The document is not a complete manual.
It describes only the differences between IGEMS_R7 and IGEMS_R8

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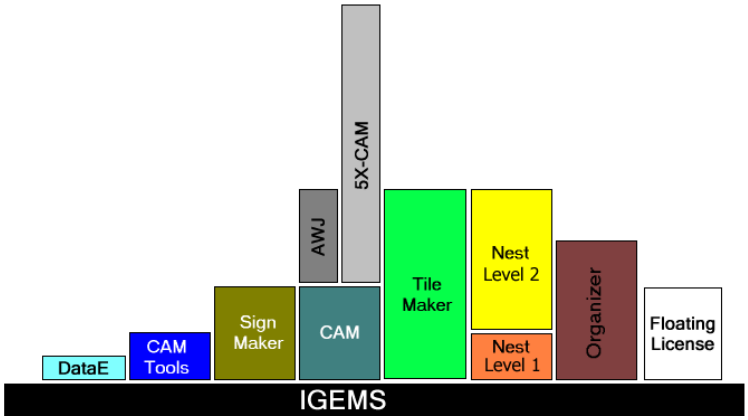
Chapter 1.
General changes

In this version we have changed the structure of the plug-ins. Our idea is that the customers should not have to buy more functionality than they have use for.



CAM-option

The old 2D-CAM-option has been divided in two separate modules: CAM-option and AWJ-option. All customers that already have the 2D-CAM-option will automatically have both when updating. Customers that do not need Advance WaterJet-option (plasma cutting for example) can now buy a cheaper CAD/CAM-package.

AWJ (Advance WaterJet)-option

This is now an extra plug-in to the CAM-option. This includes the advanced feed ramping, material database and much more.

Tile Maker-option

In this version we have added a new plug in called "Tile Maker". This is an option suitable for customers who works with tiles. You can read more about this in Chapter 3.

Nesting Level 1 Option

This is a low cost nesting solution including strategies for Single nest, Rectangle nest and Quick nest.

Nesting Level 2 Option

This level includes the Automatic True Shape nesting option. Customers that already have the nesting option in previous version of IGEMS will in future have both options.

Chapter 2.
General changes and CAD-commands

Trim

The syntax is changed for this command. After you have started the command you can immediately pick object that should be trimmed or removed. In previous versions, it was always required to press enter or select the objects that should be affected before the trim command could be used.

Extend

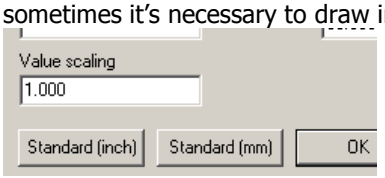
The syntax is changed for this command. After you have started the command you can immediately pick the objects that should be extended.
In previous versions, it was always required to press enter or select the objects that should be affected by the command.

Edit text

The edit text function now also works on dimension text. Note! The dimension in IGEMS R8 is not associative. It will not affect anything else but the text.

Dimensioning on scaled drawings

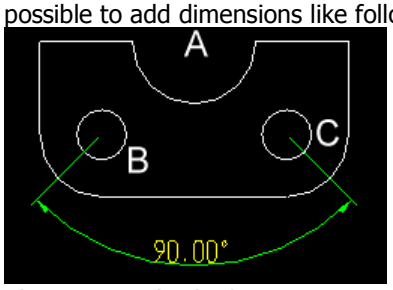
The recommendation is to always draw all objects in correct dimension. However sometimes it's necessary to draw in another scale.



By using the new variable "Value scaling" you can force IGEMS to suggest a scaled value of the dimension text. This setting can be found in the Dimension settings in the Format menu.

Angular dimension

The angle dimension command has been improved. Instead of only pick on existing object you can now pick on tree points to specify the angle. This makes it possible to add dimensions like following image:



The command asks for:
Specify first segment [three points] (press enter)
Select points: Center of A, Center of B and Center of C.

Fillet

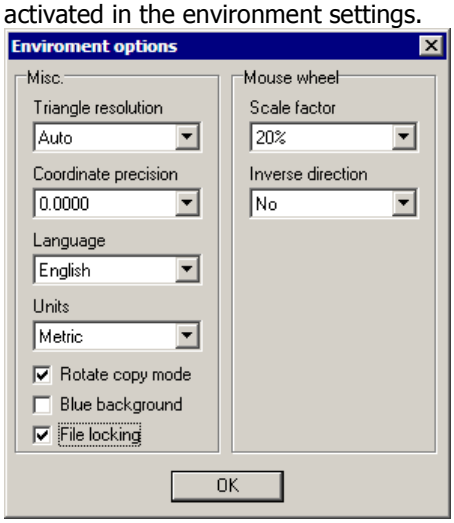
You can now create fillets between circles.

Import drawing

In IGEMS R6 we introduced the possibility to also insert the name of the drawing as an extra insert. This was possible by holding down the CTRL or SHIFT when the part was placed. With IGEMS R8, CTRL and SHIFT will have different functionality.
SHIFT: This works as in previous version.
CTRL: This option asks for the height of the text to insert.

File locking

IGEMS are now using Windows file locking system. The file locking feature can be activated in the environment settings.



The file locking can only be activated on IGEMS drawing files (ACD-files)

Automatic base point

The commands, Move, Copy, Scale and Rotate has now an automatic base point selection. It works as follows:
Select the objects.
Specify base point [Auto]
You have now two choices. You can pick a point as base point (as previous versions) or you can accept the center option by pressing space or enter. The Auto option takes the center of the bounding box of the objects.

AutoCAD 2008 DWG-Support

IGEMS R8 is now supporting the AutoCAD 2008 DWG format. This makes it possible to import the latest version of AutoCAD files.

Shape library

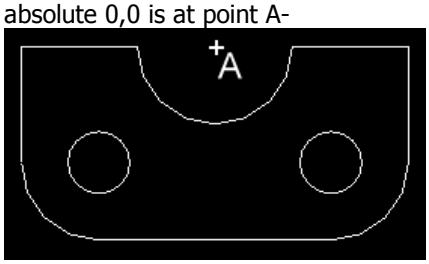
If you are frequently inserting the same shape of geometry it is now possible to store the shape in the Shape library.

