry has been cut. The reason to have this stop is to check and adjust dimensions.

Stop after first part

This option provides a stop after the first geometry has been cut. The reason to have this stop is to check the part before starting to cut next parts.

Serial production

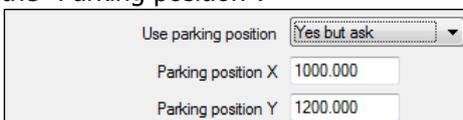
This option is only possible when cutting parts with high and low pressure. When checked the machine will end with low pressure ensuring that the pressure is correct when starting to cut with the same program again.

Stop pump automatically

This option is only enabled if the machine can be turned on/off from the CNC-file. The feature is helpful if the machine runs in unmanned mode during the night. The instruction in the CNC-file turns off the pump when the job is done.

Use parking position

This feature has the machine go to this position after the job has been done. The position will be taken from the "Parking position".



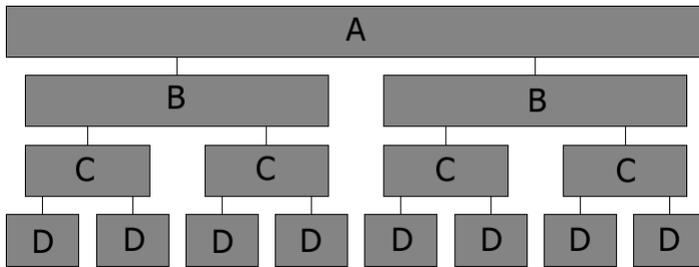
Picture 675

The coordinates are given from the "machine zero position" ensuring that the position will be the same no matter where the zero point is for the job.

15.2.3 IFPC IGEMS Flexible Process Control

Drilling, marking and pre-piercing can use a feature called IFPC (IGEMS Flexible Process Control)

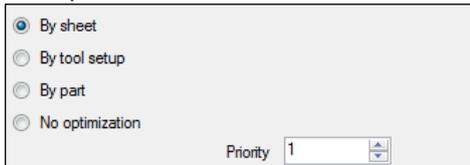
The IFPC controls when an operation should be executed. The process control can be used on drilling, marking and pre-piercings.



Picture 676

The "By sheet" (A) operations will always be executed first, then the "By tool setup" (B) operations followed by the "By part" (C) operations. The rest of the operations (D) will be done as the last step.

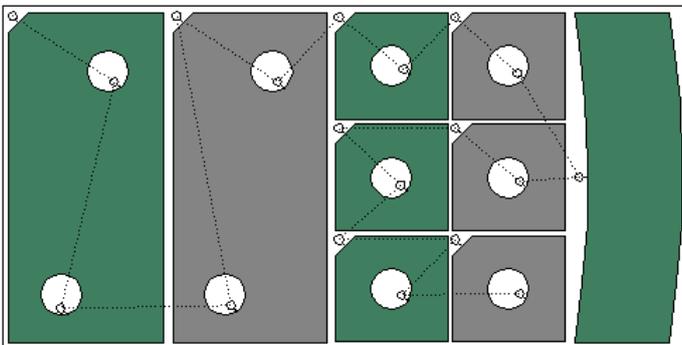
The options below are the same for drilling, marking and pre-piercing:



Picture 677

By sheet

With this option the operations will be done on the whole sheet before the other process is started.



Picture 678

When several kinds of operations (drilling, marking and pre-piercing) are set to "By sheet" then the internal order between the operations are controlled by the priority. The priority 1 will be executed before priorities 2 and 3.

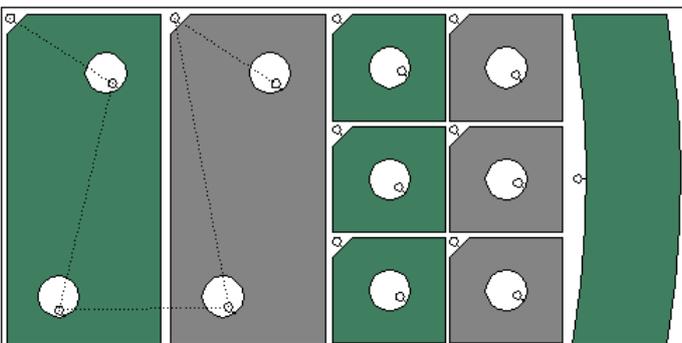
If the priority is set to the same for different operations then it will be executed internally in the order defined in the toolpath.

? Is there any typical machine configuration that should not use this optimizing?

The distance between the heads may need to be changed several times when using this method. If you don't have servo controlled tool distance on the machine then use "By tool setup" instead. That will save jobs for the operator.

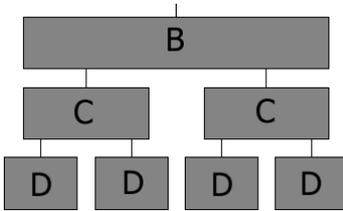
By tool setup

These options will be done after the "By sheet" step.



Picture 679

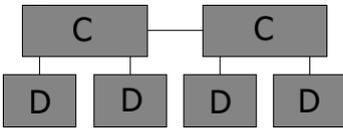
After the (B) operation is done the process goes to next step and makes all (C) and (D) operations.



Picture 680

If more than one type of operation is set to "By tool setup" then the internal order between the operations is controlled by the priority. The priority 1 will be executed before priorities 2 and 3.

By part



Picture 681

Use the "By part" to control the processing order of the remaining operations on a part level.

If more than one type of operation is set to "By tool setup" then the internal order between the operations is controlled by the priority. The priority 1 will be executed before priorities 2 and 3.

No optimization

Picture 682

The cutting process is always executed as the last operation. If another kind of operation (drilling, marking, pre-piercing) needs to be executed in the order the toolpath is added then select "No optimization".

Example 1:

First all drillings need to be made over the whole sheet, and then the markings made over the whole sheet. Then the pre-piercings are made over the whole sheet and finally the cutting is done last.

((drilling, drilling, drilling) (marking, marking, marking) (pre-piercing, pre-piercing, pre-piercing) (cutting, cutting and cutting))

Picture 683

Set drilling to "By sheet" and the priority to 1. Set the marking to "By sheet" and the priority to 2. Set the pre-piercing to "By sheet" and the priority to 3. The cutting will be done as the last step.

Example 2

In this example the optimization is as follows:

(Marking, pre-piercing, drilling and cutting together)

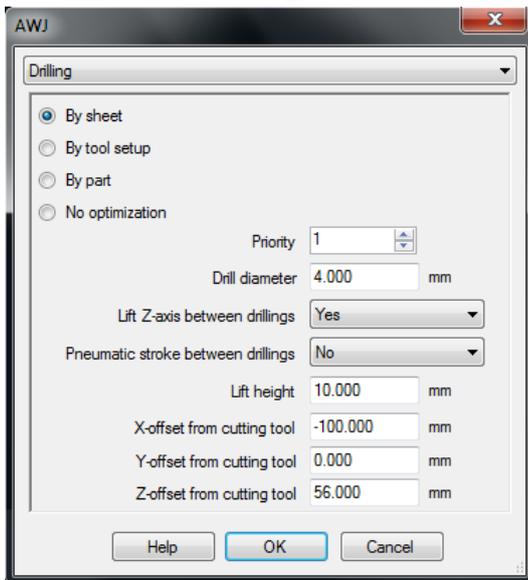
(Marking, pre-piercing, drilling and cutting together)

(Marking, pre-piercing, drilling and cutting together)

Picture 684

Set marking to "By part" priority 1. Set the pre-piercing to "By part" priority 2. Set the drilling to "No optimization".

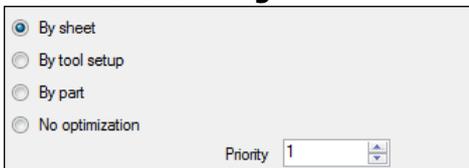
15.2.4 Drilling



Picture 685

If the machine does not have a drilling unit then this option is not in the "Strategy".

The IFPC for drilling



Picture 686

The IFPC controls when the drillings are executed in the process. The IFPC feature is described on page 159.

Drill diameter

When a drilling position thru a "Piercing" or the "Hole" command is added then this is represented as a circle on the screen. This setting controls the size of that circle.

Lift Z-axis between drillings

This option controls if the drilling tool is lifted between every hole.

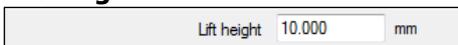
The drop down list is disabled if the machine does not have this option available.

Pneumatic stroke between drillings

This option controls if a pneumatic stroke up and down is done between every hole.

The drop down list is disabled if the machine does not have this option available.

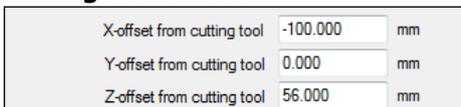
Lift height



Picture 687

This setting controls the lift height between each hole. If the machine does not have this possibility then lift height setting is disabled.

Drilling tool offset

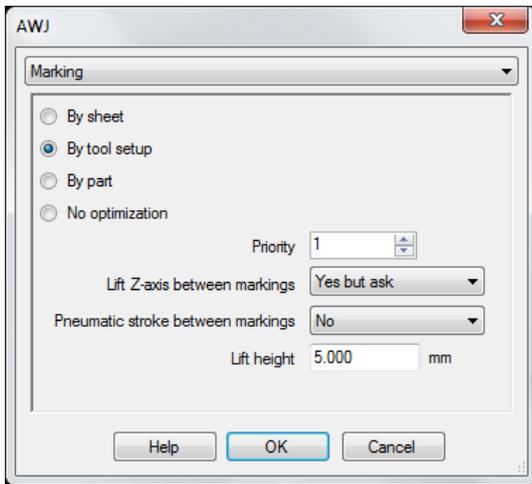


Picture 688

These values set up the distance between the cutting tool and the drill in X, Y and Z. On many machines this offset is controlled on the CNC-controller. These values are also used by the 2D-simulation to define the drilling tool position.

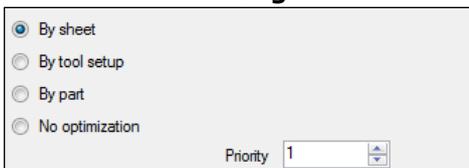
15.2.5 Marking

This option is only visible if there are any marking devices defined on the machine.



Picture 689

The IFPC for marking



Picture 690

The IFPC controls when the marking should be executed in the process. The IFPC feature is described on page 159.

Lift Z-axis between markings

This controls if the Z-axis is lifted between markings. This control is disabled if the machine cannot use this option. To move the marking tools up and down may take longer time then the marking itself.

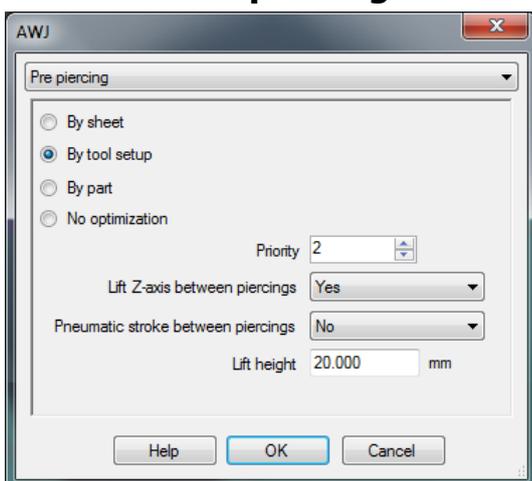
Pneumatic stroke between markings

This controls if "pneumatic stroke" should be used between the markings.

Lift height

If the machine has the possibility to move the marking tool to different heights then set this height here. The value is an incremental value from the position above the material.

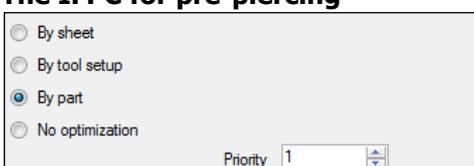
15.2.6 Pre-piercing



Picture 691

If the machine does not have possibilities to use marking then this option is not visible.

The IFPC for pre-piercing



Picture 692

The IFPC controls when the marking is executed in the process. The IFPC feature is described on page 159.

Lift Z-axis between piercings

This controls if the Z-axis is lifted between piercings. This control is disabled if the machine cannot use this option. If the IFPC makes all piercings before the cutting there is little risk to have the nozzle close to the material during pre-piercings.

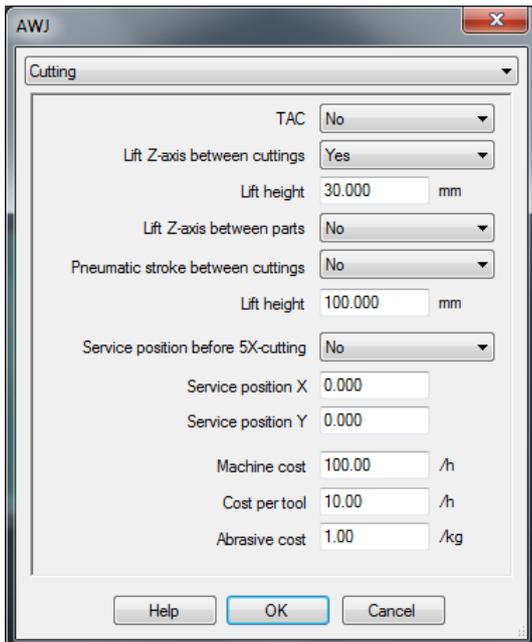
Pneumatic stroke between piercings

This option controls if a pneumatic stroke up and down is done between pre-piercing. The drop down list is disabled if the machine does not have this option available.

Lift height

If the machine has the possibility to move the marking tool to different heights then set this height here. The value is an incremental value from the position above the material.

15.2.7 Cutting



Picture 693

TAC

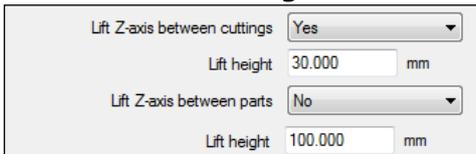
To use the TAC function the user must have a 5-axis cutting machine and a CNC-controller that can tilt the cutting head in different directions.



