IGEMS Release 8

CAD/CAM/NEST

User Manual

2008-10-09

Chapter 1.
Installation

Requirements

Before you start the installation, be sure that you have a wheel mouse and a computer with Windows 2000, Windows XP or Windows Vista. (Windows 98 and Millennium is NOT supported). We recommend a processor with at least 500 MHz and 512 MB of available system RAM. The software needs a hard disk space of about 50MB. The faster computer you have, the more you will like to work with IGEMS.

Step 1: Install the IGEMS-software

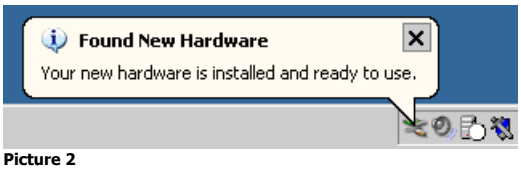
Extract all files in the installation if downloaded from our website or insert the IGEMS CD-disk if available. Run the Install.exe file.

Picture 1

Install the IGEMS Software and follow the information on the screen.

Step 2: Install Hardware lock driver

If you don't have a Dongle (Hardware lock) or if you already have installed the driver, then go to next step. Before you start the driver installation, remove any USB-dongle from the computer. Follow the instructions on the screen. Insert the USB dongle after installation of driver is completed.



Picture 2

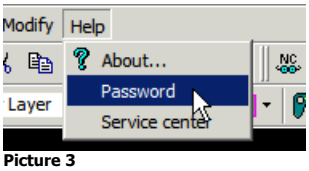
If everything is OK you should have a message like above image.

Step 3: Start IGEMS

Trial version

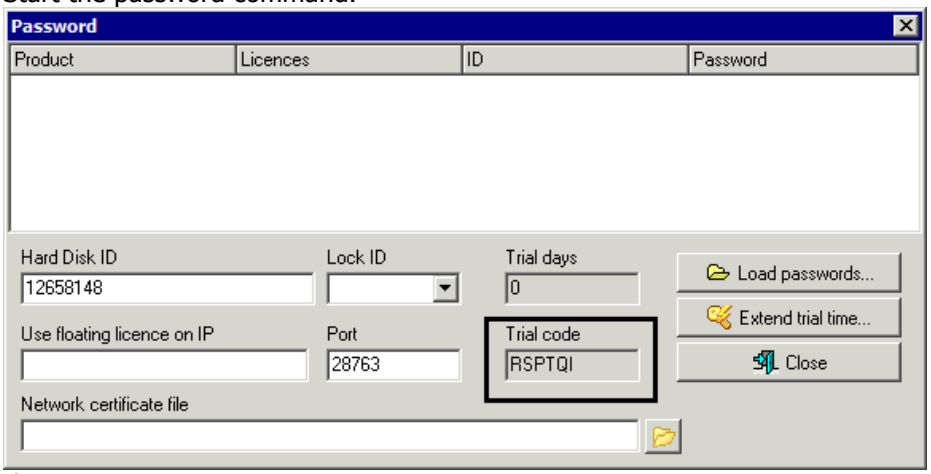
The only difference between the trial and the industrial version of IGEMS is that it is not possible to save anything in the trial version. If you do not have any password then IGEMS automatically will be running in trial mode.

Trial or industrial version



Picture 3

Start the password command.



Picture 4

Load a password file.

If you have received a permanent password file, then press the "Load passwords" button and select the file (xxx.PWD). The password file is based on information from the Hardware lock ID or the Hard Disk ID and the modules that you have.

Trial code

If you want to test IGEMS for a limited time you will need a temporary code. Send the Trial code to IGEMS Software AB and a temporary code will be returned to you. Activate the password file by pressing "Extend trial time" and enter the code.

Step 4: Restart IGEMS

Restart IGEMS to complete the installation.

Floating License Manager

Floating license is an extra option to IGEMS that makes it possible to handle all licenses from one computer used as server. Example: If you have one license of IGEMS and you have installed IGEMS on several computers then it's possible to run IGEMS from any computer, but only on one computer at a time.

Step 1: (On server)

Install IGEMS floating license in a computer that are connected to a network. You do not have to install the IGEMS Software.

Step 2:

Install the Hardware lock.

Step 3:

Start the Floating License Manager.
Write down the IP and the Port number of the server. Finally activate the toggle Auto start.

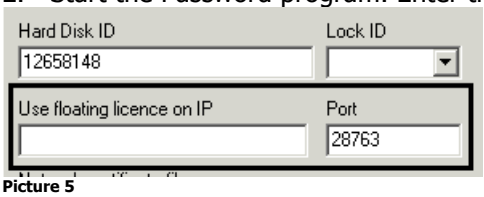
Step 4:

Load your password file for the Floating License Manager. (It's a button where this command can be started).

Step 5: (On clients)

Repeat the following procedure on all computers that should be connected:

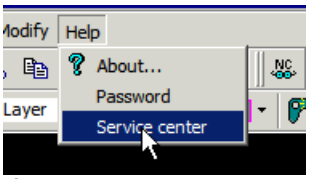
1. Install IGEMS Software.
2. Start the Password program. Enter the IP and the Port number



Picture 5

Updates

To be sure that you have the very latest version, please check for a new versions. This can be done from our service center. You need to be connected with Internet for using this service.

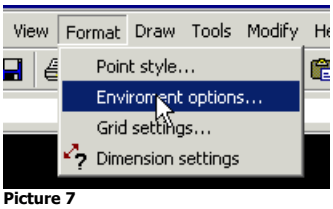


Picture 6

When you update just follow the instructions on the screen. Always install minor updates in same directory as the previous version. No information changed by the user is overwritten when making an minor update. If you install a new release (R6, R7, R8) then use a new folder. The old version is not uninstalled, you can run the old and new Releases at the same time.

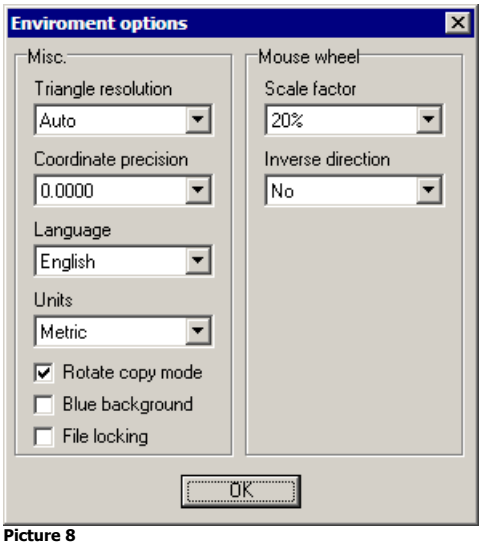
Chapter 2.
General CAD functionality

Language and units



Picture 7

From this command you can make the general settings for the CAD/CAM system like Language and Units. IGEMS currently support 12 different languages: Czech, Dutch, English, Finnish, French, German, Greek, Italian, Polish, Russian, Spanish and Swedish.



Picture 8

The file locking can be activated and it handle only the IGEMS drawing files (ACD).

Zoom and Pan

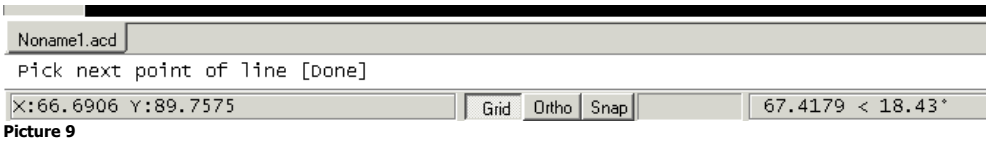
You need a wheel mouse to work properly with IGEMS.

- Zoom: You will zoom in and out by rotating the mouse wheel.
- Pan: Press and hold the mouse wheel down to pan.
- Zoom extents: If you double click with the mouse wheel, then IGEMS will zoom up the drawing in full screen.

If the mouse 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.

Command line

There is no command line in IGEMS but sometimes the program asks for additional information. Be sure to always read the information at the line.

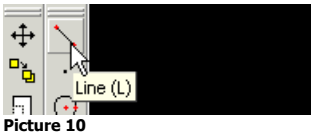


Picture 9

When something is inside brackets [Example] then this is the default value. This can be accepted by using the Space or the Enter keys.

Short keys

By holding the mouse pointer over a command, you can see the short key for this command.

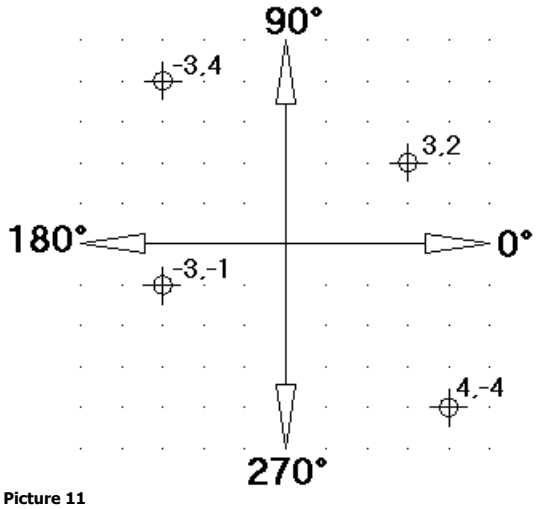


Picture 10

If you want to repeat the same command, the space or Enter keys can be used as a short key for last used command.

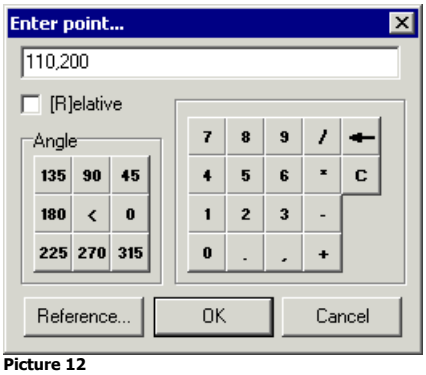
Coordinate input

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



Picture 11

If IGEMS expects coordinate input and you type digits on the keyboard the following dialog box is shown.



Picture 12

Absolute coordinates

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

Relative coordinates

This always refers to the last used point, so it is rather a distance then a coordinate. It is typed @X, Y as in following example:
@110, 218.9

Absolute polar coordinate

Always refer to IGEMS fixed zero point. It is typed DIST>ANGLE as in following example:
150<45

Relative polar coordinate

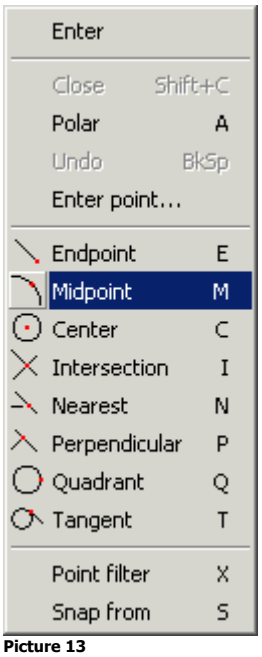
Always refer to the last used point. It is typed @DIST>ANGLE as in following example:
@180<225

Direction coordinate

This is the fastest way of enter coordinates. This is used if you just type one value @DIST or DIST. This method takes the pointing direction and it is often used together with Ortho mode. Example:
@200 or 200.

Object snap

If you have a command active that need coordinates input, then you can click on the right mouse button. This will show you the list of object snaps that can be used in IGEMS.



Picture 13

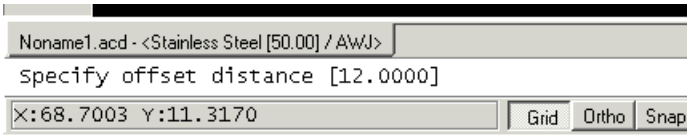
The different options finds coordinates on the drawing. It is also possible to use short key for object snap. This is done by typing E, M, C, N, P, Q or T instead of using the right mouse button.

Point filter
By using the point filter option (short key X) you can extract individual X and Y coordinates from different points on the drawing to create a new composite 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.

Distance input

Several commands in IGEMS ask for a distance value.
Example:



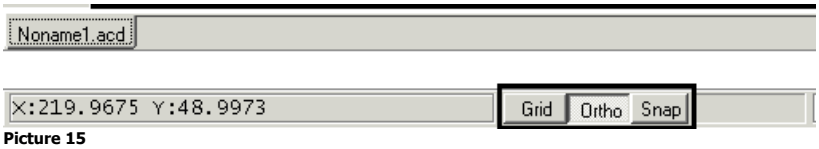
Picture 14

In this example you have 3 choices.

- You can accept the value inside brackets [12.0000] by using the space key.
- You can enter a new value.
- You can pick two points by measuring a distance on the screen.

Grid, Ortho and Snap mode

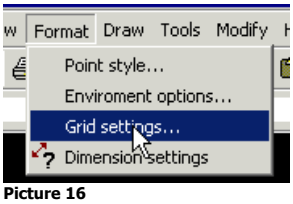
These modes can be activated or deactivated from following buttons or with the short keys F7, F8 and F9.



Picture 15

Grid mode

This mode shows a grid on the screen. The size and distance between the grid points can be modified from the grid settings command.



Picture 16

Ortho mode

Sometimes it helps a lot having the possibility to pick points that are located in vertical or horizontal direction. This can be done by activating the Ortho mode.

Snap mode

Whit this mode activated, the cursor will only snap to the grid points.

Select objects

Many commands need objects as input. For example: Erase, Move, Copy and others. The select object function IGEMS works as follows:

Select

- Select by picking: Pick on an object.
- Select by window: Needs two points. Click where there is no object, next point must be to the right side. The objects must be completely inside the window to be selected.
- Select by crossing: Needs two points. Click where there is no object, next point must be to the left side. It is enough if just a small part of the object is inside the window to be selected.

Deselect

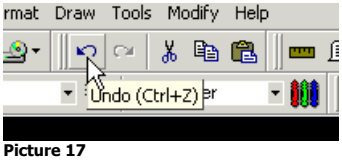
This is done in the same way as Select except from that the SHIFT key must be activated at the same time.

You can mix select and deselect, when your selection is done then press space.

Automatic base point

Many commands has an automatic base point selection:
Specify base point [auto]
You have then two choice. You can pick a point as base point or you can except the auto option by pressing space or enter. The new option take the center of the extends of all selected objects.

Undo and Redo

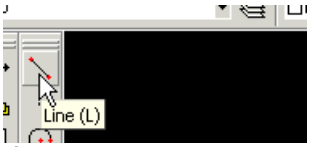


Picture 17

IGEMS has an Undo and Redo system that make it possible to Undo and Redo up to 10 steps backward.

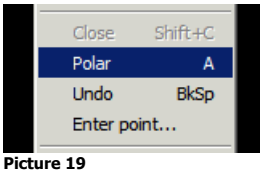
Chapter 3.
Object creation

Line (L)



Picture 18

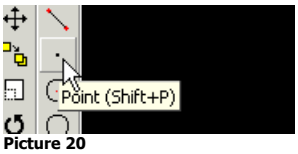
This command asks for a start point and then next point. The command must be interrupted by space, enter or escape. Backspace can be used for undo the last line segment there is also a special Polar option (Short Key A).



Picture 19

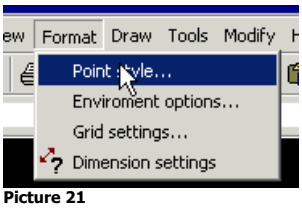
With this option you can enter a relative angle and a distance.

Point (Shift P)



Picture 20

Ask for input of point positions. The point command must be interrupted by space or escape.



Picture 21

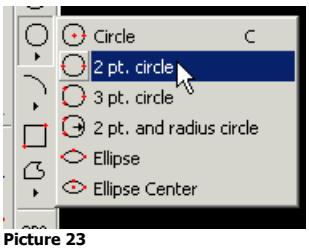
This visual presentation of the point can be modified from the Point style command on the Format menu.

Circle (C)

Picture 22

This command requires a center point and a radius/diameter. You can toggle between radius/diameter modes by using the D key on the keyboard.

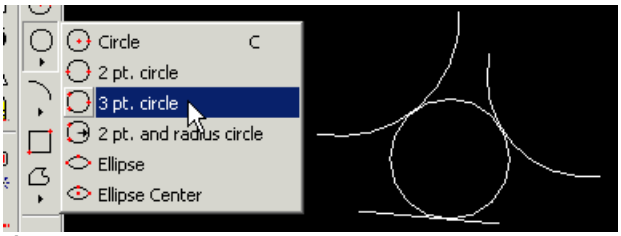
Circle by 2 points



Picture 23

This command makes a Circle that goes through two points.

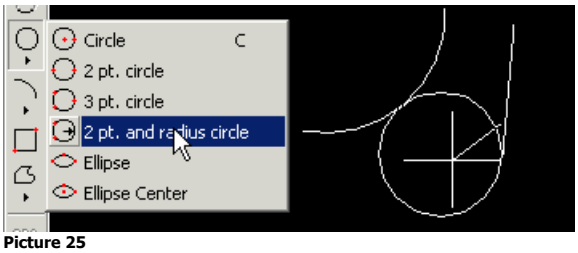
Circle by 3 points



Picture 24

This command makes a circle that goes through three points. It is often used together with the snap mode tangent. This makes it possible to find center points that are difficult to define by other methods.

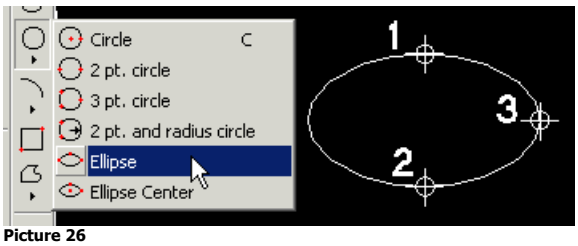
Circle by 2 points and a Radius



Picture 25

This command makes a circle with a specific radius that goes through two points.

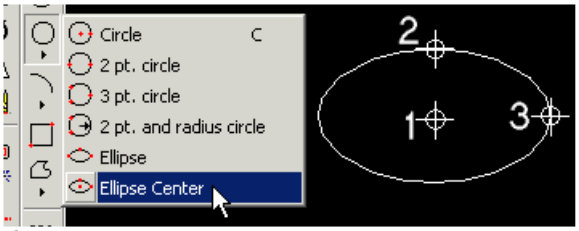
Ellipse



Picture 26

This command makes an ellipse by defining first axis diameter and then the second axis radius. The ellipse is automatically converted to a polyline.

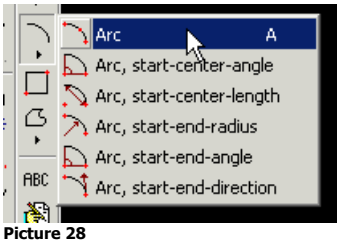
Ellipse by center



Picture 27

This command makes an ellipse by center and two axis radius. The ellipse is automatically converted to a polyline.

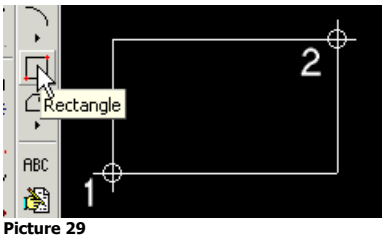
Arc (A)



Picture 28

IGEMS support many ways of creating arcs.

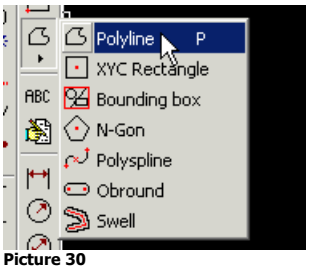
Rectangle



Picture 29

This command makes a polyline rectangle from 2 opposite points.

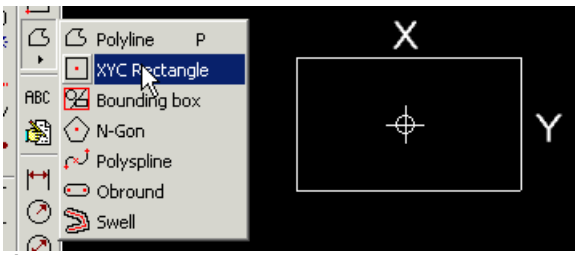
Polyline



Picture 30

A polyline is chain of lines and arcs that are joined together into one object. When drawing a polyline you can toggle between line and arc mode by typing A on the keyboard.

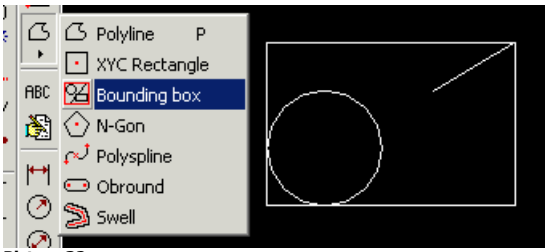
Rectangle by X, Y and center



Picture 31

This command makes a polyline rectangle by asking for X-size, Y-size and then a insertion point and rotation angle.

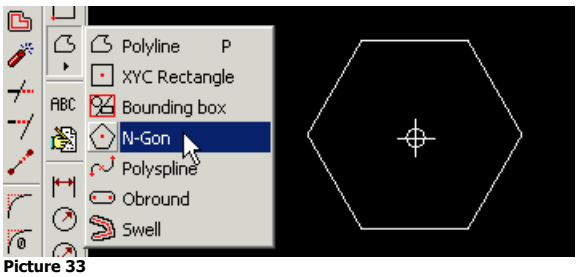
Bounding box



Picture 32

This command makes a rectangle around selected objects.

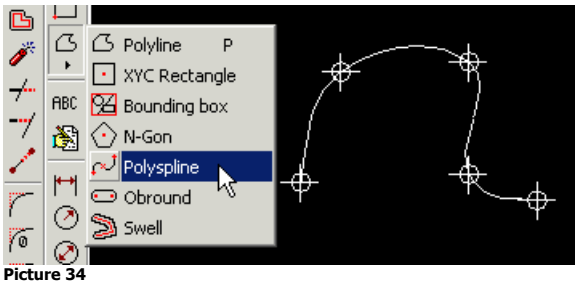
N-Gon (Polygon)



Picture 33

This command make a polygon by asking for number of sides, radius and rotation angle. You can toggle for outside or inside radius type by typing O on the keyboard.

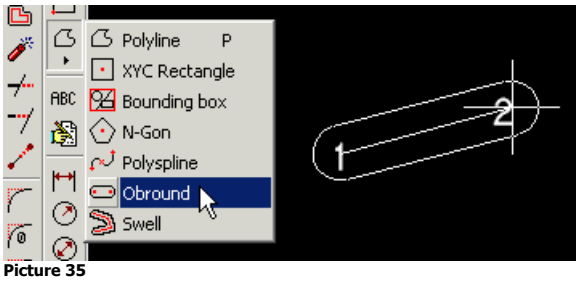
Polyspline



Picture 34

This command makes a spline that goes through a set of points. When the command is ended the spline is converted to a polyline.

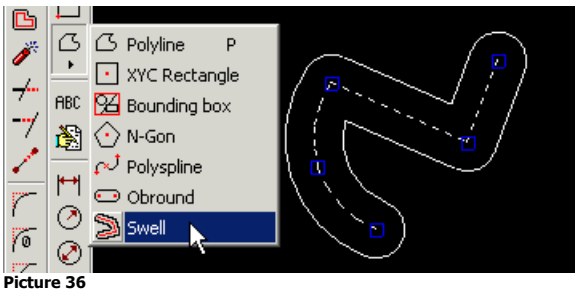
Obround



Picture 35

This command makes an obround by asking for two points and a radius.

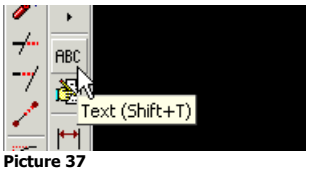
Swell



Picture 36

The Swell command makes a trimmed offset around a selected object. The command asks for objects and a radius.

Text (Shift T)

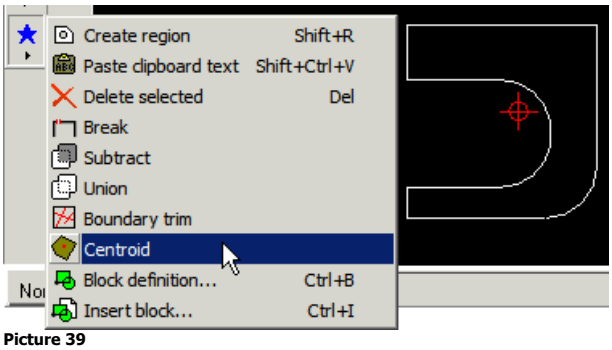


Picture 37

Before you enter the text, the command asks for insertion point and text size. By clicking right mouse button you can change rotation angle and size.

Picture 38

Centroid



Picture 39

This command inserts a point in the centroid of a closed object.

Chapter 4.
Object position

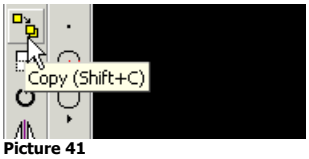
The commands described in this chapter are using functions described in Chapter 2. (Select object, Coordinate input, Object snaps and so on).

Move (M)

Picture 40

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

Copy (Shift C)



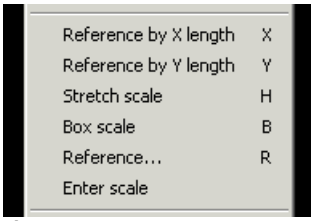
Picture 41

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

Scale (S)

Picture 42

This command asks for object to scale, then a base point and a scale factor. If you click right mouse button instead of entering the scale factor you will have following options.



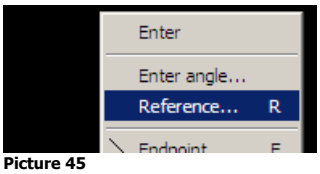
Picture 43

- Reference by X length:
  This option automatically calculates the scale factor. Enter the outermost size in X.
- Reference by Y length:
  This option automatically calculates the scale factor. Enter the outermost size in Y.
- Stretch scale:
  With this option you can enter different scale factors in X and Y.
- Box scale:
  This option allows you to enter different 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.
- Reference:
  Asks for a reference length, and then a new length. All objects will be scaled by reference to this two length.
- Value:
  Enter a scale factor (2=double size, 0.5 will scale to half size).

Rotate (R)

Picture 44

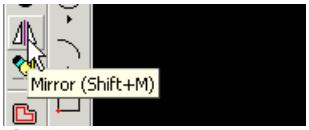
This command rotates an object around a specified point. Instead of entering the new rotation angle you can use the Reference option.



Picture 45

With this option you can make a relative rotation from a Reference angle and a new angle. As standard the rotate command is in copy mode. It is possible to turn if the copy mode from the environment settings (se Picture 8).

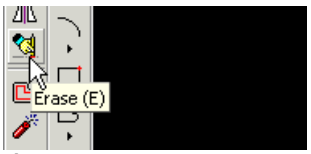
Mirror (Shift M)



Picture 46

This command asks for object and then a mirror line. The selected objects are still selected after the mirroring, if you want to delete the original then use the Delete key.

Erase (E)



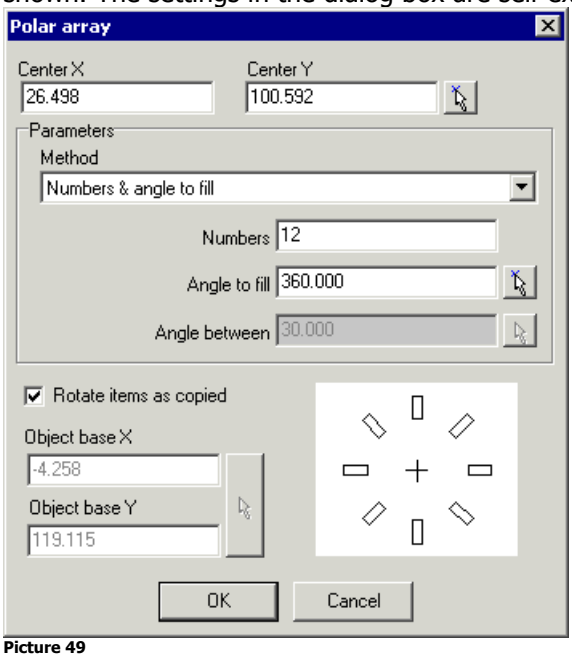
Picture 47

This command erases the selected objects from the drawing.

Polar Array

Picture 48

This command copies the selected objects in a polar array. Following dialog box is shown. The settings in the dialog box are self explained.

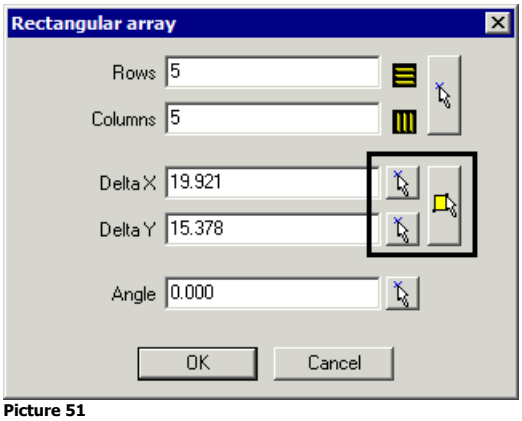


Picture 49

Rectangular Array

Picture 50

This command copies selected object in a Rectangular Array. Following dialog box is shown.



Picture 51

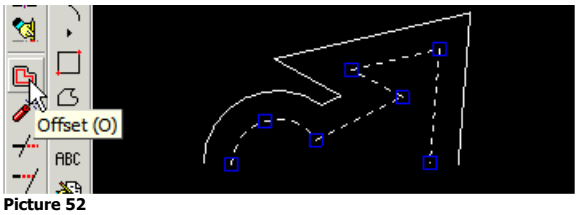
The settings in the dialog box are self explained, but here are some remarks.

Delta X and Y

The default distance is the size of the selected object in X and Y. By clicking on some of the buttons shown in the picture you can point out a distance on the drawing.

Chapter 5.
Modify objects

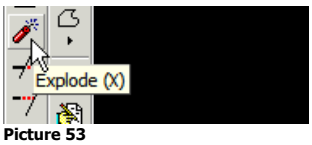
Offset (O)



Picture 52

This command makes an offset of the selected object to a specified distance. The command asks for a distance, object and a side. If you hold down the CTRL or/and SHIFT key when picking the side you can make a non trimmed offset or offset with outside arcs.

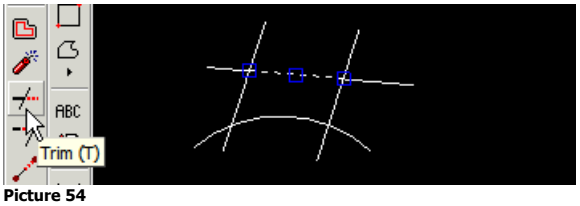
Explode (X)



Picture 53

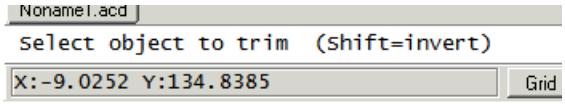
This command explodes a block or a polyline to separate objects. The opposite command to Explode is Join.

Trim (T)



Picture 54

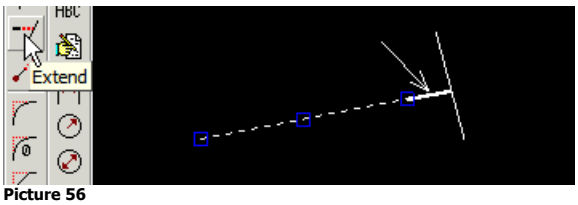
The Trim command deletes the portions of an object that intersects with other objects.



Picture 55

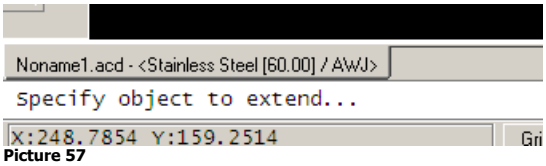
You can now click on that portion of the object that you want to delete. If you hold down the Shift key the trim function is inverted, the complete object except the portion that you click on will then be deleted.

Extend



Picture 56

The Extend command extends objects to it intersect with other boundary's.



Picture 57

When selecting the bounding edges, you can select the objects to extend.

Lengthen

Picture 58

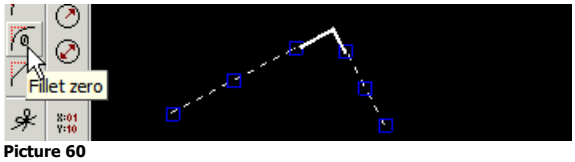
This command lengthen selected object.

Fillet (F)

Picture 59

This command makes a tangential arc between two objects. If the object is a polyline and you hold down the CTRL-key then all corners will be selected.

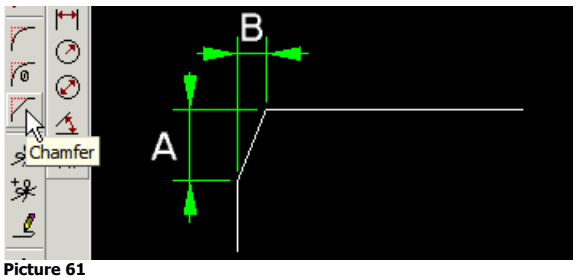
Fillet zero



Picture 60

This command does the same as Fillet except that the radius is always set to 0.

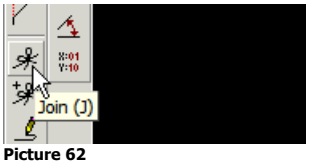
Chamfer



Picture 61

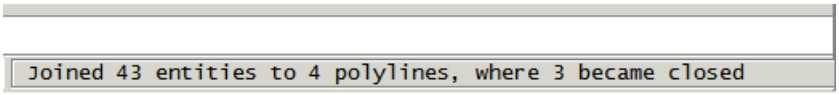
This command asks for two distances (A and B) and two objects. A chamfer line is inserted between the two objects.