Add shapes to the library

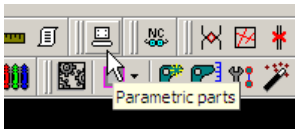
This is the workflow to add new shapes:

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

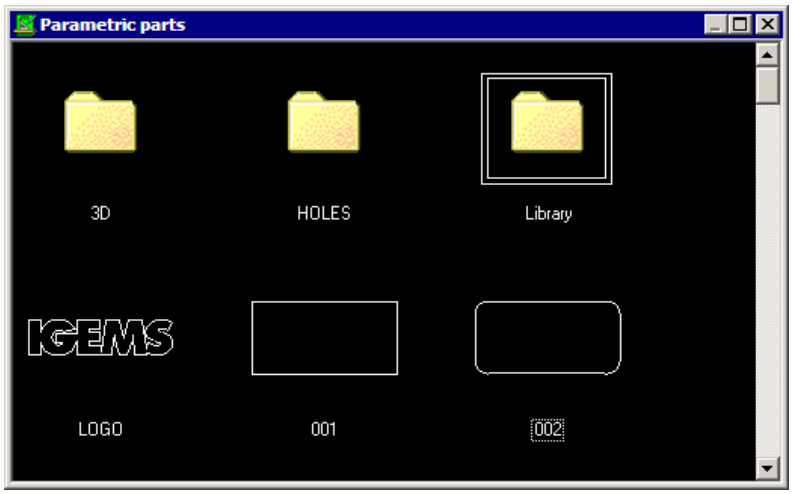
absolute 0,0 is at point A-



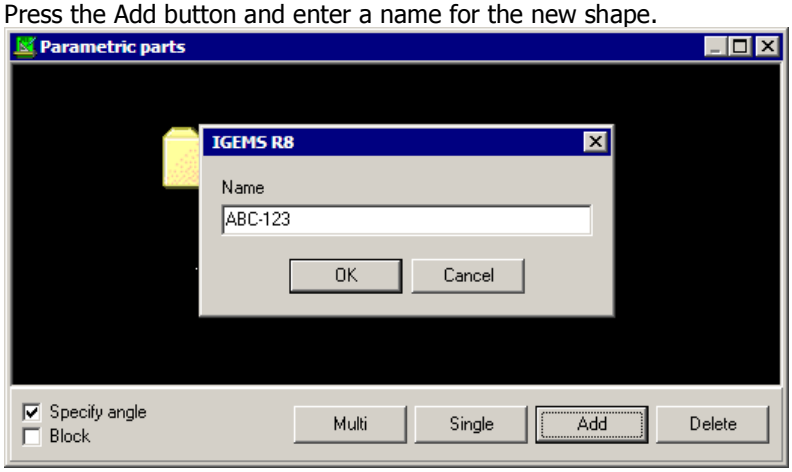
Step 2:
Start the Parametric part command



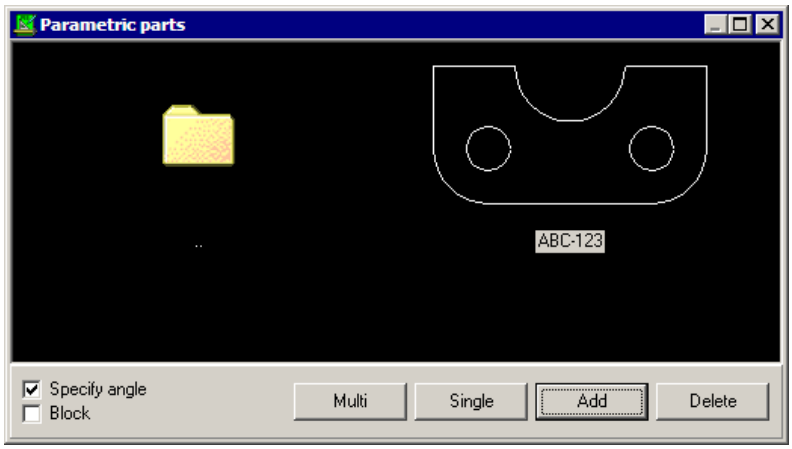
Press the Parametric part button



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

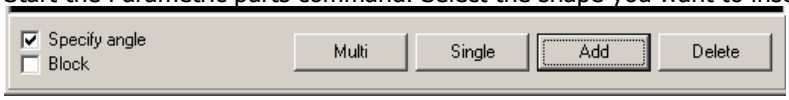


Finally select the geometry.
Select objects (Select the geometries)
The Shape is now stored in the library and can be reused at any time.



Insert shapes from the library

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

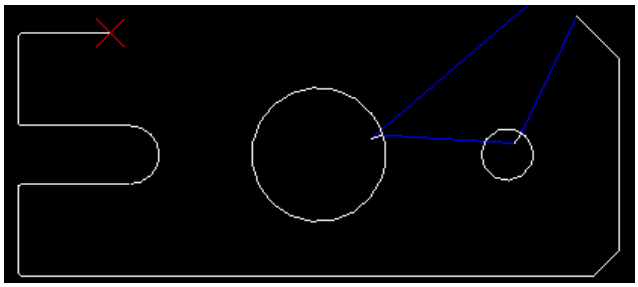


If you want to insert only one shape, then use the Singe button. If you want to insert multiple shapes then use the Multi button. From this window you can also control if you want to modify the angle of the inserted shape or if the shape should be represented as a block.

Chapter 3.
Changes in the Data exchange module

Only small changes is made the Data exchange module.

NC-Reader



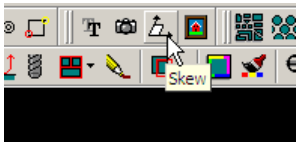
The NC-reader is now more intelligent and has now a cursor that shows the last position of the movement.