Picture 694

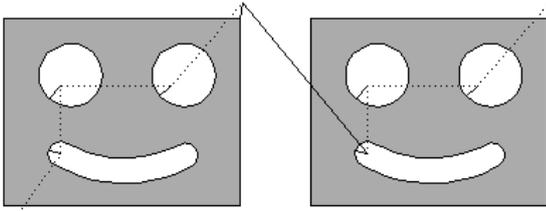
How much the nozzle should be tilted in different material can be set in the material database for each material. The functionality to use a 5-axis machine to obtain a straight cut has been used on plasma cutting machines for more than 20 years.

Lift Z between cuttings



Picture 695

If the machine cannot move the nozzle to different heights then this option is disabled. The lift settings are divided into two settings.



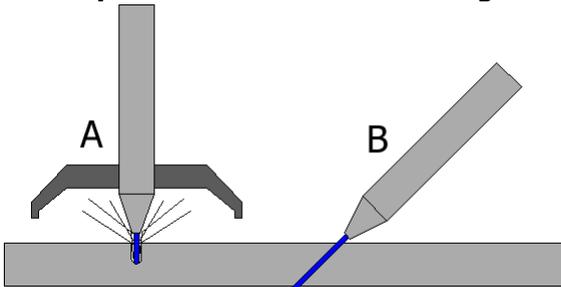
Picture 696

Control the lift height between different cuts in the parts and set the lift height between the parts. The line between the parts.

Pneumatic stroke between cuttings

This option controls if a pneumatic stroke up and down is done between pre-piercing. The drop down list is disabled if the machine does not have this option available.

Service position before 5-axis cutting



Picture 697

When cutting with tilted jets it is difficult to use a protector and is often not used in combination with 5-axis cutting. To cut without a protector will damage the equipment, add more noise and dust to the environment. Activate this feature to use a protector. The IFPC function will do all piercing before cutting starts. When it comes to the first tilted cutting then the machine will first go to the service position and stop. The operator can then remove the protector and restart the machine that will do the final 5-axis cutting.

Service position before 5X-cutting	Yes	▼
Service position X	0.000	
Service position Y	0.000	

Picture 698

The postprocessor uses these coordinates to write instruction for the movement. Use the machine coordinate system and the machine goes to the same position no matter where the local program zero point is located.

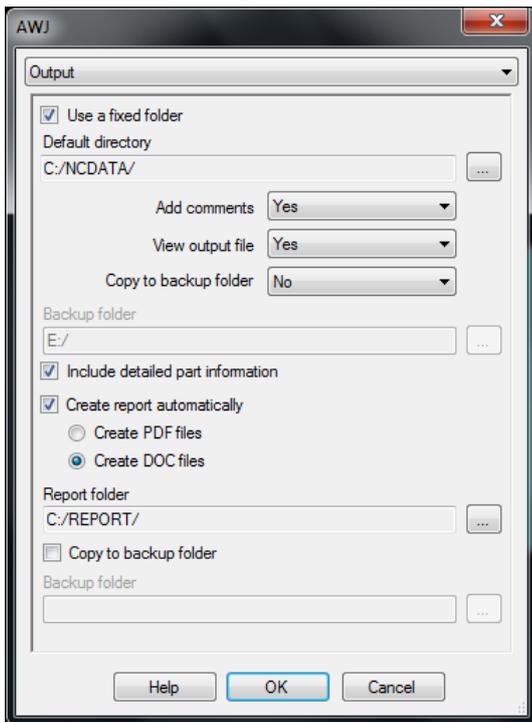
Costs

Machine cost	100.00	/h
Cost per tool	10.00	/h
Abrasive cost	1.00	/kg

Picture 699

"Cost per tool" and "Abrasive cost" is only available if the machine is an AWJ machine. The costs are used to calculate cutting costs in the "Cost estimate" function in the postprocessing.

15.2.8 Output



Picture 700

The location of the CNC-file



Picture 701

When using a fixed folder, IGEMS suggests the default directory every time a new CNC-file is created. If "Use a fixed folder" is not selected, then IGEMS remembers the last used directory.

Add comments

This setting controls if comments are added to the CNC-file.

N100 M03 (WATER ON)

N105 G04 F0.5

N110 M06 (ABRASIVE ON)

The text inside the parentheses is an example of comments that can be turned on or off if the postprocessor uses the provided MSG function.

- ? Is there any reason to not always add these comments?
It makes the CNC-file larger but that is probably not very important.

View output file

This "No-Yes" option handles if the CNC-file is viewed in the process window.

- ? Is there any reason to not always view the file?
Not really, the viewings of the file take a little longer time.

Copy to backup folder



Picture 702

This feature copies the final CNC-file to a backup folder or a USB-drive. If no USB is connected or if the backup folder does not exist then the procedure is skipped. There is not any disk error.

Settings for the reports

Include detailed part information

Create report automatically

Create PDF files

Create DOC files

Report folder
C:/REPORT/ ...

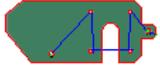
Copy to backup folder

Backup folder
C:/REPORT/ ...

Picture 703

Include detailed part information

This setting controls if the job report includes only a summary or more information about each part.

 <p>265.0 mm x 110.0 mm</p>	Name Customer Quantity Cutting length Single time Average time Total time Weight	PART-27 1 1007.9 mm 00:02:01 00:02:01 0.55 Kg
 <p>159.2 mm x 125.5 mm</p>	Name Customer Quantity Cutting length Single time Average time Total time Weight	PART-28 1 788.5 mm 00:01:32 00:01:32 0.37 Kg

Picture 704

The information above is only included if this function is activated.

Create report automatically

If this function is activated then it is not necessary to press the "Report" button to make a report as it is created automatically. Decide if the report should be generated as a DOC file or as a PDF file.

? Is there any benefit to use DOC file?

The DOC file can be opened and changed by Word, but the PDF file is read only.

Report folder

All reports are created in this directory:

Copy to backup folder

Backup folder
E:/ ...

Picture 705

This feature can copy the report file to a backup folder or an USB-drive. If no USB is connected, or if the backup folder does not exist, then the procedure is skipped. There is not any disk error.

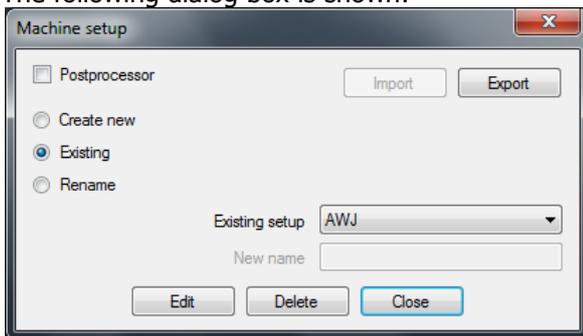
16 CAM – Machine setup

This chapter contains information how to setup the machine and the equipment. The "Machine setup" command is started from the "Machine" command in the Setup menu.



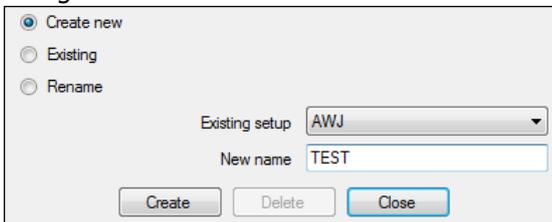
Picture 706

The following dialog box is shown:



Picture 707

NOTE! If you do not have enough knowledge about the machine you should not change any parameters on an existing machine. If you want to play with this setting then first make a test machine and make the changes in that machine.



Picture 708

Select "Create new" and set the new name to "TEST" and press the "Create" button.

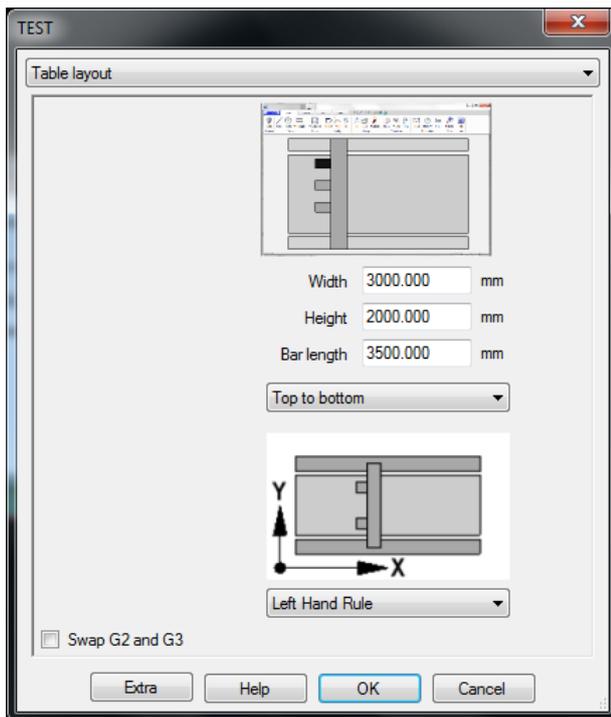


Picture 709

TEST is now the active machine.

Press the "Edit" button.

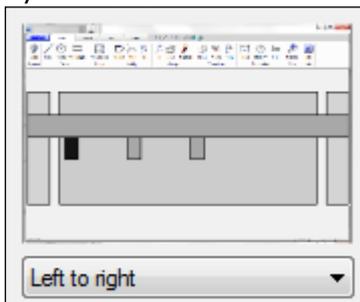
16.1 Table layout



Picture 710

Machine orientation on the screen

In IGEMS R10 the user defines the machine coordinate system independent on how the coordinate system is defined in IGEMS. Since new computer screens are much wider now it is recommend that the machine layout in IGEMS is defined so that the long side of the machine is rotated as the long side of the screen.



Picture 711

The black rectangle on the picture defines the first tool.



Picture 712

Width

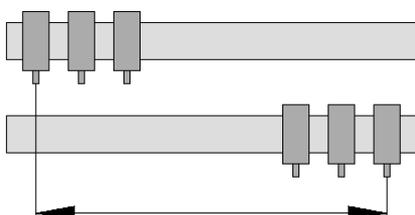
Enter the length of the side of the machine that is oriented horizontal on the screen.

Height

Enter the length of the side of the machine that is oriented vertical on the screen.

Bar length

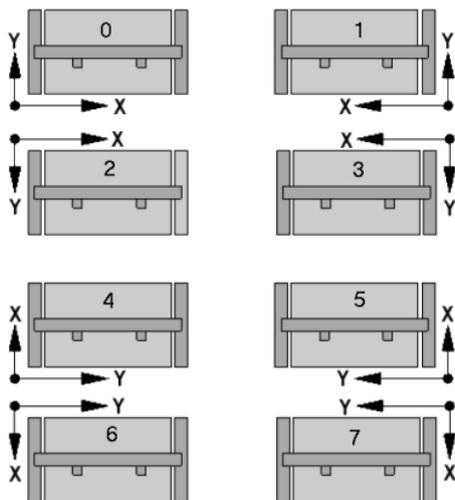
This value should be the maximum movement from the left tool in the most left position to the right tool in the most right position.



Picture 713

This value is used by the simulation functionality in the postprocessing.
 All the settings in the Table layout should now describe the fundamental dimensions of the machine.

Coordinate system



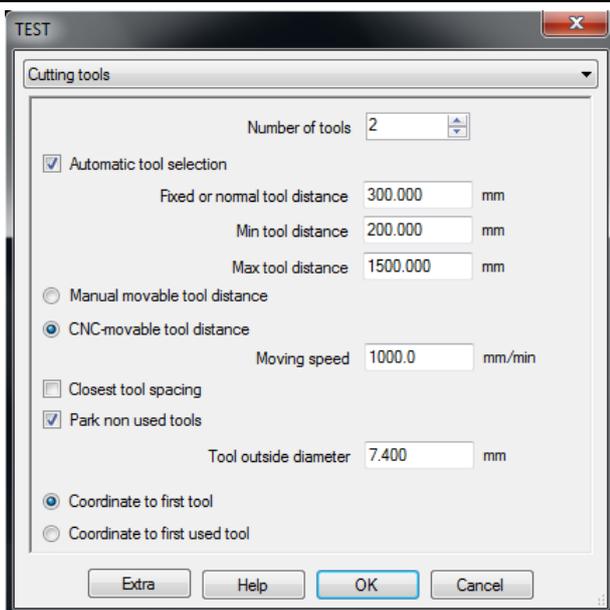
Picture 714

In IGEMS R10 there is no relation between the CAD coordinate system and the Machine coordinate system. Select the coordinate system according to the machine layout on the screen.

Swap G2 and G3

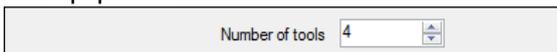
According to chosen coordinate system 0-7 the G2/G3 Arc directions are swapped automatically. To manually change the direction, activate this function.

16.2 Cutting tools



Picture 715

If the machine is equipped with more than one tool, then the postprocessor needs to know how to handle the equipment.



Picture 716

Enter the number of cutting tools. Do not include marking and drilling utilities.

16.2.1 Automatic tool selection



Picture 717

If a job handles different tools, the postprocessor needs to know how to instruct the machine as to which tool is on and off. Old machines may only have manual switches to select tools. If active tools can be changed by the CNC-file then activate this setting.

- ? If you don't have automatic tool selection what will then happen when changing tools?
Then the machine will stop (M00) and the operator must change tool manually.

16.2.2 To handle the distance between the tools

Picture 718

Fixed or normal distance

This value is used as default distance in the "Tool setup" command and in "Auto nest". Using a fixed distance is important for a machine that does not have the ability to change the tool distance.

Min tool distance

This value is used by the dynamic tool distance function in "Auto nest". "Auto nest" never uses a smaller distance than this value.

Max tool distance

This value is also used by the dynamic tool distance function in "Auto nest". "Auto nest" never uses a larger distance than this value.

Picture 719

Manual movable tool distance

Use this option to stop the machine so the operator can change tool distance manually.

CNC-movable tool distance

Select this option if the machine can move the tools on the cross beam automatically

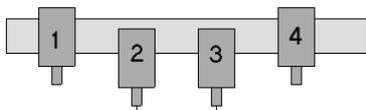
Moving speed

This is the speed that is used when the tools are moved on the cross beam if the tools can be moved by the CNC. The value can be used by the postprocessor and for the time calculations.