Join (J)



Picture 62

This command joins objects to a polyline. Only objects that has no gap or overlap are joined together.



Picture 63

The result of the Join command can be found on the information line. The opposite command to Join is Explode.

Extended Join

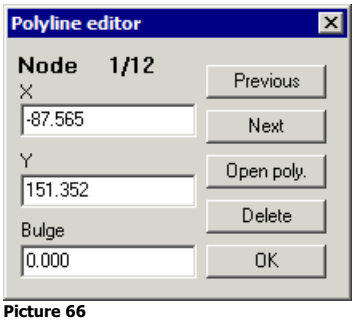
Picture 64

Extended join is similar to join but has also a variable tolerance. If there is a gap or an overlap shorter then the tolerance then the command will make a line between the objects.

Polyline editor

Picture 65

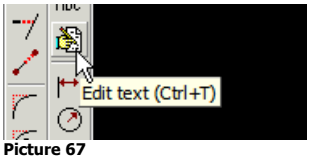
If you want to analyze or modify a polyline you can use this command.



Picture 66

You can analyze each object in the polyline by clicking Next or Previous.

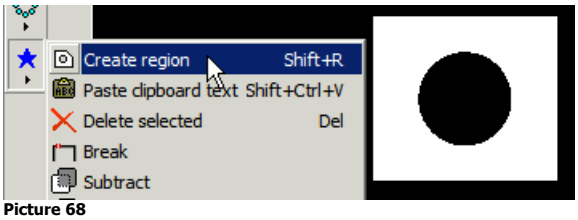
Edit text



Picture 67

This command makes it possible to change the contents of existing text.

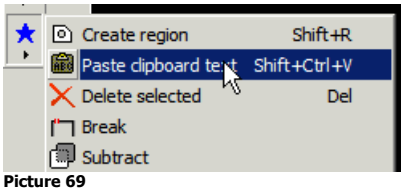
Create Region



Picture 68

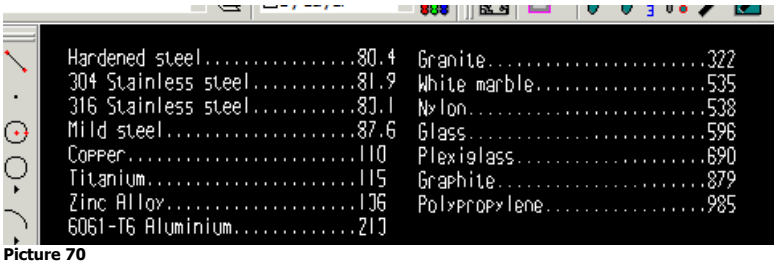
This command creates a region from one or several closed polylines.

Paste clipboard text



Picture 69

This command can paste text into IGEMS. The text is inserted as a block. If you need to modify the text then you must explode the block before.

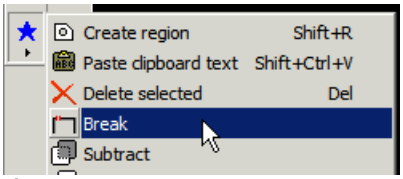


Picture 70

Delete (Del)

Delete can only be used on selected objects when no other command is used.

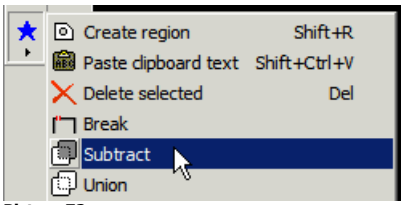
Break



Picture 71

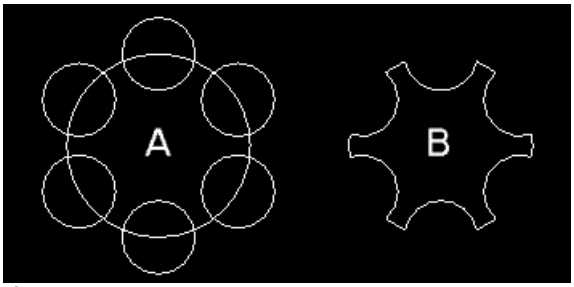
This command breaks up objects in several portions. The command asks for an object and a break point.

Subtract



Picture 72

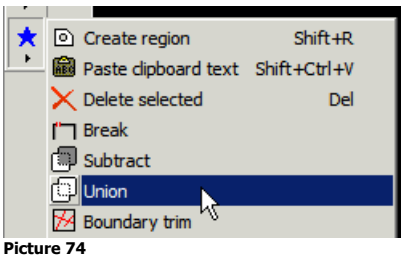
This command removes the area from a closed object by subtracting the area from other closed objects.



Picture 73

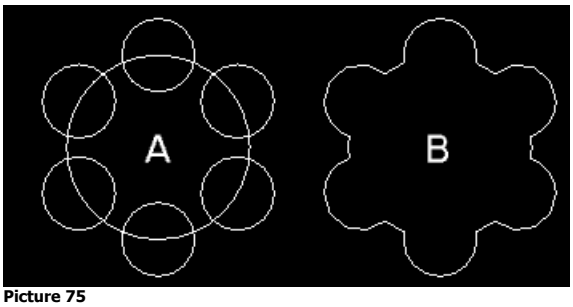
The above image shows the result of the Subtract command.

Union



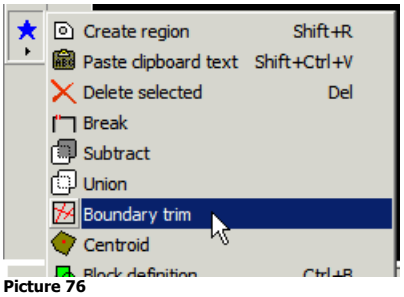
Picture 74

The Union command creates a new closed object by union all selected closed object.



Picture 75

Boundary trim

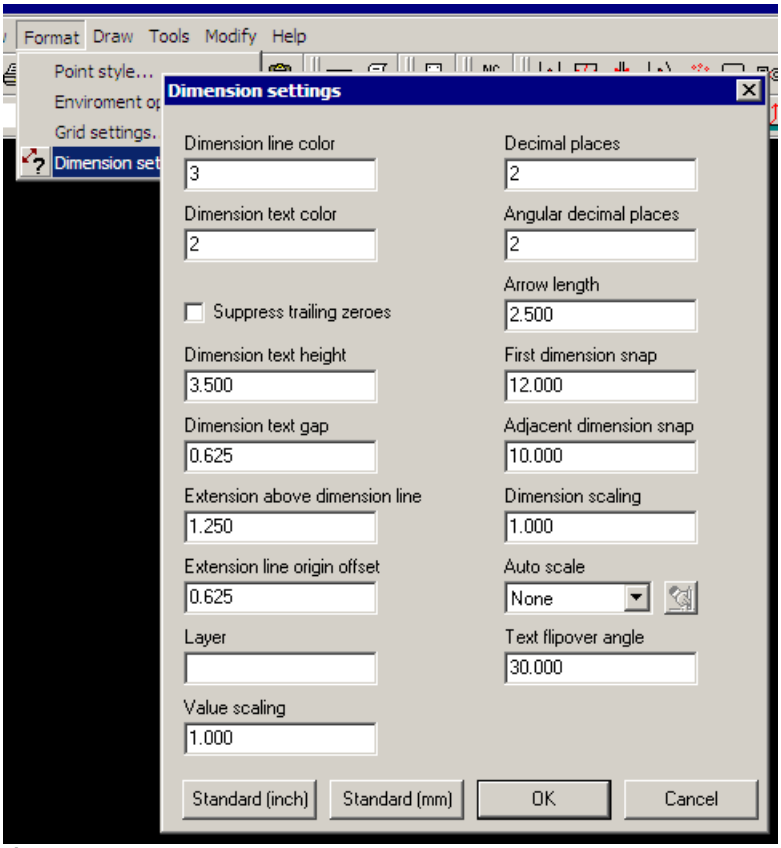


Picture 76

This command trims and erases all parts that are inside or outside a closed object.

Chapter 6.
Dimension command

The general settings for all Dimension command are handled by the Dimension Settings command.



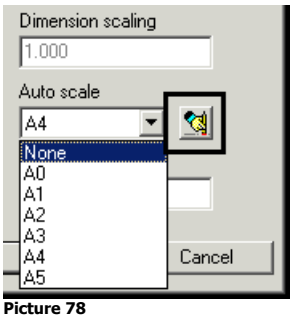
Picture 77

Most of the settings are self explained but here are some remarks.

Object color and layer

It is possible to define a color for Dimension lines and text. This color will automatically be used when using dimension command. You can also define a Layer that always will be used for the dimension object. By using a special layer it is easy to turn dimension information on and off.

Auto scale



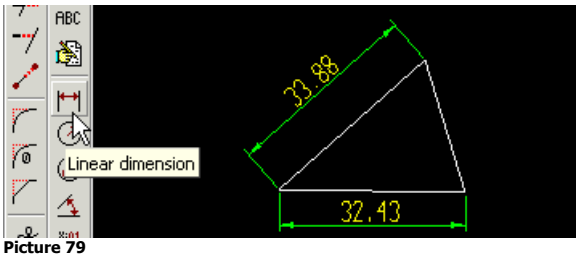
Picture 78

If you set the Auto Scale to a specific paper size, the Dimension scaling will be automatically calculated. The calculations are done when you use the first dimension command. If you press the button shown in previous picture the dimension scaling will be recalculated.

Value scaling

By using the new variable "Value scaling" you can force IGEMS to suggest a scaled value of the dimension text.

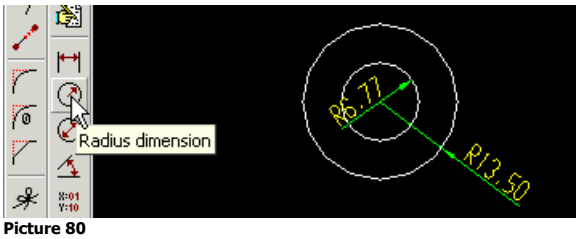
Linear dimension



Picture 79

This command makes Horizontal, Vertical or Aligned dimensions. If you want to measure an existing object, you can just pick on that object. If you for example want to measure the distance between two objects, then press space and enter two points.

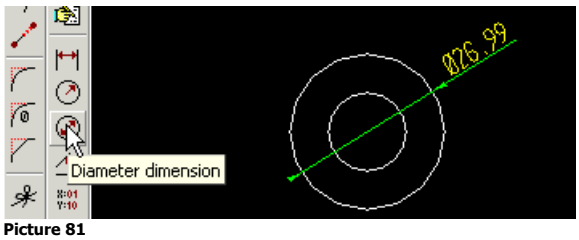
Radius dimension



Picture 80

This command makes Radius dimensions.

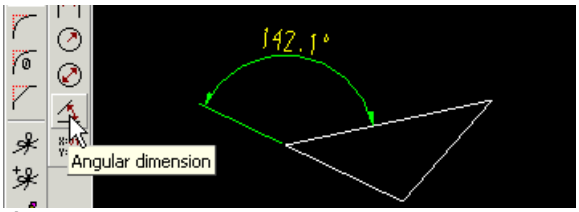
Diameter dimension



Picture 81

This command makes Diameter dimensions.

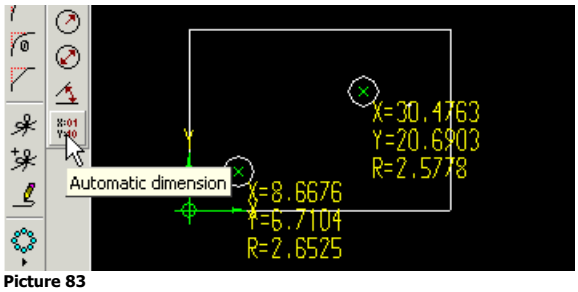
Angular dimension



Picture 82

This command makes an Angular dimension from the information of two lines or polylines. If you have no object you can press space and define the vectors by picking three points

Automatic dimension



Picture 83

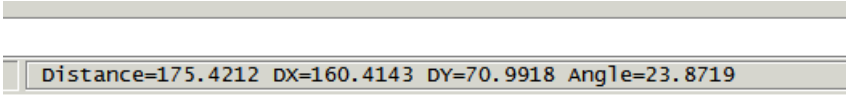
This command makes a non standard dimension information by adding position in X,Y and the radius of the objects.

Chapter 7.
Various commands

Distance

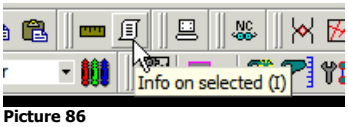
Picture 84

This command asks for two points and present information at the information line.



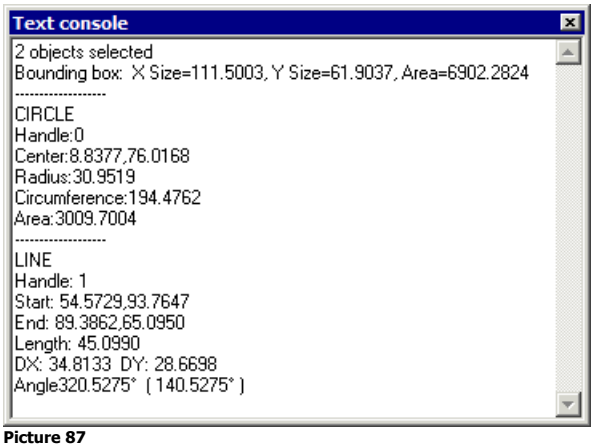
Picture 85

Info (I)



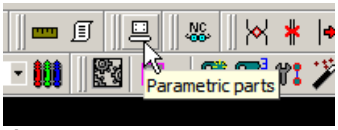
Picture 86

This command shows information about selected objects.



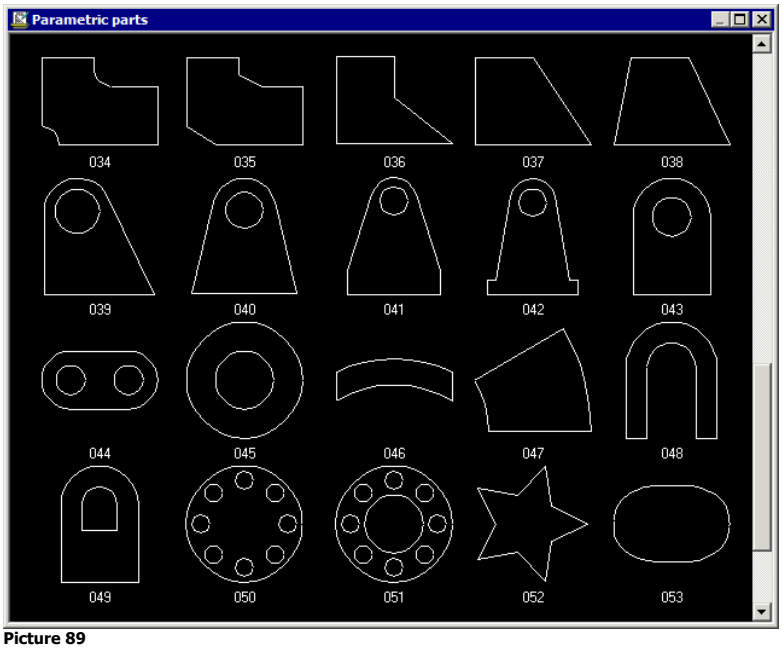
Picture 87

Parametric parts



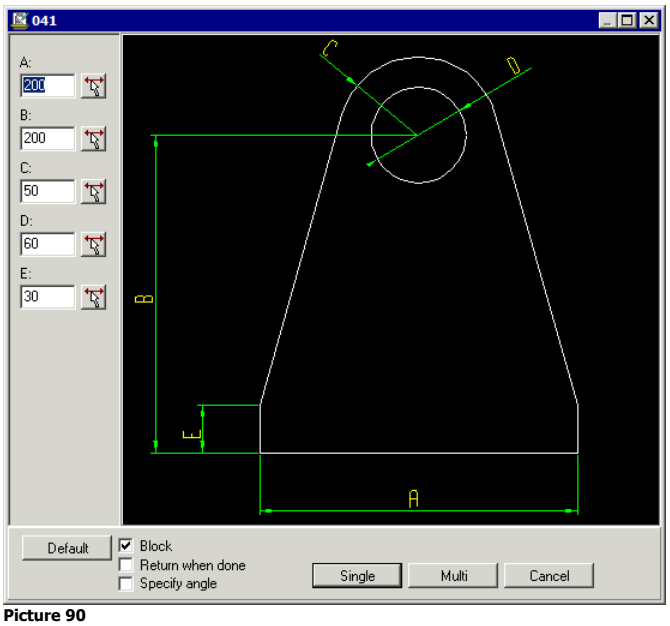
Picture 88

With this command you can create parametric geometries. It is possible to make new parametric parts templates. Contact IGEMS if you need more information on how to make your own shapes.



Picture 89

Click on a symbol



Picture 90

The geometry can be inserted as a block or as separate objects. It is also possible to insert the geometry with an optional rotation angle.

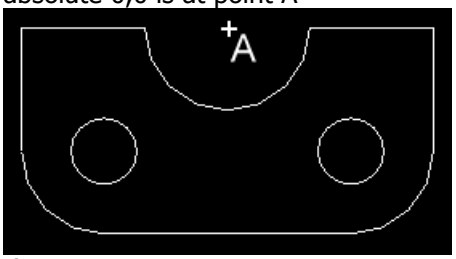
Shape library

If you are frequently inserting the same shape of geometry it is now possible to store the shape in the Shape library.

Add shapes to the library

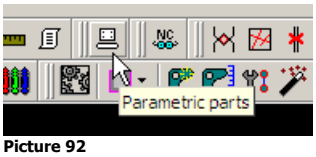
This is the workflow to add new shapes:

Step 1:
Create the geometry.
The position of the geometry is important since the absolute 0,0 will become the base point for the shape. In this example we have moved the geometry so absolute 0,0 is at point A-



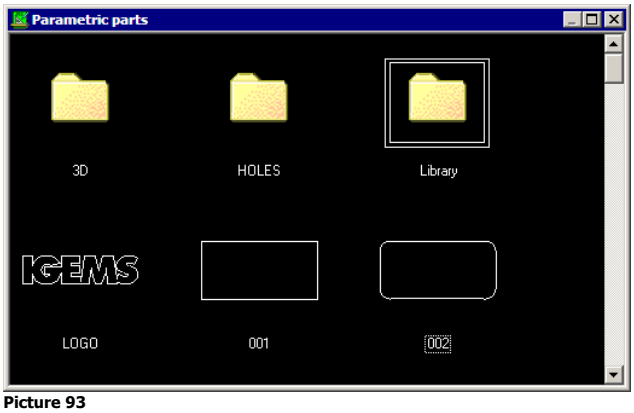
Picture 91

Step 2:
Start the Parametric part command



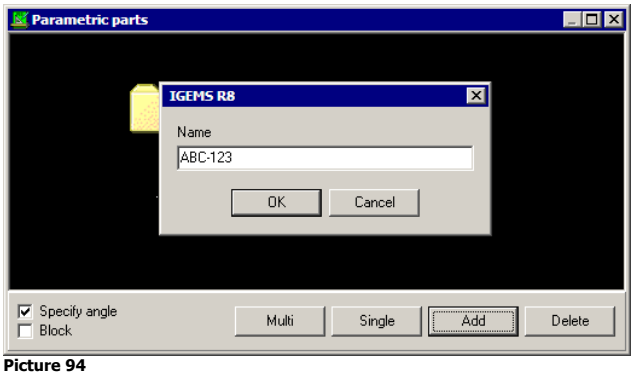
Picture 92

Press the Parametric part button



Picture 93

Press the Add button and enter a name for the new shape.

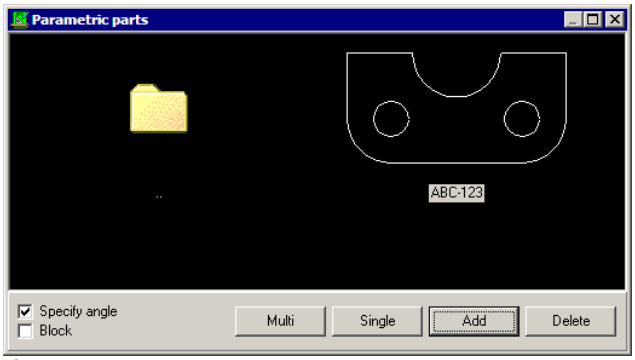


Picture 94

Finally select the geometry.

Select objects (Select the geometries)

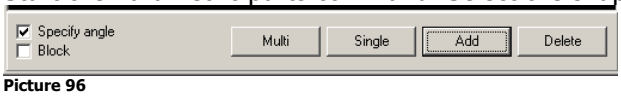
The Shape is now stored in the library and can be reused at any time.



Picture 95

Insert shapes from the library

Start the Parametric parts command. Select the shape you want to insert.



Picture 96

If you want to insert only one shape, then use the Singe button. If you want to insert multiple shapes then use the Multi button. From this window you can also control if you want to modify the angle of the inserted shape or if the shape should be represented as a block.

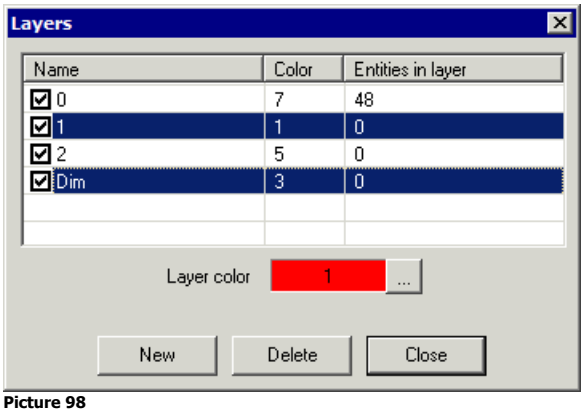
Chapter 8.
Layers, colors and plotting

Layer (Y)

Picture 97

IGEMS supports unlimited number of layers. If you should create DXF-files that should be used in other CAD systems, be sure not tu use space in layer names.

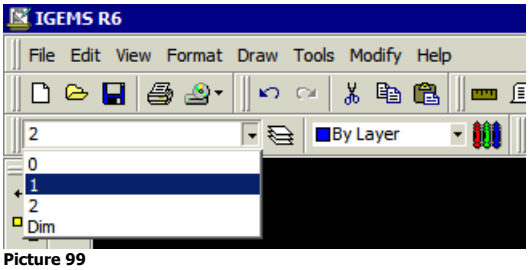
Layer options



Picture 98

From this dialog box you can create new layers, delete layers and change default color of a layer. If you delete a layer then all objects on that layer will be deleted.

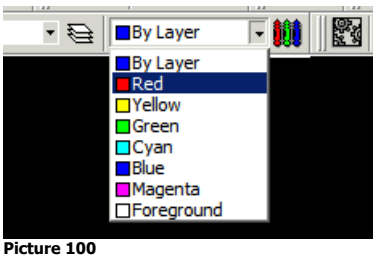
Change active layer



Picture 99

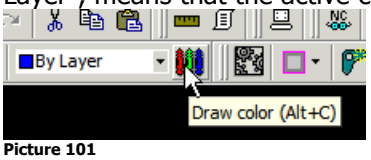
You can change active layer from the layer list. All new objects will be placed in the new layer.

Color



Picture 100

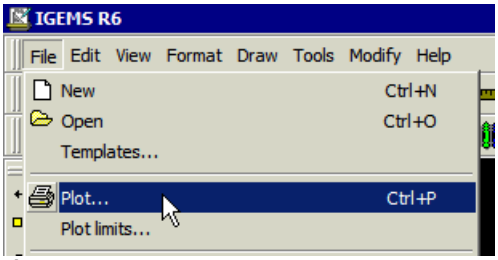
You can change color by selecting any color from the list. Setting the color to "By Layer", means that the active color will be the color defined in the layer setting.



Picture 101

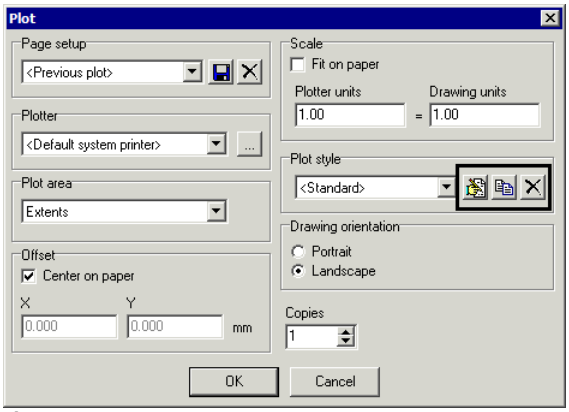
If you want to use other colors than you have in the list then press the Draw color button.

Plot (Ctrl+P)



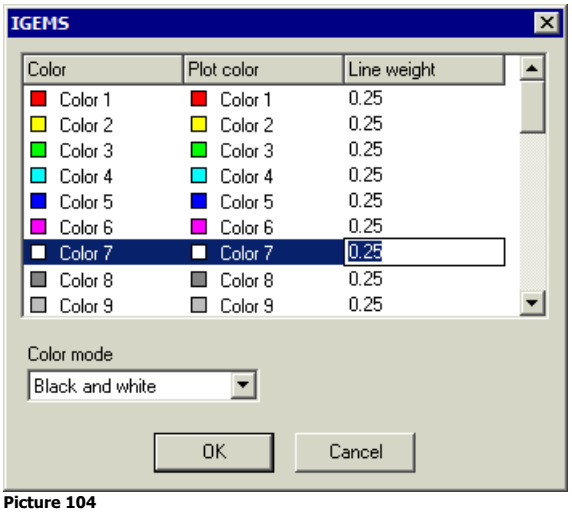
Picture 102

The plot command will open following dialog box.



Picture 103

The buttons can be used for handling different plot styles.

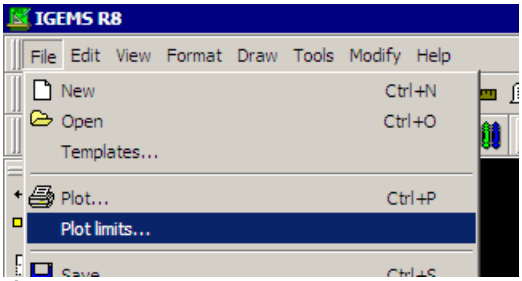


Picture 104

From this dialog box you can map the colors on the screen to different colors on the paper. You can also set up different line weights. The color mode controls how the different colors should be plotted on the paper.

Plot limits

If you always use same plotting area over and over again then it's a good idea to specify a plot limit area.

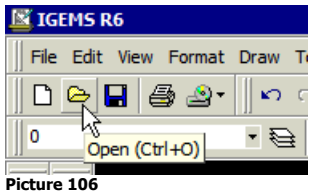


Picture 105

If you use this feature, then use the Plot area "Limit" in the Plot command.

Chapter 9.
File and Block handling

Open



Picture 106

Following files can be opened as standard in IGEMS:

- ACD (Files created by IGEMS),
- DWG (AutoCAD drawing file).
- DXF (Draing Exhange Format)

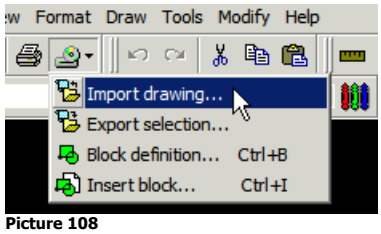
If you have the Data Exchange option then you can also open following files:

- CBF (Created by CAMbAL cut)
- GEO (Created by Tops)
- TAG (Created by Taglio)
- ORD (Created ny Omax)
- MEC (Created by Lantek)
- IGS (IGES files)
- PRT (Created by Admicut)
- PS (Postscript)
- EPS (Encapsulated postscript)
- WMF (Windows Meta File)

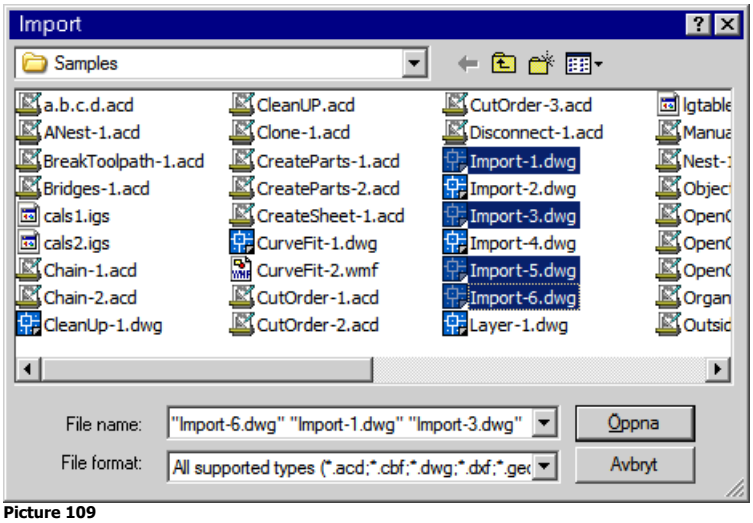
When you open a file, it will be opened in a separate drawing window. You can have many drawings open at the same time.

Picture 107

Import drawing

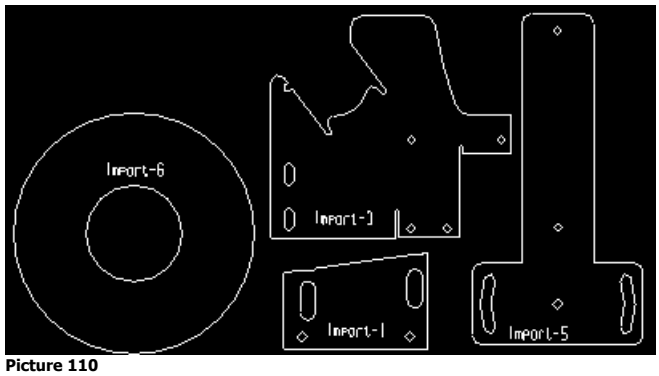


Picture 108



Picture 109

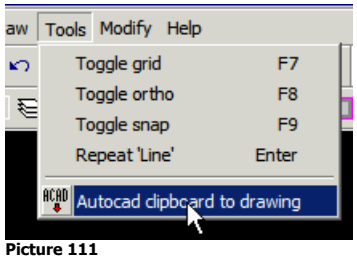
When using Import all selected files are inserted in current drawing.



Picture 110

If you hold down CTRL or SHIFT when inserting the drawings, then you will have an extra insert that contains a text with the file name.
The difference between CTRL and SHIFT is that you can modify the text size if you use the SHIFT. The text information can be used later when making parts for 2D-CAM or Nesting Option.

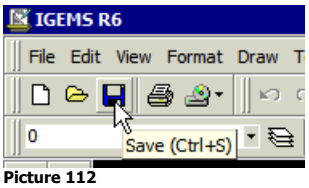
Paste from AutoCAD



Picture 111

If you want to Copy/Paste from AutoCAD then you can use this function to paste the drawing into IGEMS.

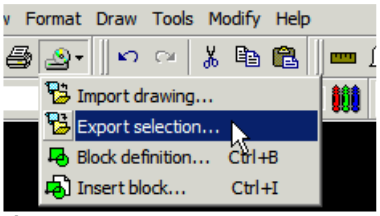
Save



Picture 112

This command saves the entire drawing. The default format for IGEMS is ACD and all information saved in this format can be reused by IGEMS. The DXF-format should only be used for communication with other software.

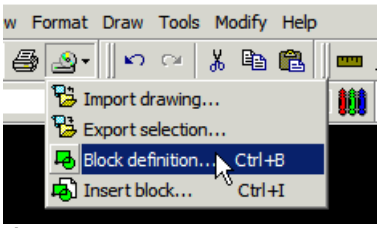
Export



Picture 113

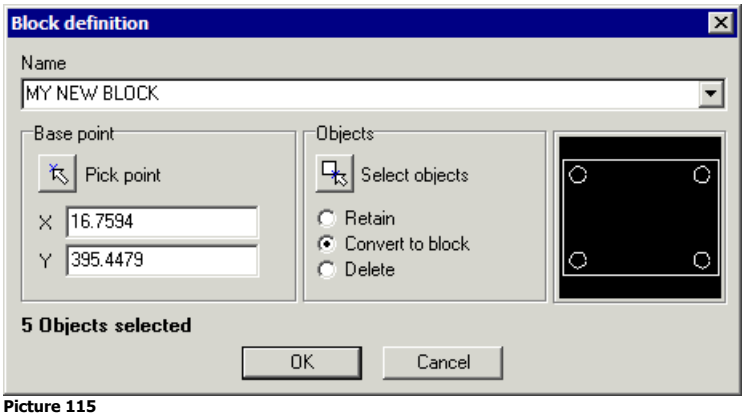
This command asks for objects to be saved. The information can be saved as an ACD or DXF-file.

Block



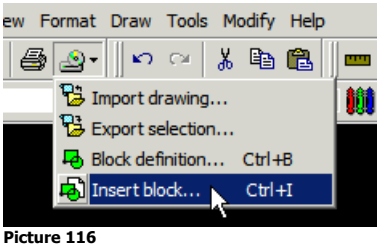
Picture 114

The block description is saved in current drawing instead of a file. If you need a set of block, it is a good idea to make the blocks and then save the drawing as a template. The Block command shows following dialog box.