Chapter 4.
Changes in the SignMaker module

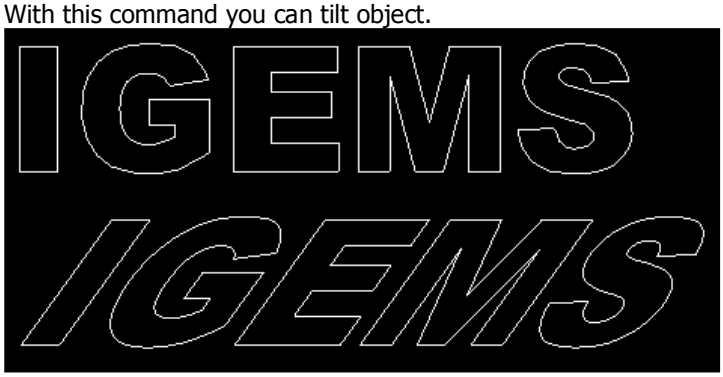
Following changes is made in the SignMaker module

Skew command

The new Skew command can be started by clicking on following button.



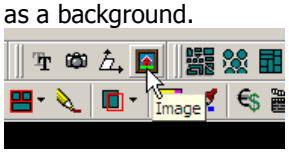
With this command you can tilt object.



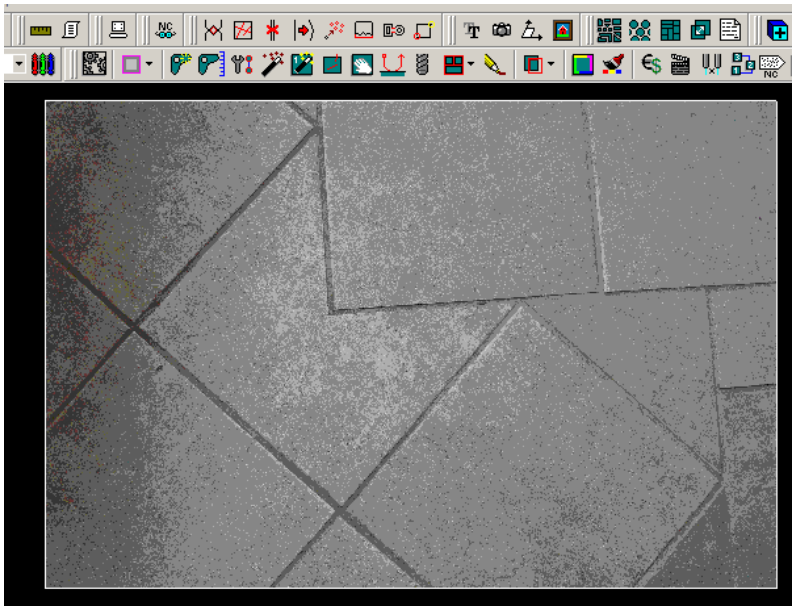
In the example above the Skew command is applied on objects from Image tracer.

Image

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



The command starts by clicking on the Image button.



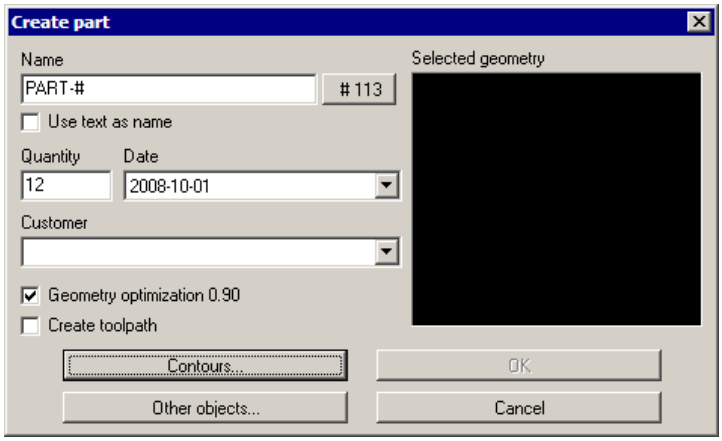
The command asks for a file and the Insertion point.

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 5.
Changes in the CAM module

Following changes are done in the CAM module. The functionality for the Advanced WaterJet has been moved to the AWJ-option. All customers that update from an older version of IGEMS will automatically get the AWJ-module.

Create object



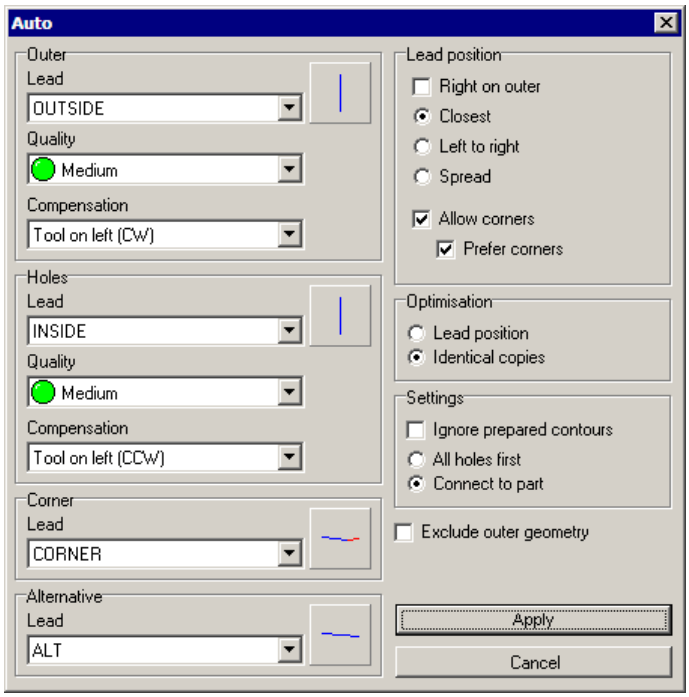
The function to create a part with several external geometries has been removed. (it now has to be done with the Join parts command).
Small internal geometries that is to small to be machined (with actual Max tool diameter) are now automatically converted to Other objects.