Picture 720

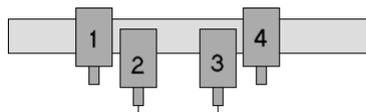
Closest tool spacing

This settings controls how non-used tools are packed on the cross beam.



If not activated then all tools on the beam will have the same distances.

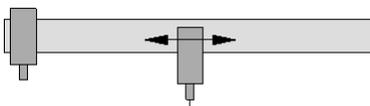
In this example T2 and T3 are used for cutting and the distance should be 500mm.



Picture 721

Park non-used tools

Some machine constructions put non-used tools in a parking position. This means that they are not moved on the cross beam if they are not in use. This information is used to make a more correct simulation.



Picture 722

Calculate tool offsets

 Coordinate to first tool
 Coordinate to first used tool

Picture 723

When using Tool 1 then the coordinates will always be calculated to this tool. When using another tool then this setting controls how the coordinates should be given.

Example:

When using tool number 3 only and the distance between each tool is 300mm the position is X1000:

Coordinates to first tool gives the value in the CNC-file: $1000-300-300=X400$

Coordinates to first used tools gives the value in the CNC-file: X1000

16.2.3 Tool outside diameter

Tool outside diameter mm

Picture 724

This value is used in the "Tip up avoidance" feature in the "Order" command. It has no affect on the final CNC-file or the postprocessor.

16.3 Z-axis control

Picture 725

IGEMS R10 supports five different methods of controlling the Z-axis. Select all methods supported by the machine. In the "Strategy" command select which method to use depending on the actual material and working strategy.

16.3.1 Fixed Z-axis

 Fixed Z-axis

Picture 726

Check this method if the machine cannot move the Z-axis from instructions in the CNC-file or if the cutting tool is to be in the same Z-axis position all time.

16.3.2 CNC-controlled

CNC-controlled

Automatic
 By M-codes
 By Z-coordinates

Max Z-axis movement mm

Number of movable Z-axis

Z-axis speed mm/min

Acceleration mm/s²

Picture 727

The CNC-controlled method has three options:

Automatic

This option is used if the tools go up and down with the same codes for cutting on and off. This means that the machine will always move the tool to the upper position defined by the "Max Z-axis movement" between each cut. The different settings in the strategy that control the lifting of the tools are disabled.

The reports calculate the time depending on the "Max Z-axis movement", "Z-axis speed" and "Acceleration".

By M-codes

Use this if separate M-codes are used to move the cutting equipment up and down. The strategy controls if the tool lifts between different operations or not. The user cannot control how much the tool lifts. The reports calculate the time depending of the "Max Z-axis movement", "Z-axis speed" and "Acceleration".

By Z-coordinates

Activate this if the machine supports movement with Z-coordinates. These settings allow the user to have different Z-axis values for different operations in the "Strategy" settings (Drilling, Marking, Pre-piercing and Cutting and movements between parts). The reports calculate the time based on all given information.

Max Z-axis movement mm

Number of movable Z-axis

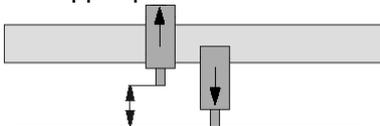
Z-axis speed mm/min

Acceleration mm/s²

Picture 728

Max Z-axis movement

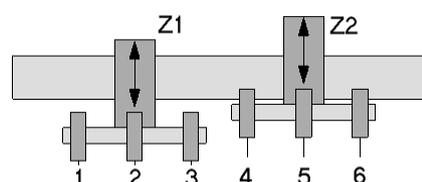
This is the limit setting for the Z-axis value. The value should be measured from the table surface up to the most upper position. To not over travel in Z, decrease this value by some millimeters.



Picture 729

The maximum Z-axis movement that is used by the machine is this value minus the material thickness.

Number of movable Z-axis



Picture 730

It is possible to add more than one tool on each Z-axis. Picture 730 shows 6 cutting heads, but only two movable axes. If the number of tools and the number of Z-axis is different, there is a manual tool distance movement.

Z-axis speed

The Z-axis speed (axis up and down) is used in the CNC-file. It is also used by the simulation and the time calculation.

Acceleration

The acceleration for the Z-axis is used to get a more exact time calculation. It does not affect the CNC-file.

16.3.3 By height sensor

<input checked="" type="checkbox"/> By height sensor
<input type="radio"/> Automatic
<input type="radio"/> By M-codes
<input checked="" type="radio"/> By Z-coordinates
Min Z-axis movement <input type="text" value="50.000"/> mm

Activate this option if the machine has a Height sensor that controls the Z-axis. This option has three choices:

Automatic

This option is used if the height sensor is activated and deactivated with the same codes for cutting on and off. This means that the machine will always move the tool to the upper position defined by the "Max Z-axis movement" between each cut. The different settings in the strategy that control the lifting of the tools are disabled. The reports calculate the time depending of the "Max Z-axis movement", "Z-axis speed" and "Acceleration".

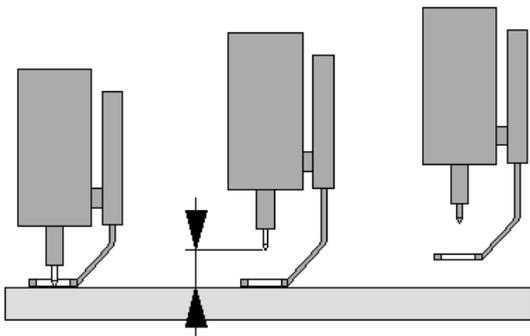
By M-codes

This code is used if separate M-codes activate and deactivate the height sensor. The strategy controls if the height sensor is turned on and off between different operations. The user cannot control how much the tool lifts. The reports calculate the time depending of the "Max Z-axis movement", "Z-axis speed" and "Acceleration"

By Z-coordinates

Activate this if the height sensor can be used together with Z-coordinates. These settings allow the user to have different Z-axis values for different operations in the "Strategy" settings (Drilling, Marking, Pre-piercing and Cutting and movements between parts). The reports calculate the time based on all given information.

Min Z-axis movement



Picture 731

Set this value to the distance the Z-axis must move up before the sensor starts to lift up from the material. Depending on the postprocessor this value is also used when the Z-axis is moved down.

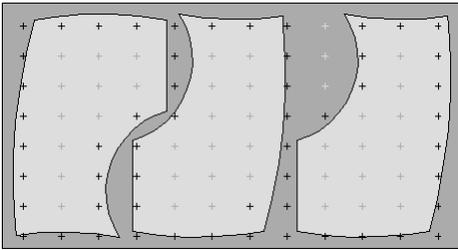
16.3.4 Measuring in advance

This feature is mainly developed for 5-axis machines that cannot be equipped with a height sensor and when controlling the height during the cutting is needed.

<input checked="" type="checkbox"/> Measuring in advance
<input checked="" type="radio"/> By sheet
<input type="radio"/> By part
Max number of measure points <input type="text" value="200"/>
Distance between measure points <input type="text" value="100.000"/> mm
X-offset from cutting tool <input type="text" value="200.000"/> mm
Y-offset from cutting tool <input type="text" value="0.000"/> mm

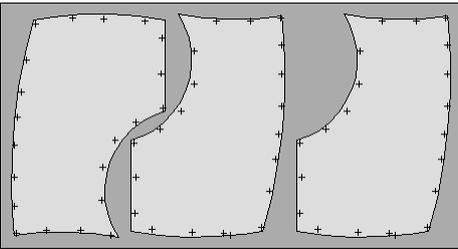
Picture 732

This option is used on a machine that has no height sensor to use during the cutting.

By sheet

Picture 733

This method measures the whole sheet in advance. It only measures points that are close to the cutting geometry and that will be used by the cutting.

By part

Picture 734

This method is a combination of measuring and cutting. It measures the geometry and then cuts the geometry; then it measures again and then cuts again, and so on.

Measure data

Max number of measure points	200
Distance between measure points	100.000 mm

Picture 735

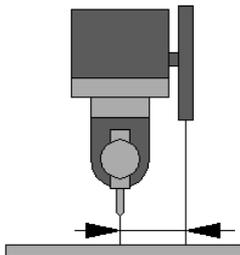
Max number of measure points

This value affects the size of the measuring grid when using the "By sheet" option. 200 points means a grid of 10x20 or 12x16 points, depending of the ratio of the sheet.

Note! The system only measures the points that are closest to the geometry.

Distance between measure points

This value sets up the maximum distance between measuring parts when using the "By part" option.

Offsets

Picture 736

IGEMS can adjust distance from the measuring equipment to the cutting position. This is important to get the measurement or correct positions.

X-offset from cutting tool	200.000	mm
Y-offset from cutting tool	0.000	mm

Picture 737

If these distances are controlled by the controller then set these values to zero.

16.3.5 Pneumatic stroke

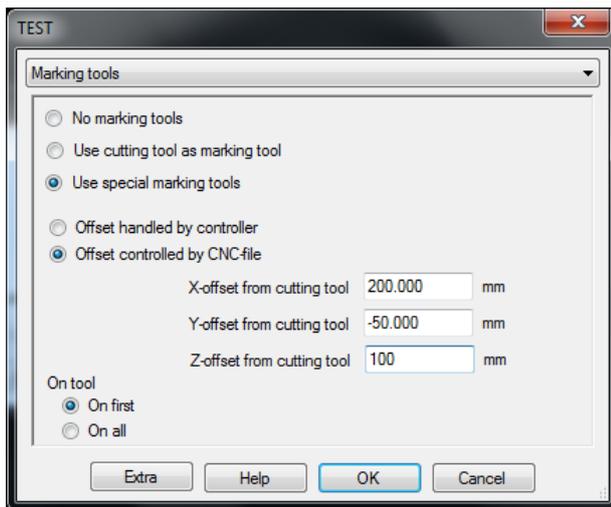
Some machines are equipped with a "pneumatic stroke" that moves the cutting equipment very fast up and down.

<input type="checkbox"/> Pneumatic stroke

Picture 738

The "pneumatic stroke" can be combined with all other measurement methods above.

16.4 Marking tools



Picture 739

Markings can be used in combinations with cutting. Marking is useful for identifications, bending lines, welding position and much more. The following options are available:

No marking tools

IGEMS will skip all marking information in the CNC-file. Use this for toolpath that contain markings but there is no marking equipment.

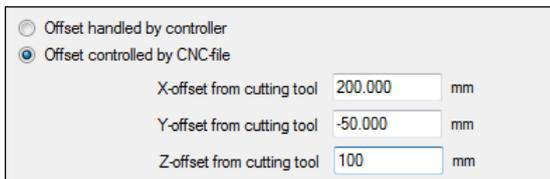
Use cutting tool as marking tool

This is the most common alternative that is used on water jet machines and lasers.

Use special marking tools

Use this option if the marking equipment does not use the cutting head as the marking tool. The marking equipment can be powder, inkjet, etc.

Offsets



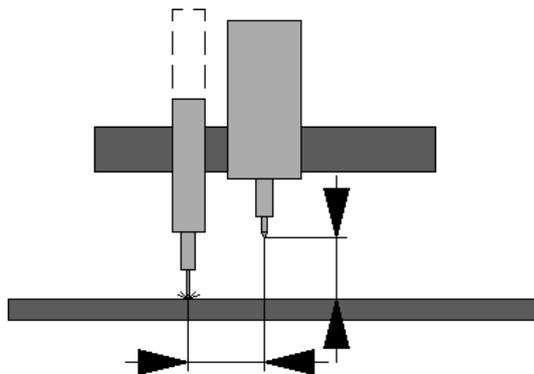
Picture 740

Offset handled by controller

Use this option if the offset from the cutting head to the marking unit is handled by the controller.

Offset controlled by the CNC-file

Use this option if the CNC-file should write information that handles the distance between the cutting head and the marking unit.



Picture 741

X-offset from cutting tool

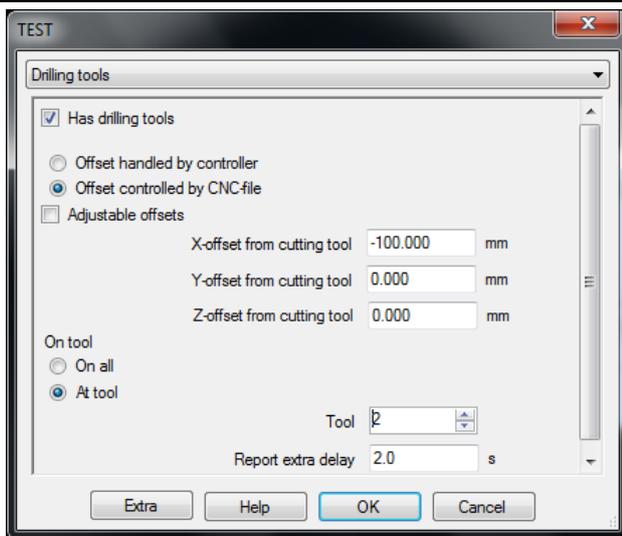
Enter the value related to how the machine is oriented on the screen.

Y-offset from cutting tool

Enter the value related to how the machine is oriented on the screen.

Z-offset from cutting tool

Enter the Z-axis difference between the cutting tool and the marking equipment.

16.5 Drilling tool

Picture 742

If the machine has drilling units then activate the checkbox "Has drilling tools".

Offsets**Offset handled by controller**

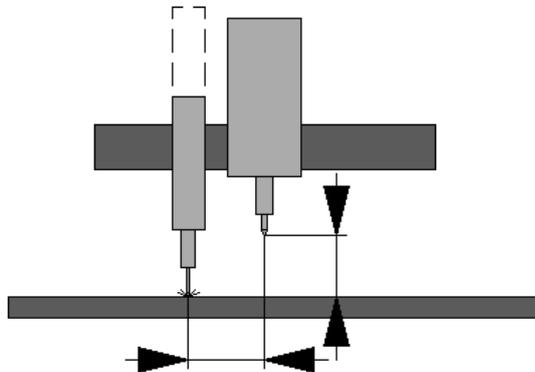
Use this option if the offset from the cutting head to the drilling unit is handled by the controller.

Offset controlled by the CNC-file

Use this option if the CNC-file should write information that handles the distance between the cutting head and the top off the drilling unit.