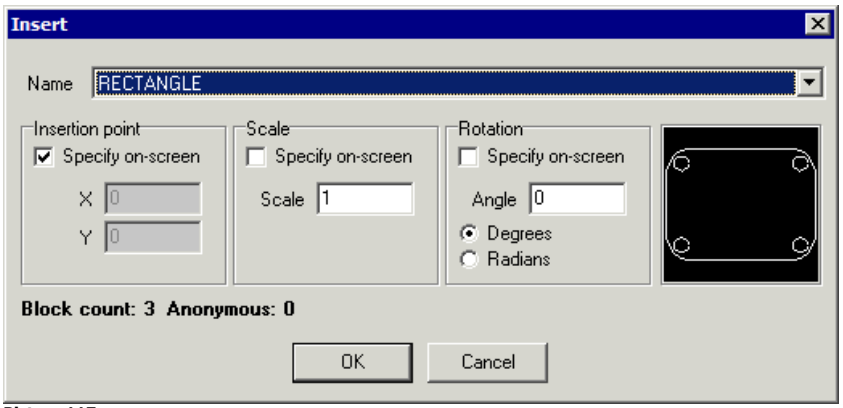
Picture 115

Insert



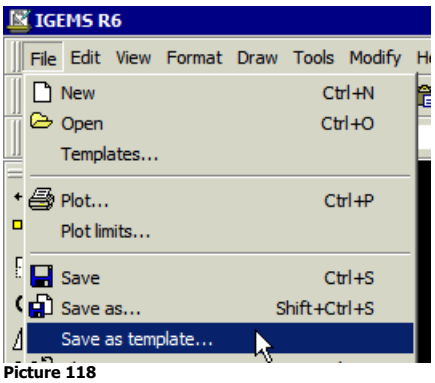
Picture 116

The command shows all blocks available in the drawing.



Picture 117

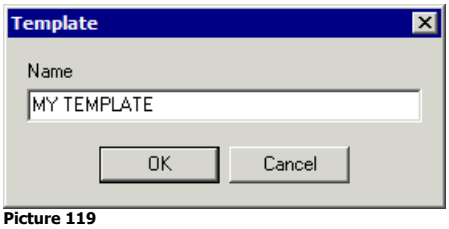
Save as template



Picture 118

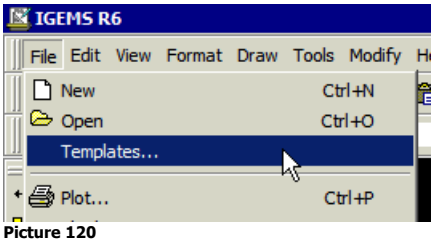
A template is a drawing that will be used every time you create a new drawing. If you should make a template do as follows:

- Start a new drawing
- Define layers and colors.
- Define blocks.
- Draw objects.
- Save as template, and enter a name for the template.



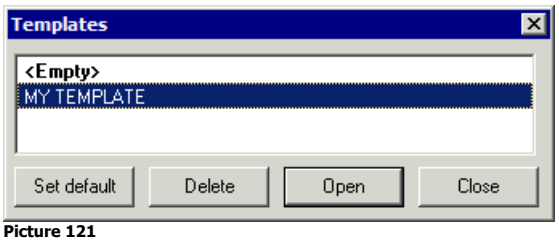
Picture 119

Select a template



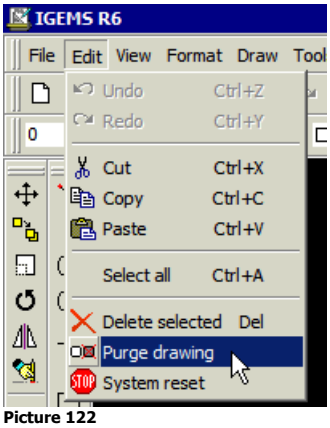
Picture 120

Select a template and press "Set default" button.



Picture 121

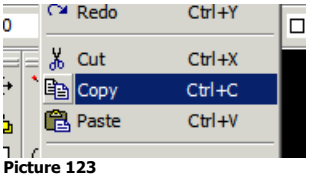
Purge



Picture 122

This command will remove all unused block and layer on the drawing.

Cut and Paste

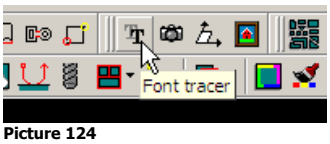


Picture 123

IGEMS Supports the standard Windows Cut and Paste commands. These commands is perfect to use for copy objects between drawings.

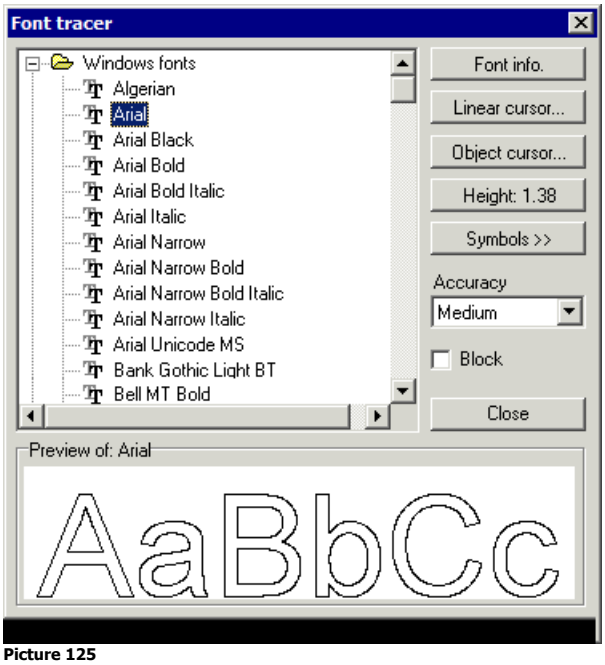
Chapter 10.
SignMaker option

Font tracer



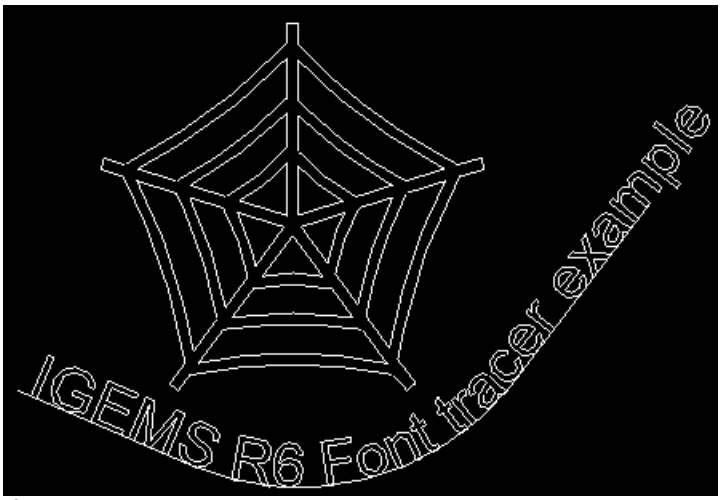
Picture 124

The Font tracer command can import TrueType fonts and convert the information into CAD-geometry.



Picture 125

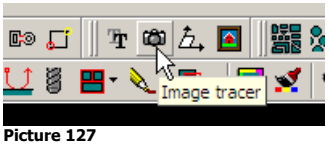
If you want to add more fonts, you can add fonts to the standard windows font directory or to the …"IGEMS_R8/Plugins/Signmaker/FonTracer/Fonts" directory.



Picture 126

The result is a polyline with lines and arcs.

Image Tracer



Picture 127

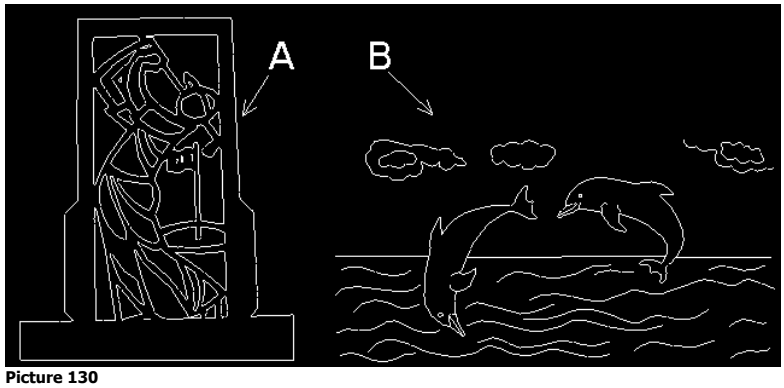
This command converts bitmap files to CAD vectors.

Picture 128

The command has several options that can be used for optimizing of the result.

Picture 129

Click on the button shown in previous picture.



Picture 130

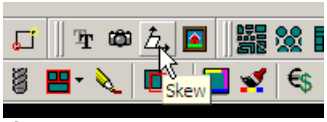
Outline

In example A you, can see the result of the outline method. This method should be used if you have solids areas of black and white. This option creates closed polylines around black areas.

Curves

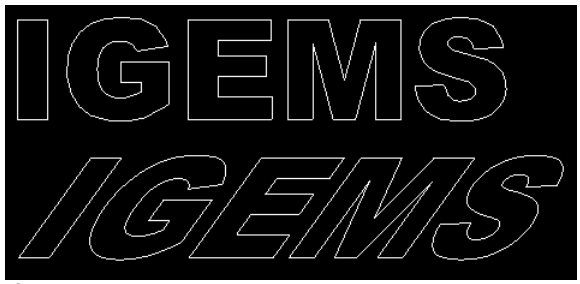
In example B you can se the result of the curve method. This method should be used if you have lines that describe the geometry. This option makes a polyline in the center of the lines.

Skew



Picture 131

With this command you can tilt object.

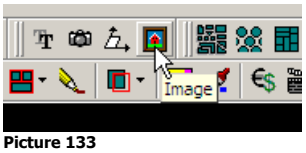


Picture 132

In the example above the Skew command is applied on objects from Image tracer.

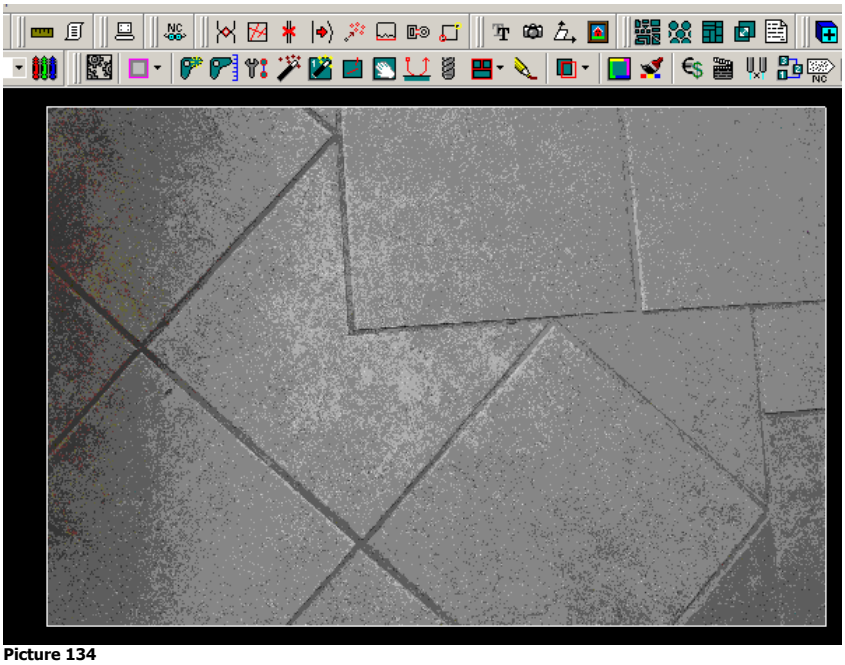
Image

The Image command makes it possible to insert and use a bit map image as a background.



Picture 133

The command starts by clicking on the Image button.

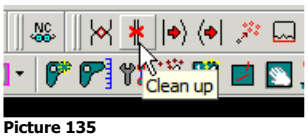


Picture 134

JPG and BMP files can be used.
After insertion it's a good idea to use the Scale command to get the image in correct size. The Image can be copied, scaled, rotated or mirrored like all other drawing objects in IGEMS.

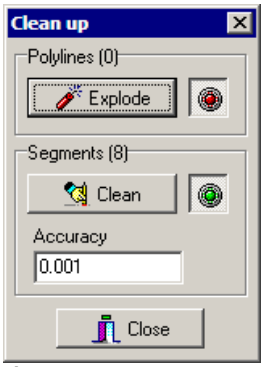
Chapter 11.
CAM-Tools option

Clean Up



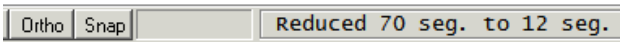
Picture 135

This command removes double objects, overlapping geometry and joins gaps in the geometry.



Picture 136

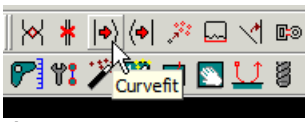
All objects that should be cleaned must be lines, arcs and circles. If you have polylines then you can click the explode button. The Delete buttons is not very often used. The Clean button executes the command.



Picture 137

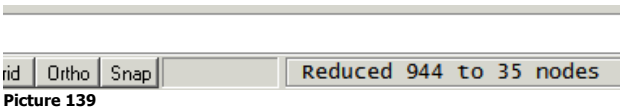
On the information line you can se the result of the Clean up command.

Curve fit



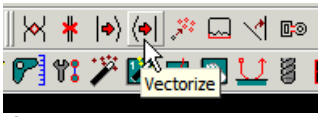
Picture 138

All objects that should be used by this command must be polylines. This command optimizes the polyline. Short line segment will be converted to longer lines or arcs. The command asks for a tolerance, the result of Curve fit can be found on the information line.



Picture 139

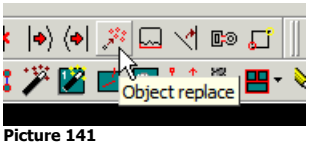
Vectorize



Picture 140

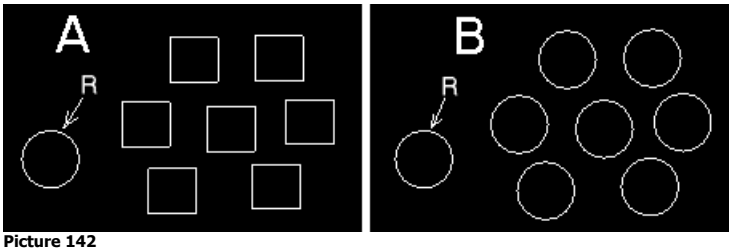
This command works the opposite as the Curve Fit command. It converts all arcs to short linear vectors. If you need this feature you must add this button to a toolbar.

Replace



Picture 141

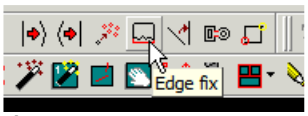
This command asks for a replacement object, then object to replace. The selected object will be replaced by the replacement object.



Picture 142

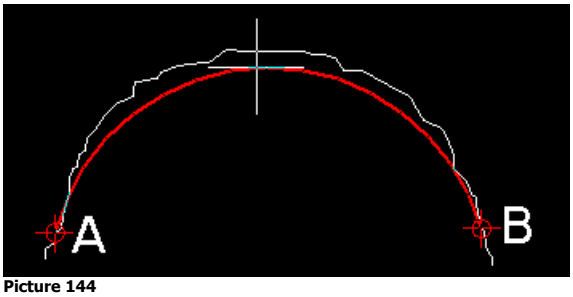
A is before and B is after the command. The R object is the replacement object and the rectangles are the object to replace.

Edge fix



Picture 143

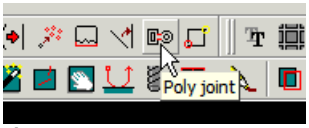
Edge fix can replace a portion of a polyline with an arc or a line.



Picture 144

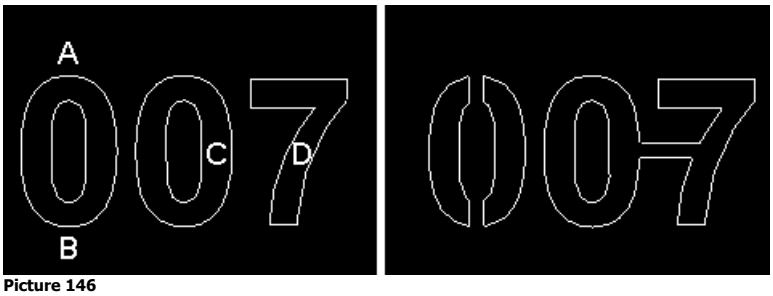
Edgefix ask for a start point (A) and an end point (B). The portion between the points will be replaced by an arc if you pick a point or with a line if you press the space button.

Poly joint



Picture 145

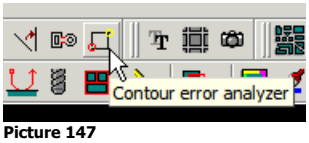
This command joins two closed polylines to one, or divide one closed polyline into two.



Picture 146

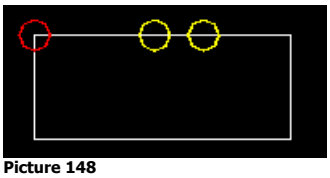
If you click on two points that are outside the closed object (A and B) then the object will be divided. If you pick two points that are inside closed objects (C and D) then the objects will be joined.

Contour analyzer



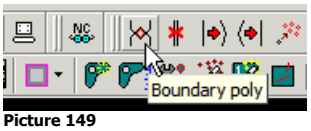
Picture 147

This command makes red circles on gaps between objects and yellow circles if the objects are overlapped.



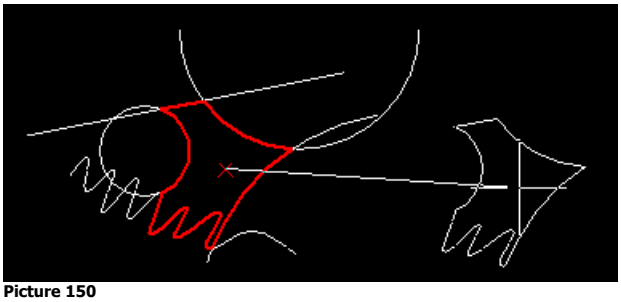
Picture 148

Boundary polygon



Picture 149

This command can create closed polylines from enclosed areas.



Picture 150

Click inside where you want the new polyline, press space to move the polyline, press space again to end the command.

Chapter 12.
The workflow in 2D-CAM

The workflow for creating a CNC-file in IGEMS can be described in 5 steps.

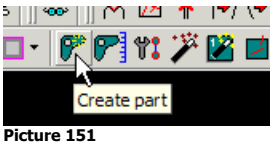
Step 1: Create the geometry

A part must be created from existing geometry. This geometry can be created in 3 different ways.

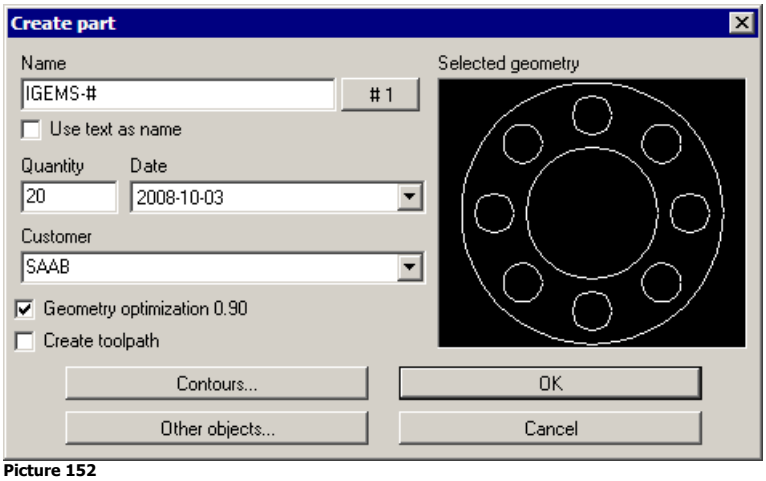
1. It can be drawn using the different CAD-command in IGEMS
2. It can be imported from other CAD systems.
3. It can be automatically created from our parametric parts library.

The geometry that describes the outside and inside contour must be free from gaps and overlapping objects.

Step 2: Create a part



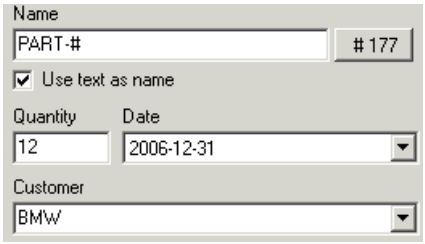
Picture 151



Picture 152

By clicking on the button "Contours" you can select the objects describing the boundary geometry. By clicking on "Other objects" you can select objects that should be used for other purposes like marking for example.

Non geometrical information



Picture 153

This information can be used by the Organizer module. It can also be printed on reports.

1. Name: Is used for identification of the part. The letter "#" can be used as a count up number.
2. Quantity: Number of parts that should be produced.
3. Date: The date can be printed out on reports.
4. Customer.

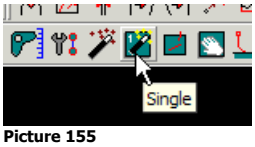
Geometry optimization

Picture 154

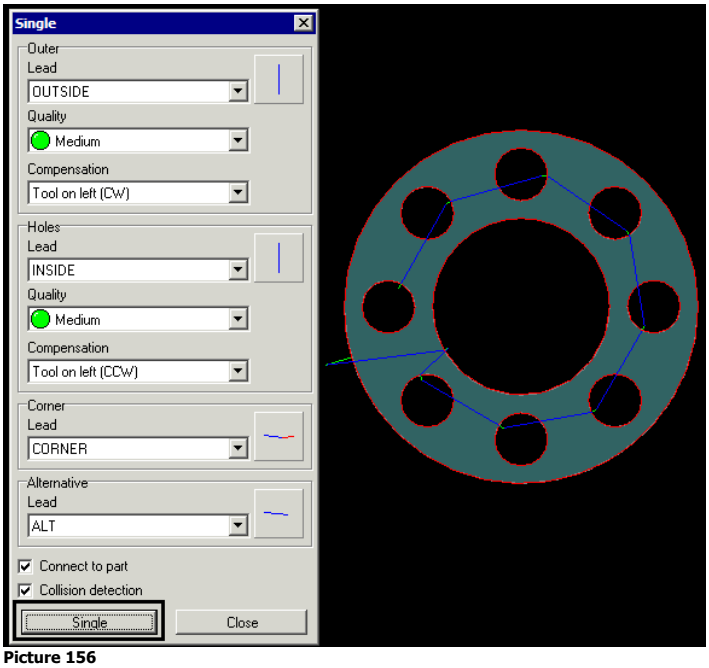
We recommend that you always use the geometry optimization. The value of the geometry optimization must always be larger than the maximum tool radius compensation that should be used in the machine.

Step 3: Add toolpath

There are different commands to apply a toolpath. One of them is the Single command.



Picture 155

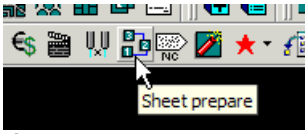


Picture 156

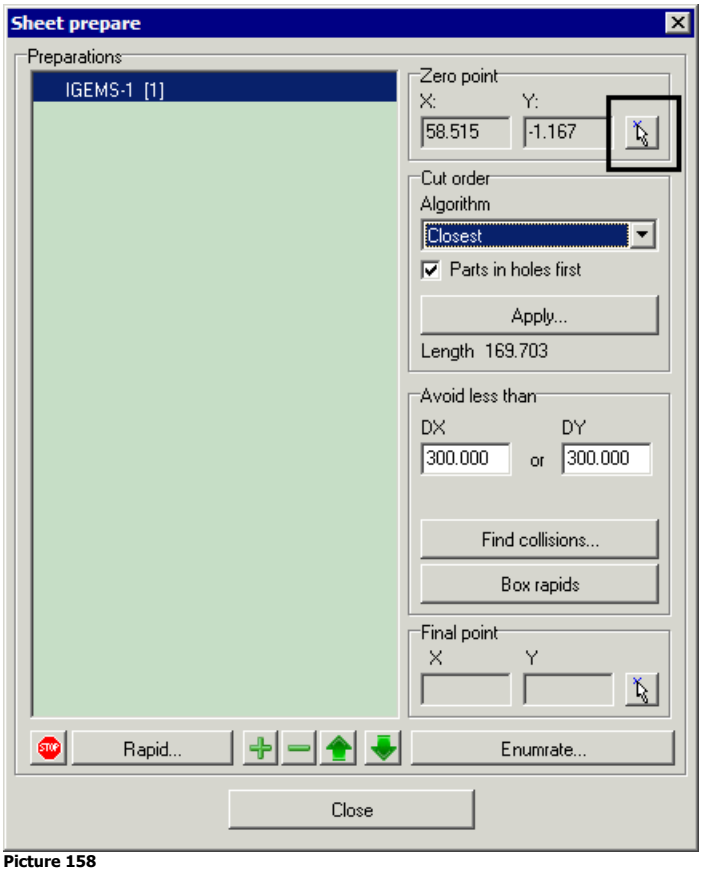
Press the Single button and create the toolpath for each starting position. Start with the geometry that should be cut first.

Step 4: Add the cut-order

The Sheet prepare command makes an object called Cut order. The cut order has information about the origin point and the order between parts. (On this example there is only one part).



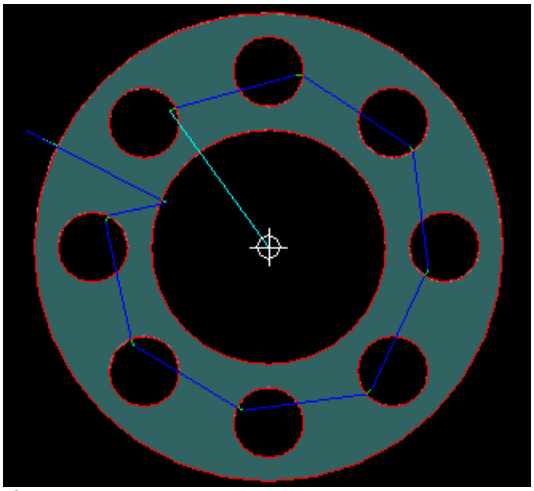
Picture 157



Picture 158

Set the zero point

You can set the "Zero point" to an optional position on a part.

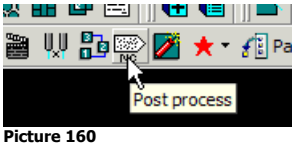


Picture 159

Other options in this command will be described later in this manual.
It's also possible to go direct to Step 5. In that case the step 5 will use the default values from Sheet prepare.

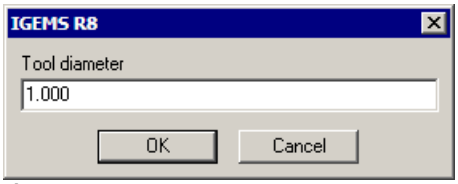
Step 5: Postprocessing

Last step is to create the CNC-file.



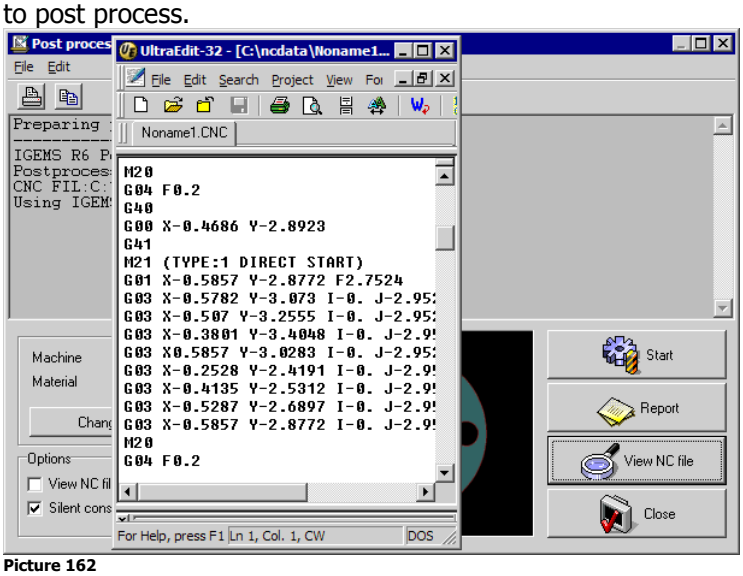
Picture 160

If you are using IGEMS tool radius compensation, then the command ask for a tool diameter.



Picture 161

If you have not made Step 4 (Sheet prepare), then the command will ask for parts to post process.



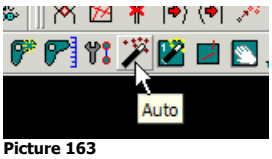
Picture 162

The Post processing is executed by clicking on the button start. The result will be a CNC-file and a report file. These files can be viewed by clicking on the Report or the View NC-file.

Chapter 13.
Create toolpath

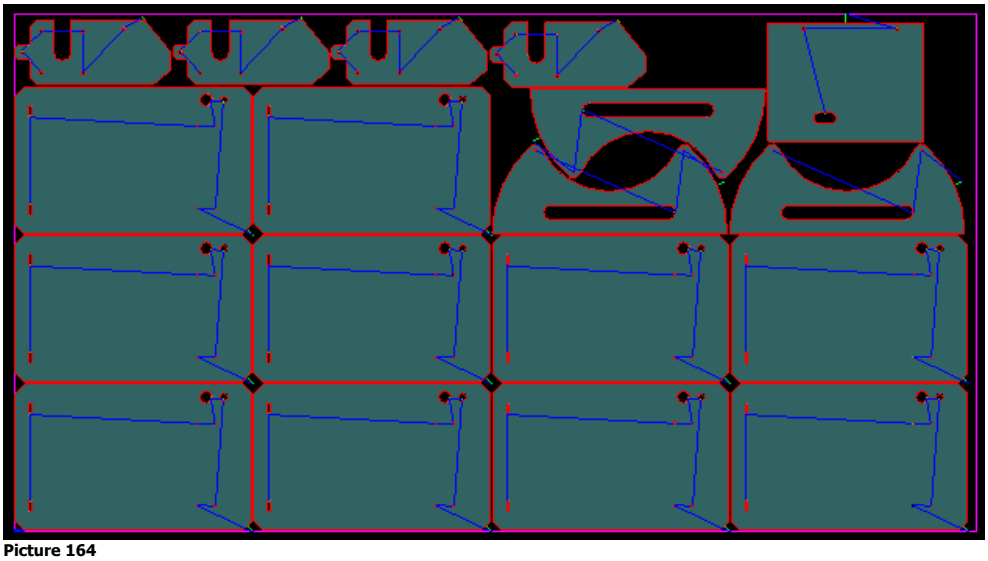
There are different commands that can create toolpaths. The commands has a different level of automation.

The Auto command

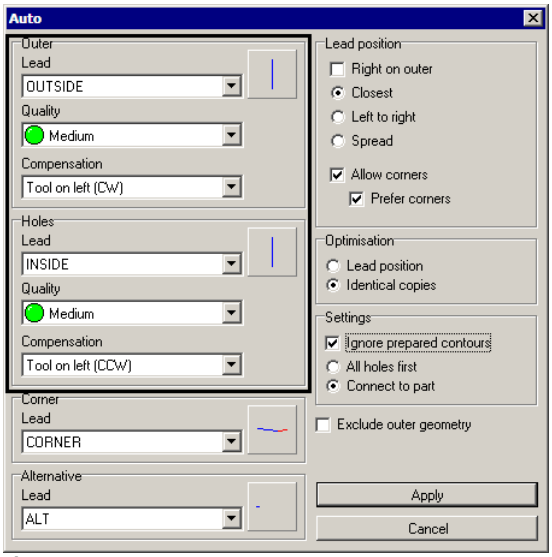


Picture 163

This command automatically creates a toolpath for one ore several parts at the same time. This command is perfect to use on parts that have been nested without any toolpath on them.



Picture 164

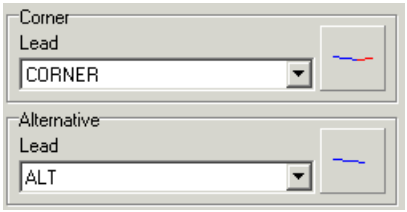


Picture 165

Outside and Inside settings

All the values on the left side are used for controlling the leadin/out, piercing type, cutting quality and the tool radius compensation.

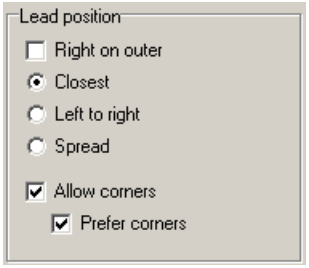
Special leads



Picture 166

In many cases it is a good idea to put the leads at the corners of the geometry. This lead can be defined as corner lead. If there is no place for the lead then the command will test if it is possible to use an alternative lead. This lead geometry should be defined with the piercing type that starts on the geometry.

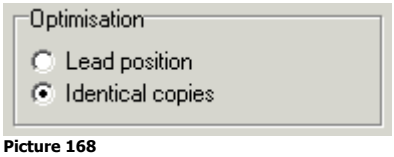
Leads position



Picture 167

These settings control the internal cut order between the holes. It also controls where the leads should be placed.

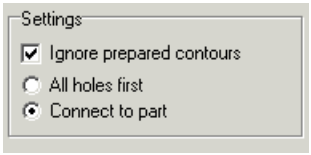
Optimization



Picture 168

If your machine has unlimited memory space or you have a sequential postprocessor (long file) then it will be a better result if you use optimization by "Lead position".
If you have an older NC-machine with limited memory space and your postprocessor creates a NC-file that are built up by main and subroutines, you should optimize it for identical copies. This will make the NC-file much smaller.

Settings



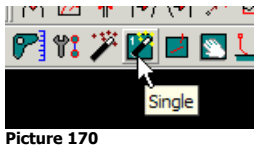
Picture 169

Ignore prepared contours
If you already have added a toolpath on some parts, then the settings "Ignore prepared contours" can be used for avoiding that the toolpath are re-done.

All holes first or Connect to part
If you are using the option All holes first, then the toolpath will do all holes first on the selected parts. This option is mostly used for oxyfuel cutting. This will create a disconnected toolpath. In all other cases the "Connect to part" option is the most common setting.