Text object

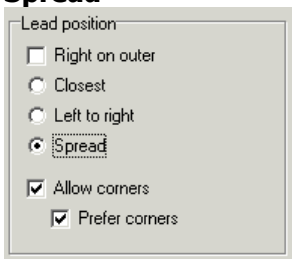
In previous version the color of text objects was always the same even if it was used or not. This is now changed so that the objects change color when toolpath is added.

Auto

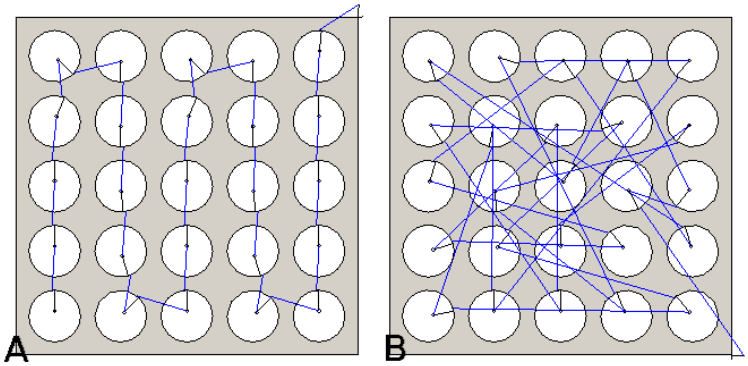
The design of the dialog box is easier and more intuitive to understand. Some of the settings have been removed since more intelligence is built in to the command.



Spread

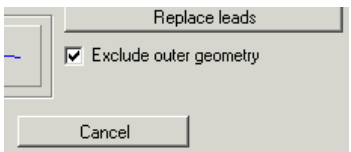


The new spread option is mainly developed for plasma cutting. This option will use a cutting order between holes in a part in such an order that it will distribute the heat from the cutting more evenly.

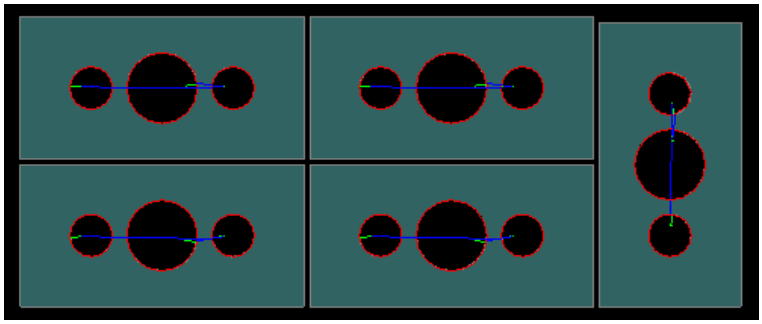


Example A is done with the "Left to right" and example B with the Spread option.

Exclude geometry



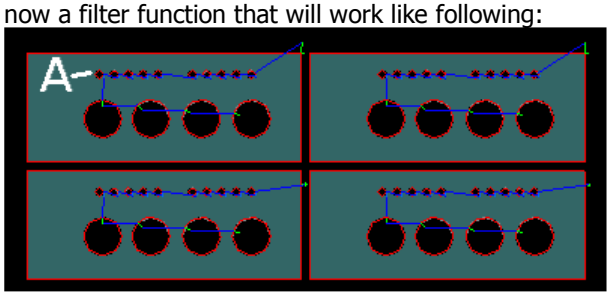
This option is developed to be used together with the common cut line commands.



The reason to use the Auto for the internal geometry is that this command uses the standard tool radius compensation. Common cutline will then detect and ignore the already used contours.

Lead

The Replace option on the lead command has been improved. The command has now a filter function that will work like following:



Imagine a large nesting where you want to replace all the leads on small holes. The replace will ask:

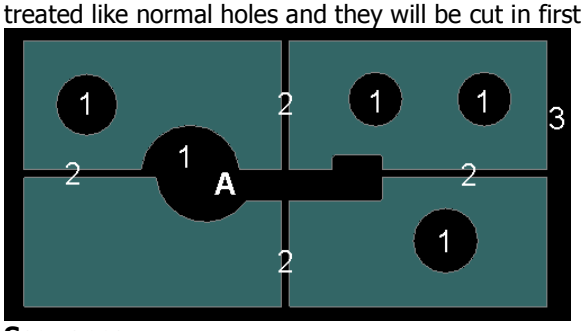
Select filter object [None]: *Pick on the lead (A)*
Select leads to replace: *Select all parts*
All selected leads with the same name as selected (on A) will now be replaced.

Common cut line

The common cut line now has an option for cutting the islands between parts in a later sequence.

Islands before separation

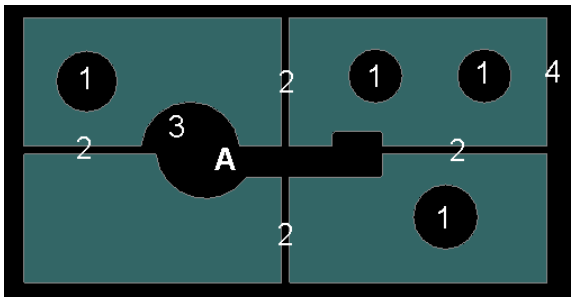
If you activate the checkbox "Islands before separation" then islands (A) will be treated like normal holes and they will be cut in first priority.



Sequence
1. Cut all holes and the islands
2. Cut the common cut lines
3. Cut the outside

This was the method used by IGEMS R6 and R7.

If you are not using the "Islands before separation" then the cutting order will be like follows



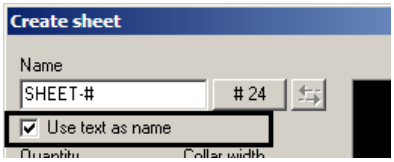
Sequence

1. Cut all holes
2. Cut the common cut lines
3. Cut the islands
4. Cut the outside

The idea behind this method is that the parts are separated from each other in a later sequence that they are still holding together by the island. This is a big advantage on huge common cut line layouts.