Adjustable offsets

If the drilling tools are adjustable, then it is important that the offsets can be adjusted easily. If this checkbox is activated then the offsets can also be adjusted from the "strategy" window. If not activated, then the offsets can only be changed from the "machine setup".



Picture 743

X-offset from cutting tool

Enter the value related to how the machine is oriented on the screen.

Y-offset from cutting tool

Enter the value related to how the machine is oriented on the screen.

Z-offset from cutting tool

Enter the Z-axis difference between the cutting tool and the marking equipment.

Location

Picture 744

IGEMS supports drilling tools placed on all heads or one head. The single drilling tool can be located on an optional head.

Report extra delay

Report extra delay	2.0	s
--------------------	-----	---

Picture 745

When calculating the drilling time IGEMS uses the information from different speeds related to the X, Y movement and the actual Z-axis options. The "Drilling delay" is used to fine-tune the estimated cutting time.

16.6 Speeds

Picture 746

16.6.1 Rapid

To have a better estimation of the cutting time provide the "Rapid speed" and the "Acceleration and deceleration" parameters.

Rapid speed	10000.0	mm/min
Acceleration and deceleration	250.0	mm/s ²

Picture 747

Rapid speed

Set the value of the rapid speed supported by the machine. The values entered here do not affect the final CNC-file and is only used for calculations.

Acceleration and deceleration

If X and Y axis have different values, then use the smallest value of the two.

16.6.2 Cutting speed

Max cutting speed	4000.0	mm/min
Min cutting speed	1.0	mm/min

Picture 748

Max cutting speed

Set this value to the maximum cutting speed desired. If the database generates a faster speed then IGEMS reduces the speed to the "Max cutting speed" value.

Min cutting speed

If the database generates values that are too low then IGEMS will round up the speeds to this value.

16.6.3 Max speed for 5-axis heads

Max speed Primary axis	60.0	°/s
Max speed Secondary axis	60.0	°/s

Picture 749

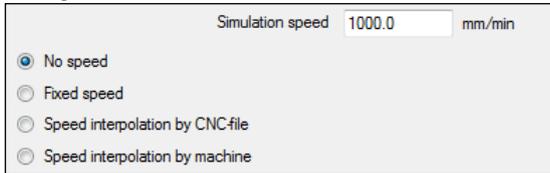
In IGEMS the primary axis is always called A and the secondary axis called B. The A and B speeds are calculated from the motion and the speeds of X and Y (A and B axis are slaves to the X and Y movements).

Kinematic constructions (when the primary axes are in vertical direction) may sometimes need extremely high rotation speed. By specifying the maximum A and B speed it is possible for IGEMS to calculate a XY speed that never exceeds the maximum speed for A and B.

16.6.4 Speed modes

IGEMS works with 4 different speed modes:

No speed



Picture 750

This mode is used when the CNC-controller calculates all speeds. IGEMS internally uses a speed called "Simulation speed". This speed is the same and never changes, no matter if material or cutting quality changes. IGEMS needs this speed to present reports and for making simulations.

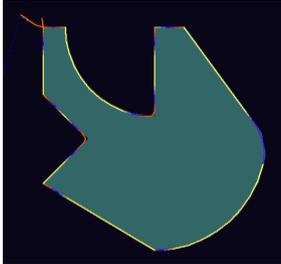
- ? Are there any disadvantages to calculating the speed in the controller?
Yes, you can't make cost estimations and other reports in advance.

Fixed speed

The "Fix speed" value can be stored on every thickness for every material in the material database. When this mode is selected IGEMS uses this speed. The speed will not change depending on the cutting quality.

Speed interpolation by CNC-file.

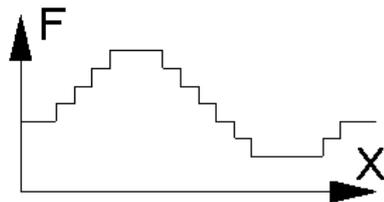
To obtain a good cutting result with a fast production then the cutting speed must be optimized for the material, thickness and the geometry. When speed optimizing is used IGEMS takes care of all speed optimizing. This gives a fast production, a very good cutting result and also gives correct cutting times, reports and simulations.



Picture 751

When using "speed optimizing" IGEMS controls all kind of speeds in the CNC-file.

To be able to have a smooth acceleration and deceleration linear and circular movements are split up into shorter movement, each movement changes the speed a little step.

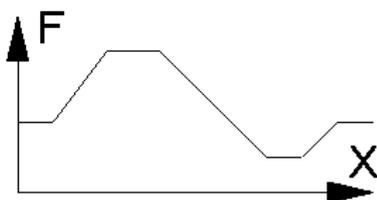


Picture 752

To make a smooth acceleration the movements are divided into shorter steps. In the example above it will need 13 lines in the CNC-file to create this movement. More or less all controllers can use this method.

Speed interpolation by machine

Some controllers have a functionality to make it possible to interpolate with the speed.

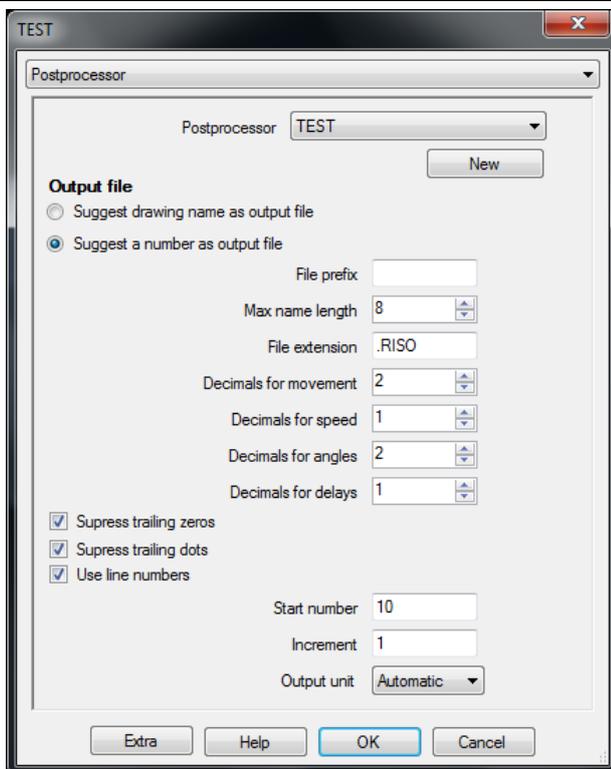


Picture 753

IGEMS' technology for feed interpolation is working on Siemens FLIN/FNORM functions but other controllers provide similar functionality. The result will be smoother movements, shorter CNC-files, better cutting results and shorter cutting time.

Note! This speed mode is unusable if the machine does not support cutting speed interpolation.

16.7 Postprocessor



Picture 754

These settings handle the fundamentals in the postprocessing, such as naming the file and different formats.

16.7.1 Postprocessor selection

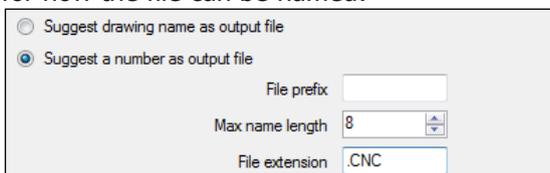


Picture 755

The postprocessor is an open text file written in ILISP syntax. The ILISP documentation has detailed information on how to change or develop postprocessors. To make a new postprocessor press the "New" button and enter a name of the new postprocessor.

16.7.2 Output file

The postprocessor creates a file that can be used by the CNC-machine. The settings below describe the rules for how the file can be named:



Picture 756

Suggest drawing name as output file

In this case IGEMS will suggest that the output file name will be the same name as the actual drawing. The extension of the file is defined below.

Suggest a number as output file

Many users want to have the output files named as a number (100.CNC, 101.CNC, 102.CNC). With this setting the actual number will be increased by one for every new output file created.

File prefix

Specify here if there should be any prefix added before the number.

Max name length

Specify longest acceptable file name.

Note! This controls only the name not the extension of the file.

File extension

This is the extension of the output file.

The following table describes the result of different settings:

File prefix	Max name length	Number	Extension	Result
	8	100	.CNC	100.CNC
P	8	101	.CNC	P101.CNC
P000000000	8	102	.CNC	P0000102.CNC
P00	10	103		P00103
	3	1010	.CNC	010.CNC
O	6	104	.DNC	O104.DNC

Add information where the CNC-file will be saved in the "Strategy" command.

16.7.3 Format

Decimals for movement: 3

Decimals for speed: 1

Decimals for angles: 2

Decimals for delays: 1

Suppress trailing zeros

Suppress trailing dots

Picture 757

Decimals for movements

This setting controls how many decimals are used to present the movements in the output file. The value is generated by the (RTS <value>) function in the postprocessor.

Decimals for speed

This setting controls how many decimals are used to present the speed in the output file. The value is generated by the (RTF <value>) function in the postprocessor.

Decimals for angles

If using a 5-axis machine this value controls the number of decimals. The value is generated by the (ATS <radian>) function in the postprocessor.

Decimals for delays

This controls how many decimals are used for delays in the output file. The value is generated by the (RTT <second>) function in the postprocessor.

Suppress trailing zeros

Suppress trailing dots

Picture 758

Suppress trailing zeros

If this checkbox is activated then all trailing zeros will be removed.

Example: 10.000 will be 10. and 10.100 will be 10.1.

Suppress trailing dots

This feature is only useful if "Suppress trailing zeros" is activated.

Example: 10.000 will be 10 and 10.100 will be 10.1.

16.7.4 Line numbers

Use line numbers

Start number: 10

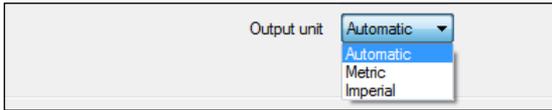
Increment: 1

Picture 759

This option controls the ILISP function (NTXT). The function generates a text with increasing numbers.

Example: N10, N15, N20, N25 and so on. The numbers are used on many controllers. Check the box to control if the line numbers should be used. Set the start value and the increment.

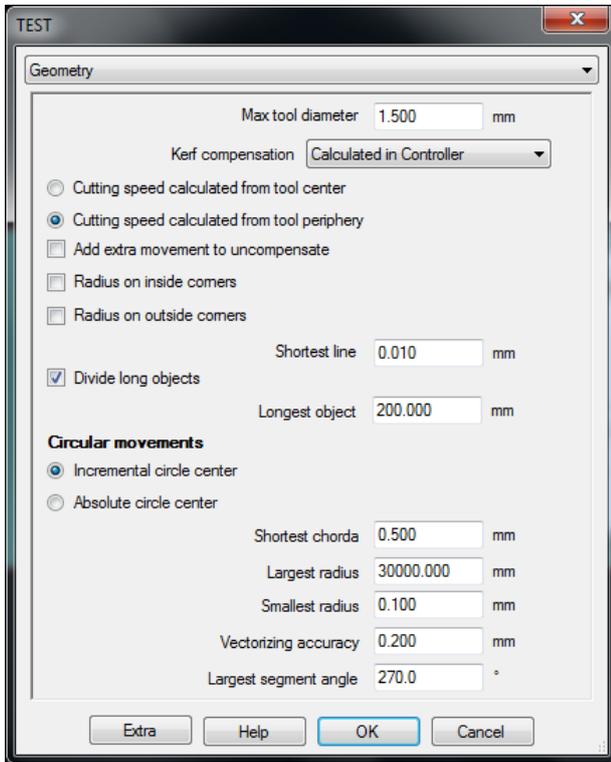
16.7.5 Output units



Picture 760

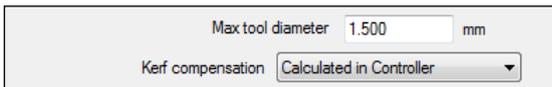
By setting this to "Automatic" the output file is in the same units as the drawing. The output can also be set to Metric or Imperial. If this setting is different than the units on the drawing then IGEMS will scale the output dimensions.

16.8 Geometry



Picture 761

16.8.1 Tool radius compensation



Picture 762

Max tool diameter

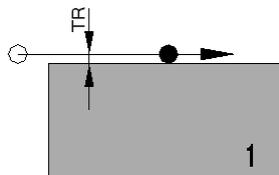
This is a very important setting that controls IGEMS geometry optimizing. The value should always be larger than the largest tool diameter used on the machine.

- ? What will happen if this value is smaller?
The machine will stop with a contour error.

Kerf compensation

IGEMS supports three different options:

Calculated in IGEMS

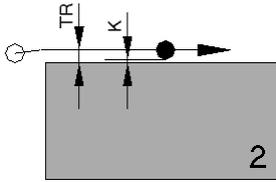


Picture 763

With this option the toolpath is internally offset by IGEMS. The size of the compensation is the same as the Tool Radius (TR). There are no kerf instructions in the CNC-file (it will use G40 mode or similar).The

coordinates in the CNC-file is the same as the movement of the machine. To change the measurements of the parts run the postprocessor again.

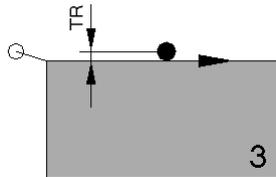
Calculated in IGEMS and Controller



Picture 764

With this option the toolpath is internally offset by IGEMS. The size of the compensation is the same as the Tool Radius (TR). The postprocessor adds kerf instructions (G41, G42 or similar). The idea behind this is that IGEMS controls the major part of the kerf compensation (TR) and the machine controller adds a smaller kerf (K) just to fine-tune the measurements. This option makes it possible to adjust the size of the part without creating a new CNC-file.

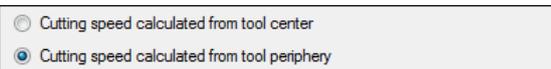
Calculated in Controller



Picture 765

When using tool radius compensation Right or Left (G41 or G42) the coordinates in the CNC-file describe the size of the part (not the machine movement). The Kerf (K) will be added in the CNC-controller. This option makes it possible to adjust the size of the part without creating a new CNC-file.