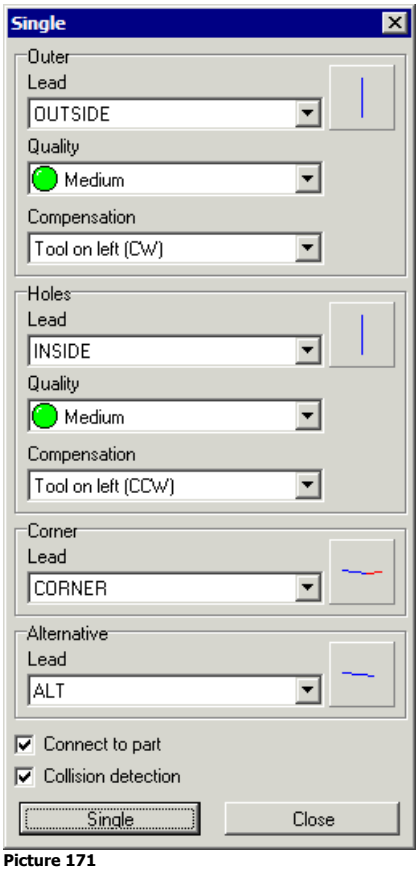
Picture and information is deleted

The Single command



Picture 170

The Single command requires more input, but gives you more control over the lead positions and the internal cut order. The single require one click for each part on the part.

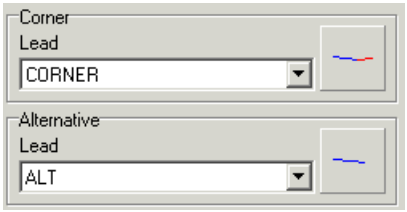


Picture 171

Outside and Inside settings

All the values on the left side are used for controlling the lead selection. It also controls the cutting quality and the tool radius compensation that should be used.

Special leads



Picture 172

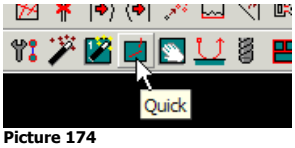
In many cases it is a good idea to put the leads at the corners of the geometry. This lead can be defined as corner lead. If there is no place for the lead then the command will test if it is possible to use an alternative lead. This lead geometry should be defined with the piercing type that starts on the geometry.

Various

Picture 173

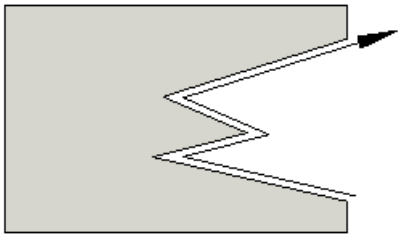
With these settings you can control if the toolpath should be connected to the part and if collision detection should be activated.

The Quick command

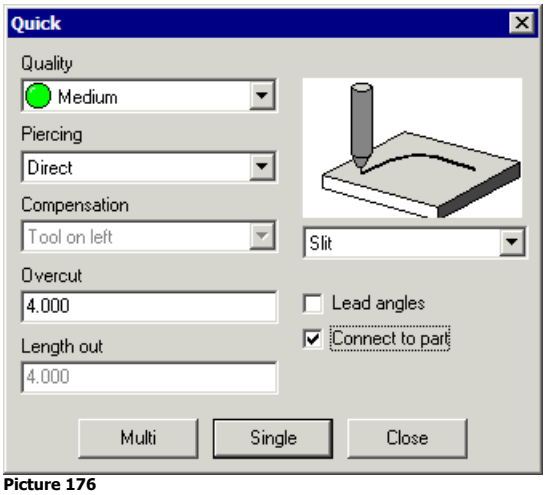


Picture 174

The Quick command can be used on all kind of geometries but is mostly used when it is necessary to cut only some segments of the part.

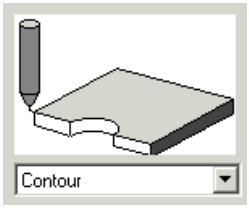


Picture 175



Picture 176

By selecting the "Slit" option, you can add toolpath to open parts that are located on the solid part. The lead will always follow the geometry, and the over cut distance will be cut two times.

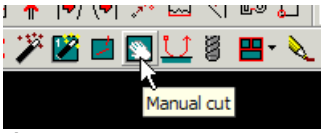


Picture 177

By selecting the "Contour" option, you can add toolpath to the inside and outside geometry.

Lead angle
This option can be used only together with the contour option. When activated the command also asks for the lead in and lead out angels.

Manual



Picture 178

This is the most flexible command for creating a toolpath, but it is also the command that require more input from the user.

Picture 179

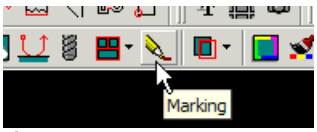
By using this function you can create a toolpath step by step. All movements in the machine will be in the same order as you enter the sub-commands.
Following sub-commands are available:

1. Rapid: Turns the cutting off and add a rapid transport.
2. Start: Start the cutting process.
3. Line: The cutting will continuo linear to a selected point.
4. Next: The cutting will proceed to next object.
5. Follow to: The cutting will proceed all the way to it reach selected position.
6. Quality: The cutting quality can be changed between different objects.
7. Event: It is possible to add different event that can control the Postprocessing process. The postprocessor must be adapted for this function.
8. Undo: Undo the previous command.
9. Loops: Can activate or deactivate corner loops.
10. Done: Finish the manual command.
11. Cancel: Do not save the manual toolpath.

The command line

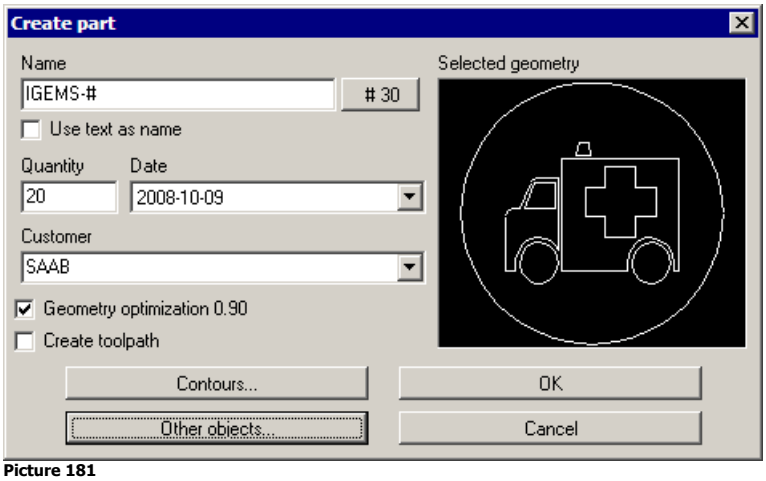
A good idea is to look at the command line. In many cases you can use the defaults. The defaults input are inside brackets [example] and are activated by the space bar on the keyboard.

Marking



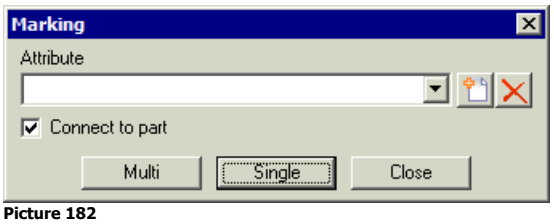
Picture 180

The marking command activates functions on the machine that mark the surface of the material. In a Waterjet machine it is possible to use pure water without abrasive for marking materials.



Picture 181

If you have markings that should be connected to a part, then the marking object must be selected with the "Other object" button in the Create part command.



Picture 182

The marking command can be used on all normal objects as polylines, arcs, and lines. But it can also be used for points and texts objects.

Attribute

The attribute is optional text information that can be used for machines with more than one marking method. The attribute will then inform the postprocessor about equipment to be used.

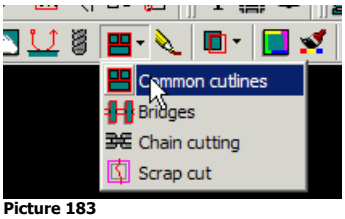
Multi or Singe

If you select the Multi button, then you can select multiple objects at the same time. If you select the Single button then you must click on each object. The start point is at that endpoint closest to you click point.

Chapter 14.
Disconnected toolpath

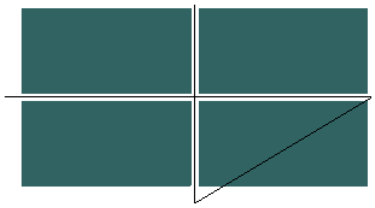
A toolpath that is connected to the part are very simple to handle. If you move a part then you also move the toolpath or opposite. But sometimes it is necessary to create toolpath that are disconnected from parts. Following commands creates disconnected toolpaths.

Common cut line



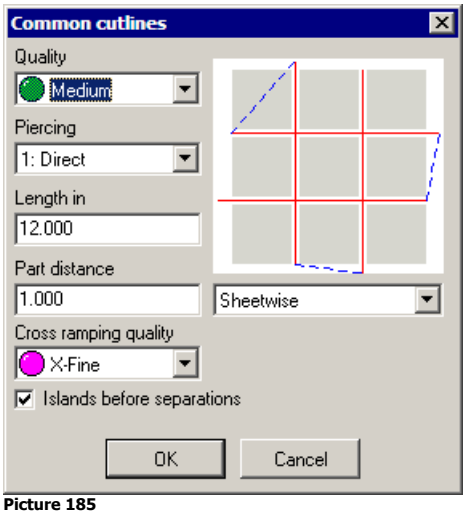
Picture 183

Depending on the geometry of the parts you can sometimes save a lot of machining time by using common cut line.



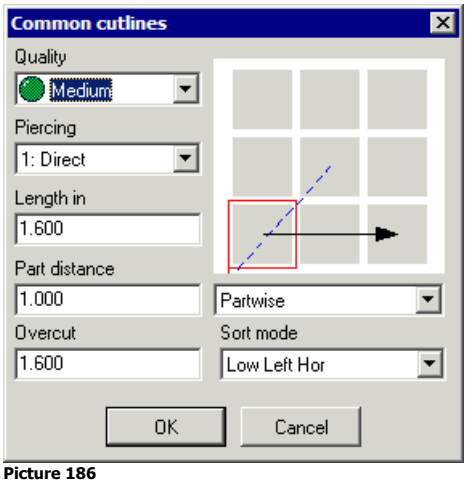
Picture 184

The common cut line can be started by clicking on this button.



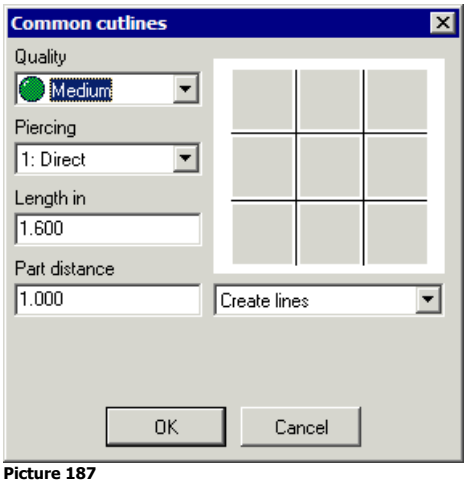
Picture 185

This dialog box shows the option "Sheetwise". This method is cutting in as long straight lines as possible. The Part distance must be the same as the diameter of the jet. If not, the parts will have incorrect dimensions or may not be machined.



Picture 186

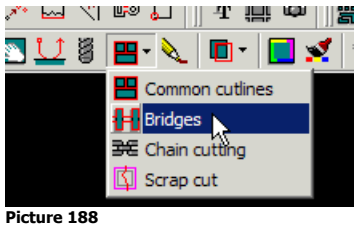
The option "Partwise" is completing each part before starting to cut next part.



Picture 187

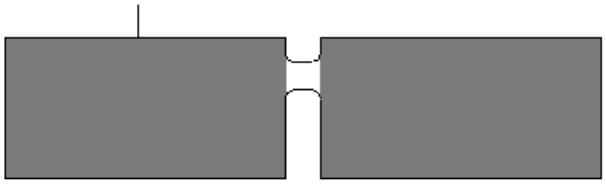
The option "Create lines" creates only drawing objects. These objects can be used with the quick or manual command for creating the toolpath.

Bridge

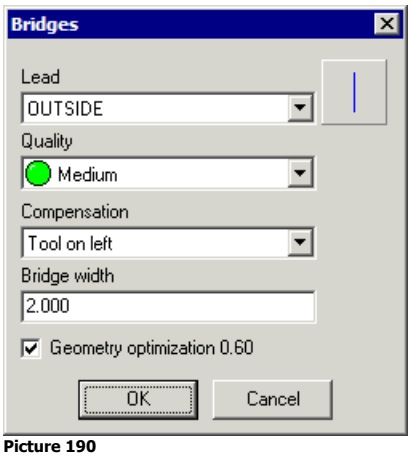


Picture 188

The Bridges command joins two or more parts with a small tab between. This command is often used when cutting small parts to avoid them to disappear.



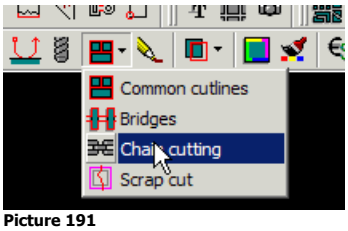
Picture 189



Picture 190

When you press OK then you can insert the starting lead and the tabs.

Chain cutting

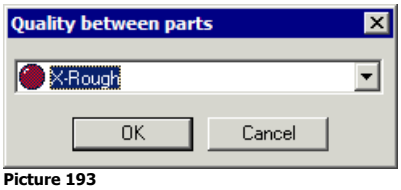


Picture 191

The chain cut command makes a toolpath with only one piercing for all selected parts. This command is often use in materials which are difficult to pierce.

Picture 192

This command can only be used on parts that already have a toolpath. The command joins the different toolpaths together. Before you try this command use the Single command and place the lead on good positions.



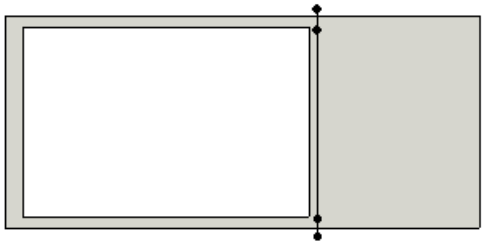
Picture 193

For saving time it is better to use as high cutting speed as possible. When you press OK the command asks you for a fence line. The toolpath will be connected in that order the fence line intersect with the parts.

Scrap cut

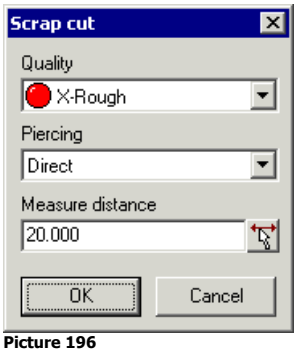
Picture 194

This command is specially designed for making cut offs. The command can control where the height sensor should be activated and deactivated.



Picture 195

The settings are controlled in following dialog box.

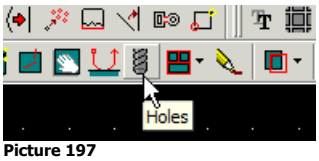


Picture 196

The measure distance control the distance between the measuring points (height sensor on/off points).

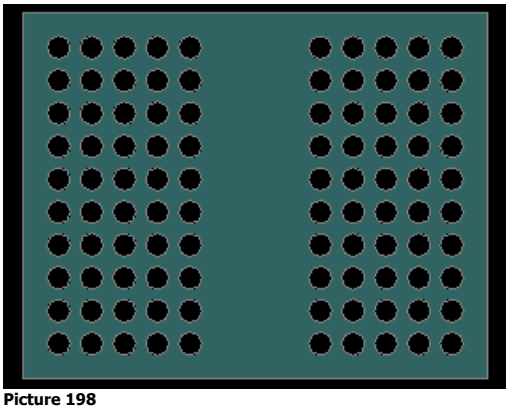
Chapter 15.
Various command

The Hole command



Picture 197

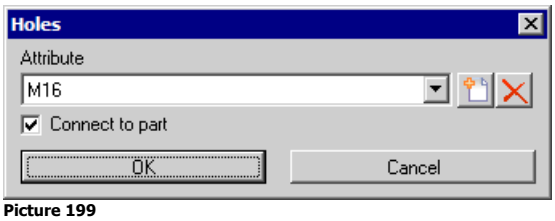
This command can be used for various things. The result from using this command is controlled by the postprocessor.



Picture 198

Some example:

1. Can be used for cutting small holes.
2. For controlling drilling units.
3. For controlling multi operation macros like drilling and the tapping.
4. Can execute machine macros defined by the machine.

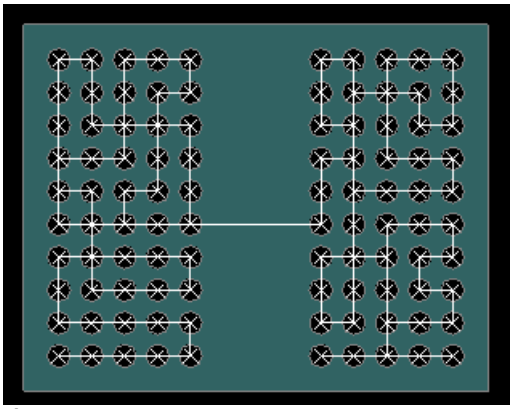


Picture 199

The command first asks for a filter object. By using the filter, you can easily select circles with a specific radius or a block with a specific attribute.
The command then takes following information to the postprocessor:

1. X and Y position of block, circles and points.
2. The attribute.
3. The radius if you select a circle.
4. The angle of the block.

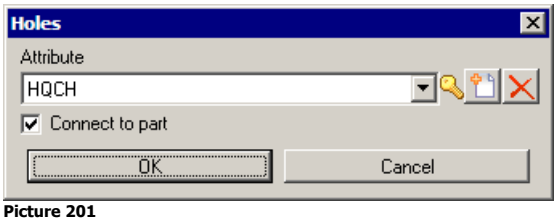
Again! This is a command made for customization, what should happen when you use this command is depending on the postprocessor.



Picture 200

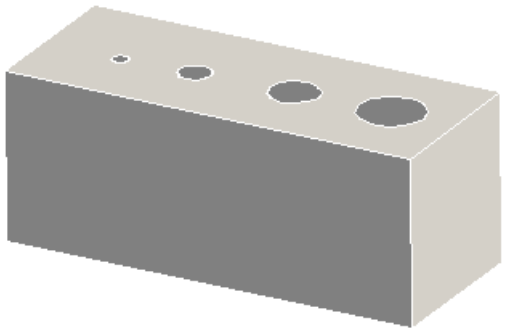
HQCH High Quality Circular Hole

The feature HQCH is a feature to make High Quality Circular Holes (HQCH)
This option is activated by the Hole command.



Picture 201

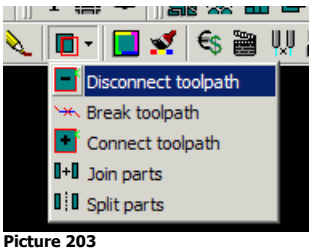
The name of the fixed attribute is HQCH. By using this attribute you can cut small circular holes with very high quality. Valid diameter starts at 1.2*Max tool diameter and up.



Picture 202

The HQCH feature takes longer time than the standard cutting, but it gives a much better result.

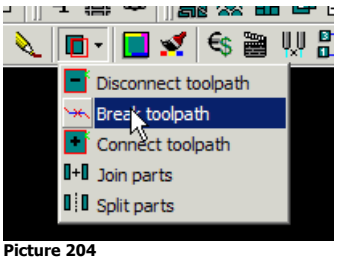
Disconnect toolpath



Picture 203

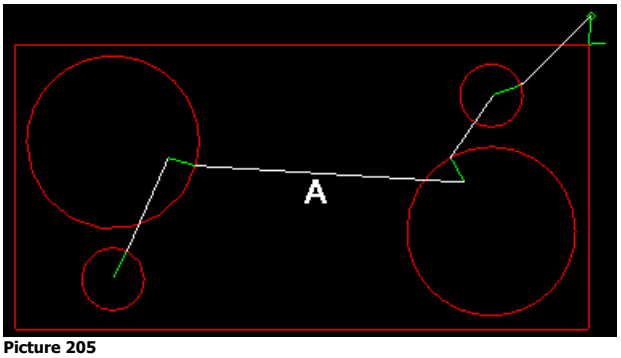
This command can be used to convert a connected toolpath into a disconnected toolpath. The command asks for one ore several parts that have a toolpath. The result will be separated parts and toolpaths.

Break toolpath



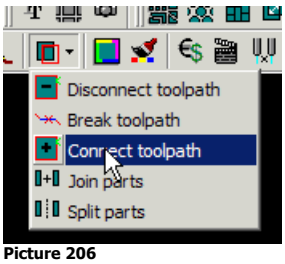
Picture 204

This command breaks up a disconnected toolpath into two parts.
By clicking on the rapid transport (A) the toolpath will be divided.



Picture 205

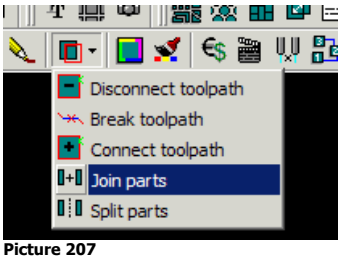
Connect toolpath



Picture 206

This command connects an optional toolpath to a part.
Select the part, then the toolpath.

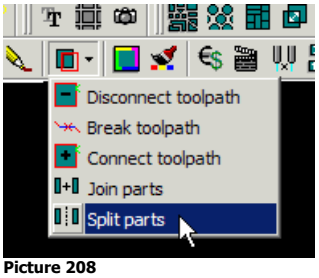
Join Parts



Picture 207

With this command you can join several parts to one part.
Select first the main part, then the other parts to join. The name quantity, date and customer information will be taken from the main part. This command can create a part from several external geometries.

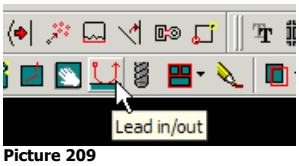
Split part



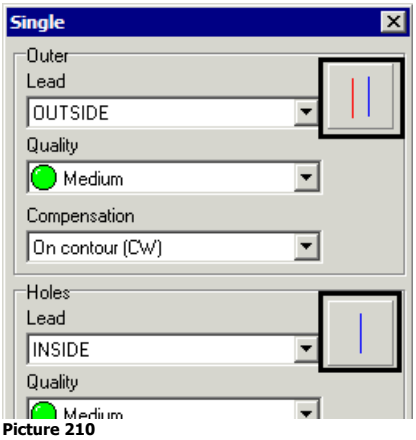
Picture 208

This command can only be used on joined parts.
If the part has a toolpath then the toolpath will be removed before the parts are split.

Lead settings

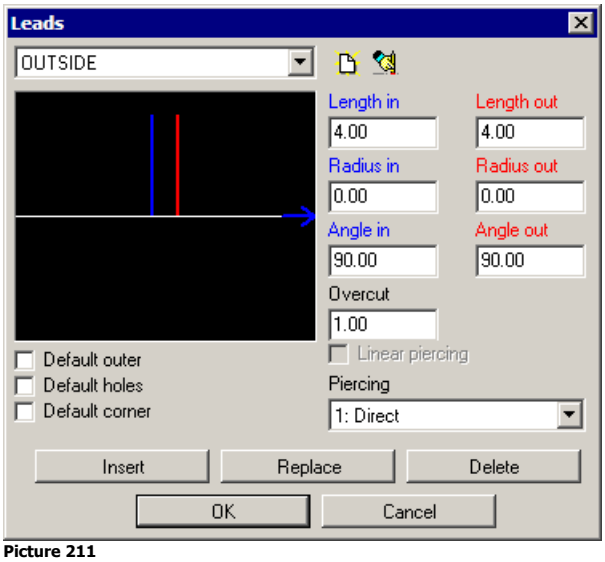


Picture 209



Picture 210

The lead settings command can also be activated in the Single and Auto command by clicking on one of the rectangular buttons.

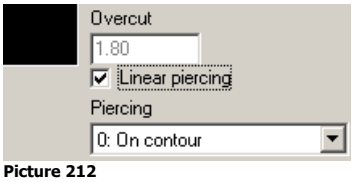


Picture 211

Overcut

The geometry of the leads can be adjusted by changing the values shown in last picture. The overcut value can also be negative.

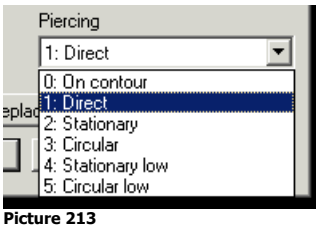
Linear piercing value



Picture 212

When using piercing type 0 it's possible to use the Linear piercing distance from the material database as a over cut value.

Piercing

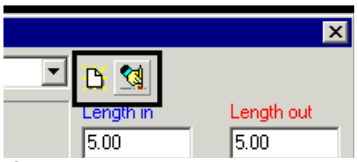


Picture 213

The different piercing is following:

- On geometry, means that the start of cutting will be activated after the lead in. This will result in an invisible lead. This means that the lead it self does not have to be collision detected. This piercing is perfect to use on small holes or when other leads do not fit.
- Direct piercing, do not use any delay, the piercing will be linear during the lead.
- Stationary piercing is not often used. The time for the piercing is controlled by each material.
- Circular piercing, means that the jet will make circular movements during the piercing. The piercing diameter and number laps is controlled by each material.
- Stationary low is not often used. The method makes a pre-piercing of all holes on the sheet, before start cutting.
- Circular piercing in low pressure, is pre-pierceing all holes before start cutting. The piercing diameter and number of laps is controlled by each material.

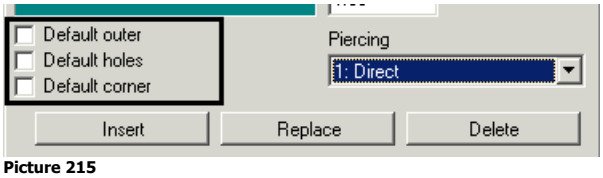
Save and delete



Picture 214

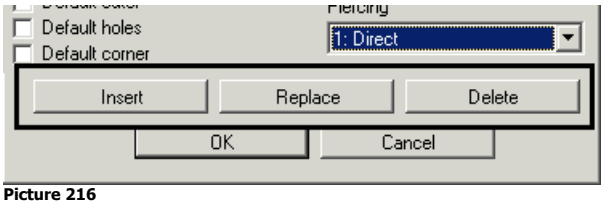
These buttons are used for saving and deleting leads from the lead library.

Connect lead and material



Picture 215

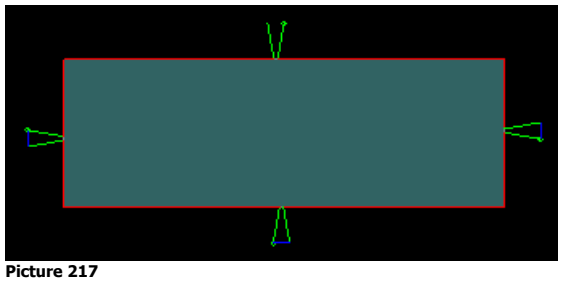
If you activate these toggles, this lead will always become standard for the active material. This is an important function if you must have a special designed lead for a special material.



Picture 216

Insert, Replace and Delete

These buttons can not be activated if you have started the lead library from Auto or Single. By using this command you can Insert, Replace or delete leads on the part of the drawing.

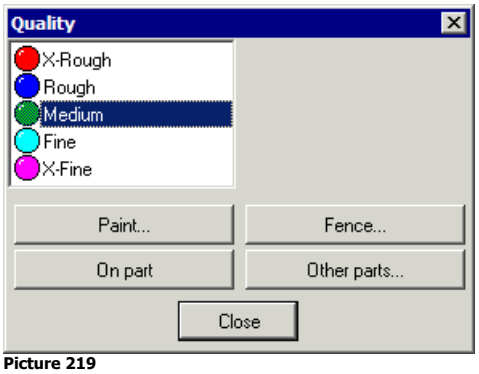


Picture 217

Cutting Quality

Picture 218

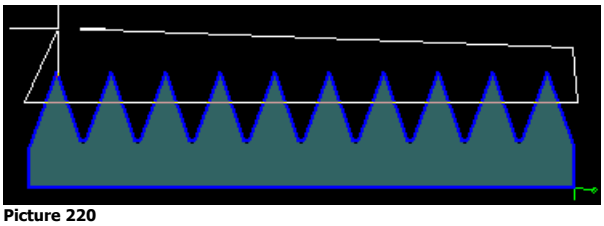
On most command it is possible to set the cutting quality while creating the toolpath. By using the Quality command you can also set the quality afterwards. Press the Quality button and select the part you want to change.



Picture 219

Following options are available:

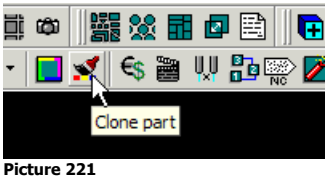
- Paint, makes it possible to change portions of a geometry by clicking on two points and a side. If you double click on the geometry the entire geometry get the same quality.
- On part, the entire part gets the selected quality.
- Fence, makes it possible to make a fence around an area. Everything inside that area will have the selected quality.
- Other parts, makes it possible to select other part on the drawing. The part you select will get the selected quality.



Picture 220

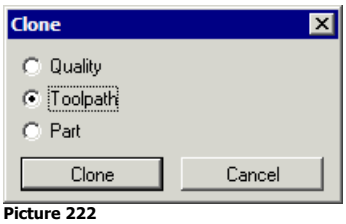
In the example above everything inside the fence will have the selected quality.

Clone



Picture 221

By using the Clone command you can transfer properties from one part to another. Select the part with correct information.

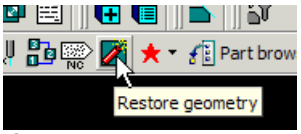


Picture 222

Following options are available:

- Quality, in this case only the quality information will be cloned to other part.
- Toolpath, this clones the lead positions, and also the quality.
- Part, this option clones the complete part.

Restore geometry

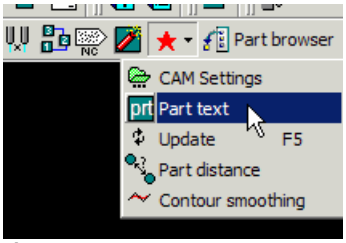


Picture 223

It is sometimes necessary to convert a part to CAD-geometry. Following picture shows a part before and after using Restore geometry.

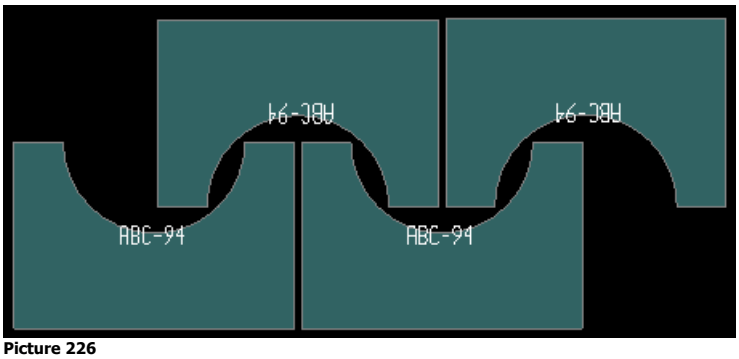
Picture 224

Part text



Picture 225

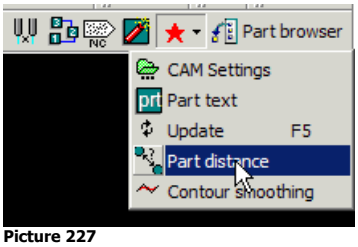
This command display the name of the part on the part.
Select the parts and enter the text height.



Picture 226

If you set the text height to zero, then you can remove the text.

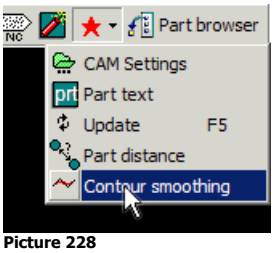
Part distance



Picture 227

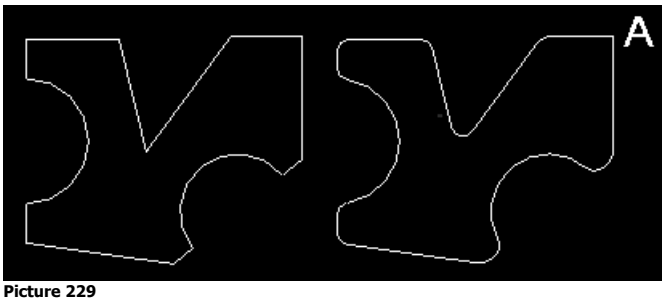
This command shows the closest distance between two parts. Start the command and select two parts. The distance is shown on the information line.

Contour smoothing



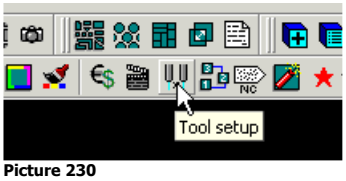
Picture 228

This command was developed for laser cutting in thick material. The command creates fillet radius in all corners on a closed polyline.
If you hold down the Shift key, all corners will be rounded except the closest.



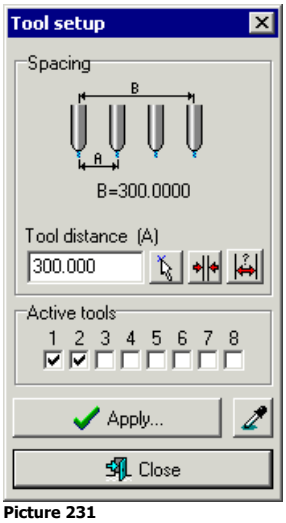
Picture 229

Tool setup



Picture 230

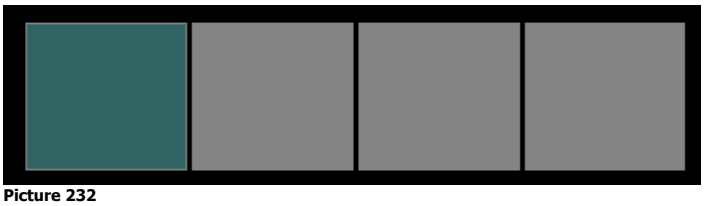
Many machines are equipped with more than one cutting tool. The tool settings can be controlled with the tool setup command.
The command shows following dialog box.