Create sheet

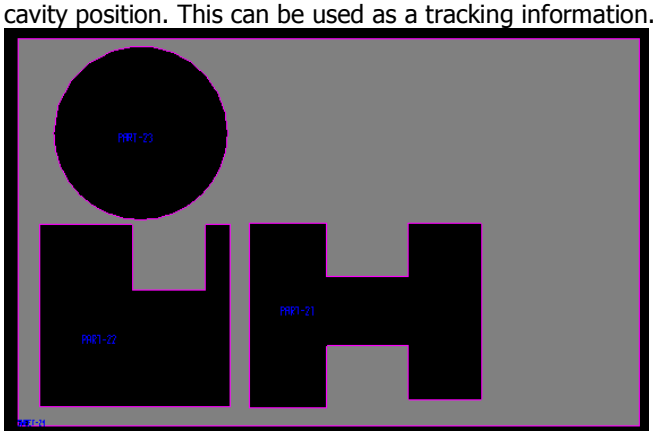
The Create sheet command is now using the identical functionality for naming sheets as the Create part command do for naming parts.



If you have a text inside the sheet then the text is used as name of the sheet. This feature has been implemented to make it easier to import sheet geometries from other system.

Lock sheet

When locking a sheet with parts IGEMS will now write the name of the parts in the cavity position. This can be used as a tracking information.



After selecting the parts the command asks for text height:
Specify text height [2.000]
If you specify the value 0.0 then no text is placed.

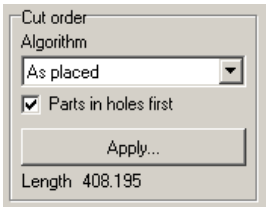
Inventory

When using the Draw option in the Inventory command, rectangles in different sizes are placed on the sheet. In IGEMS R8 these rectangles are placed in layer INVENTORY. This makes it easier to remove them afterwards.

Sheet prepare

Following changes has been made in the Sheet prepare command.

New cutting order algorithm

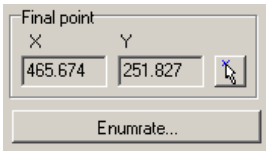


The new option "As placed" cuts the parts in the same order as they are placed by the nesting module. Sometimes this option can give a better result on rectangular parts.

Multiple final points

In previous versions if IGEMS it was possible to specify a final point. This point was used to make an extra movement after the cutting. In IGEMS R8 it is possible to specify a path of points. This feature has been an important request from customers with old machines that do not have any Z-axis. With this function they can now make manual movements without risking collision with parts or other equipment.

This feature requires a postprocessor developed for IGEMS R8.



Move parts in cutting order list

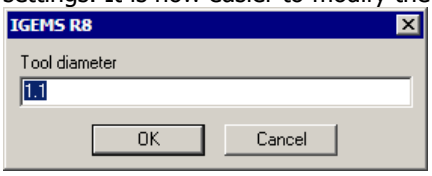
You can now more easy move parts to the beginning or the end of the cut order list. This can be done by clicking on one of these buttons.

Post processing

Following changes has been added to the post processing.

Improved inbuilt tool radius compensation

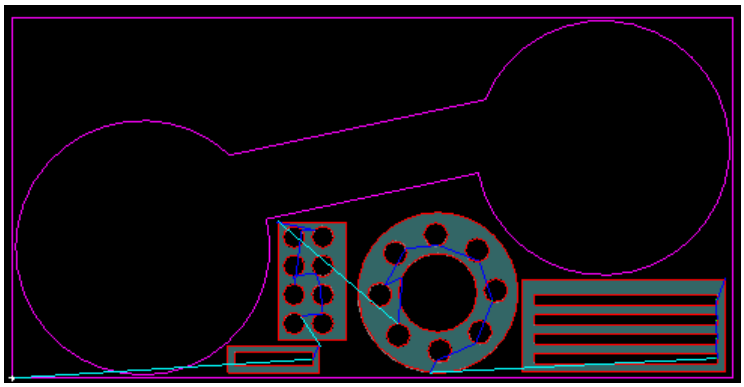
IGEMS has a built in tool radius compensation that can be activated from machine settings. It is now easier to modify the size of this tool radius compensation.



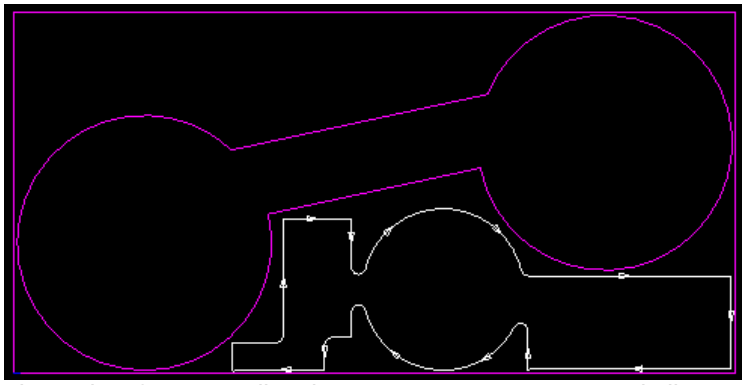
When you start the post processing you will now have a question about the diameter of the jet. The default value is from the machine setup. The value must be less than the "Max tool diameter" specified on the machine setting.

Outline

It is now possible to activate an outline function in the postprocessor. The outline function makes a linear outline movement around all parts. This function is developed to be used on scrap sheets. Here is an example:

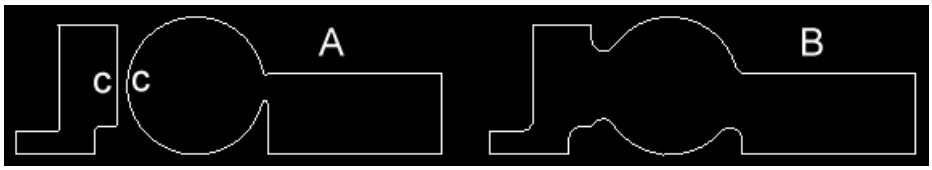


The parts displayed above should be machined out of this rest sheet. To be sure that the sheet geometry is correct and that all parts are located on material, the Outline function is activated.