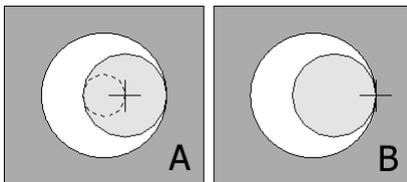
16.8.2 Cutting speed on circular movements



Picture 766

When the kerf is calculated in IGEMS the speed is always calculated from the tool center. When using tool compensation in the machine then the controller has two choices.

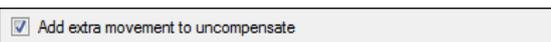
In the examples the hole has diameter 1.5 and the jet diameter 1.0.



Picture 767

In example A the programmed path speed is on the tool center point (dashed circle). In example B the programmed path speed is set to the work piece contour. This programmed path is three times longer. To have an optimal result it is important for IGEMS to know which method the controller will use.

16.8.3 Cancel kerf compensation



Picture 768

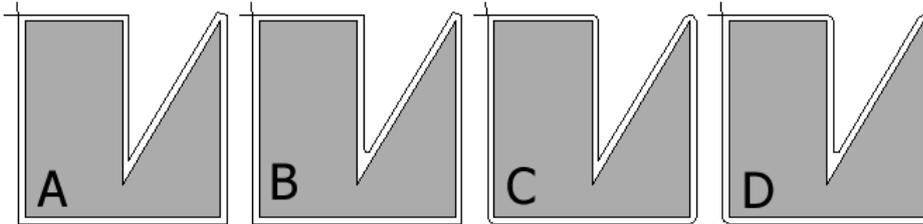
Some controllers need to make an extra movement to cancel the kerf compensation. If this option is activated, then an extra movement will be added after the kerf compensation has been turned off. The length of the movement is the same as the tool diameter and the speed is the same as maximum cutting speed.

16.8.4 Corner radius



Picture 769

The following pictures show the center of the tool. When the kerf compensation is calculated in the machine then the machine will always make radius on outside corners:



Picture 770

No radius (example A)

Only use this solution when the kerf compensation is calculated in IGEMS.

Radius on inside corner only (example B)

If the kerf is calculated in IGEMS then the inside radius size will be half of the difference between "Max tool diameter" and the actual tool diameter. If the kerf is calculated in the controller then the radius will be half of "Max tool diameter".

Radius on outside corner only (example C)

If the kerf is calculated in the controller then the machine will always make this arc. If the kerf is controlled in IGEMS then turn this radius on or off.

Radius on inside and outside (example D)

See example B and C

? Which combination will give the best cutting result?
Example A: Do not use any radius on inside or outside.

? What is the other alternative used for?
IGEMS supports many machines and some machines need the other options.

16.8.5 Remove short lines

Picture 771

Using very short lines in combination with using kerf compensation can sometimes cause geometry problems. Linear movements shorter than this value will be removed from the CNC-file.

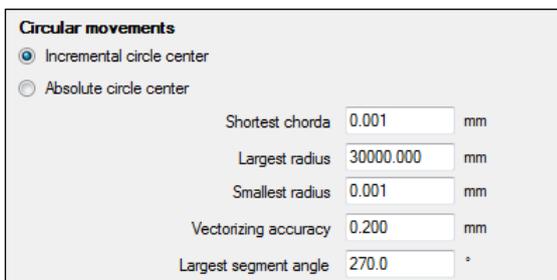
16.8.6 Divide long objects

Picture 772

This feature splits up long lines or circular movements into shorter steps. It should be easier to restart the machine on a good position if the machine is stopped.



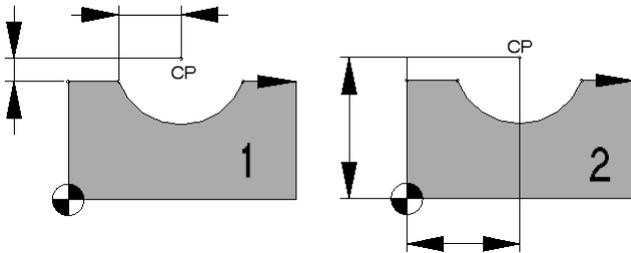
Picture 773

16.8.7 Circle movements

Picture 774

Circle center

The postprocessor gives the possibilities to hardcode one of these two options. If the machine supports both of these methods, then select which method to use.



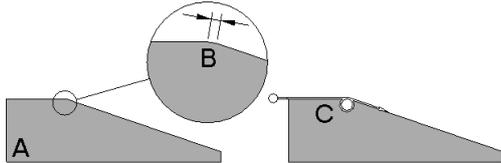
Picture 775

Example 1 describe the "Incremental circle center" and example 2 the "Absolute circle center".

On many controllers the circle center can be described by the radius. This method is also available in the postprocessor.

Shortest chord

Almost all CNC controllers are using the same G-code (G02 and G03) to describe an arc and a circle. If the distance from the start point to the end point is close then the machine will make a circle.



Picture 776

If (B) is a short arc then the machine may make an unwanted circle movement (C).

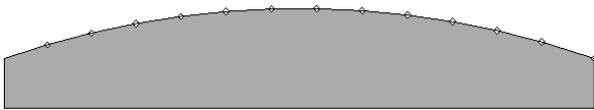
The value of the shortest chord must be high enough so no unwanted circles are created.

Chords that are too short will be replaced by a linear movement.

Largest radius

Enter the size of the largest arc or circle that is supported by the CNC-controller. If the radius is larger, then it will be vectorized.

Vectorizing accuracy

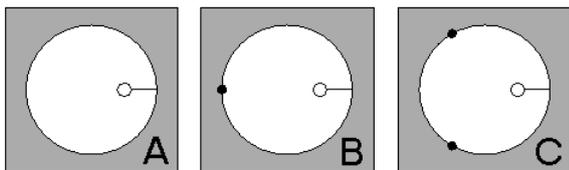


Picture 777

The differences between the original arc geometry and the lines will not exceed the vectorizing accuracy.

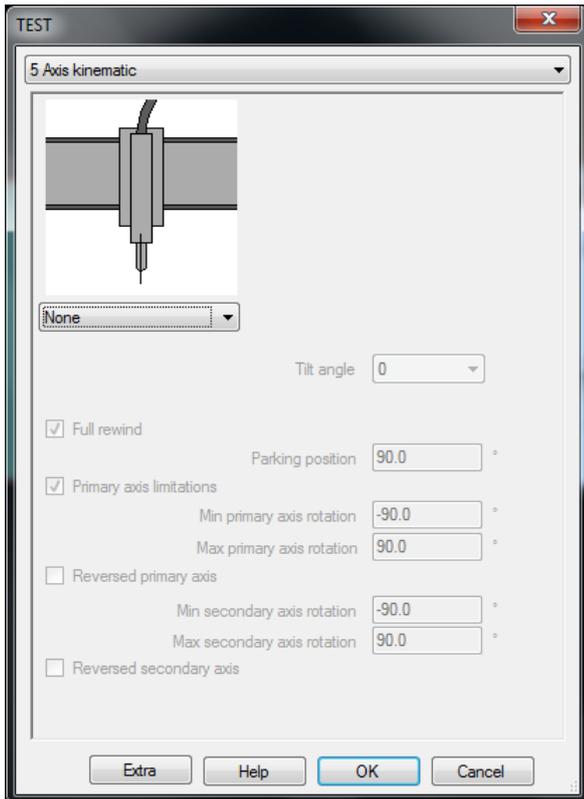
Largest segment angle

This value controls the largest segment angle that will be generated by the postprocessor. The value can be in range 1 to 360 degree. If the segment angle is larger then this value the arc will be divided into parts.



In the example (A) the value is set to 360 degrees and the circle movement is not divided. In example (B) the value is set to 180 degrees and in example (C) the value is set to 120 degrees.

16.9 5-Axis kinematic



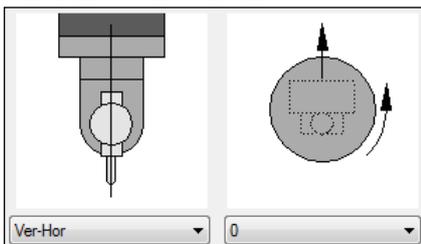
Picture 778

Set the choice to "None" if the machine does not a 5-axis cutting head. The arrows describe the positive axis directions.

16.9.1 Supported 5-axis heads

IGEMS has an inbuilt kinematic model for most kinds of 5-axis cutting heads. Other constructions than below can be defined in the postprocessor.

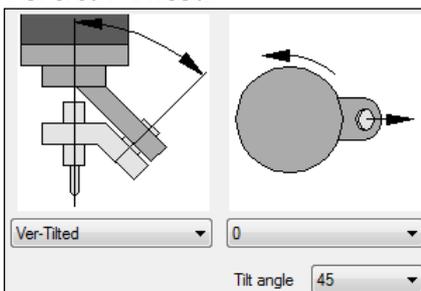
Vertical-Horizontal



Picture 779

In this construction the primary rotation is vertical and the secondary horizontal. Set the zero rotation of the vertical axis as the layout and axis direction are defined on the screen.

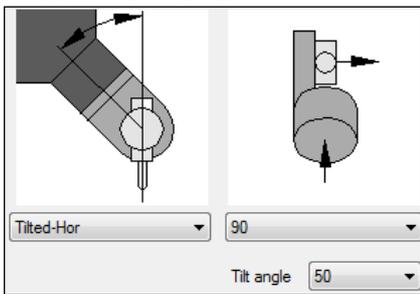
Vertical-Tilted



Picture 780

In this construction the primary rotation is vertical. The secondary angle can be set to 45 or 30. It is possible to define other angles in the postprocessor. Set the zero rotation of the vertical axis as the layout and axis direction are defined on the screen.

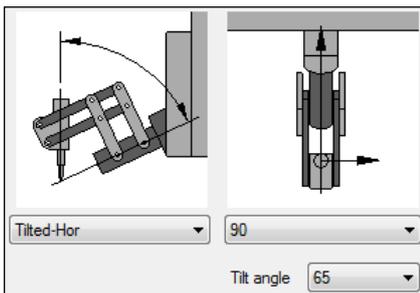
Tilted-Horizontal



Picture 781

In this construction the primary axis is tilted and the second is horizontal. The tilt angle from vertical is 50 degree. Other tilt angles can be defined in the postprocessor. Set the direction of the primary axis as the layout and axis direction are defined on the screen.

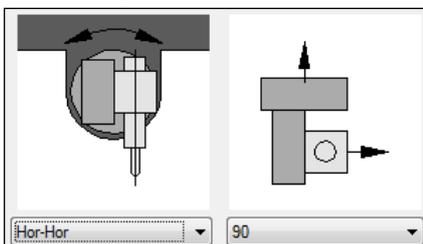
I-HEAD



Picture 782

In this construction the primary axis is tilted and the second is horizontal. The tilt angle from vertical is 65 degree. Other tilt angles can be defined in the postprocessor. Set the direction of the primary axis as the layout and axis direction are defined on the screen.

Horizontal-Horizontal



Picture 783

In this construction both axis are horizontal. Set the direction of the primary axis as the layout and axis direction are defined on the screen.

Other settings

<input checked="" type="checkbox"/> Full rewind	Parking position	<input type="text" value="0.0"/>	°
<input checked="" type="checkbox"/> Primary axis limitations	Min primary axis rotation	<input type="text" value="-365.0"/>	°
	Max primary axis rotation	<input type="text" value="365.0"/>	°
<input type="checkbox"/> Reversed primary axis	Min secondary axis rotation	<input type="text" value="-90.0"/>	°
	Max secondary axis rotation	<input type="text" value="90.0"/>	°
<input type="checkbox"/> Reversed secondary axis			

Picture 784

Full rewind

When the primary axis has rotation limits it may be necessary to make a rewind when the rotation reaches the limits. There are two options to make 360 degree or 180 degree rewind. Check "Full rewind" for 360 degrees.

? Which is best?

This depends of how much you can rotate. Full rewind gives better re-position of the nozzle and gives best quality and accuracy.

Parking position

 A screenshot of a software control field labeled "Parking position" with a numerical input box containing the value "0.0" and a degree symbol (°) to its right.

Picture 785

This setting is only important if the primary axis is vertical. It controls the angle the primary axis should be when in "parking position" (when it is not in use).

16.9.2 Primary axis

 A screenshot of a control panel for "Primary axis limitations". It features a checked checkbox labeled "Primary axis limitations". Below it are two input fields: "Min primary axis rotation" with the value "-365.0" and "Max primary axis rotation" with the value "365.0", both followed by degree symbols. At the bottom is an unchecked checkbox labeled "Reversed primary axis".

Picture 786

If the construction has rotation limits for the primary axis then activate the checkbox. Set the max and minimum rotations for the axis. The system will automatically generate a rewind if needed. The axis direction should always be given as "Right hand rules" as the layout and axis direction are defined on the screen.

Reverse direction

If the primary axis is defined in the opposite direction in the controller then activate this checkbox.

16.9.3 Secondary axis

 A screenshot of a control panel for "Secondary axis rotation". It features two input fields: "Min secondary axis rotation" with the value "-90.0" and "Max secondary axis rotation" with the value "90.0", both followed by degree symbols. At the bottom is an unchecked checkbox labeled "Reversed secondary axis".

Picture 787

Enter the "minimum" and "maximum" angle for the secondary axis. If the axis direction is defined in opposite directions then check the "Reverse secondary axis".

16.10 Cutting equipment

 A screenshot of a dialog box titled "TEST" with a close button (X) in the top right corner. The dialog has a dropdown menu labeled "Cutting equipment" currently showing "Abrasive waterjet". Below the dropdown are five radio button options: "Abrasive waterjet" (selected), "Pure waterjet", "Plasma", "Oxyfuel", and "Laser". Below these is a text input field for "General machine speed factor" with the value "1.00". At the bottom are four checkboxes: "Pump on/off controlled from CNC-file" (unchecked), "Pressure handled manually" (unchecked), "High and Low pressure control" (selected), and "Variable pressure" (unchecked). At the very bottom are four buttons: "Extra", "Help", "OK", and "Cancel".

Picture 788

IGEMS is designed for Abrasive waterjet but also supports other kinds of cutting machines. Select what machine type the machine should be set up for.