Picture 231

Set active tools

By activating different tools 1-8 and then press the Apply button and select parts, you can make master and slave parts. The distance between all tools will be the tool distance value.

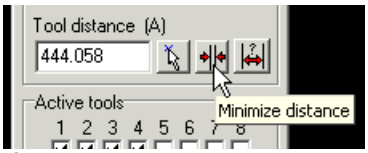


Picture 232

The master and the slave parts have different colors.

Minimal distance

By using this option the tool distance will be calculated automatically.



Picture 233

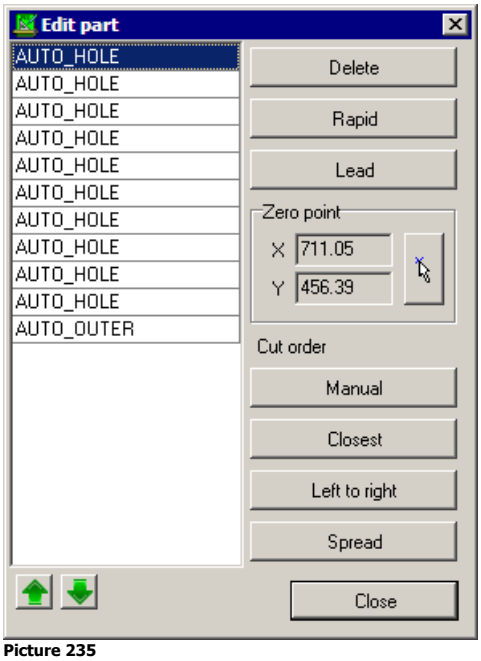
Equal distance

This option asks for a sheet. The sheet will be divided in as many areas as you have active tools.

Edit part

Picture 234

With the Edit part command you can change parameters related to the toolpath of the part.



Picture 235

Internal cut order

With the Manual, Closest, Left to Right and Spread option you can change the internal cut order between the parts.

Picture 236

You can also change the internal cut order by select geometry and then use the up and down arrow buttons.

Other options

This command can also do following:

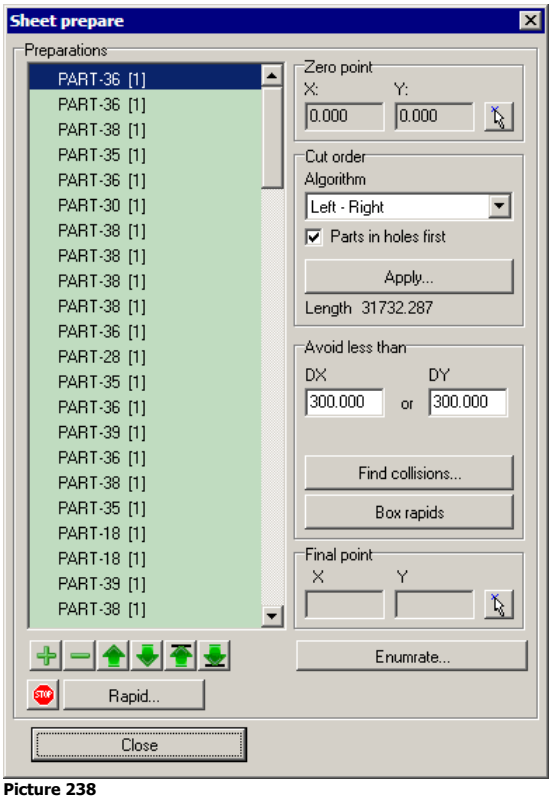
- Delete: This option delete the selected geometry.
- Rapid: Creates additional rapids between two geometries.
- Lead: Makes it possible to change the lead geometry and the piercing.
- Zero point: Normally the zero point of the part is to the lower left corner. It can be changed by this option.

Chapter 16.
Cut order and Postprocessing

Sheet prepare (Cut order)

Picture 237

You must have a toolpath before you can use this command. Select all parts that should be included in the cut order. Following dialog box is shown:



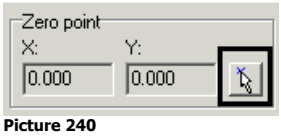
Picture 238

By clicking on following pictures you can remove or add parts that should be included in the cut order.

Picture 239

Zero point

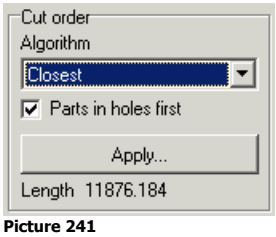
The program has a default location of the zero point. By clicking on following button you can change the zero point to an optional position.



Picture 240

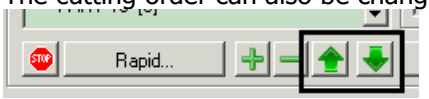
Cut order

This settings control the cut order between the parts.



Picture 241

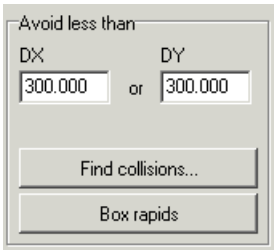
Select algorithm and then press "Apply.." button.
The cutting order can also be changed by using following buttons.



Picture 242

Extra rapid movements

It is possible to detect and create a toolpath that moves the tool around parts that are smaller than a specified size in X and Y.



Picture 243

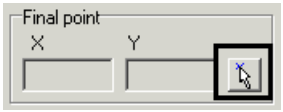
By clicking the "Find collisions" button you can make a rapid toolpath around parts that are too small.
With the "Box rapids" button you can automatically create a toolpath without diagonal rapid transport between the parts.

Picture 244

By clicking on the Rapid button, you can create extra rapid transports, between different parts.

Final point

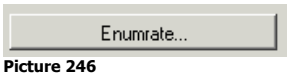
It is sometimes important to make a final transport of the jet. This can be done by clicking on following button.



Picture 245

Enumerate

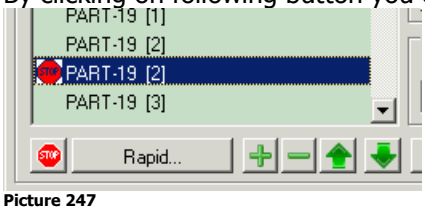
This function inserts number on all parts. The number sequence is the same as the cutting order.



Picture 246

Stop

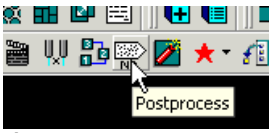
By clicking on following button you can add a stop before the part is cut.



Picture 247

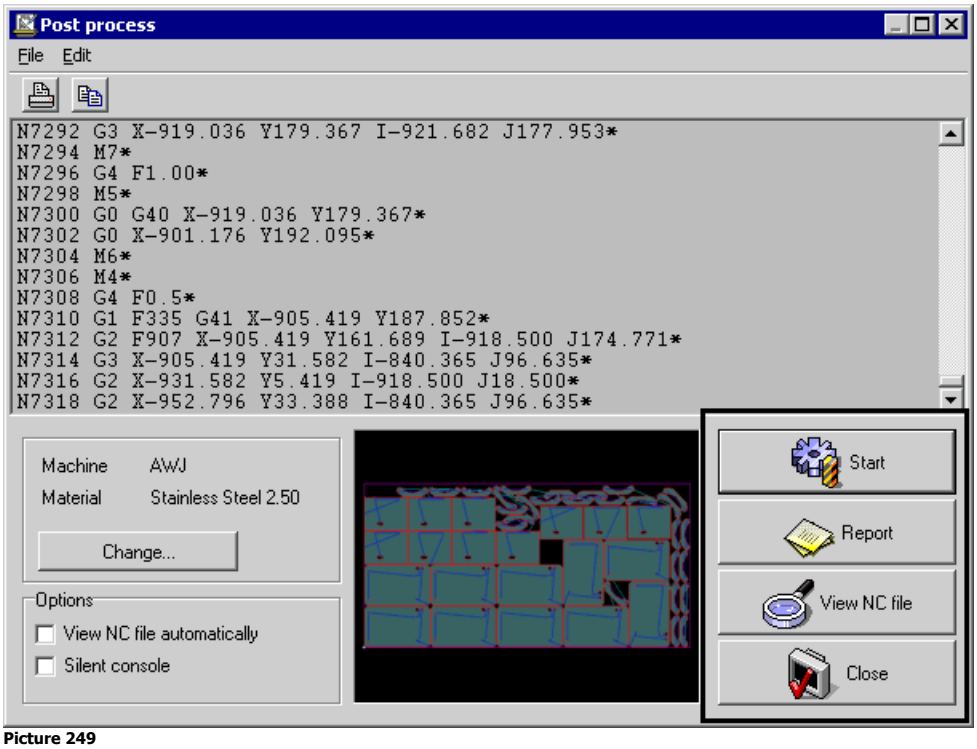
Postprocessing

The command is started by clicking on following button. If you don't have a cut order, then the command ask for parts to cut and then create cut order.



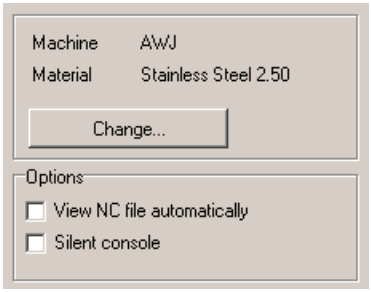
Picture 248

If you have more than one cut order you must select which cut order that should be used. In the CAM-Settings described on page 111 you can activate an automatic interference check before the posting is started.



Picture 249

You can start post processing, view CNC-file or report file by clicking on the buttons on the right side of the dialog box.



Picture 250

Other settings

If you temporary want to change machine or material, you can click on the change button. If you have a very long CNC-file, then the post processing will go faster if you activate the Silent console.

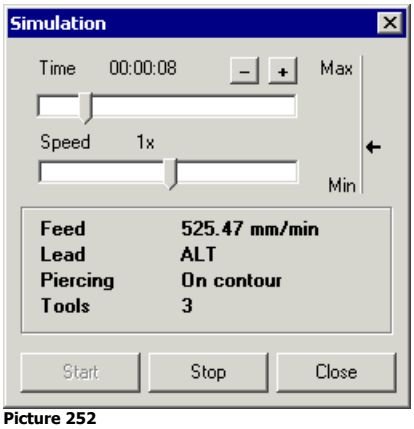
Chapter 17.
Simulation and cost calculation

You have following option available in IGEMS.

Simulation

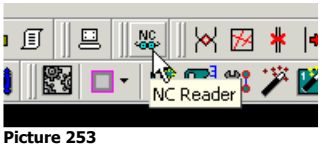
Picture 251

You must have a toolpath or a cut order to simulate a toolpath. You can control the simulation speed and positions in following dialog box.



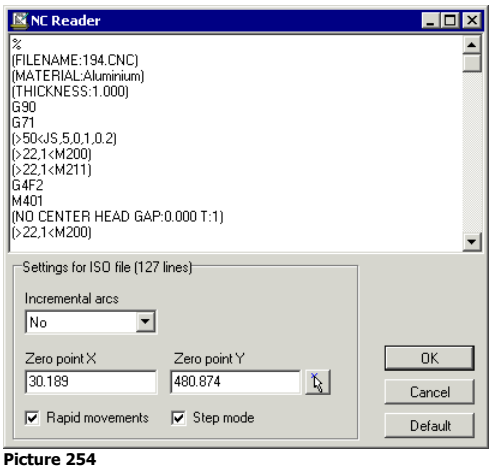
Picture 252

NC-Reader



Picture 253

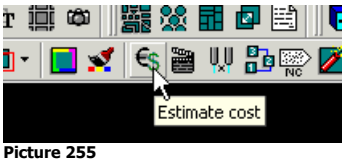
Only standard ISO and ESSI files are supported in this command.



Picture 254

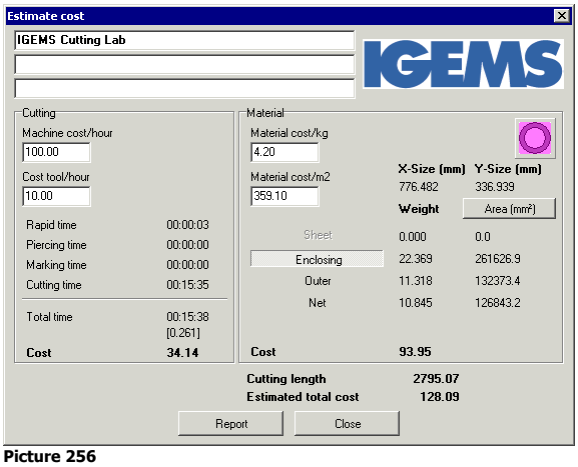
If you get a strange result, then you may need to change the settings of "Incremental arcs". You can change the zero point. You have an option if you want to view rapid transport and you can also use Step mode. If you use the step mode you can step forward by clicking on space or escape.

Estimate cost



Picture 255

Select parts for cost calculation.

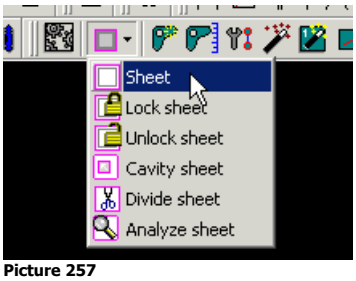


Picture 256

In this dialog box you can find the cost for the selected parts. By clicking on the report button you can make a report.

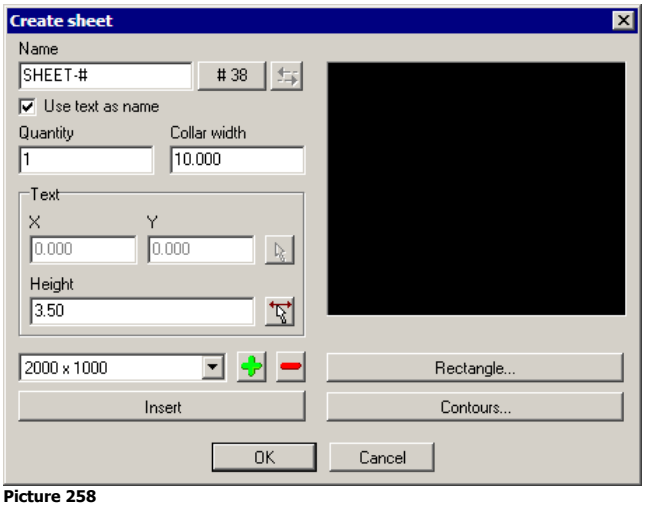
Chapter 18.
Sheet commands

Create sheet



Picture 257

All closed geometry can be used as sheet.



Picture 258

This command has following options:
Name: This value is printed out on reports and it is also used by the Organizer module.
Quantity: This value is only used by the Organizer module.
Collar width: This is the area around the sheet where no parts will be placed by the nesting module.
Text: This value controls where the name of the sheet will be printed out on the sheet. If you have a text height of 0, then no text is used.

Predefine sizes

Picture 259

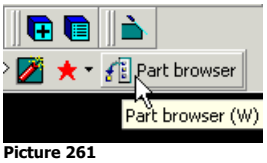
By using the + and – button you can add or remove predefined sheet sizes. By pressing the Insert button you can insert the sheet on the drawing.

Define geometry

Picture 260

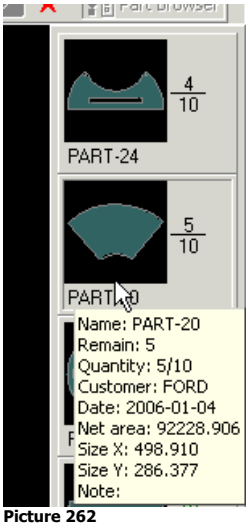
You can define the geometry when you start the command. You can also define a rectangle or use a polyline as geometry.

Part browser



Picture 261

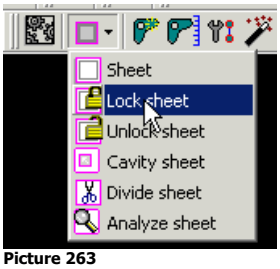
All parts have a property called Quantity. By using the part browser you can se how many parts that have been placed and the quantity of all parts on the drawing.



Picture 262

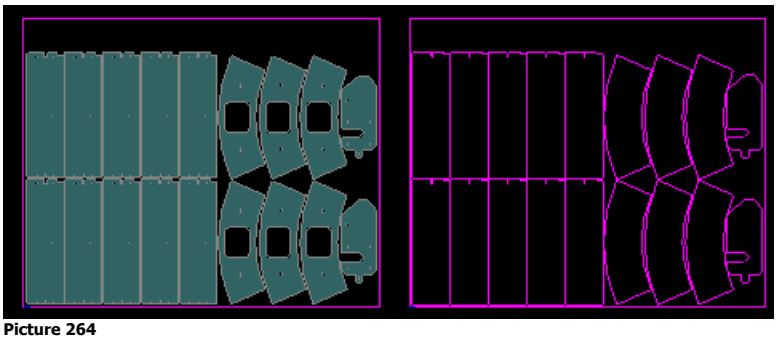
In the picture above, 4 out of 10 and 5 out of 10 has been placed on the sheet. If you hold down the right mouse button on a part you can se more information that is stored on the part.

Lock sheet



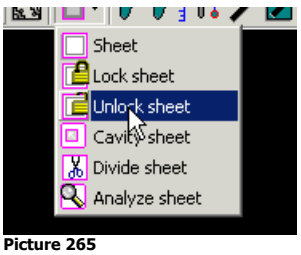
Picture 263

When you lock the sheet, all parts are converted to holes on the sheet. On the picture below you can see the result of locked sheet.



Picture 264

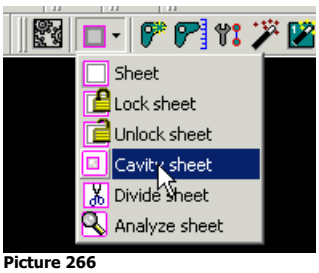
Unlock sheet



Picture 265

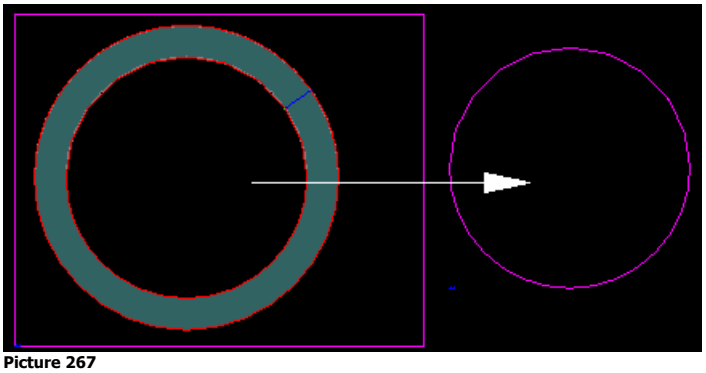
It is sometimes important to change the geometry on an existing sheet. By using the unlock command you can explode the sheet, and then move or delete the geometry on the sheet. As long as you do not change the external geometry of the sheet, all properties (name and quantity) are left unchanged.

Cavity sheet



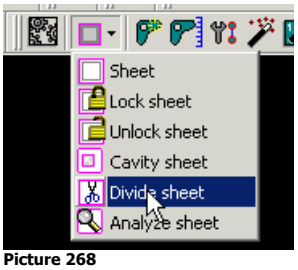
Picture 266

If you have parts with large internal geometries, it is then often a good idea to keep the internal areas as rest sheets. With this command you can do this. The new sheets will inhibit all the properties from the original sheet.



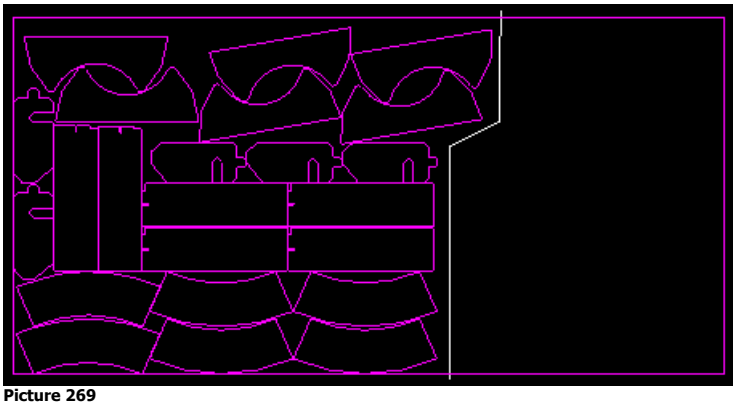
Picture 267

Divide sheet



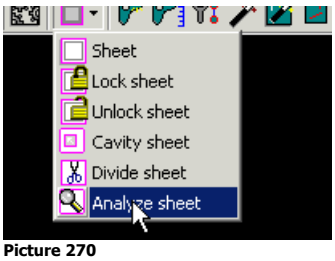
Picture 268

With this command you can divide a locked sheet in two or more sheets. You must have a toolpath (scrap cut) or a polyline that describe where the sheet should be divided.



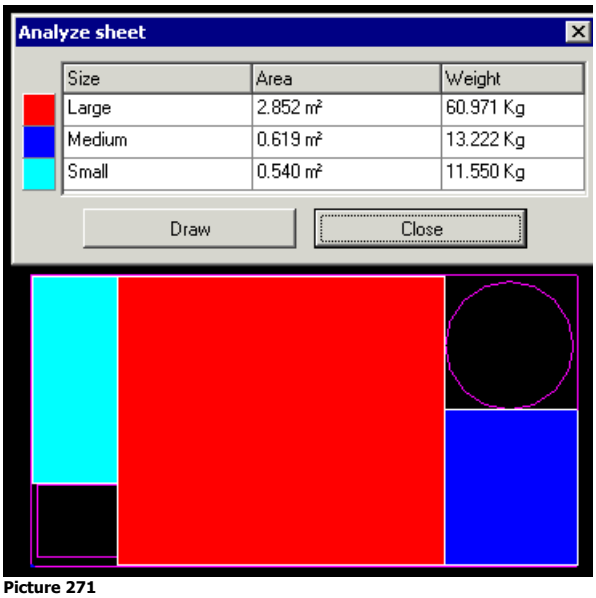
Picture 269

Analyze sheet



Picture 270

This command is for automatically making inventories of sheets. The information will be used by the Organizer module.

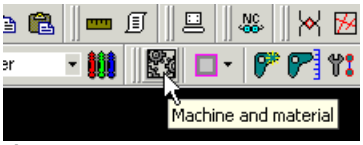


Picture 271

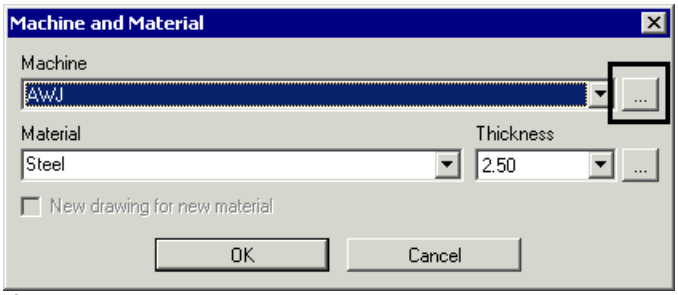
In the CAM-Settings command on page 111 you can change the definition of Large, Medium and Small area.

Chapter 19.
Machine settings

The main focus with IGEMS is Waterjet and Plasma/Oxyfuel cutting. For supporting customers that also have other machines we also support pure water and Laser machines. The AWJ option (Advance WaterJet Option) can only be selected if you have this option.

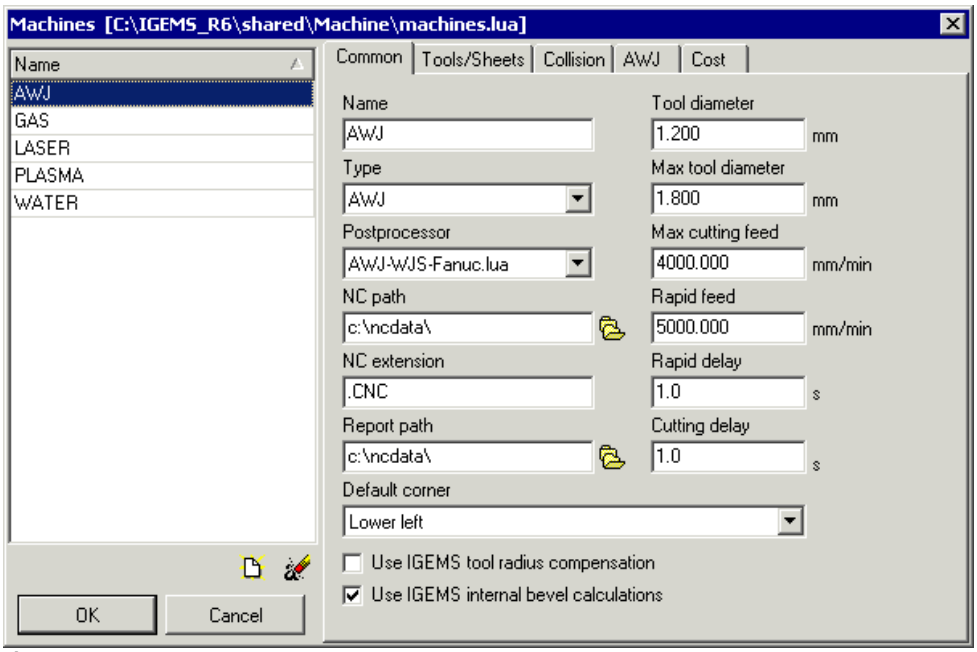


Picture 272



Picture 273

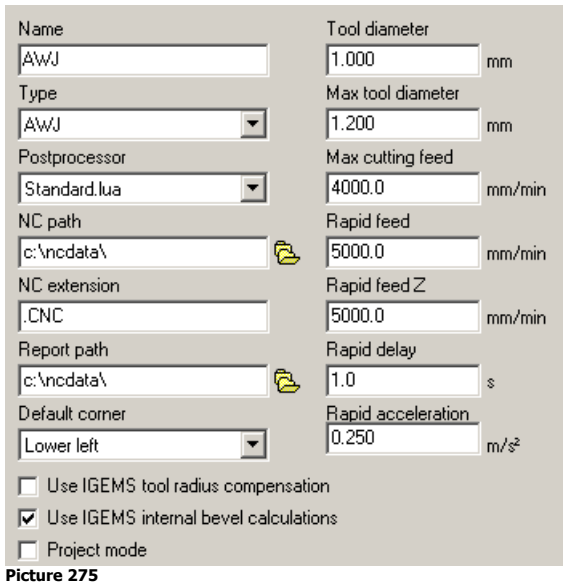
IGEMS supports unlimited number of machines. Before you make the toolpath you should select a suitable machine. If you need to define a new machine or change the settings of an existing machine then click on the button shown in picture above.



Picture 274

By using the list of machines to the left and the button at the end of the list, you can select, make new and delete machines.

Common settings

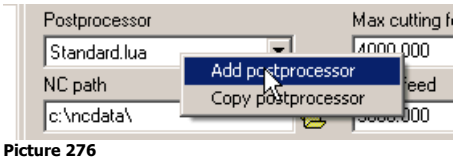


Picture 275

These settings are for the same kind of cutting machines. Most of the settings are self explained but here are some remarks.

Postprocessor

Be sure that you are using a postprocessor that is made for your cutting machine. If you want to add or copy a postprocessor you can use the left mouse button and click on the postprocessor list.



Picture 276

Default corner

This setting controls the position of the zero point on the Sheet prepare command.

Use IGEMS tool radius compensation

If you are using this option, the toolpath is internally offset by IGEMS. The size of the compensation is the same as the tool diameter. If you are not using this option then the toolpath will be offset by the machine (using G41/G42 codes).

Use IGEMS internal bevel calculations

This is a settings that is only used by IGEMS 5X-axis option. The settings are depending on machine and postprocessor. When using IGEMS internal bevel calculations then the tool compensation are calculated by IGEMS.

Project mode

If you activate the Project mode, then the NC-path and the Report path will change to the same path as used for the drawing file. If you try to post process and you have an unsaved drawing then IGEMS first ask you to save the drawing file.

Tool diameter

This setting is used by IGEMS internal tool radius compensation. It is also used as a default value for the nesting system and for the Lock sheet command.

Max tool diameter

Important! This value must be larger than the maximum tool diameter (size of jet) that will be used on the machine.

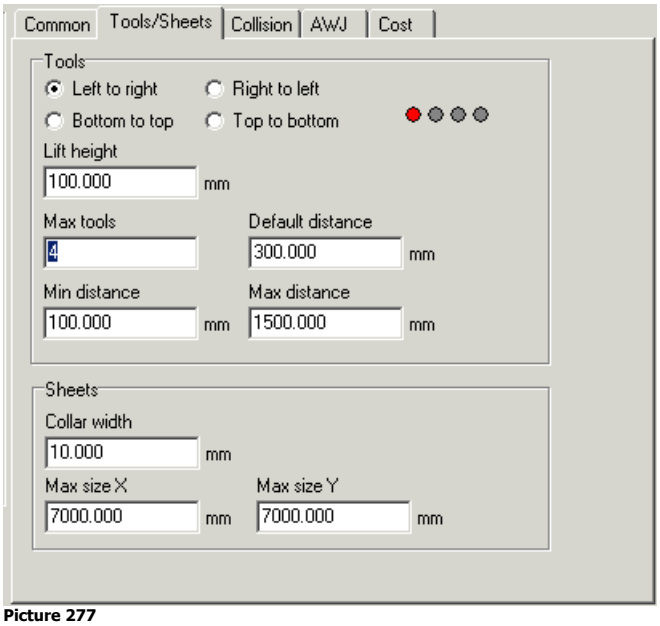
Rapid feed Z

This information is used for calculation of the time it takes for moving the cutting tool up and down. The tool up/down distance is still controlled by the Lift height value.

Rapid acceleration
The acceleration parameter is for the rapid transport in XY-plan and Z-axis. If you have a part with many holes (many rapids) then this value is important to get an accurate cutting time.

Rapid delay
The value is only used for fine tuning the time calculation. This value is added after every cut off.

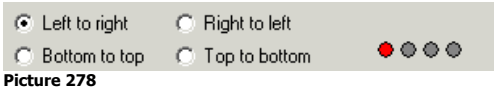
Tools and Sheet setting



Picture 277

Most of the settings are self explained but here are some remarks.

Tool alignment



Picture 278

This setting controls the tool alignments and which tool will be the master tool.

Lift height

If you have a machine equipped with a Z-axis control. Then this value is used for controlling the Z-axis rapid plane between parts.

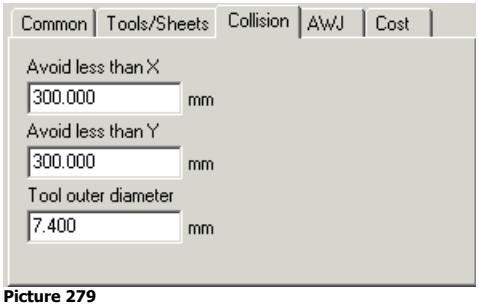
Max tools

Set this value to the maximum number of cutting tools on your machine.

Tool distances
Set the minimum, default and the maximum distances between the cutting head.

Collar width
This is the area around the sheet where no parts will be placed by the nesting module.

Collision settings

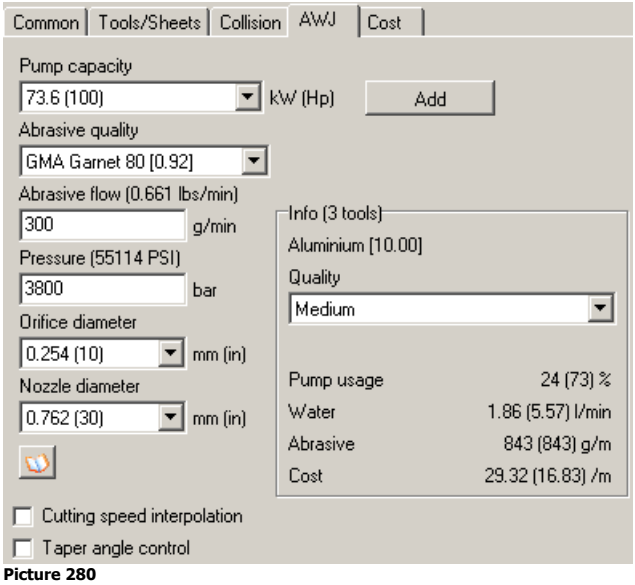


Picture 279

Here you can change the default value for the collision settings in the Sheet prepare command.

Advance WaterJet Settings

Most settings are self explained but here are some remarks.

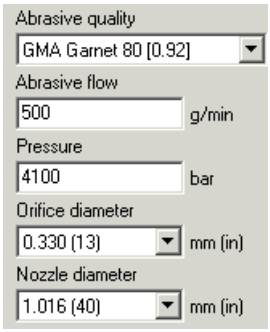


Picture 280

Pump capacity

This value does not affect the cutting speed that is used in the program. It is only used for calculating the pump usage. You can add other pump capacity by clicking on the add button.

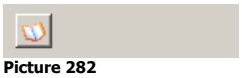
Abrasive parameters



Picture 281

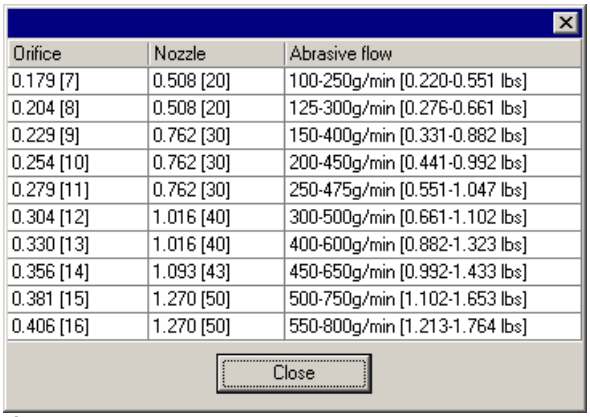
All values above affects the cutting speeds. Be sure to set them to correct value.