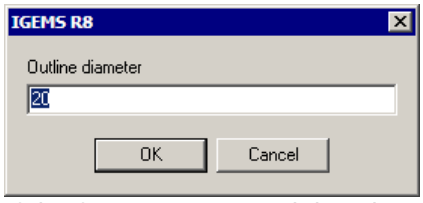


The outline function will make an extra movement around all parts before the cutting starts. The operator can during this movement check if all parts are place correctly. The outline movement will not show any internal movements.

Outline diameter



You can think about the outline diameter as a circle that you try to move between the parts. If the distance between is smaller than the circle then it will not be possible to move the circle between the parts. In example A the outline diameter is set to 10 and on example B is the outline diameter set to 40.



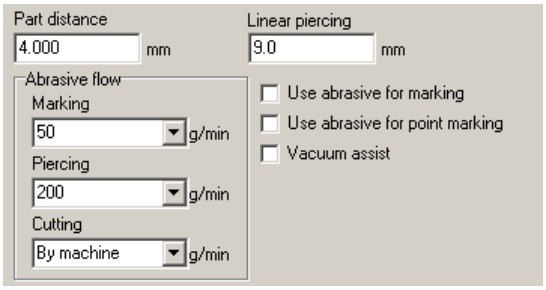
If this function is activated then the postprocessor asks for the Outline diameter. The default value is set to 3 times part distance.

Note! The Outline command needs modification of the postprocessor.

Chapter 6.
Advance Water Jet option

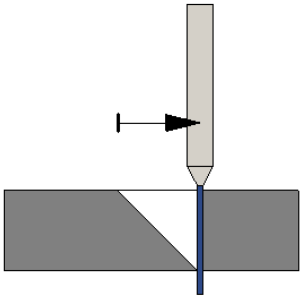
This is a new option in IGEMS. All customers with an older version of IGEMS will have the AWJ-option included if they have the previous 2D-CAM module

Material settings for AWJ-cutting



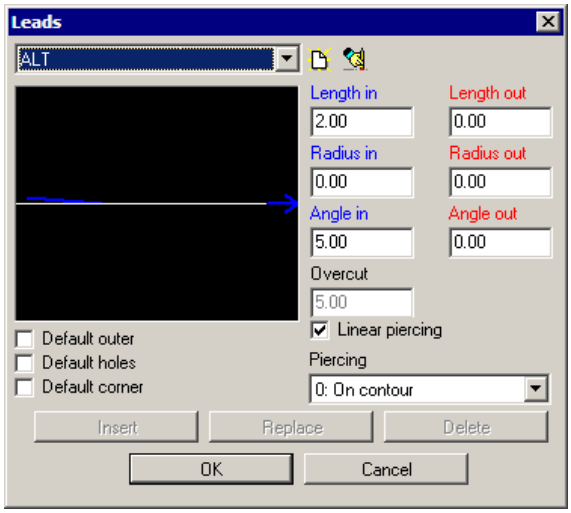
Abrasive flow
In previous version of IGEMS the amount of abrasive has been controlled from the Machine settings. This was a disadvantage since it was impossible to use different abrasive for different operations. In IGEMS R8 you can have different amount of abrasive for Marking, Piercing and Cutting. If you want to use the setting specified from the machine setting then set the value to "By machine".
Note! This feature needs modification of the postprocessor.

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).

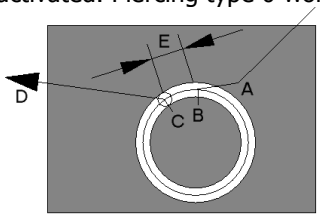


Linear piercing is the most time saving method to pierce through the material. To use this method in previous versions of IGEMS it was necessary to change the overcut distance (for piercing type 0) and the lead-in length (for piercing type 1). This had to be done every time you changed material or thickness to get an optimal result. Whit IGEMS R8, this value is stored in the material database and is automatically taken care of.

Example for piercing type 0 (Start on geometry)
In this example the linear piercing value is set to 5mm in the material database.



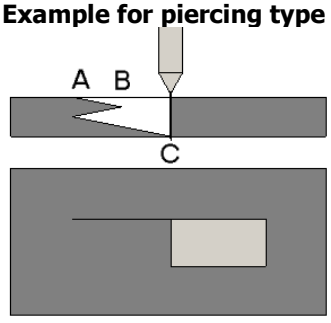
On the "On contour" leads, the piercing type 0 is activated. The overcut value will always be the same as the Linear piercing if the Linear piercing checkbox is activated. Piercing type 0 works like follows:



1. Rapid transport to point A
2. Tool compensation is activating and the jet moves to point B
3. The cutting process starts with no delay.
4. Circular movement 360 degree to point B again.
5. The circular movement continues to point C (Linear piercing distance from B) This is the linear piercing distance (overcut).
6. The cutting process turns off.
7. Rapid movement to next geometry.

Linear piercing is a static value. The value is set when used by the Single, or Auto commands. If you change the length value in the material setup, you will be required to redo the lead in with Auto or Single command.
This feature does not require any modification of the postprocessor.

Example for piercing type 1 (Direct start):



In this example the linear piercing is 20mm but the lead length is only set to 10mm.
When the Linear piercing length is longer than the Lead-in then the jet will go back and forward as many time as needed to allow piercing through the material.

1. Rapid transport to point A
2. The cutting process start, no delay.
3. Piercing to point B
4. Piercing back to point A.
5. Activate tool compensation

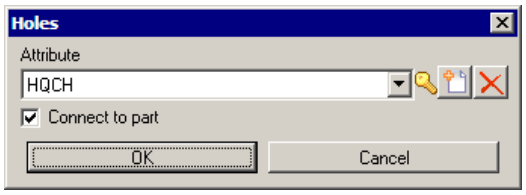
6. Piercing (cutting) to point C

The Linear piercing for this piercing type is used by the postprocessor. If you change the value in the material settings you only need to re-post to have the new value.