16.10.1 Abrasive waterjet

General machine speed factor

Pump on/off controlled from CNC-file

Pressure handled manually

High and Low pressure control

Variable pressure

Reduce by Bleed out valve

Reduce by extra piercing

Pump size kW

Max pump pressure bar

Min pump pressure bar

Pressure loss bar

Abrasive handled manually

High and low abrasive control

Variable abrasive control

Water and Abrasive controlled by separate codes

Min abrasive g/min

Max abrasive g/min

Picture 789

General machine speed factor

If it seems that all cutting values in the material setting are too low or too high then change them globally by changing the "General machine speed factor".

The example below shows High speed and Low speed for Stainless steel 12mm.

	X-rough	Rough	Medium	Fine	X-fine
High speed	258.6	185.7	116.5	83.7	64.8
Low speed	82.0	71.8	57.2	47.3	40.2

Picture 790

When the "General machine speed factor" is changed to 1.20 the speed will be increased by 20%.

General machine speed factor

Picture 791

The values below are for the same material and cutting parameters.

	X-rough	Rough	Medium	Fine	X-fine
High speed	310.3	222.9	139.8	100.4	77.7
Low speed	98.4	86.1	68.6	56.7	48.2

Picture 792

? Why can't they all use the same speed factor?

There are many manufacturers of cutting equipment on the market and there is a big difference in qualities.

16.10.2 Pump and pressure settings

Pump on/off controlled from CNC-file

Pressure handled manually

High and Low pressure control

Variable pressure

Reduce by Bleed out valve

Reduce by extra piercing

Pump size kW

Max pump pressure bar

Min pump pressure bar

Pressure loss bar

Picture 793

Pump on/off controlled from CNC-file

Check this option if the system can turn on or off the pump from the CNC file. It is possible to control whether or not this should be done in the "strategy".

Pressure handled manually

If the system cannot change the pressure then the machine will stop and the operator can change pressure manually.

High and Low pressure control

Many systems have two levels of pressure - High and Low. Check this option if the system can switch between these two by using M-codes.

Variable pressure

Select this option if the system can handle the pressure steeples.

Reduce pressure
 Reduce by Bleed out valve
 Reduce by extra piercing

Picture 794

Check this option if the system reduces the pressure by a "Bleed out valve".

If the system does not have any equipment to release the pressure then it is possible to reduce the pressure by making an extra piercing. The extra piercing will be done in an already pierced position.

Pump size

Pump size	73.6	kW
Max pump pressure	4000	bar
Min pump pressure	100	bar
Pressure loss	100	bar

Picture 795

This refers to the motor size; the value is not used for the moment.

Max pump pressure

Set the value to the highest supported pressure. It will not be possible to set a higher pressure than this in the "Strategy" or at the "Material" settings.

Min pump pressure

Set the value to the lowest supported pressure. It will not be possible to set a lower value than this in the "Strategy" or at the "Material" settings.

Pressure loss

Estimate the pressure loss between the pump and the nozzle.

Some examples:

Bar	Heads	Orifice	Pipe	Length	Loss
3800	1	0.25 (10)	¼	10	6 bar
4100	2	0.25 (10)	¼	10	23 bar
4100	4	0.36 (14)	¼	30	825 bar

16.10.3 Abrasive settings
 Abrasive handled manually
 High and low abrasive control
 Variable abrasive control
 Water and Abrasive controlled by separate codes

Min abrasive	0	g/min
Max abrasive	1000	g/min

Picture 796

Abrasive handled manually

Check this option if the system cannot change the amount of abrasive from instructions in the CNC-file. To change amount of abrasive the machine will stop and the operator can manually adjust the abrasive.

High and low abrasive amount

Check this option if the system can switch between two different amounts of abrasive.

Variable abrasive control

Check this option if the system can handle the amount of abrasive steeples.

Water and Abrasive controlled by different codes.

Check this option if using separate M-codes for water and abrasive.

Min and Max abrasive

Min abrasive	0	g/min
Max abrasive	1000	g/min

Picture 797

Enter the "minimum" and "maximum" amount of abrasive that can be used on the system.

Orifices, mixing tubes and abrasive qualities

To avoid selecting unsupported equipment, check the available items in the list below.

Available orifices		
<input type="checkbox"/> 0.060 (2.4)	<input type="checkbox"/> 0.254 (10)	<input type="checkbox"/> 0.458 (18)
<input type="checkbox"/> 0.081 (3.2)	<input checked="" type="checkbox"/> 0.279 (11)	<input type="checkbox"/> 0.483 (19)
<input type="checkbox"/> 0.101 (4)	<input checked="" type="checkbox"/> 0.305 (12)	<input type="checkbox"/> 0.505 (20)
<input type="checkbox"/> 0.127 (5)	<input checked="" type="checkbox"/> 0.330 (13)	
<input type="checkbox"/> 0.153 (6)	<input checked="" type="checkbox"/> 0.356 (14)	
<input type="checkbox"/> 0.179 (7)	<input type="checkbox"/> 0.381 (15)	
<input type="checkbox"/> 0.204 (8)	<input type="checkbox"/> 0.406 (16)	
<input type="checkbox"/> 0.229 (9)	<input type="checkbox"/> 0.432 (17)	
Available mixing tubes		
<input type="checkbox"/> 0.152 (6)	<input checked="" type="checkbox"/> 0.635 (25)	<input checked="" type="checkbox"/> 1.092 (43)
<input type="checkbox"/> 0.203 (8)	<input checked="" type="checkbox"/> 0.762 (30)	<input checked="" type="checkbox"/> 1.143 (45)
<input type="checkbox"/> 0.305 (12)	<input checked="" type="checkbox"/> 0.889 (35)	<input type="checkbox"/> 1.270 (50)
<input type="checkbox"/> 0.381 (15)	<input checked="" type="checkbox"/> 1.016 (40)	<input type="checkbox"/> 1.397 (55)
<input checked="" type="checkbox"/> 0.508 (20)	<input type="checkbox"/> 1.050 (41)	<input type="checkbox"/> 1.524 (60)
Available abrasive qualities		
<input checked="" type="checkbox"/> Barton Gamet 80	<input type="checkbox"/> Crushed Glass	
<input checked="" type="checkbox"/> GMA Gamet 80		
<input type="checkbox"/> Indian Gamet 80		
<input type="checkbox"/> Olivine 120		

Picture 798

Marking delays

This information is needed if the cutting tool is used as a marking tool.

Marking	
Between water and abrasive on	1.1 s
Between water and abrasive off	1.1 s
Before moving	1.1 s

Picture 799

The first two values are needed only if abrasive is used for marking. "Before moving" is a delay time between Jet On and the first move.

Pre-piercing delays

Pre-piercing	
Between water and abrasive on	0.5 s
Between water and abrasive off	0.5 s

Picture 800

These values handle the delay between water and abrasive On/Off.

Cutting delays

Cutting	
Between water and abrasive on	0.0 s
Between water and abrasive off	0.0 s
Before moving	0.0 s

Picture 801

These values handle the delay between water and abrasive On/Off. "Before moving" is the same as "minimum piercing time".

? Why are there separate delays for pre-piercing and cutting?

Pre-piercing often uses other pressure and abrasive settings. For this reason it is common to use different values.

Various

Various	
Report extra	1.0 s
Pressure up	9.0 s
Pressure down	10.0 s
After cutting	1.2 s

Picture 802

Report extra

This is extra time added to the cutting time for every marking, pre-piercing and cutting. This gives the possibility to fine-tune the estimated cutting time.

Pressure up

This is the delay time used if the pressure is increasing.

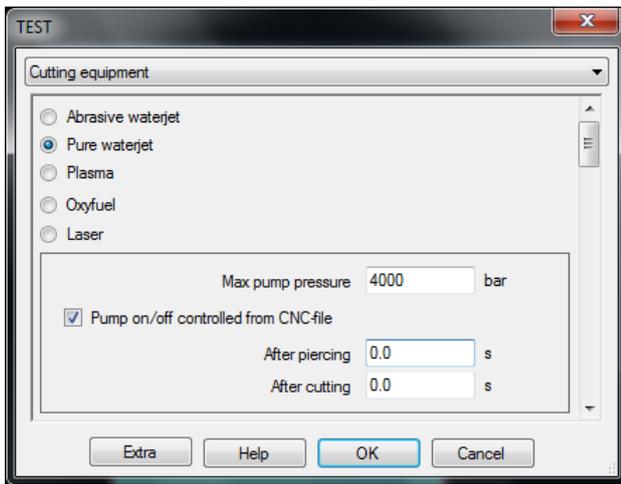
Pressure down

This is the delay time used when the pressure is decreasing.

After cutting

This is the delay time between Jet off and the rapid movement.

16.10.4 Pure waterjet



Picture 803

This machine type does not have as many settings.

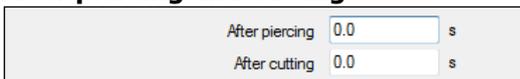
Max pump pressure

Set the value to the highest possible on the machine.

Pump on/off controlled from CNC-file

Check this option if the system can turn the pump on/off from the CNC file. Control if this is done or not in the "Strategy".

After piercing and cutting



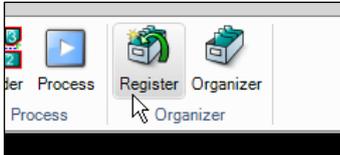
Picture 804

"After piercing" is the same as "minimum piercing time". "After cutting" is the delay time between Jet off and the rapid movement.

17 Organizer

"Organizer" is a separate module in IGEMS used to store part files and relevant information tied to the parts. Information about the parts is stored in an embedded SQL database. The parts itself are stored in the Parts directory in the Shared folder. (See 3.6).

17.1 Register



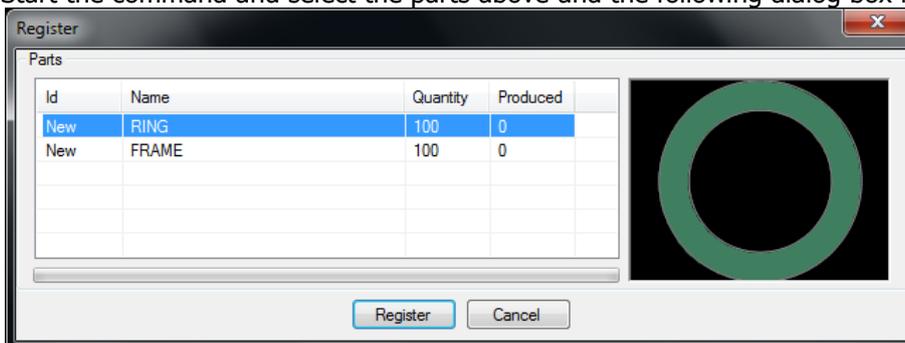
Picture 805

The "Register" command adds information to the database and saves the part as a separate DIG file.



Picture 806

Start the command and select the parts above and the following dialog box is shown:



Picture 807

Press the "Register" button if the information is correct.



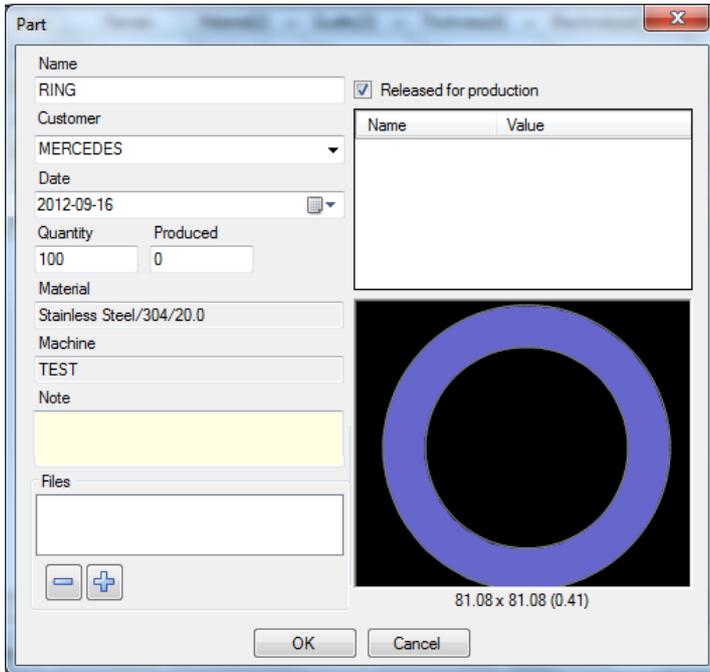
Picture 808

Registered parts are then displayed in blue color.

- ? Can a part be registered with an attached toolpath?
Yes, but you will probably have a better nesting result if you have no toolpath attached.

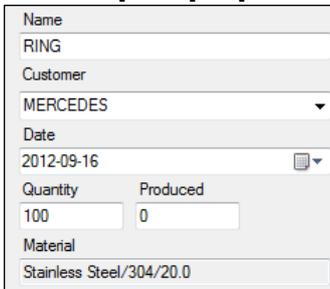
17.1.1 Part information

Edit the part information by double clicking on a registered part.



Picture 809

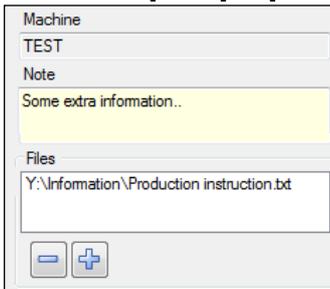
Normal part properties



Picture 810

Except for the field "Produced" the information is identical for non-registered and registered parts. The "produced" value will be a count up if a Cut order is included with the registration.

Extended part properties



Picture 811

Add extra information to the parts.

Note! This is free text information that can be used by the reports or by the postprocessors.

Files

It is possible to add links to files located on the computer or on the network. Double click on the file and the file type is associated to a program. Then start programs from here.

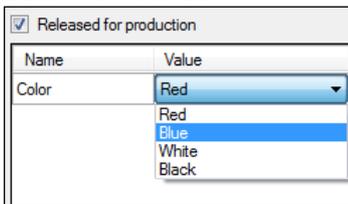
Release for production



Picture 812

It may be efficient to register a created part for estimation of prices. If a confirmation is needed before cutting starts, uncheck this box. Check the box when the part is ready for production.