Hint



Picture 282

By clicking on this button you can se recommended mixed of Orifice, Nozzles and Abrasive flow.

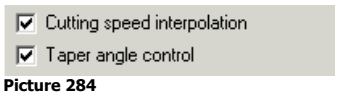


Orifice	Nozzle	Abrasive flow
0.179 [7]	0.508 [20]	100-250g/min [0.220-0.551 lbs]
0.204 [8]	0.508 [20]	125-300g/min [0.276-0.661 lbs]
0.229 [9]	0.762 [30]	150-400g/min [0.331-0.882 lbs]
0.254 [10]	0.762 [30]	200-450g/min [0.441-0.992 lbs]
0.279 [11]	0.762 [30]	250-475g/min [0.551-1.047 lbs]
0.304 [12]	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]

Close

Picture 283

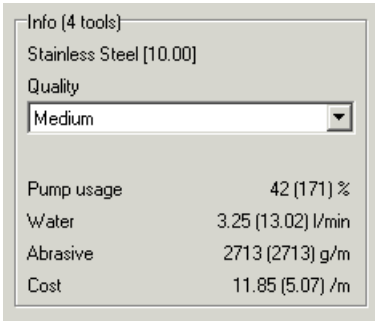
Cutting speed control



Picture 284

Some controllers has the possibility to interpolate with the cutting speed. (For example the FLIN function in Siemens 840D). Use the "Cutting speed interpolation" only if you have this kind of controller.
Taper angle control can only be used if you have a 5-axis cutting machine. This activates taper angle controls that are depending on the cutting speed.

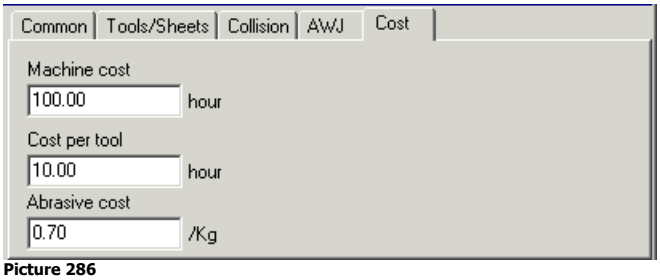
Info



Picture 285

This area shows the result of the cutting parameters. Information inside parentheses () are the values for maximum number of cutting tools.

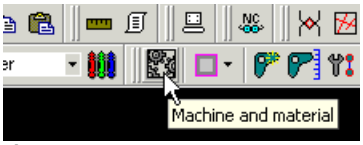
Cost setting



Picture 286

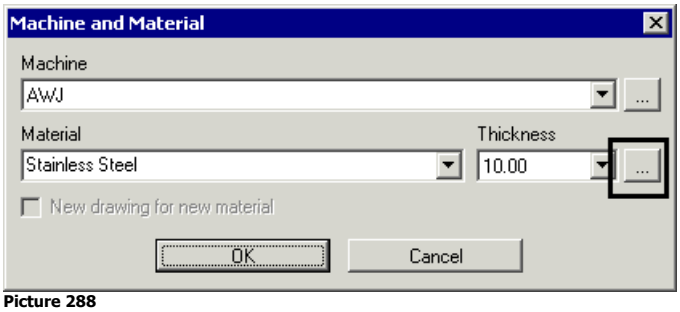
These settings are the default settings for the Estimate cost command. The settings also affects the info in the AWJ Settings.

Chapter 20.
Material setting



Picture 287

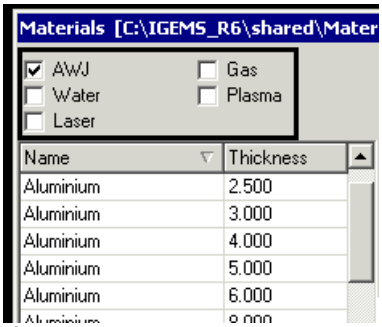
The toolpath must have a default machine and material. If you want to select or change material, then select material and thickness from the list. It is possible to activate an option that automatically creates a new empty drawing if you change active material (see page 111). If you want to change the material settings, then press the button shown in below image.



Picture 288

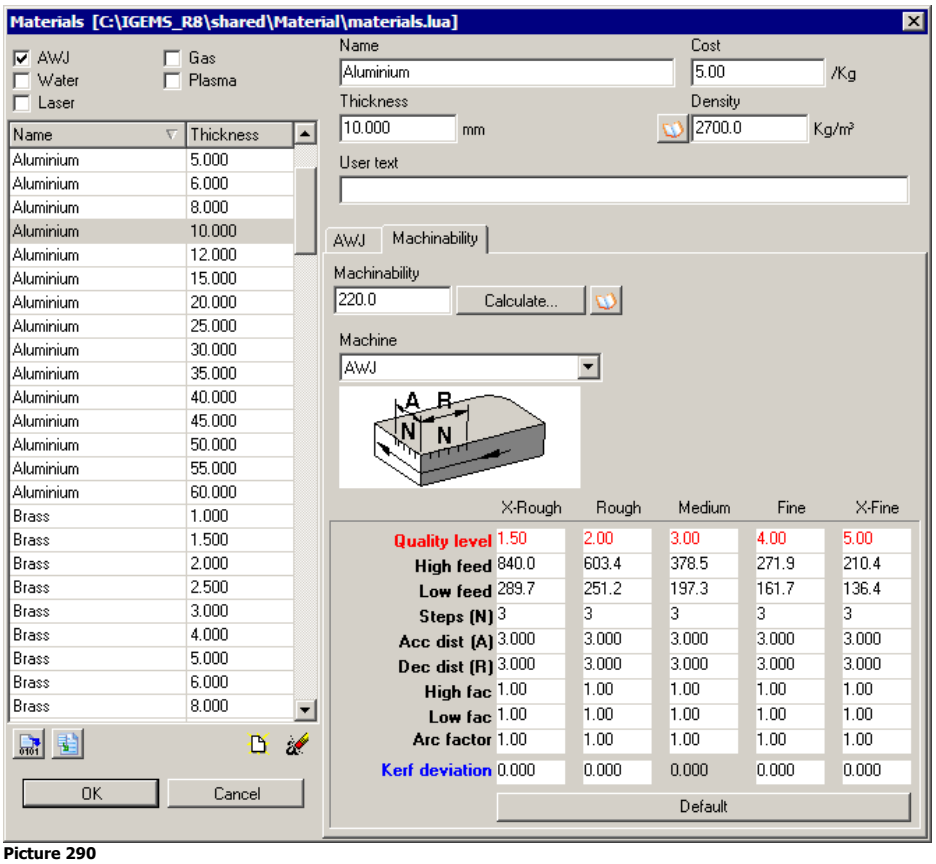
Most of the settings are self explained but here are some remarks

Machine bound materials



Picture 289

Some materials are impossible to cut with some machine types. This setting controls the connection between the machine and materials. If you only have one machine type then you can deactivate all other checkboxes.

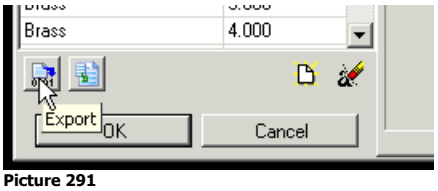


Picture 290

Multi select

You can select one or multiple materials on the list. The command supports Windows standard use of CRTL and SHIFT. By using this feature you can easily handle many materials at the same time.

Material library

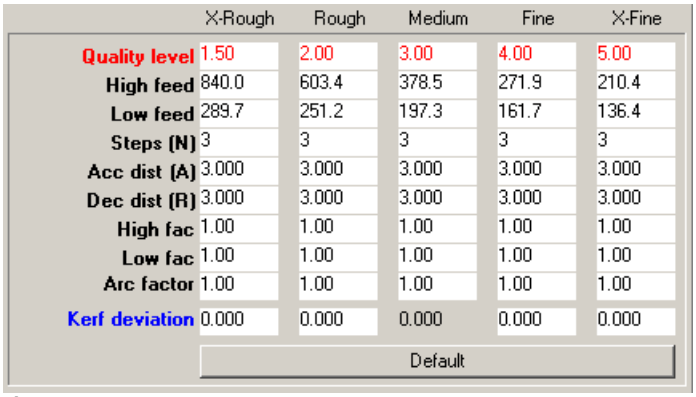


Picture 291

By using this four buttons you can export, import, copy and delete materials.

Machinability settings

Cutting speed for Advance Water Jet



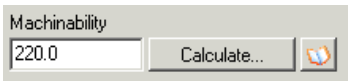
Picture 292

In this database you can define the cutting parameters for all quality. The cutting parameters are as follows:

- Quality level: This is a global setting, a low value is a rough quality and a higher value is a fine quality. Normal values of these settings are 1.5 to 5.
- High feed: This feed is used by straight lines and arc radius larger than the thickness of the material.
- Low feed: This feed is used in sharp corners, if the angle is more than 90 degrees.
- Steps: This is the number of ramping steps.
- Acceleration distance: This is the total length of all acceleration step.
- Deceleration distance: This is the total length of all deceleration step.
- High factor: This value is in a relation to a formula that calculates the cutting speed. If the value is 1.0 then the value is according to the formula.
- Low factor: This controls the relation between the formula and the low feed.
- Arc factor: This value controls the cutting speed in the arcs. If you set this value to a higher value then the small arcs will use a higher speed. Normal value is 1.0.
- Kerf deviation: This is only supported by some machines. You can use different tool radius compensation on different quality.
- Default button:
  The Default button corrugate all settings to default values

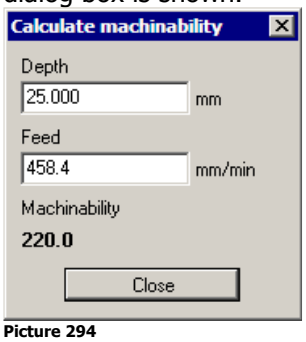
Machinability value

The Machinability value indicates how fast the material can be cut with abrasive water jet.



Picture 293

If you change the Machinability for a material then the cutting speeds in the database is recalculated. If you click on the "book" button you will have a list of different materials and Machinabilitys. If you click on the Calculate button following dialog box is shown.

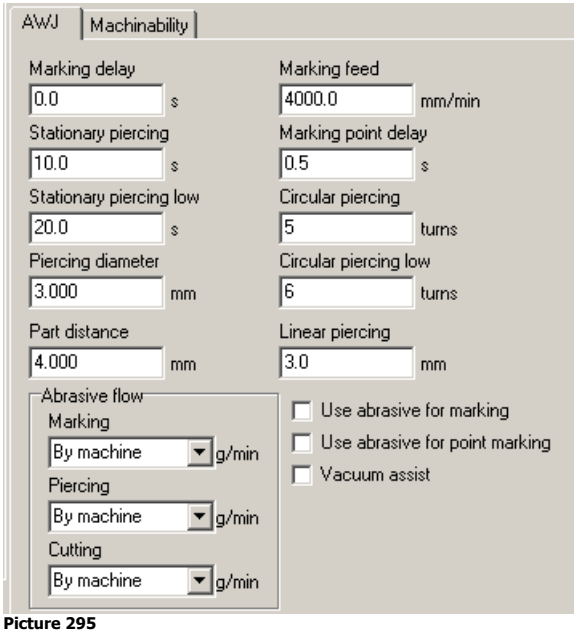


Picture 294

By making a test cut and measure the cutting depth you can calculate the machinability factor for a specific material.

Advance Water Jet Settings

Most settings are self explained but here are some remarks

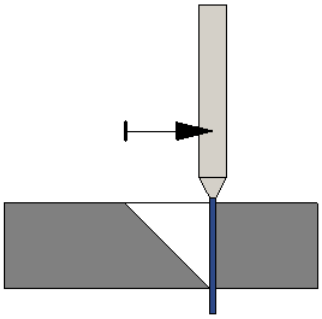


Picture 295

The piercing values are individual for all materials and thicknesses.

Abrasive Flow
In IGEMS R8 you can have different amount of abrasive for Marking, Piercing and Cutting. If you want to use the setting specified from the machine setting then set the value to "By machine".

Linear piercing
The Linear piercing value is the length needed for cutting through the material in low speed medium cutting quality. This kind of piercing is used by the piercing type 0 (Start on geometry) and 1 (Direct start).

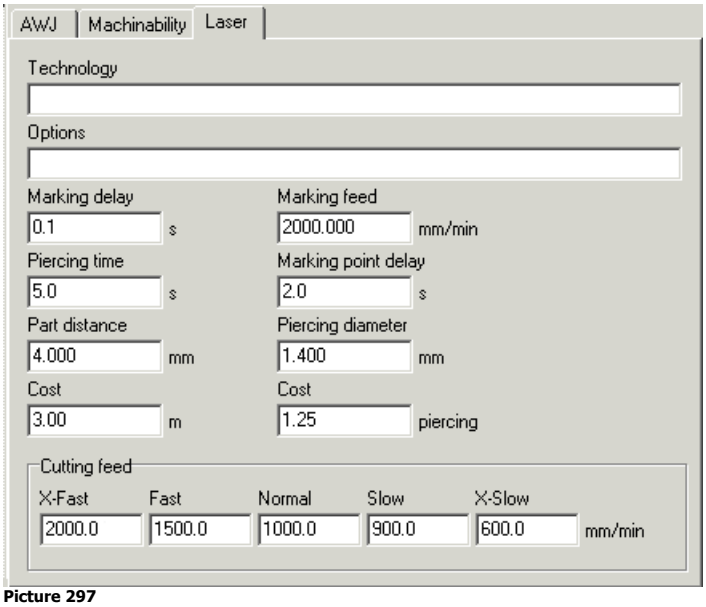


Picture 296

Linear piercing is the most time saving method to pierce through the material.

Part distance
The Part distance is used as default value by the nesting commands.

Laser Settings

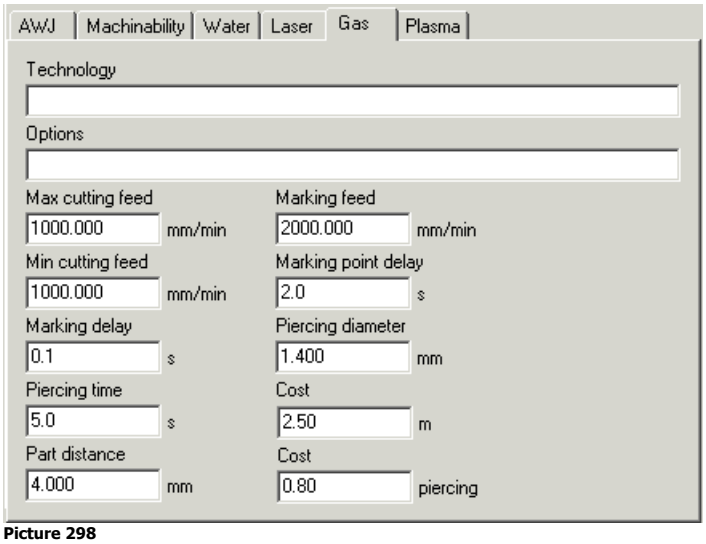


Picture 297

Most settings are self explained but here are some remarks.

- The technology and options information can be used for various things. The value can be used by the postprocessor.
- The Cost values are for consumption of gas and other cost that are related to cutting length and piercing.
- The cutting feed is related to the cutting quality.

Water, Gas and Plasma settings



Picture 298

The settings for these three machines are identical. Most of the settings are self explained but here are some remarks.

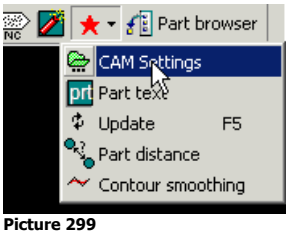
- The technology and options information can be used for various things. The values can be used by the postprocessor.
- The Cost values are for costs that are related to cutting length and piercing.
- If you have different Max and Min cutting speed, then these values are used by the quality command for calculating different speeds.

Chapter 21.
Shared folder and CAM-Settings

If IGEMS are installed on several computers at the same company, then it is a very good idea to share the same materials, leads and other settings. If you want to use a shared folder then follow these steps.

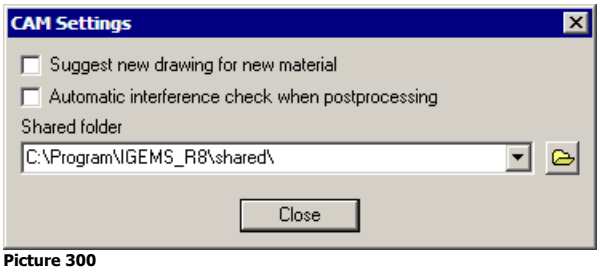
Step 1:
Set up all machines, materials, and leads as you want them to be in the first computer.

Step 2:
Make a new folder on the network, be sure that you have rights to read and write in this folder.

Step 3:


Picture 299

Start the CAM Settings command.



Picture 300

Select the empty folder from Step 2:
All files will now be copied to the empty folder.

Step 4:
Change the Shared folder on all other computers that will be used by IGEMS.

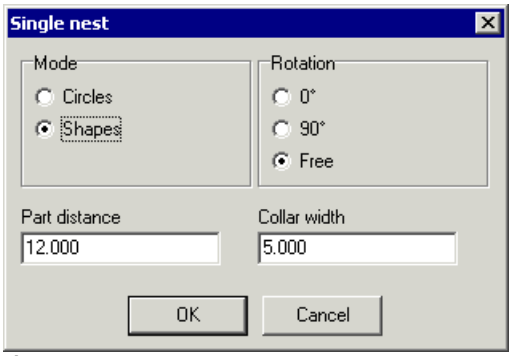
Chapter 22.
Nesting Level 1

Nesting level 1 have three different nesting commands.

Single nest

Picture 301

This nesting command can nest parts and closed objects. It runs in two different modes depending of the geometry to nest.



Picture 302

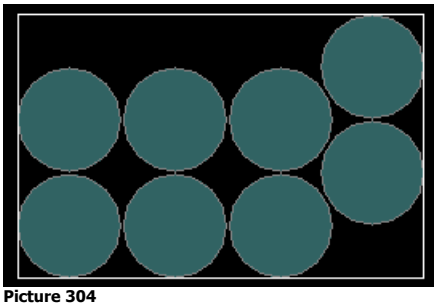
When you press OK, you should define a rectangular nest area.

Shape mode

Picture 303

This mode makes a rectangular grid.

Circle mode



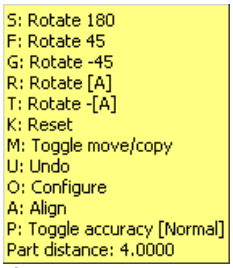
Picture 304

This mode do not nest in a grid, it uses an optimal method for finding the maximum number of single parts in an area.

Quick nest

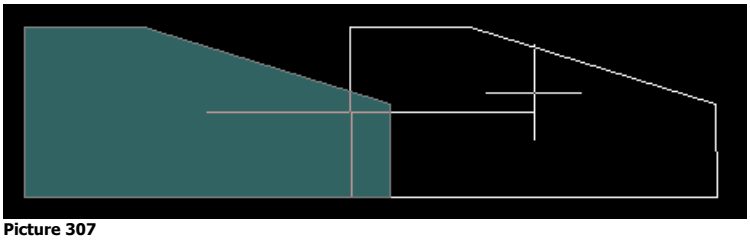
Picture 305

This command positions individual parts/geometry with a pre specified distance in relation to other parts/geometries. Quick nest can work with parts or closed geometries.



Picture 306

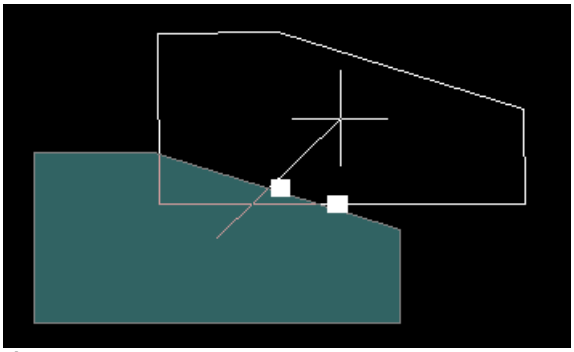
You can use following sub commands while using the Quick nest.



Picture 307

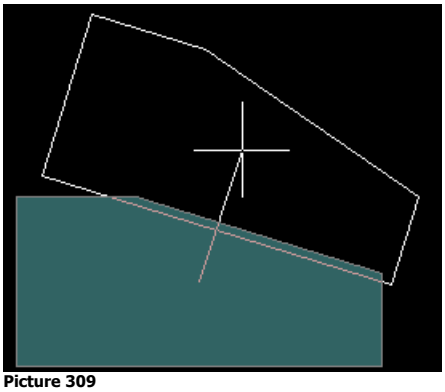
The idea is to position one part overlapping another part. When you place the part it will bump out to the specified Part distance.

Align parts



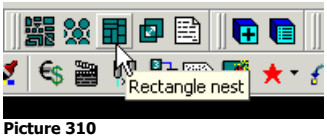
Picture 308

If you hold the part and two midpoints close to each other, use the Align option, then the part edges will align with each other.



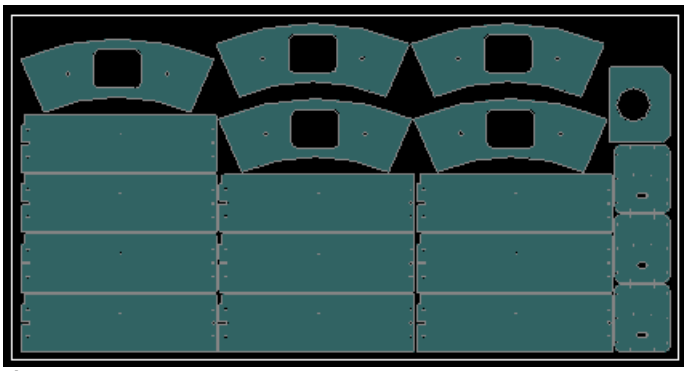
Picture 309

Rectangle nest



Picture 310

This command works only with parts, but it can use multiple parts at the same time. Select the parts and enter the Part distance. The command nests dynamically as many as needed of each part.



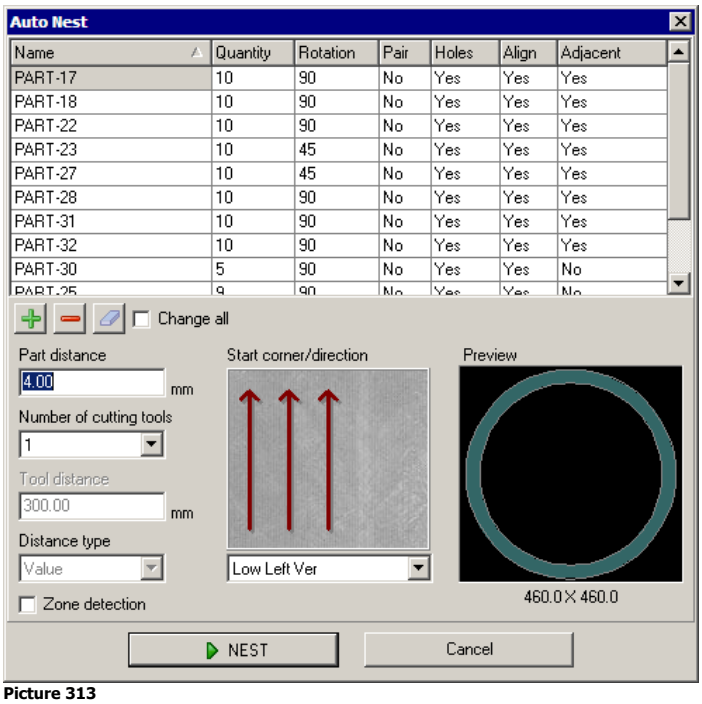
Picture 311

This command is fast and simple to use. The nest algorithm is based on rectangles. If the part has a rectangular shape it gives a very good nesting result.

Chapter 23.
Nesting Level 2

Picture 312

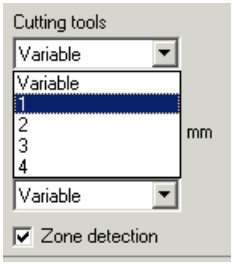
This command automatically nests parts on sheets.



Picture 313

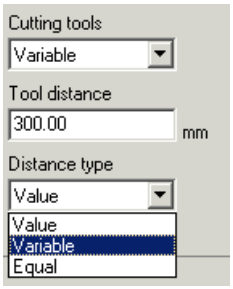
Some of these settings need to be explained.

Advance tool selection



Picture 314

These settings controls how many tools that should be used for the nesting. If you set the value to "Variable" then the nesting nests as many parts as possible with as many tools as possible. If it is not possible to use all tools then the nesting will try to use less tools and finally use only one tool.



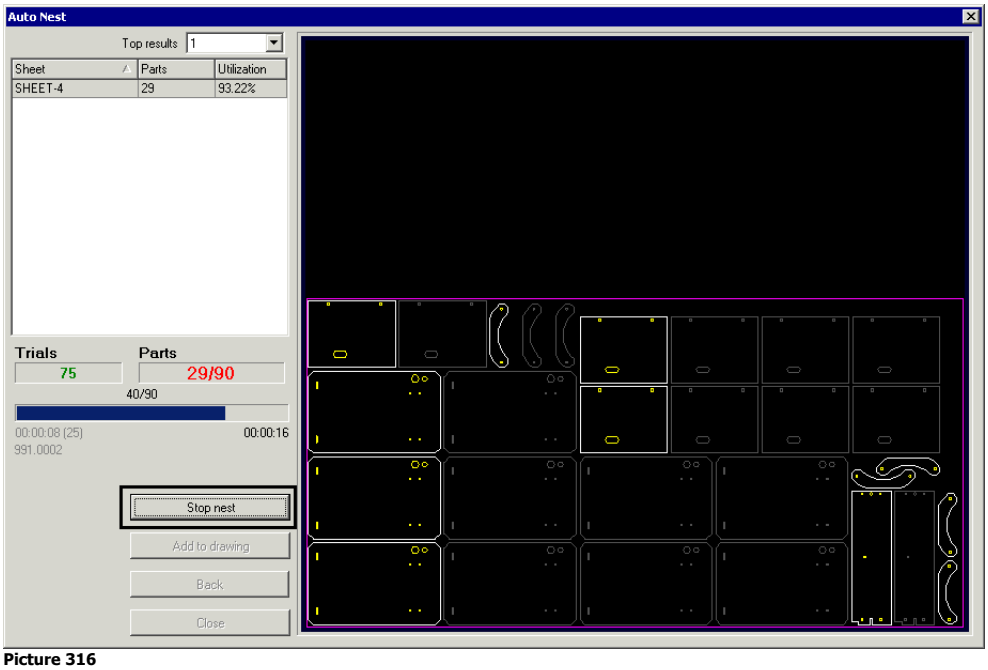
Picture 315

If the Distance type is Value then the nesting will use the specified value. If it is Variable then it will use variable distances defined in the machine settings. If it is set to Equal then the Sheet will be divided in selected number of tools.

Add parts and groups is deleted

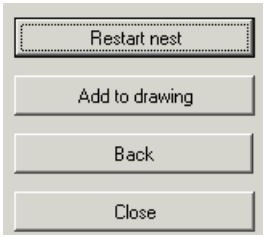
Execute nest

When you press the Nest button the calculations start and you will se following dialog.



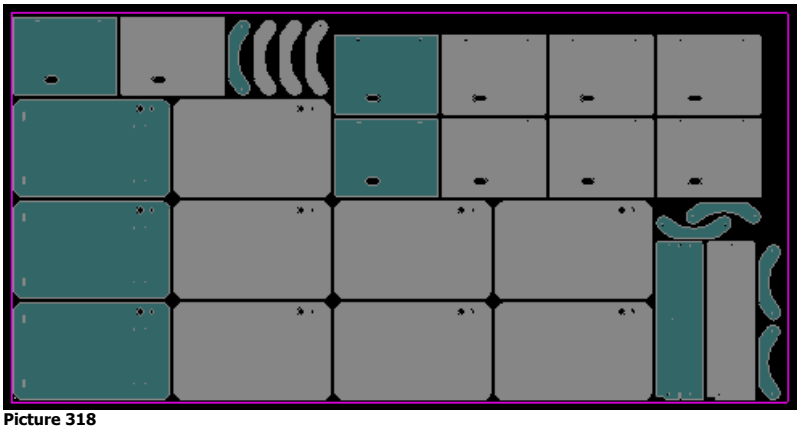
Picture 316

The nest will calculate until you press the stop nest.



Picture 317

This can be the result of an Automatic nesting with variable number of tools.

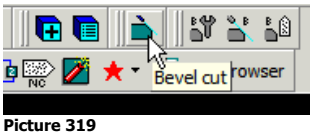


Picture 318

Chapter 24.
Bevel cutting on standard parts

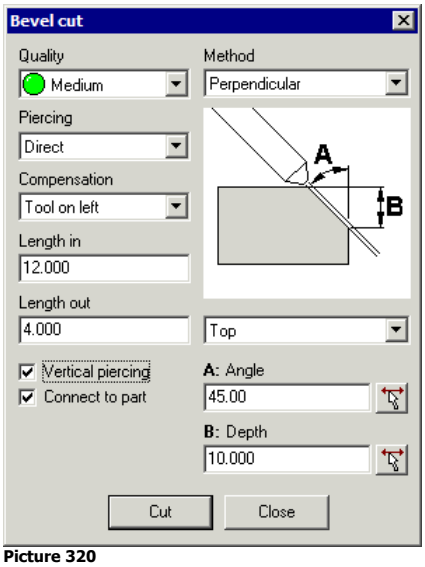
This option can only be used if you have IGEMS Bevel cutting option and a 5-axis controlled machine.

Bevel cut



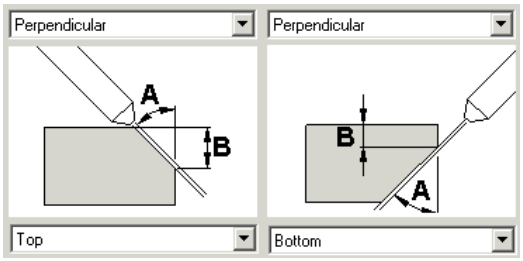
Picture 319

This command will make a bevel cut on a standard parts. The same postprocessor is used for this type of Bevel cutting. The command in following dialog box is self explained but here are some remarks.



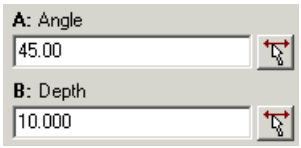
Picture 320

Perpendicular



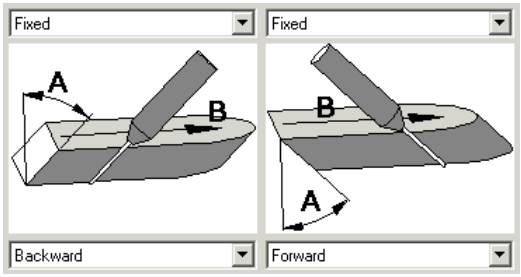
Picture 321

The perpendicular method makes a cut that are perpendicular to the edge of the part. You can cut the top or the bottom, following settings control the cutting angle and depth (se also page 100).



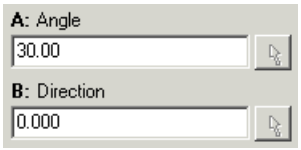
Picture 322

Fixed angle



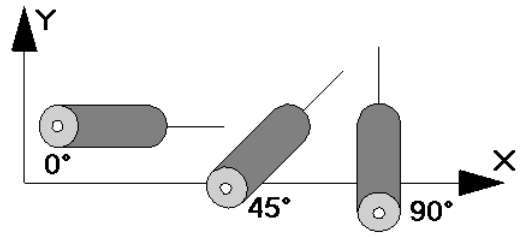
Picture 323

When using the Fixed angle method, then the same tool angle will be used for the whole toolpath. You can control the cutting angle and the direction of the cut with following settings.



Picture 324

The meaning of direction is the direction of the jet when looking on the jet from Z-axis.

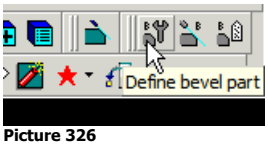


Picture 325

Chapter 25.
Bevel cutting on advance geometry

In IGEMS it is also possible to create more complicated geometry for 5-axis cutting. This option creates a special part. The part can not be used for nesting.

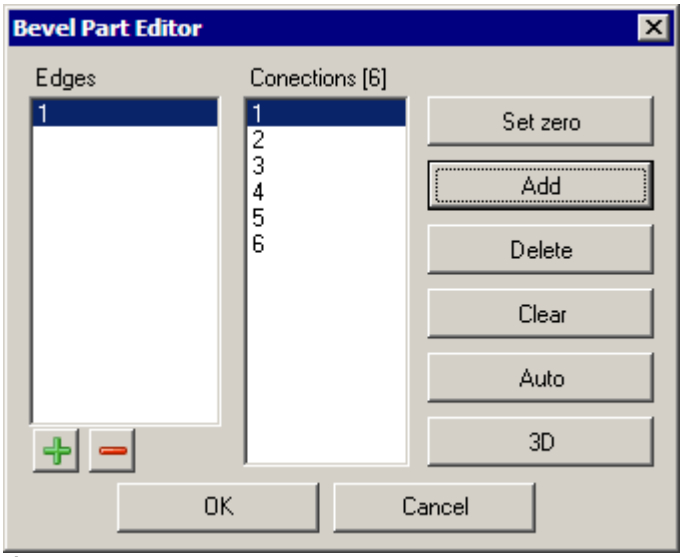
Define Bevel part



Picture 326

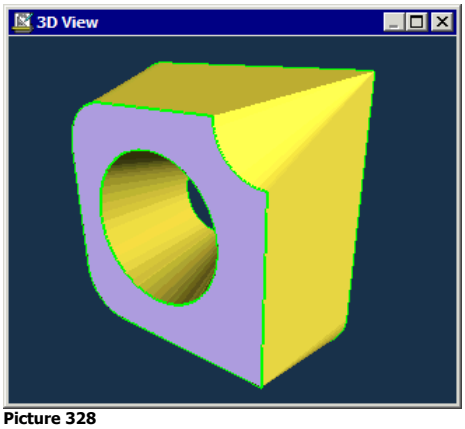
The Bevel part must be defined by closed objects for the top side and the bottom side. Each pair of top/bottom definitions is called Edges. When you start this command you must start with select the top of the external geometry then the bottom of the external geometry.