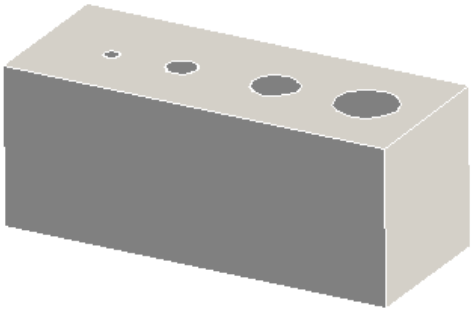
Note! This piercing needs modification of the postprocessor.

HQCH High Quality Circular Hole

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



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.



The HQCH feature takes longer time than the standard cutting, but it gives a much better result. Example to cut a hole with diameter of 2.5mm in 20mm thick Stainless steel is possible with a good result.

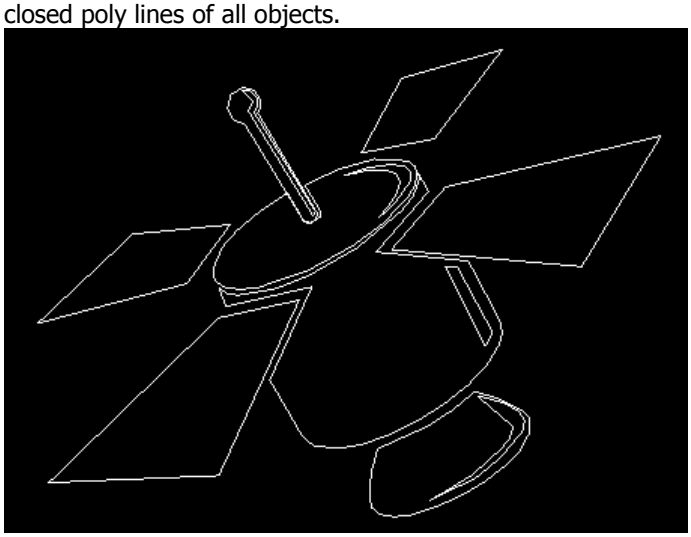
NOTE!
This feature needs modification of the postprocessor

Chapter 7.
Tile Maker option

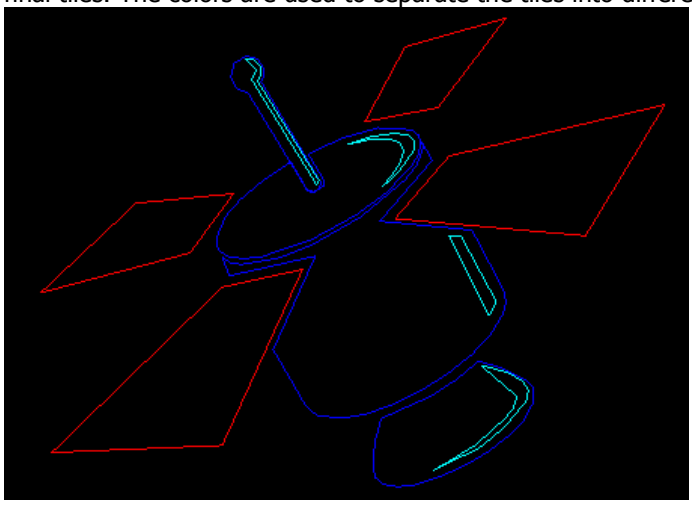
This is a new option in IGEMS that makes it possible to easier produce tiles than before. Following describes the workflow:

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.

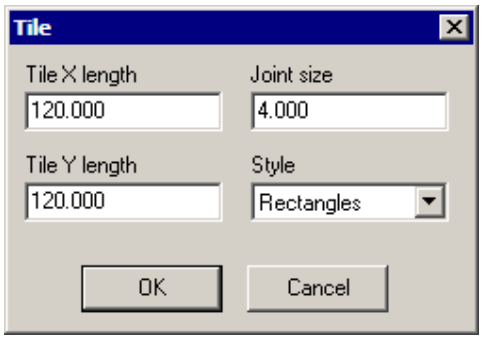


Next step is to set the colors. The color does not have to be the same as on the final tiles. The colors are used to separate the tiles into different bundles.

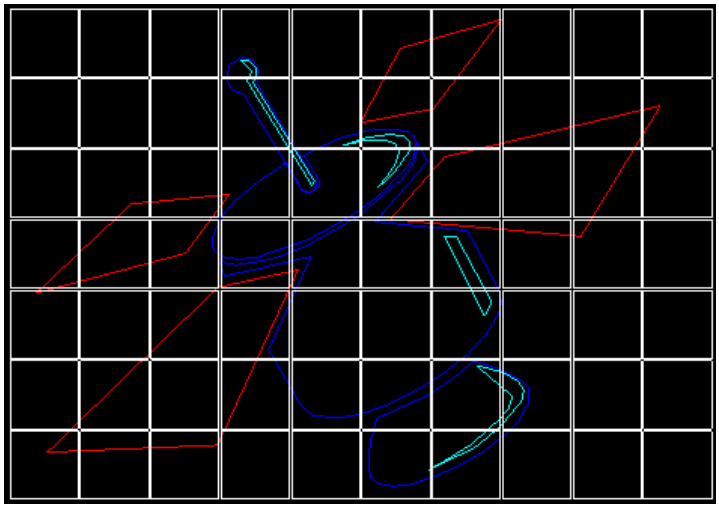


Insert the Tiles by using the Tile command.

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



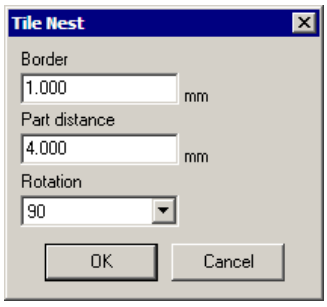
Specify the Tile data.
The Tile X length and Tile Y length is the size of the tiles.
The Joint size is the distance between each tile.



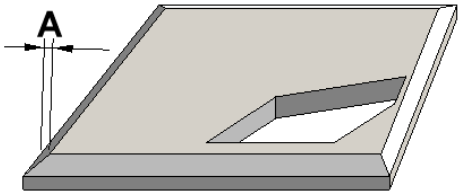
Step 2: Tile nest

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

The Tile nest has following settings.