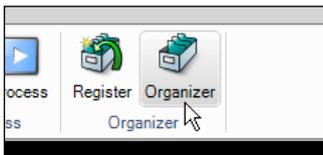
User variables



Picture 813

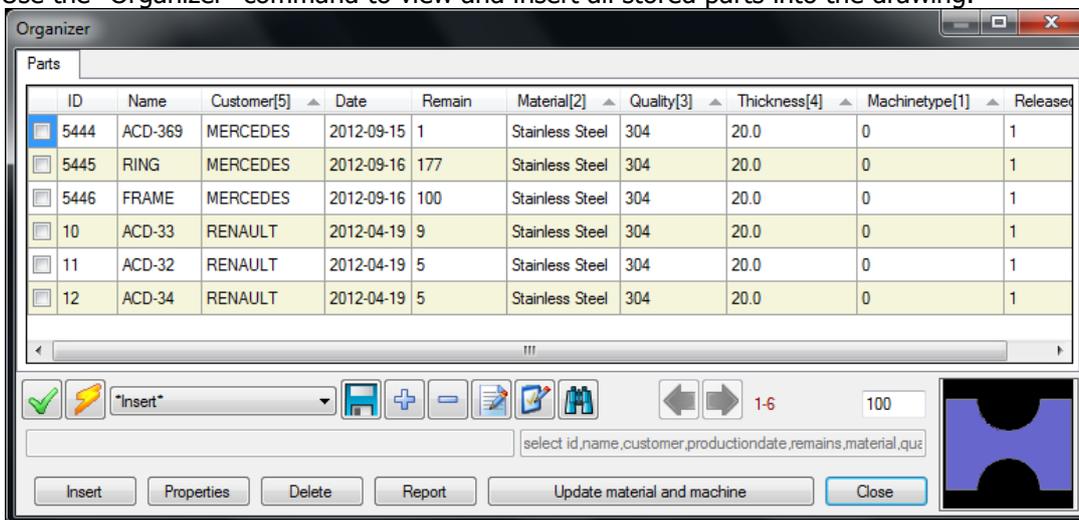
The "User variables" can be defined in the "Advance" options in the "Organizer" dialog box. The "user variables" contains 5 different variable types: Lists, real's, integers, strings and dates.

17.2 Organizer



Picture 814

Use the "Organizer" command to view and insert all stored parts into the drawing.



Picture 815

This dialog box handles all the features in "Organizer".

17.2.1 Predefined profiles



Picture 816

Filters are used to quickly sort out part information with certain criteria such as material, thickness or production dates. The profile contains rules for which parts should be shown, which columns should be visible and the sorting order. Profiles that start and end with an "*" are predefined and cannot be changed. The following two profiles are predefined:

Show all

This profile shows all parts in the database.

Insert

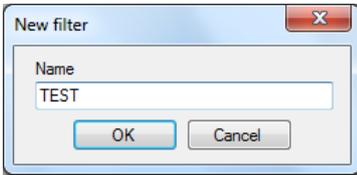
This profile shows all parts with the same material activated in IGEMS to be produced.

17.2.2 User defined profiles

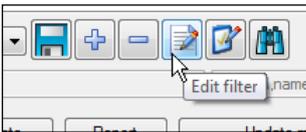


Picture 817

Click on the "plus" button to create a new filter profile.



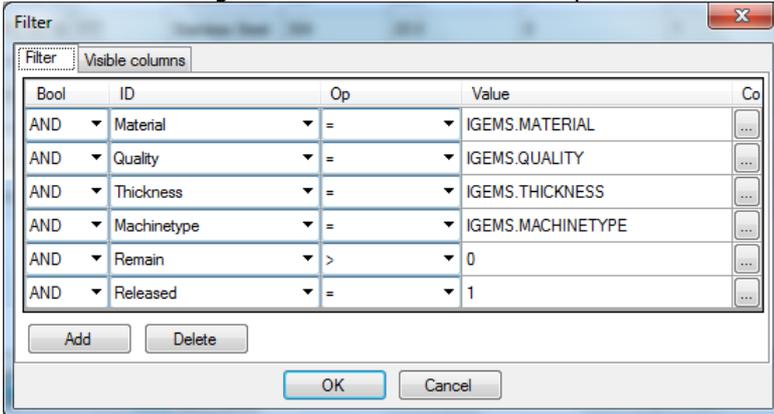
Picture 818
Enter the name of the new filter profile.



Picture 819

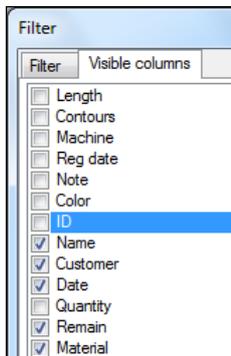
Filters

Use the filter dialog box to define rules for what parts should be visible on the screen.



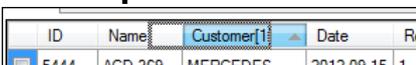
Picture 820

Visible columns



Picture 821
Use the checkboxes to control which columns are visible.

Column position



Picture 822
Hold down and move the mouse pointer at a column to modify the order of the different columns.

Sorting

	A			B				
ID	Name	Customer[1]	Date[3]	Remain	Material[2]	Quality	Thickness	MacI
5445	RING	MERCEDES	2012-09-16	177	Stainless Steel	304	20.0	0
5446	FRAME	MERCEDES	2012-09-16	100	Stainless Steel	304	20.0	0

Picture 823
Sort the list by clicking on a column. Click on the same column several times to switch between increasing or decreasing order. Hold down the CTRL key while clicking on another column to define sub-sorting orders. If some values in column (A) are the same then these will be sorted by column (B) and so on.

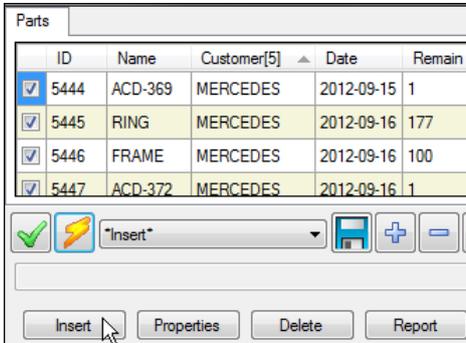
Save



Picture 824

Press the "Save" button when the new profile is defined.

17.2.3 Insert parts to the drawing



Picture 825

Check the parts to insert and then press the "Insert" button.



Picture 826

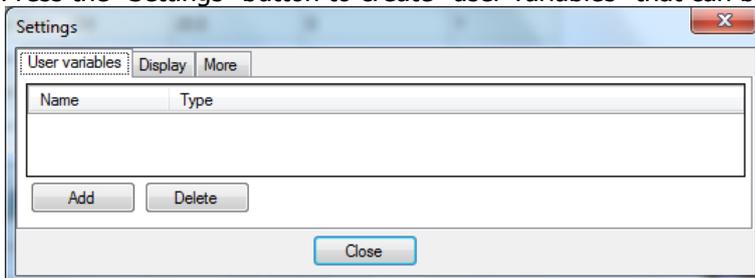
IGEMS will prompt to select the insertion point for each part.

17.2.4 User variables



Picture 827

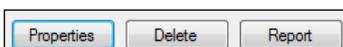
Press the "Settings" button to create "user variables" that can be located in the parts.



Picture 828

These settings are currently not described in this manual. Please contact IGEMS for information about user variables.

17.2.5 Various



Picture 829

Properties

Clicking on the "Properties" button is the same as double clicking on a part in the list. This will show information about the part.

Delete

Parts can be deleted from the database when checked in the list. However it is recommended to never delete parts from the database.

Report

The "Report" button creates a report with all checked parts.

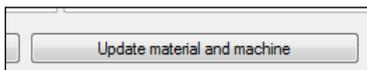
New database

"Organizer" is pre-installed with a sample database. To start with a new empty database, delete the complete "Organizer" directory. A new empty database will then automatically be created when starting IGEMS. The "Organizer" directory is located in the Shared folder. See Section 3.6 for more information.

Backup

It is recommended to backup a copy of the "Organizer" database on a regular base. Copy the entire "Shared" folder and store it in a safe place. See Section 3.6 for more information.

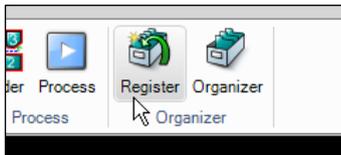
17.2.6 Update material and machine



Picture 830

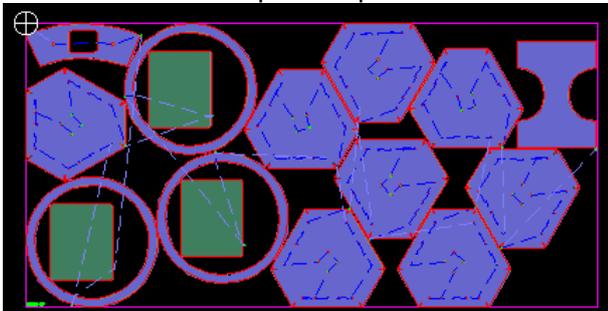
This function changes the selected parts to the material and machine that are activated in IGEMS. The function is useful the name of a material is changed in IGEMS or if updating from an older "Organizer".

17.3 Automatic count down



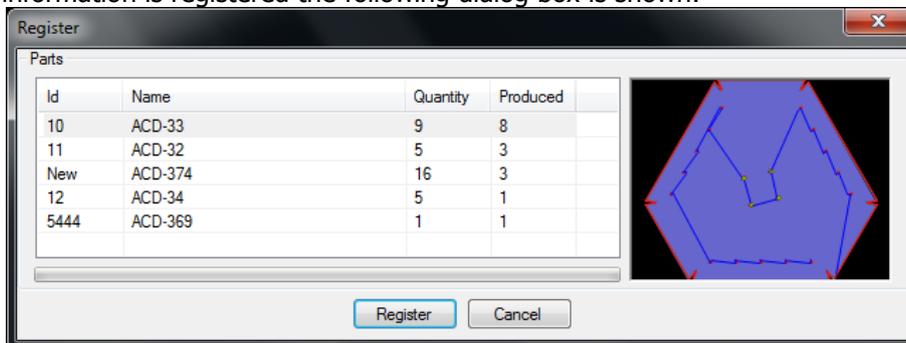
Picture 831

Include a cut order (see Section 14.2) while selecting the parts to have the "Organizer" make an automatic countdown of remain parts to produce value.



Picture 832

This example shows a mix of parts. Some are already defined in "Organizer" while others are not. When this information is registered the following dialog box is shown:



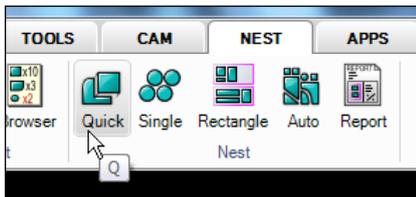
Picture 833

This will make a mix of new parts that will be registered and it will count up the produced value on these parts.

18 Nest commands

The "Nest" commands are used to automatically or semi-automatically place parts on a sheet of material and minimize the material waste.

18.1 Quick nest

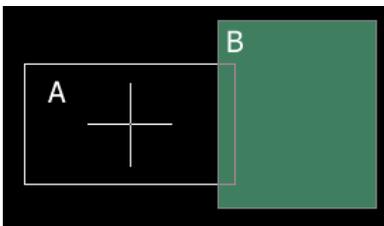


Picture 834

Shortcut "Q"

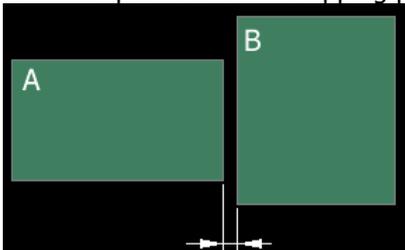
This command positions individual parts/geometry with a pre-defined distance in relation to other parts/geometries. "Quick nest" works with parts or closed geometries.

Start the command and select the part that should be moved (A).



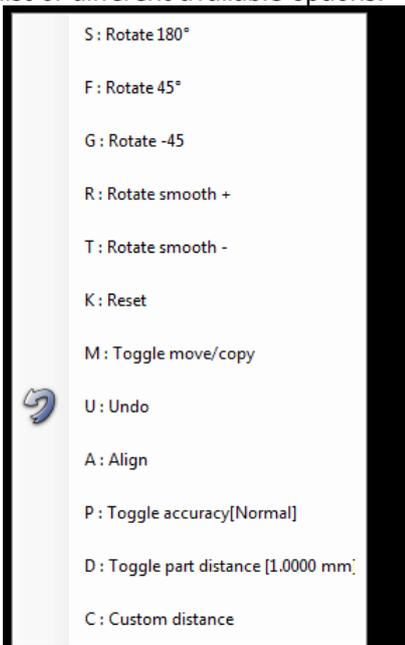
Picture 835

Move the part that it overlapping part (B).



Picture 836

Part (A) automatically moves out from part (B) to a correct distance. Click on the right mouse button for a list of different available options.



Picture 837

Most of these options are self describing.

K: Reset

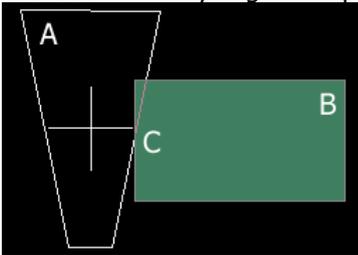
When a part is rotated, this command rotates the part to its original angle.

M: Toggle move/copy

The normal mode is copy. Use this command to switch between these two modes.

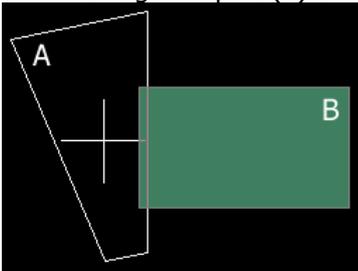
A: Align

This automatically aligns the part to an edge.



Picture 838

Hold the edges of part (A) on the edges of part (B) at the point (C).



Picture 839

Press the shortcut A and the two parts will be aligned.

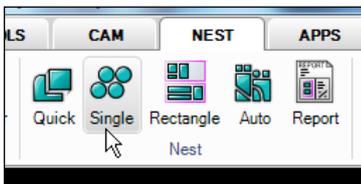
D: Toggle part distance

The material settings define a part distance (0) and there can be a custom distance. The D-key will toggle between these two distances.