Select top contour:
Select bottom contour:



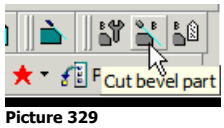
Picture 327

Each edge must have correct connections between the top and bottom definition. Use different sub command for defining the correct connections.
To check the connection it is possible to use the 3D-viewer.



Picture 328

Cut bevel part

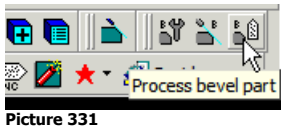


Picture 329

You must define the Bevel part before you can use this command. The method for this command is similar to the manual command, but the toolpath should be done between the connections.

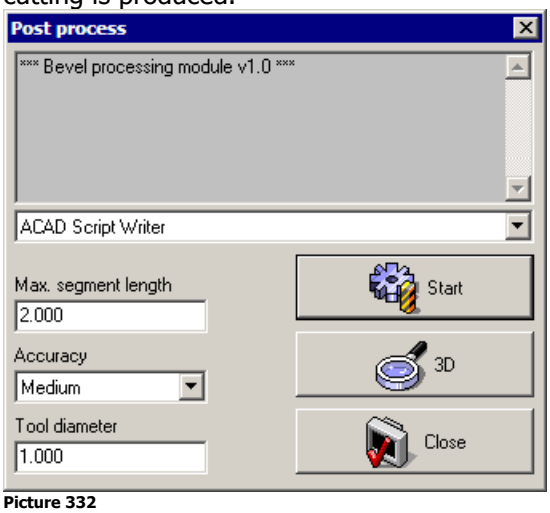
Picture 330

Process Bevel part



Picture 331

You must create a toolpath on the bevel part before you can make the CNC-file. The command shows following dialog box. By pressing OK a CNC-file for 5-axis cutting is produced.



Picture 332

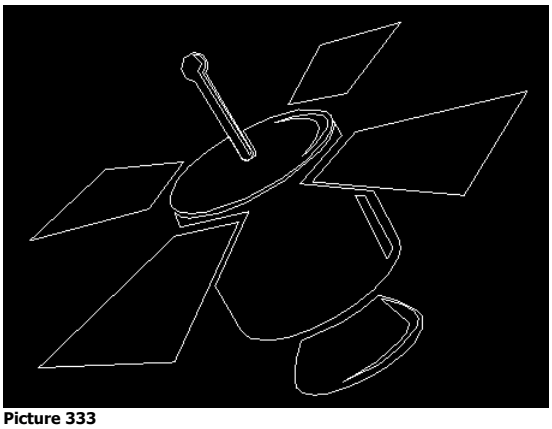
The postprocessor for the Advance Bevel is located in …"IGEMS_R8/Plugins/CAMNest/Bevel/Proc" directory

Chapter 26.
Tile Maker option

This is an option in IGEMS that makes it possible to easy produce tiles. This is the workflow:

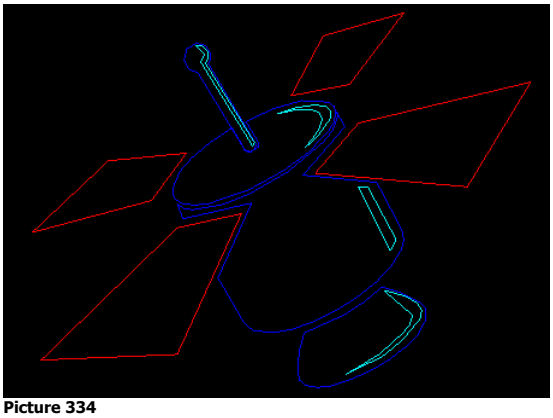
Step 1: Generate the drawing

The first step is to create the drawing. In this example we used Font Tracer and geometry from the web dings true type font. Use the Join command to create closed poly lines of all objects.



Picture 333

Next step is to set the colors. The color does not have to be the same as on the final tiles. The colors are used to separate the tiles into different bundles.

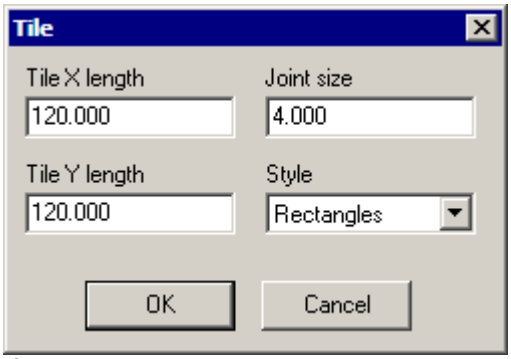


Picture 334

Insert the Tiles by using the Tile command.

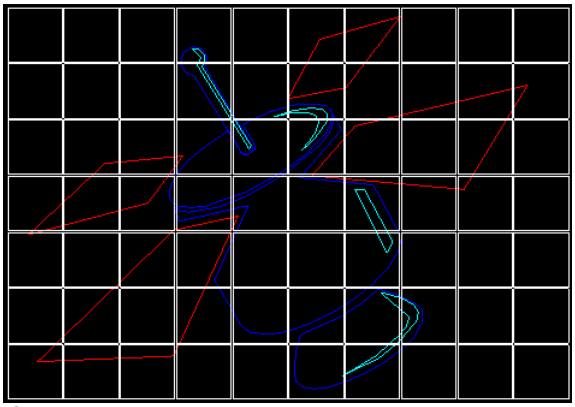
Picture 335

The tile command asks for first and second corner of the area that should be covered by tiles.



Picture 336

Specify the Tile data.
The Tile X length and Tile Y length is the size of the tiles.
The Joint size is the distance between each tile.



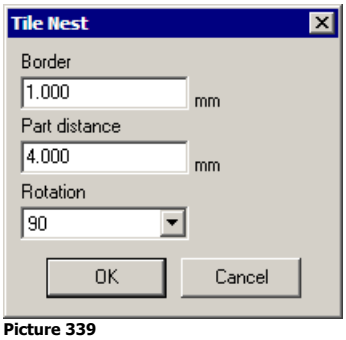
Picture 337

Step 2: Tile nest

The Tile will automatically nest geometry on tiles. Start the command by clicking on following button.

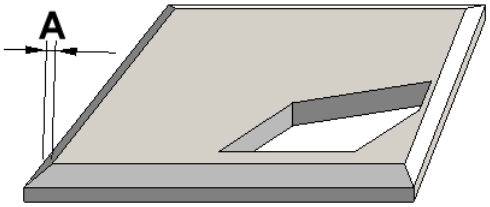
Picture 338

The Tile nest has following settings.



Picture 339

Border



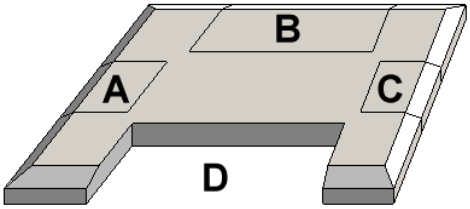
Picture 340

Tiles have sometimes a chamfer around the outer contour. This chamfer will be treated like a border and the nest will not place any center part on the border.

Part distance

This setting controls how close parts can be nested on the same tile.

Rotation



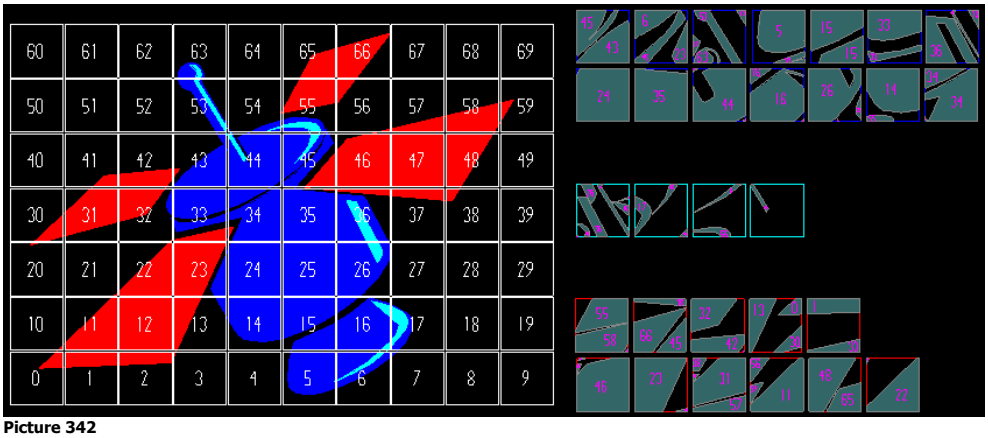
Picture 341

The rotation can be either No rotation, 90 or 180 degrees.

- None: If the part should be placed only in the same angle as it is on the drawing. This setting requires probably more tiles than other options.
- 180 degrees: If it's OK that D also can be placed on position B.
- 90 degrees: If the tiles are symmetric in geometry, color and in structure then the best option is 90 degree. This means that the part can be placed on optional edge of the tile (A, B, C or D).

When you press OK the command asks for following:

Select filter object: (Pick on one tile)
This command will put a number on all tiles.
Select polys: (Select the tiles and all inside geometry)



Picture 342

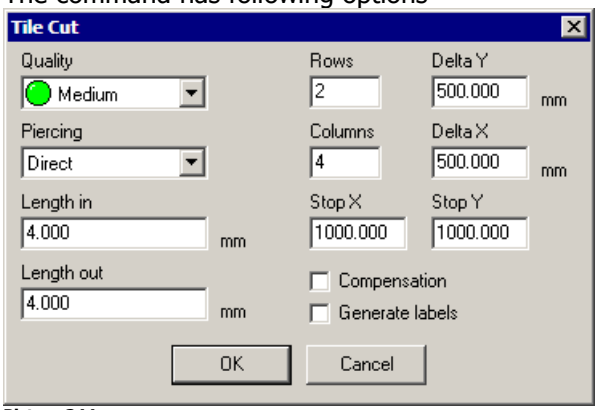
The Tile nest will only nest parts that need to be cut.

Step 3: Tile cut

This command automatically adds the tool path and the cutting order between the parts.
The Tile cut command starts by clicking on following button:

Picture 343

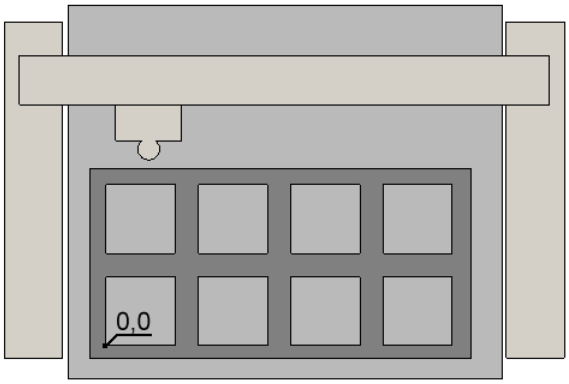
The command has following options



Picture 344

Cutting parameters
You can control the cutting parameters like quality, piercing and lead in length normal way.

Fixture parameters
This command needs a fixture in the machine.

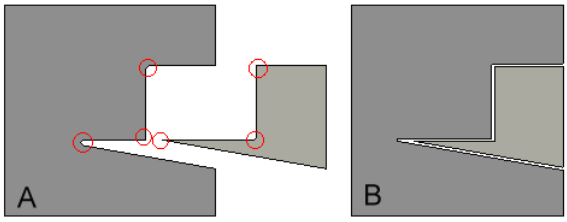


Picture 345

The value Rows, Columns, Delta Y and Delta X is related to the measurements and design of the fixture. The base point for the CNC-is automatically set to the Lower Left corner.

Stop
After cutting the first 8 tiles (in this example) tiles, the machine will go to a parking position. This can be defined by the Stop X and Stop Y values.

Compensation

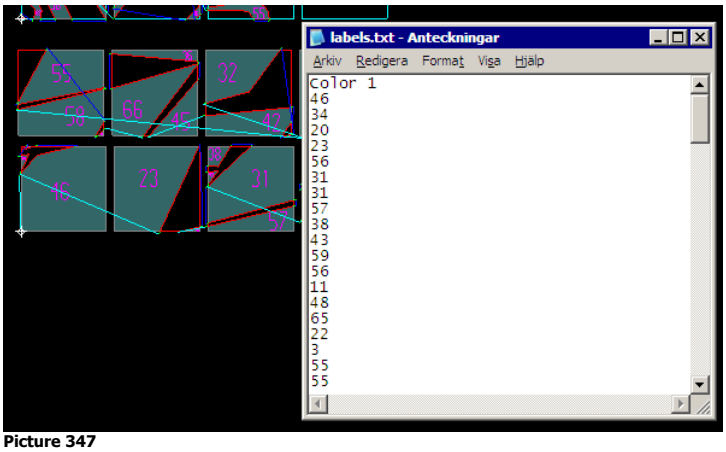


Picture 346

If you cut with tool compensation (Example A) then you can make tiles without getting any grout. To cut without a grout size means that you will needs manual modification of the machined tiles. The reason is because all inside corners will have the same radius as the jet, but the outside corners become sharp.
The most common way is to cut tiles without tool compensation (Example B). When you cut without tool compensation then the grout size between the tiles will be the same as the Jet diameter. This way of cutting tiles does not need any manual modification to fit.

Labels

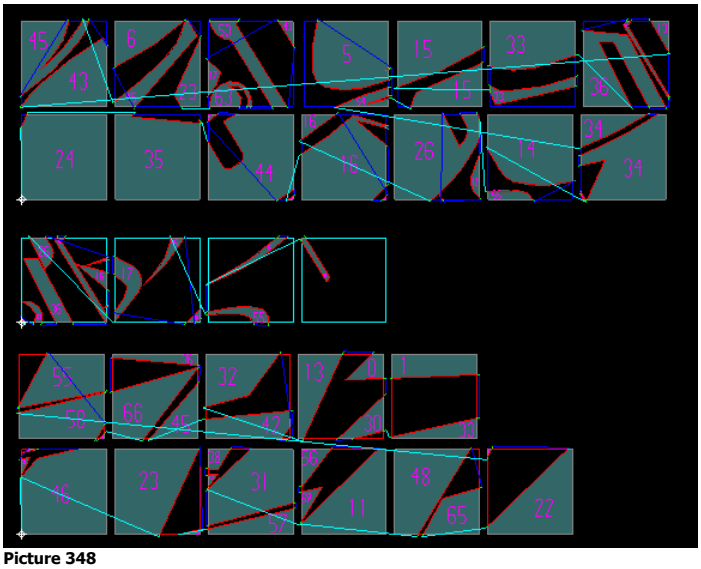
The command creates a text file that indicate the number position of each the tile on the layout.



Picture 347

This file may be used in excel or similar and be used to print out labels on each part of the tiles.

Tool path and cut order



Picture 348

When you press the OK button the command will ask following:

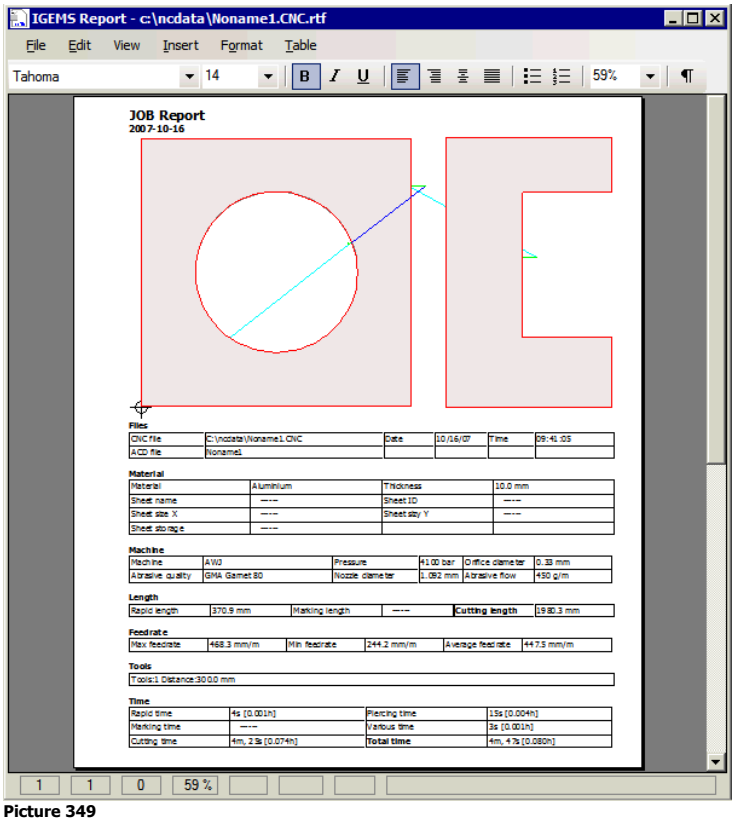
Select parts (Select all parts)

The command will now add tool path and cut orders automatically. You can generate the CNC-file by clicking on the post process button as usual.

Chapter 27.
The Report system

Chapter 28.
New Report system

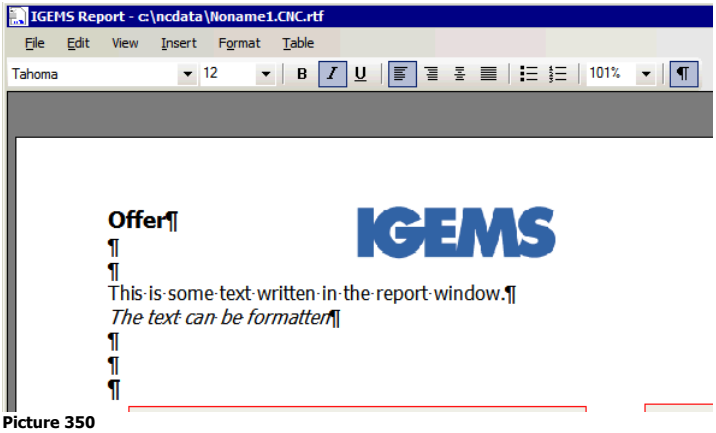
IGEMS are using the same report system in all kind of reports.



Picture 349

Dynamic document

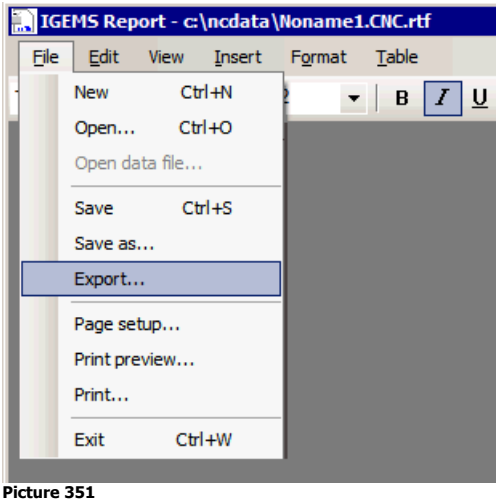
The report system is at the same time a simple word processor where you can add your own text, pictures and tables.



Picture 350

Save and export

If you save the document it will be saved as a RTF file. The RTF file is supported by the windows operating system and can also be imported to other program. Of this reason you do not need a special application for read and viewing reports made in IGEMS.



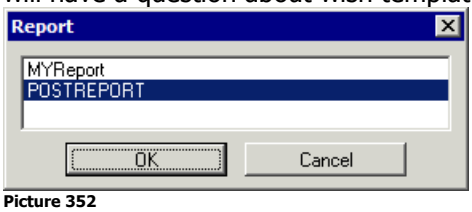
Picture 351

By the Export function you can save the reports in optional format like DOC, PDF and other.

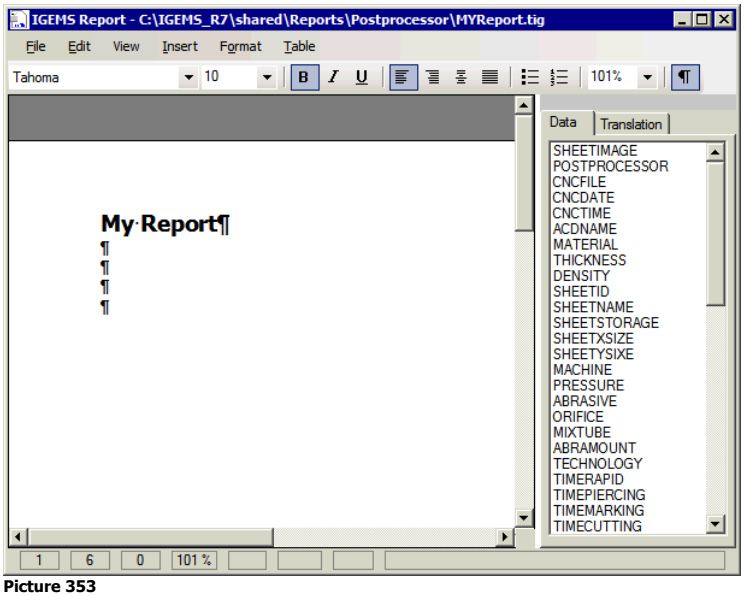
Design mode

By pressing the F5-key you can switch between Normal mode and Design mode. When using design mode you will design template files that are used in the normal mode when viewing the information.

The standard templates are read only. If you want to make your own templates then you must save the files in a different name. If you have several templates you will have a question about wish template to use when starting the report window.



Picture 352



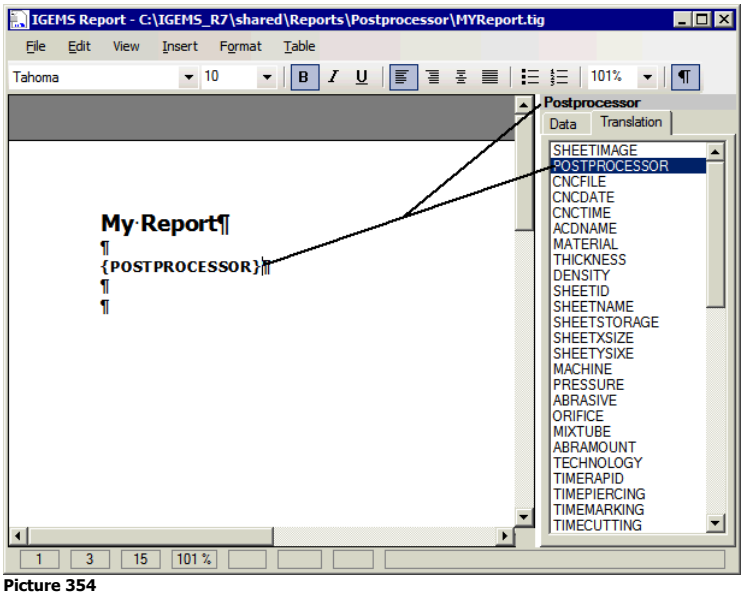
Picture 353

Plain text (Static text)

You can write any text in the document. This text will not be changed when you switch to normal mode. An example is the text "My Report" on the picture above.

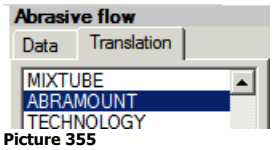
Translations (Dynamic text)

Only variables defined in the Translation field can be used as dynamic text. Dynamic text is only required when making reports that should be used in multiple languages. Put the cursor on an optional position and double click on an variable in the Translation field.



Picture 354

The variable inside the {curly braces} will be replace with the translation of the word in that language currently used by IGEMS.

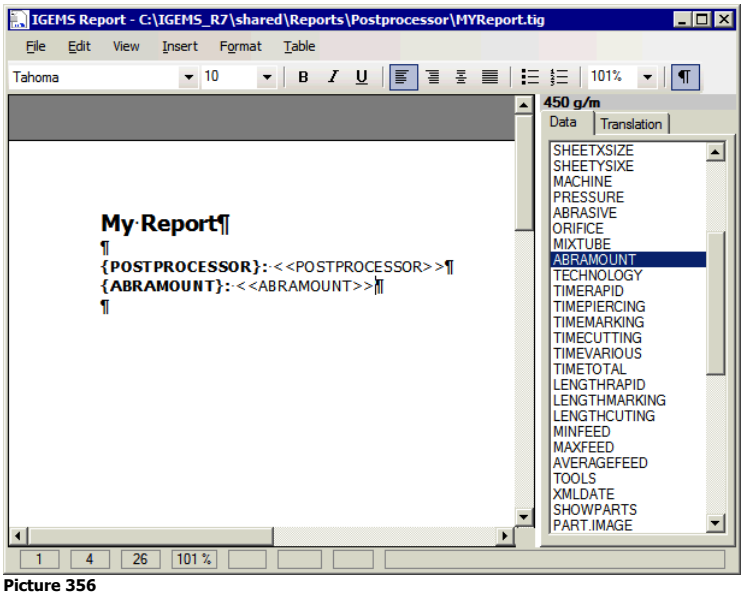


Picture 355

Above the Data and Translation field you can se the value of the selected variable.

Data variables

The variable inside the double << and >> will be replaced with the values of the variable. You can insert the variable at the same way as the Translation. The tags can (if you like) be formatted with different text styles.

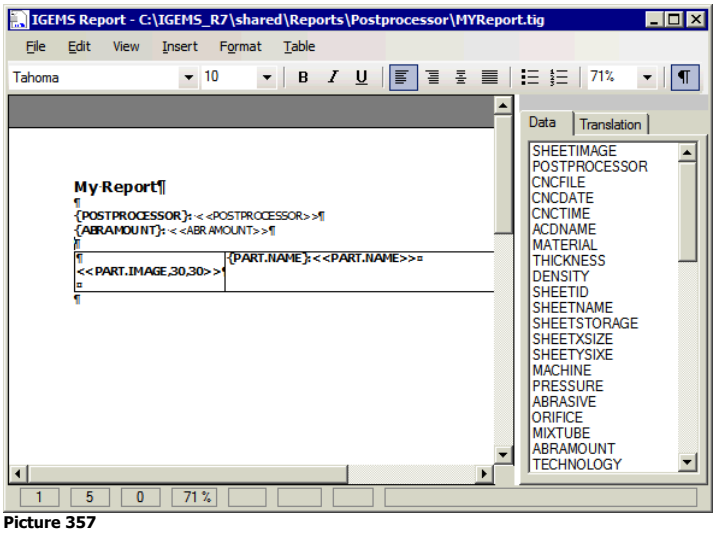


Picture 356

By pressing the F5 key you can switch back to normal mode and se the result.

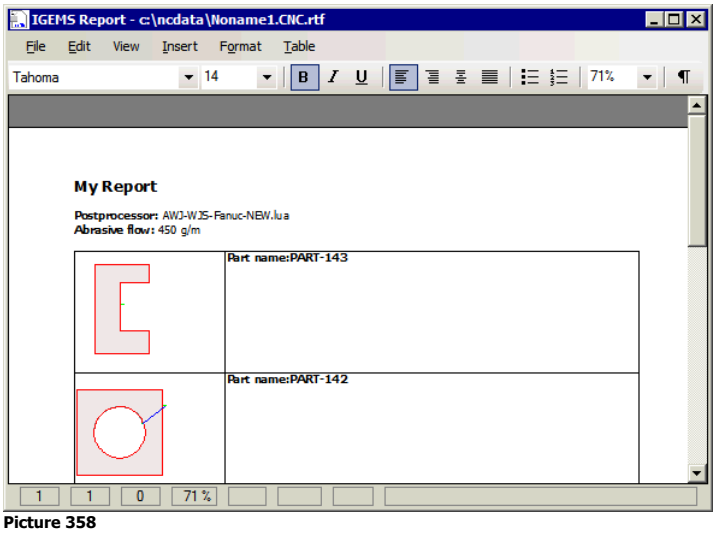
Lists

The variable name that include a dot is used for multiple parts. By using a list you can view separate information on multiple parts.



Picture 357

The number of lines in the list should always be one. It will be expand to the number of parts handled by the report. When using an image the size of the image must be included as extra argument. This is the result of the template above.



Picture 358

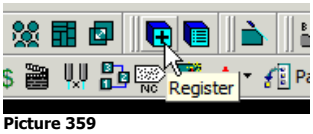
Creating reports outside IGEMS

Every time you create a new CNC-file, the postprocessor also create a data file. The file is saved in the directory specified by the Report path in the machine settings (or in the drawing directory if you are using the Project mode). The Data file has the same name as the CNC-file, but wit an extra extension ".RIG". It stands for (Report IGems). By double click on the RIG file you can start the Report program. To be able to do this you must install IGEMS in the actual computer. You do not need a license for using the reports.

Chapter 29.
Organizer module

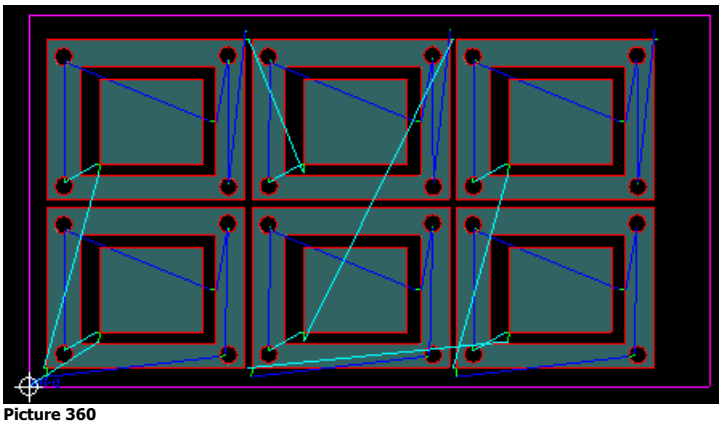
The Organizer module is an IGEMS option module. When you use the Organizer you never have to manually save parts and sheets, this will be done automatically by the software.

Register



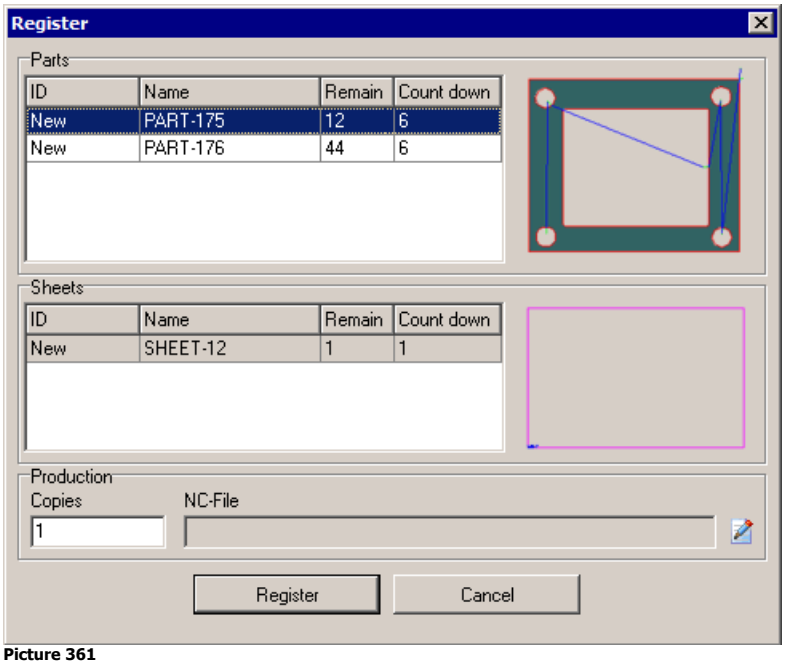
Picture 359

When using register, all files will be saved for each new part and sheet. The files are saved in the Shared folder. At the same time information about the part (customer, quantity and so on) will be saved in a database.



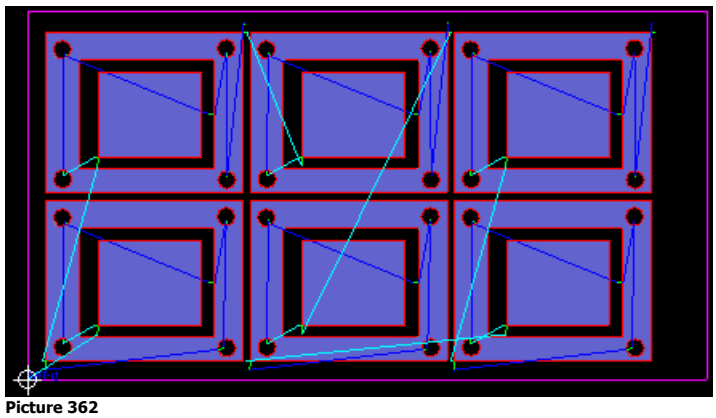
Picture 360

In previous picture you can se 12 parts and a sheet. When register you will see following information.



Picture 361

There a two new parts, the quantity remain to produce is 12 and 44. The quantity to produce will be counted down with 6 for each part. There is one new sheet, this will be registered as a new sheet, it will be counted down with one, there are no sheets left after the registration.
The count down value is only used if you register a Cut order. If you register without a Cut order then the parts and the sheets are saved only.