C: Custom distance

Use this option to define the custom distance.

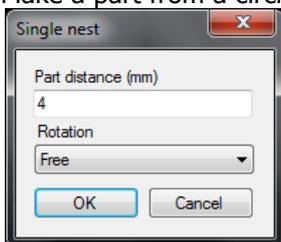
18.2 Single nest



Picture 840

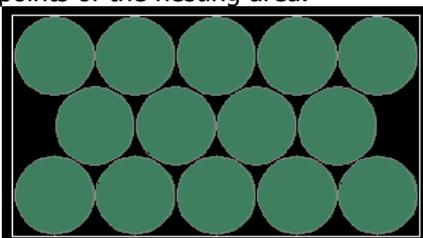
The "Single nest" nests one type of geometry on a rectangular area described by two points. It is possible to nest parts and closed geometries. There is no counter; the command will nest as many as possible.

Make a part from a circle. Start the command and select the part.



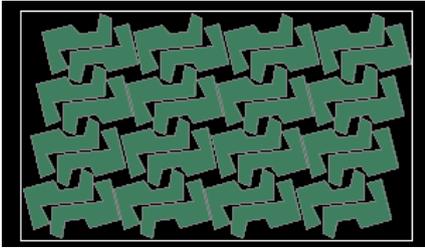
Picture 841

Set the wanted distance and rotation steps (not important for a circle). Press "OK" and define two corner points of the nesting area.



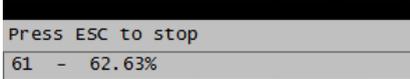
Picture 842

This could be the result when nesting disks.



Picture 843

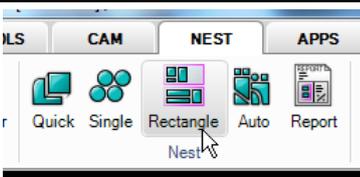
Press ESC when nesting irregular geometries to interrupt the nesting function.



Picture 844

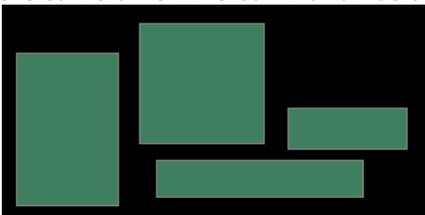
Information about the number of parts and the utilization inside the nesting area is in the lower left corner.

18.3 Rectangle nest



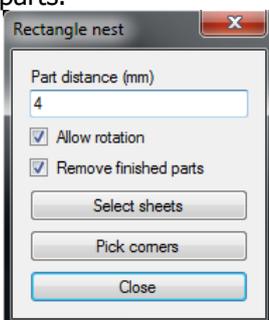
Picture 845

"Rectangle nest" is especially designed to work well when nesting rectangular parts. Select several parts at the same time. The command has a counter to avoid too many parts being nested.



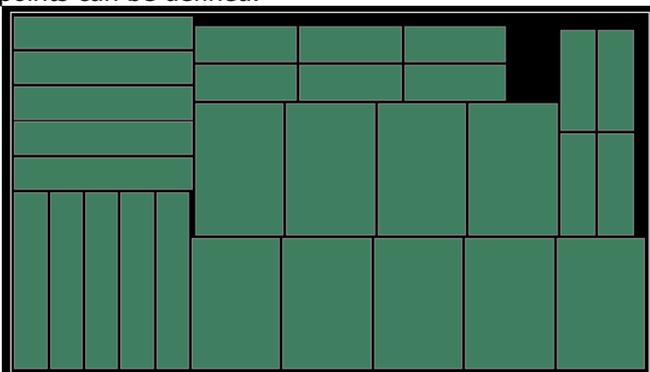
Picture 846

This example uses these four parts. The quantity is set to 10 on all parts. Start the command and select the parts.



Picture 847

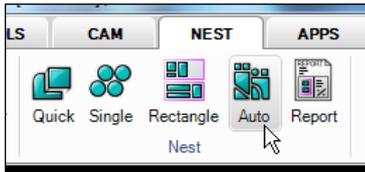
With the "Auto nest" module select one or several sheets. Otherwise only a rectangular nesting area by two points can be defined.



Picture 848

The lower left corner of the screen will show how many parts were nested and a utilization percentage based on the enclosed rectangular area used. Press ESC to interrupt the nesting.

18.4 Auto nest



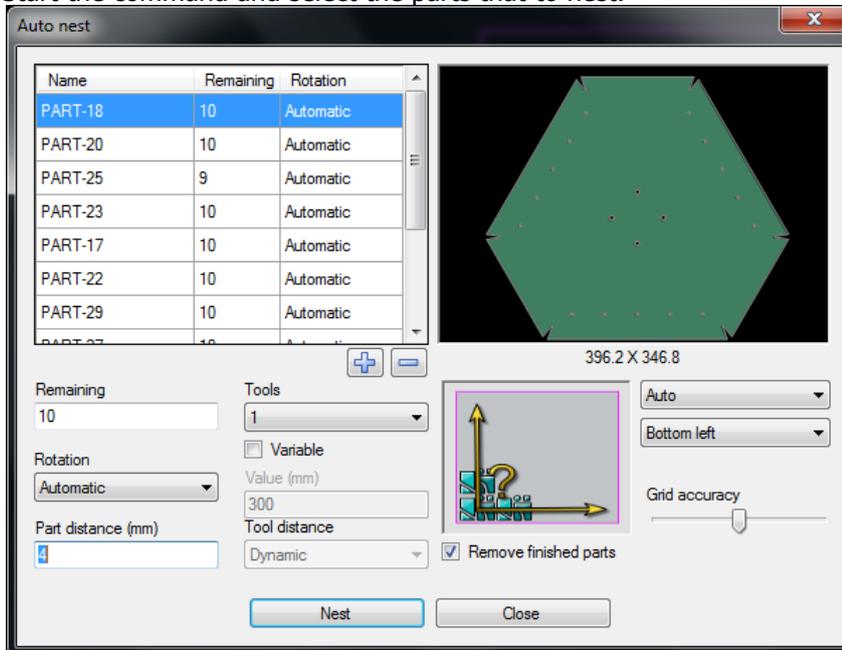
Picture 849

"Auto nest" is a true shape nest module that automatically nests multiple parts to a sheet. "Auto nest" also supports nesting with multiple heads in different combinations. It can nest parts in holes of other parts.



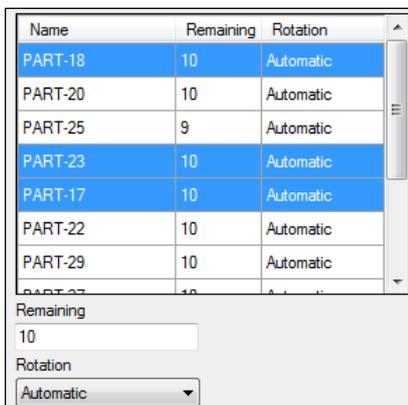
Picture 850

Start the command and select the parts that to nest.



Picture 851

18.4.1 Part settings



Picture 852

The following parameters can be changed individually on each part. Select one or several parts in the list (by CTRL or SHIFT key)

"Remaining": This shows the number of parts that still remains to nest out of the requested quantity.

"Rotation": IGEMS sets the rotation to Automatic as a default rotation. Change the rotation from the drop down menu.

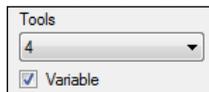
18.4.2 General settings

The following settings are common for all parts and sheet used in the nesting:

Part distance

This command always suggests the part distance specified by the material.

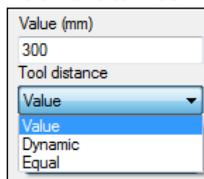
Number of tools



Picture 853

Select the number of tools to use for this nest. To use a variable number of tools set the maximum number of tools to use. If the "variable" number of tools will be used then some parts may be cut with 4 tools, other with 3, 2 or 1 tool. The algorithm always tries to use as many tools as possible.

Tool distance



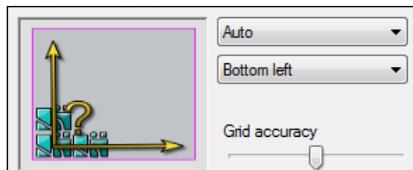
Picture 854

"Value": The same distance will be used the whole time and the distance can be set manually.

"Dynamic": A variable distance between the minimum and maximum limitation described in the machine setting.

"Equal": The sheet is divided into the number of tools and the distance is the result.

Start corner



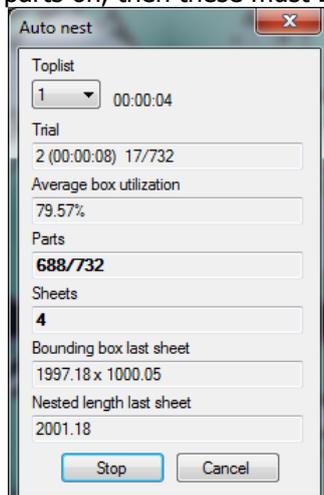
Picture 855

Various

"Grid accuracy": Auto nest internally creates a square grid on the area used for nesting. A high value means that the nesting takes longer time but part distances may be more accurate. A low value gives a quick nesting but part distances may not be as accurate.

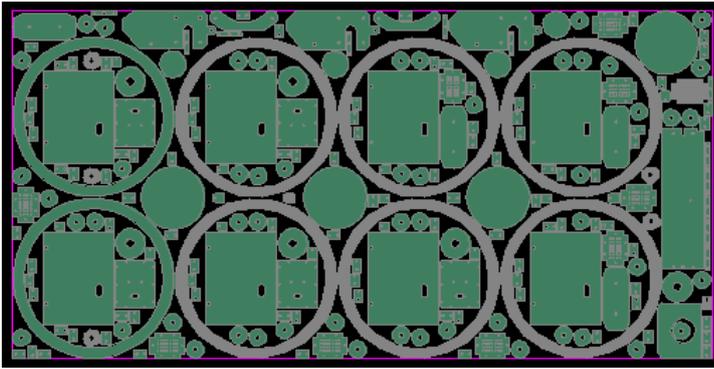
18.4.3 Execute nest

Press the "Nesting" button and select the sheets the parts are placed on. If some sheets already have some parts on, then these must be selected as well.



Picture 856

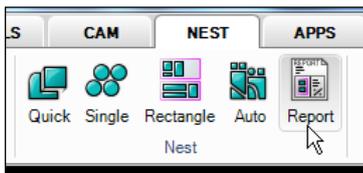
The nesting result is updated in real time. When the optimal result is reached press the "Stop" button.



Picture 857

Picture 857 shows a possible nesting result.

18.5 Nest Report



Picture 858

The "Nest report" is one of several reports generated from IGEMS. Start the command and select the sheet and the parts to generate a report.

19 APPS



Picture 859

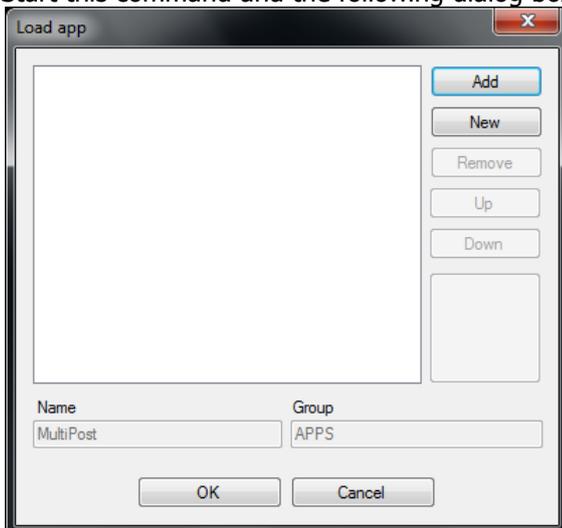
This section can be used to develop special applications and commands for IGEMS by using the programming language ILISP as developer tools. The information on how to write applications will only be available in English and is not covered in this documentation. Only information on how to load applications is provided:

19.1 Load app



Picture 860

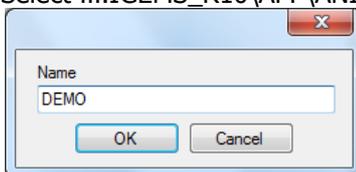
Start this command and the following dialog box is shown:



Picture 861

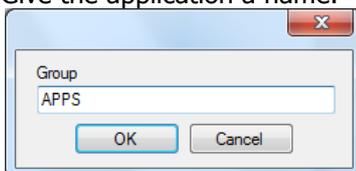
Press the "Add" button and select the LSP file to add. A small sample is in the APP folder in IGEMS program directory.

SelectIGEMS_R10\APP\ANIMALDEMO\ANIMALDEMO.LSP



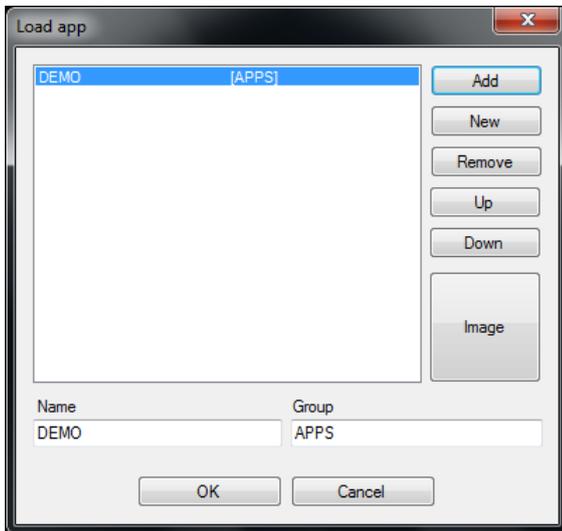
Picture 862

Give the application a name.



Picture 863

The applications can be grouped together on the menu.



Picture 864

Press the "Image" button and select an image in the same folder as the LSP file. Click "OK" to load the first application.



Picture 865

This is just an example on how to load apps.



Picture 866

If interested in how to make applications, download ILISP programming handbook from www.igems.se

20 Final words

This manual describes the software at the time of publishing. Our goal with the software is to have a high development pace. If you have an older version of the software there may be parts that do not coincide with this manual.

September 16, 2012

Copyright IGEMS AB
Sweden