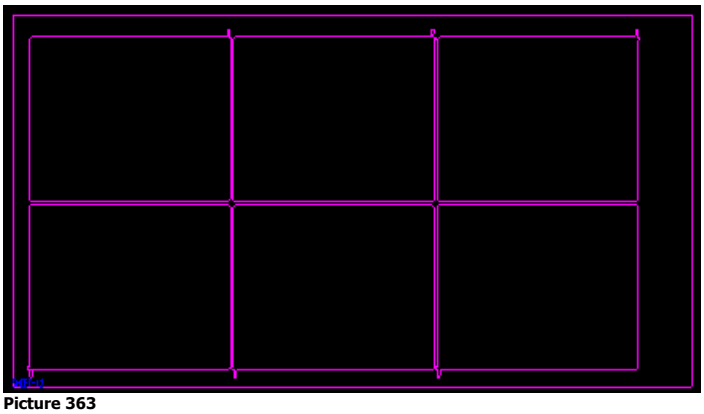


Picture 362

Registered parts are blue instead of grey.

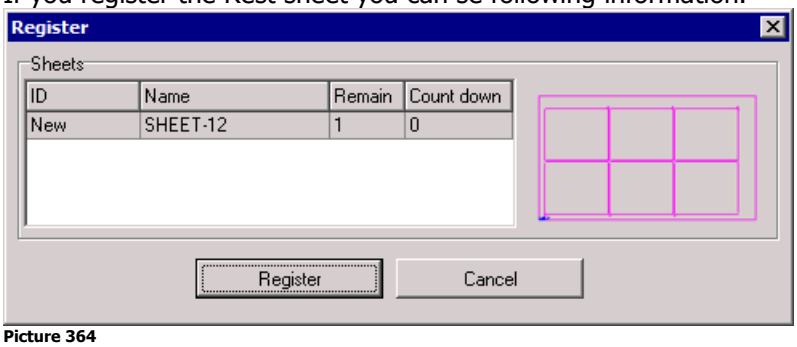
Register a rest sheet

If you want to save the rest sheet then you must lock the sheet before register it (see page 95).



Picture 363

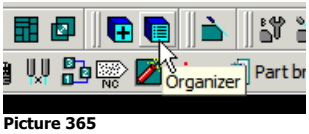
If you register the Rest sheet you can se following information.



Picture 364

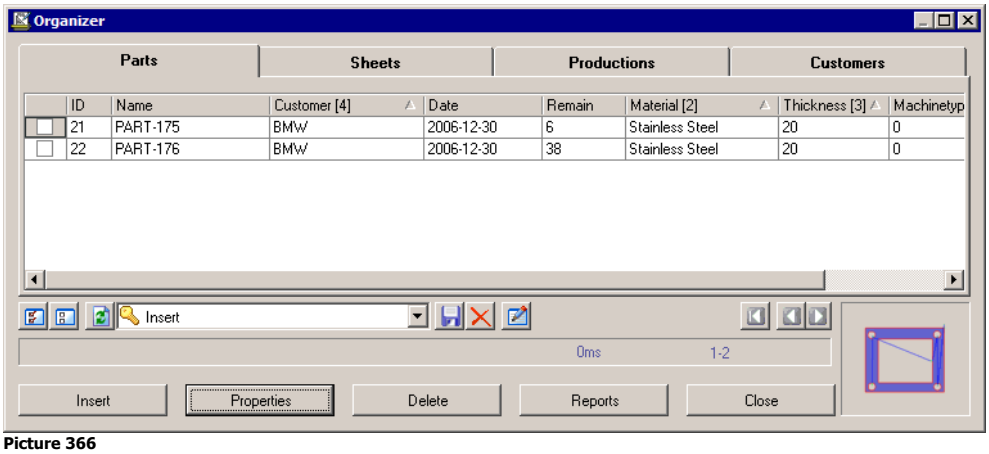
The rest sheet will be defined as a new sheet in the database. This sheet can be found in the database and re-used later.

View information



Picture 365

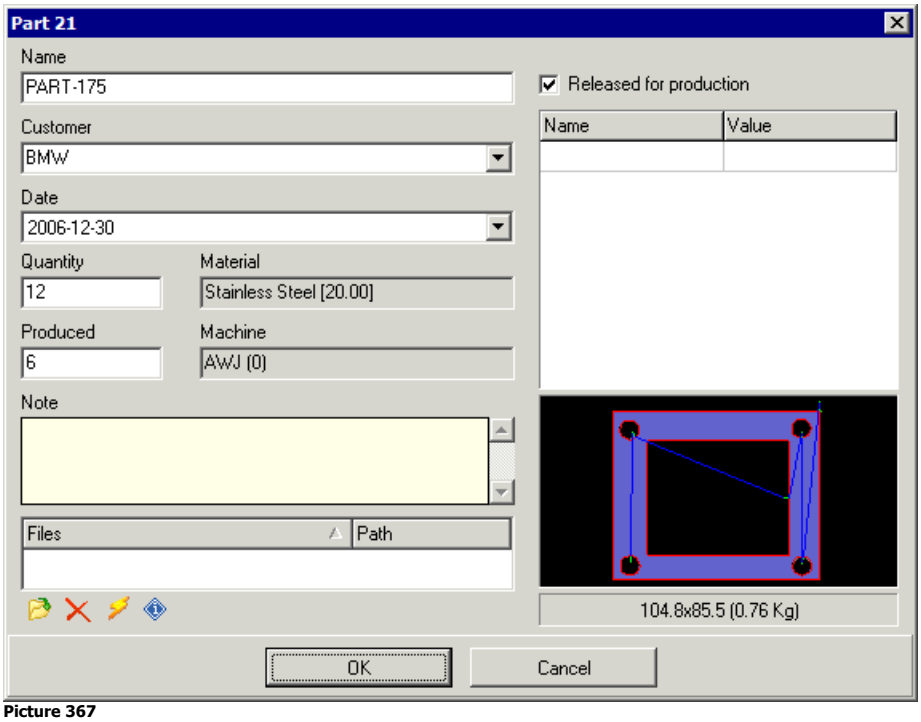
From the Organizer window you can se all you parts and sheets. You can make filter, insert parts and add information.



Picture 366

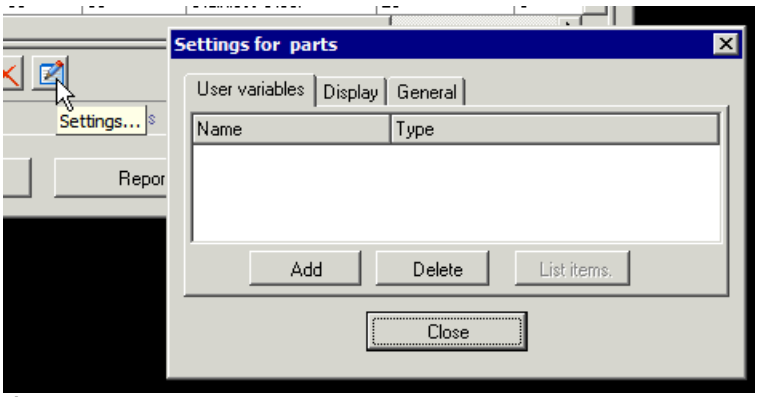
Properties

By double clicking on a line you can save more information about each part or sheet. Following dialog box is shown if you double click on a part.



Picture 367

User variables can be defined from the advance button in the main window.

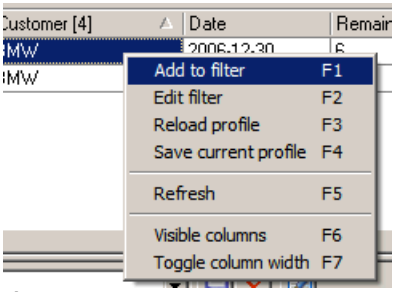


Picture 368

Profiles

Picture 369

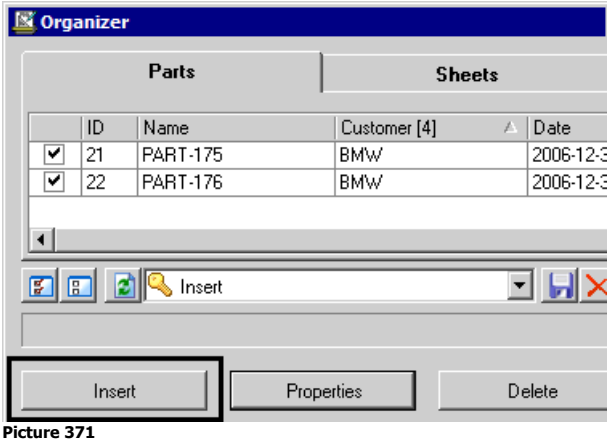
A Profile is contents of a filter, sort order and visible columns. By using the right mouse button on a column you get several choices to setup your own profile that can be saved for later re-use.



Picture 370

There are already some standard filters that can be used. The Insert filter shows you parts with the same material and thickness that is active in IGEMS.

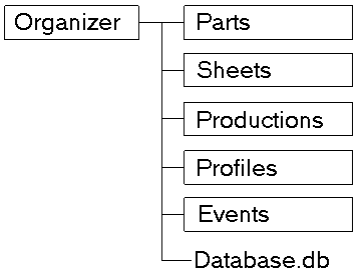
Insert parts and sheets



Picture 371

By selecting objects in the list and then press Insert you can insert parts and sheets to IGEMS.

The Organizer directory



Picture 372

All information is saved in different sub-directories of the Organizer directory.

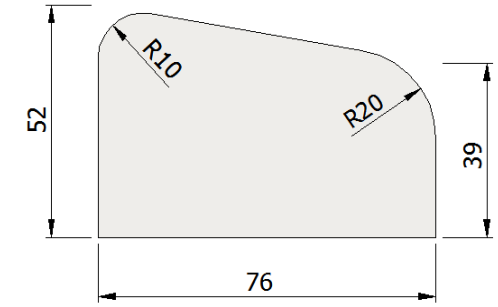
A new database

The Organizer is pre installed with a sample database with parts and sheets. If you want to start with a new empty database, then delete the complete Organizer directory. A new empty database will then be created automatically.

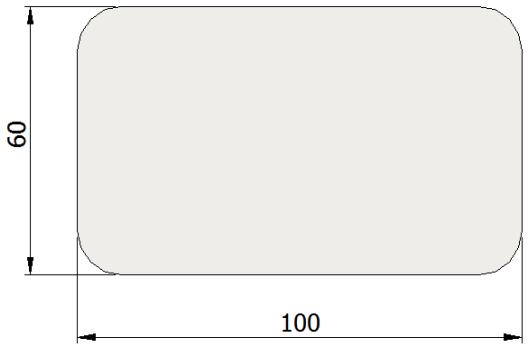
Backup

If you want to make a backup of your information, take a copy of the complete shared folder.

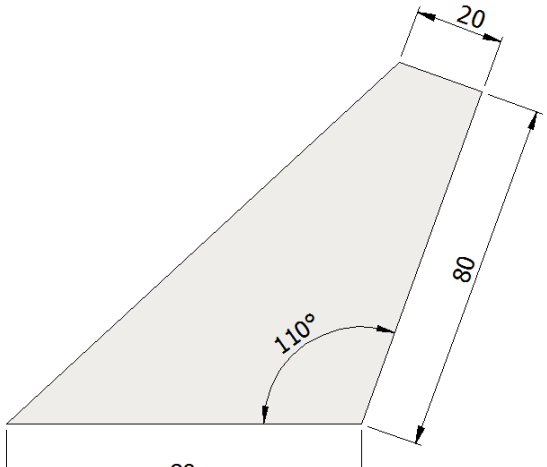
Chapter 30.
Drawing examples



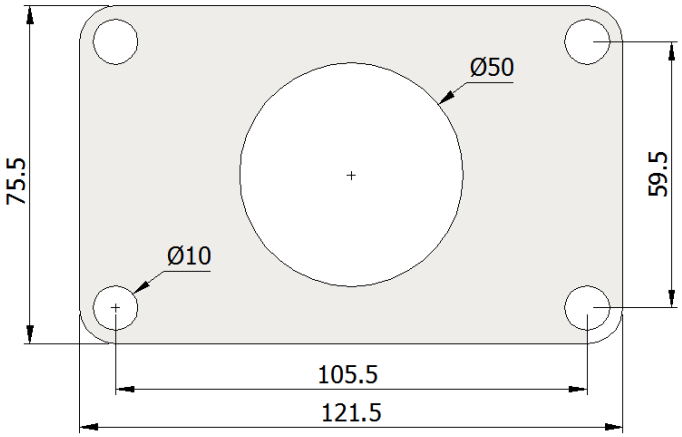
Drawing 1



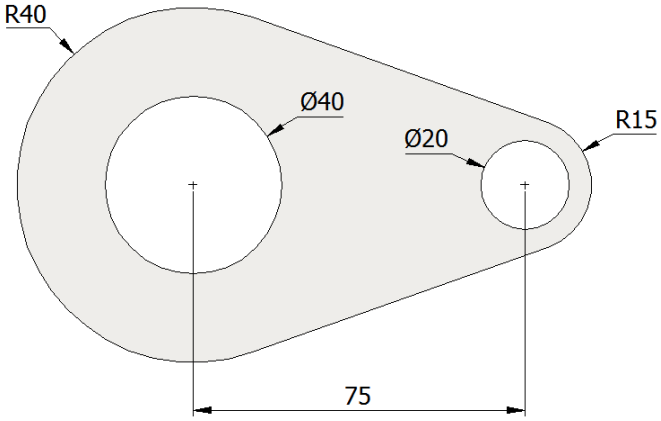
Drawing 2



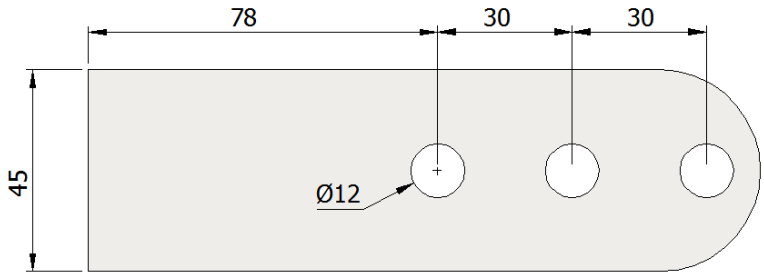
Drawing 3



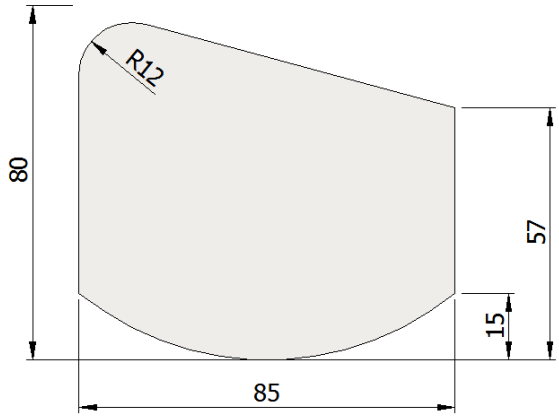
Drawing 4



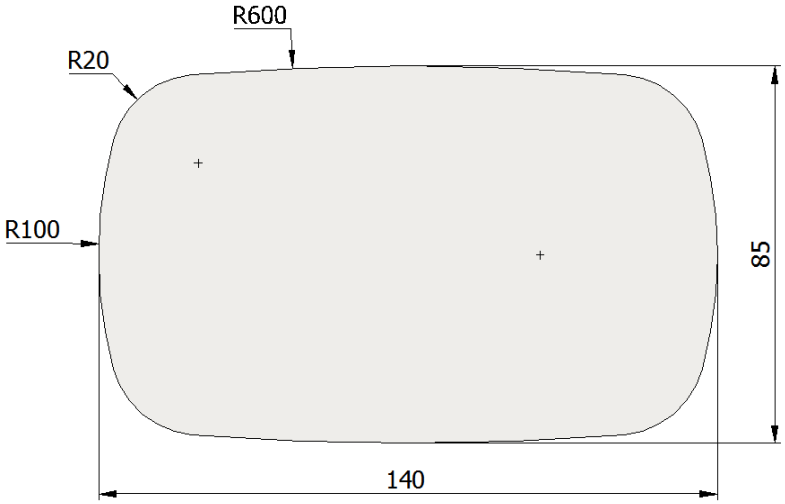
Drawing 5



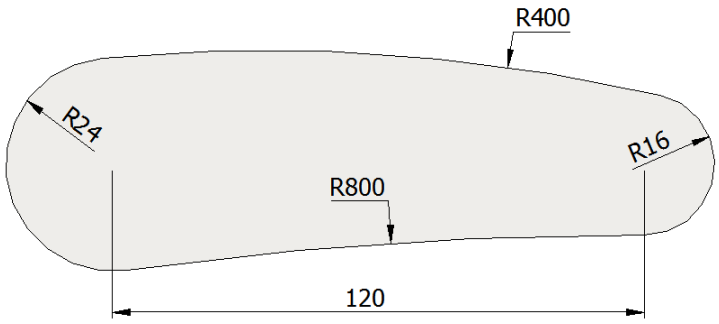
Drawing 6



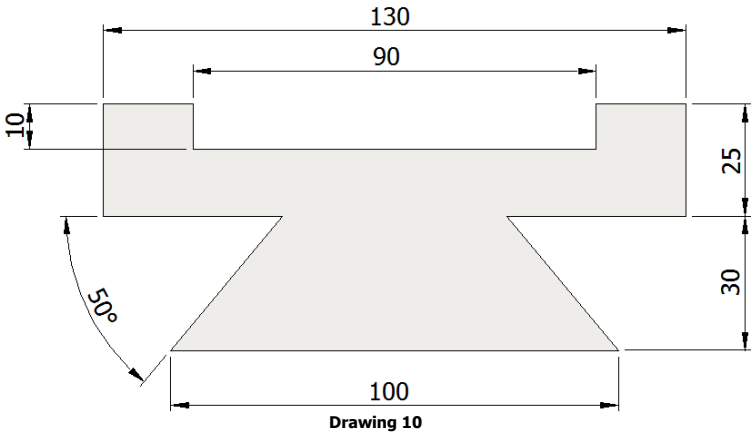
Drawing 7



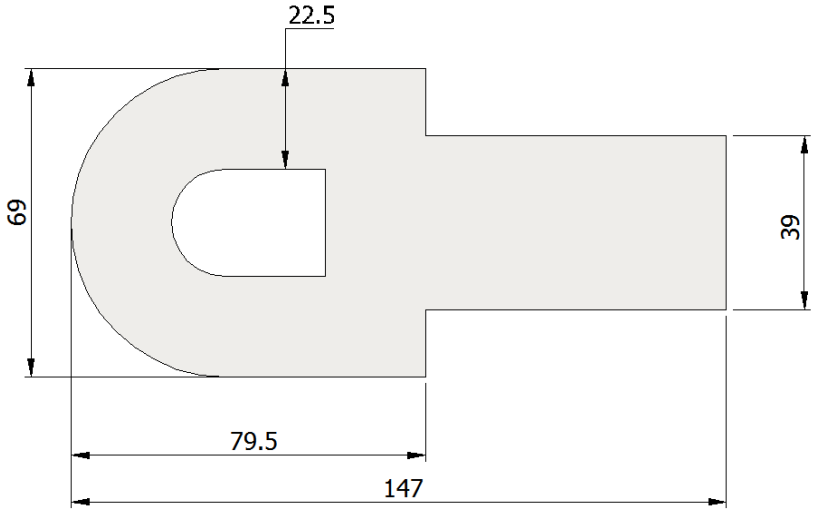
Drawing 8



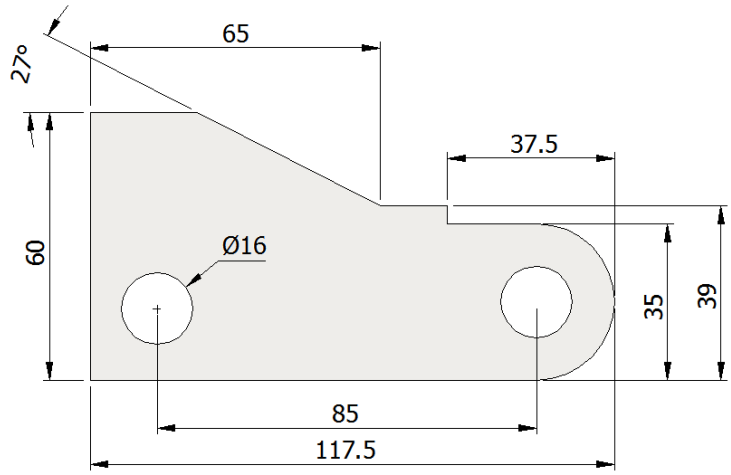
Drawing 9



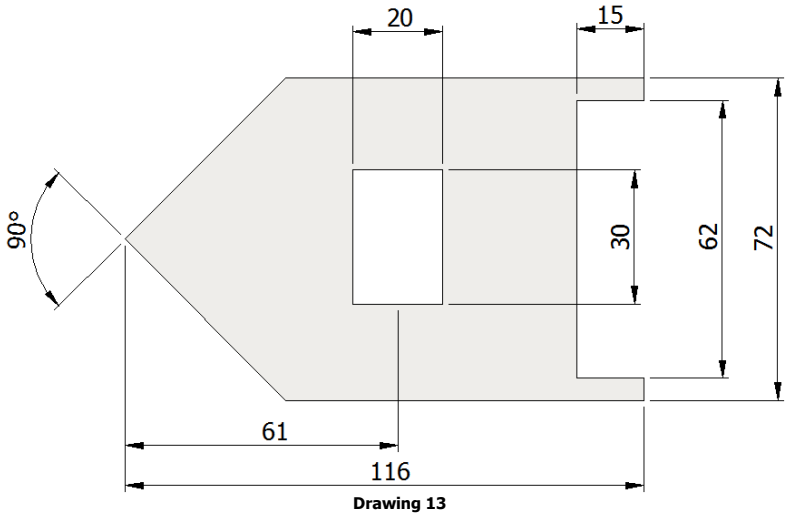
Drawing 10



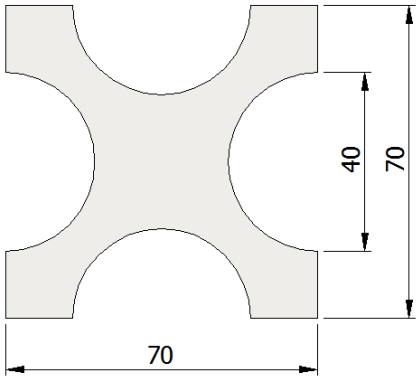
Drawing 11



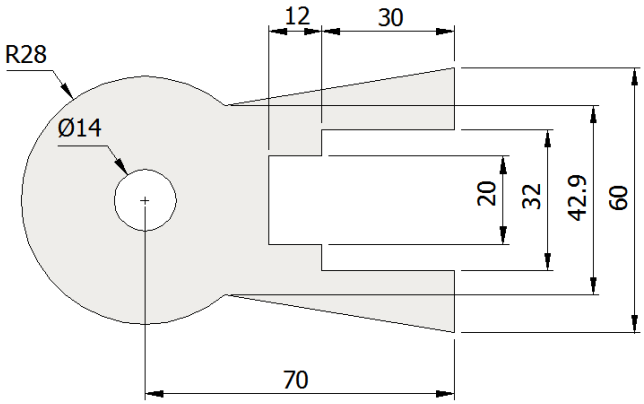
Drawing 12



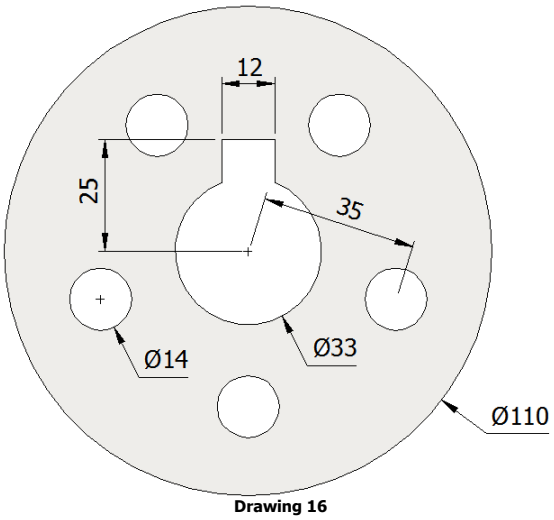
Drawing 13



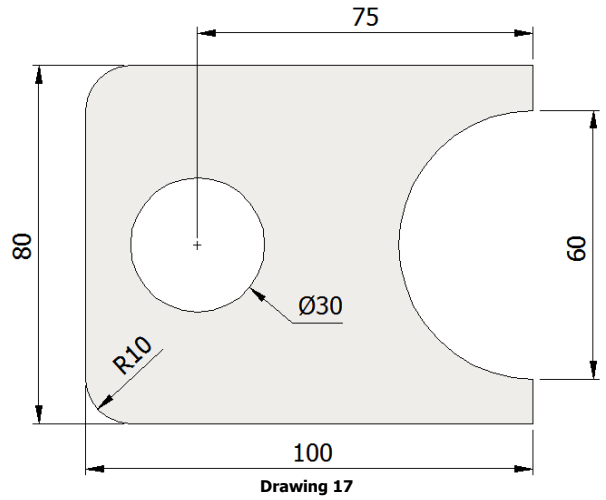
Drawing 14



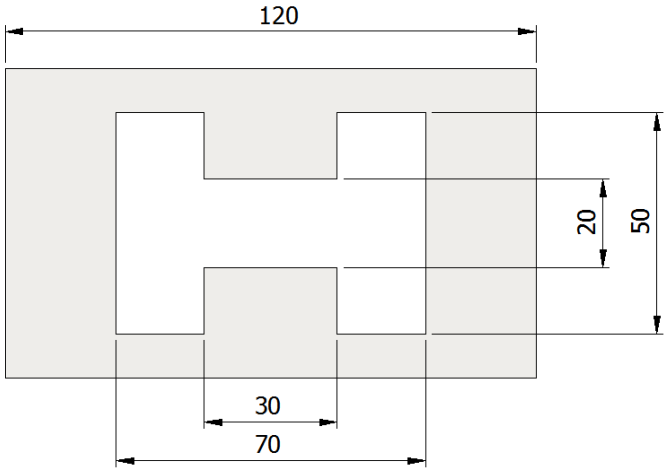
Drawing 15



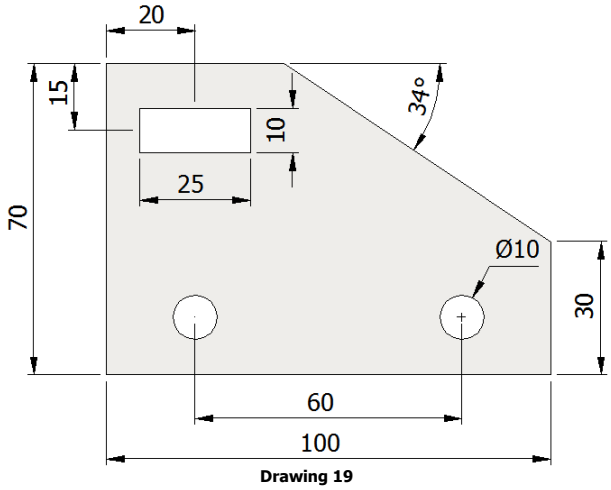
Drawing 16



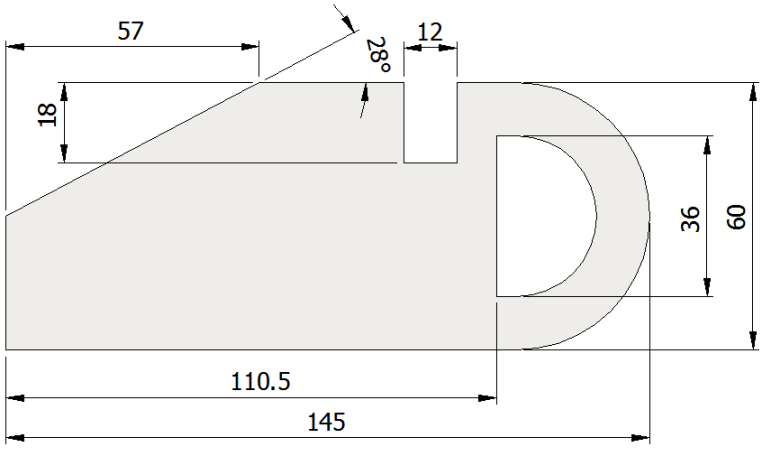
Drawing 17



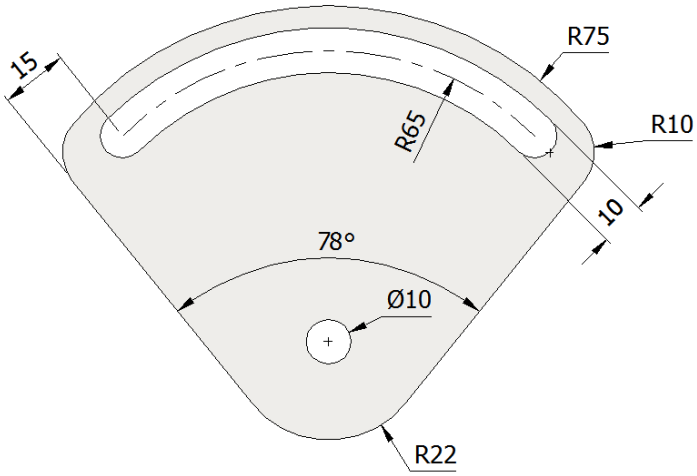
Drawing 18



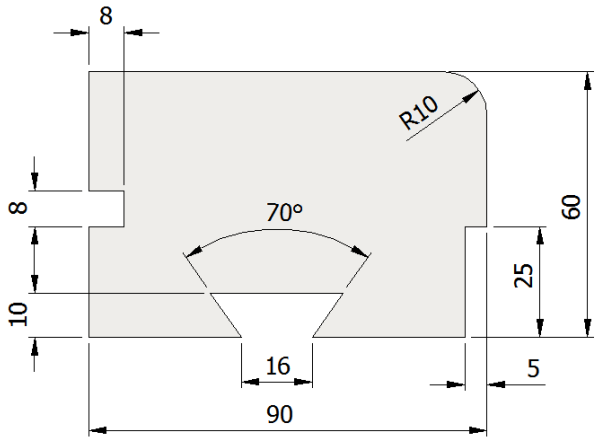
Drawing 19



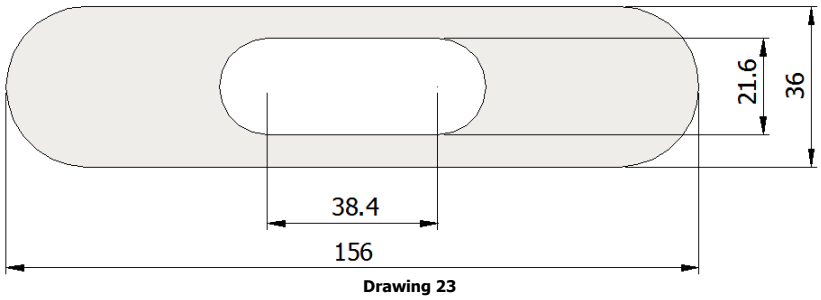
Drawing 20



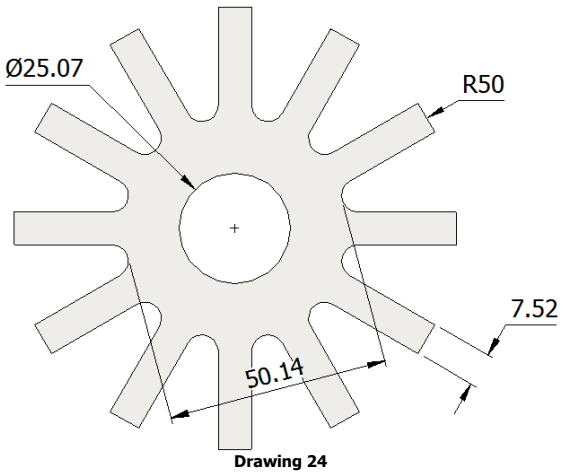
Drawing 21



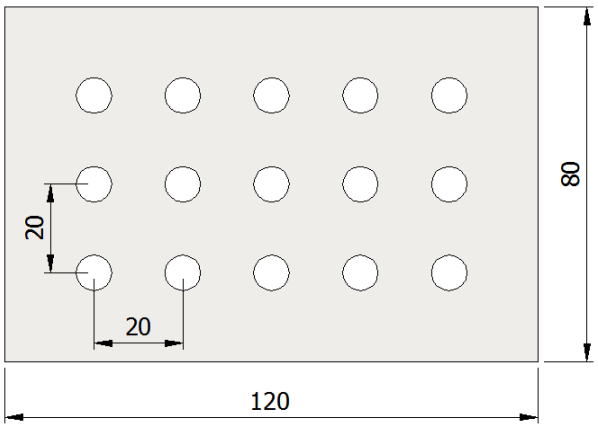
Drawing 22



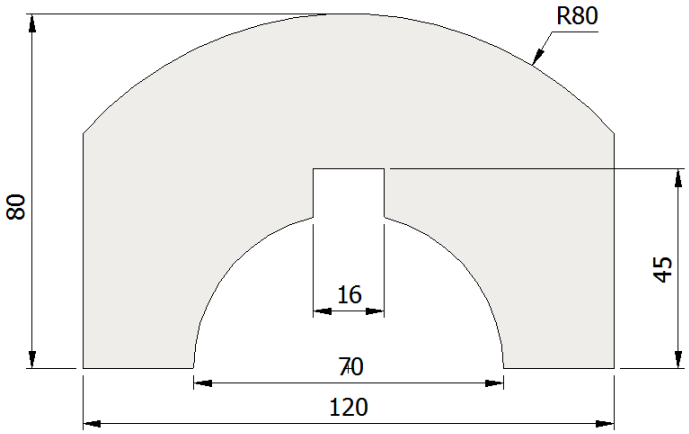
Drawing 23



Drawing 24



Drawing 25



Drawing 26

October 9, 2008