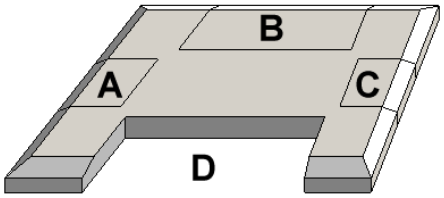
Border



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.

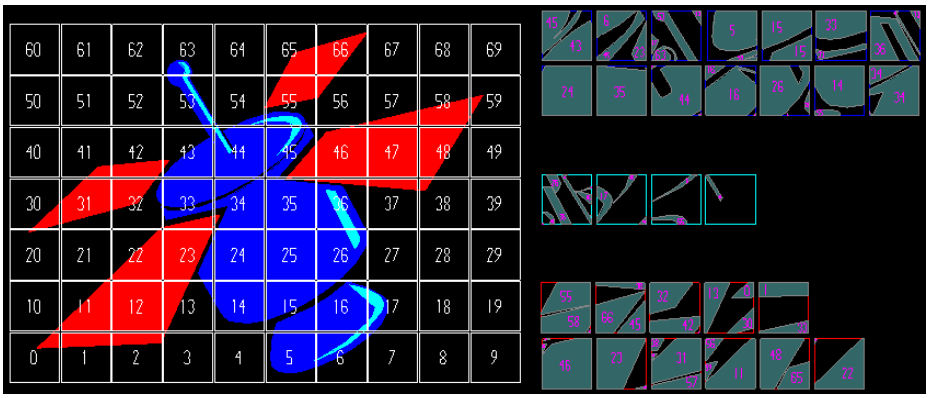
Part distance
This setting controls how close parts can be nested on the same tile.

Rotation



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)

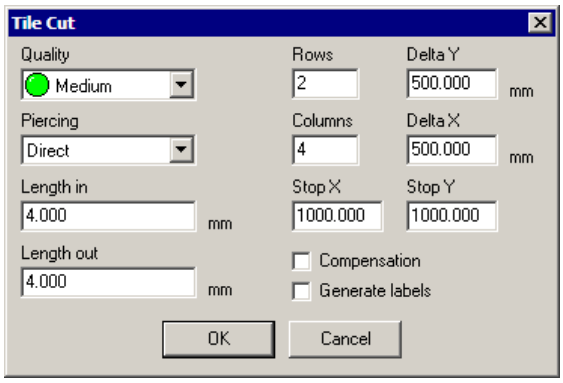


The Tile nest will only nest parts that need to be cut.

Step 3: Tile cut

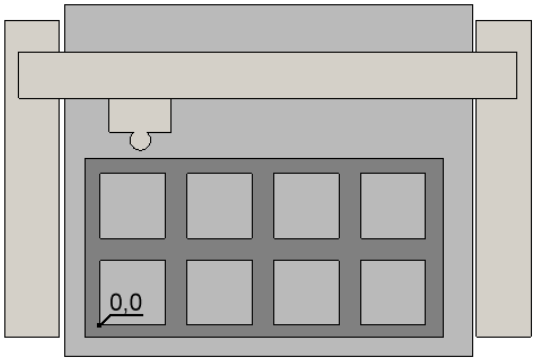
This command automatically adds the tool path and the cutting order between the parts.
The Tile cut command starts by clicking on following button:

The command has following options



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

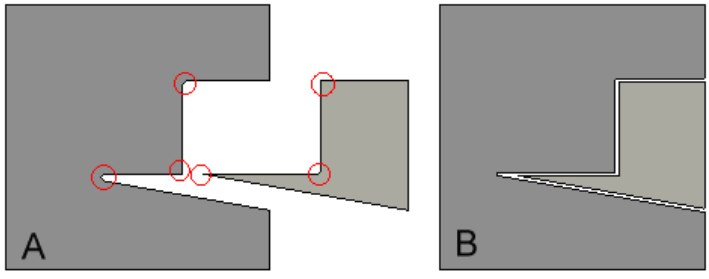
Fixture parameters
This command needs a fixture in the machine.



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.

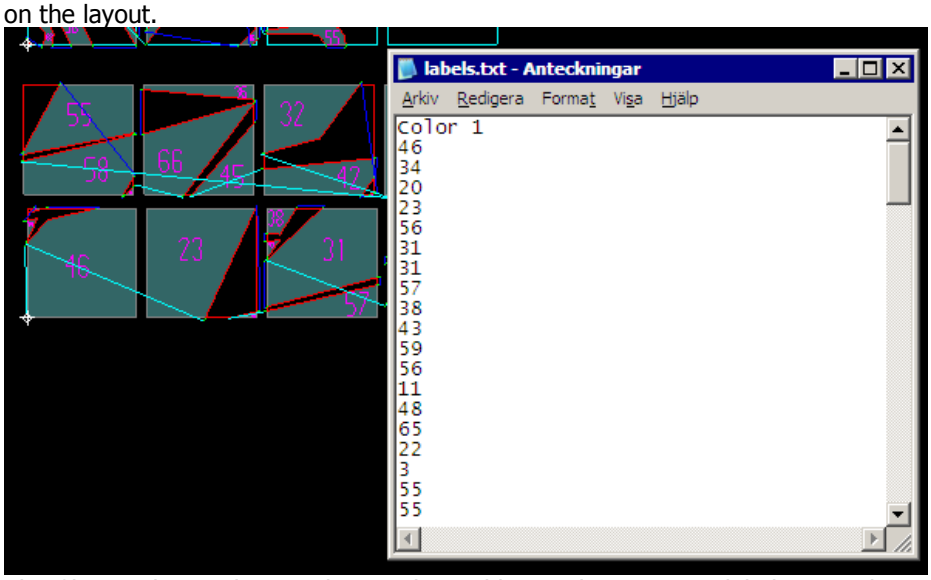
Compensation



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.

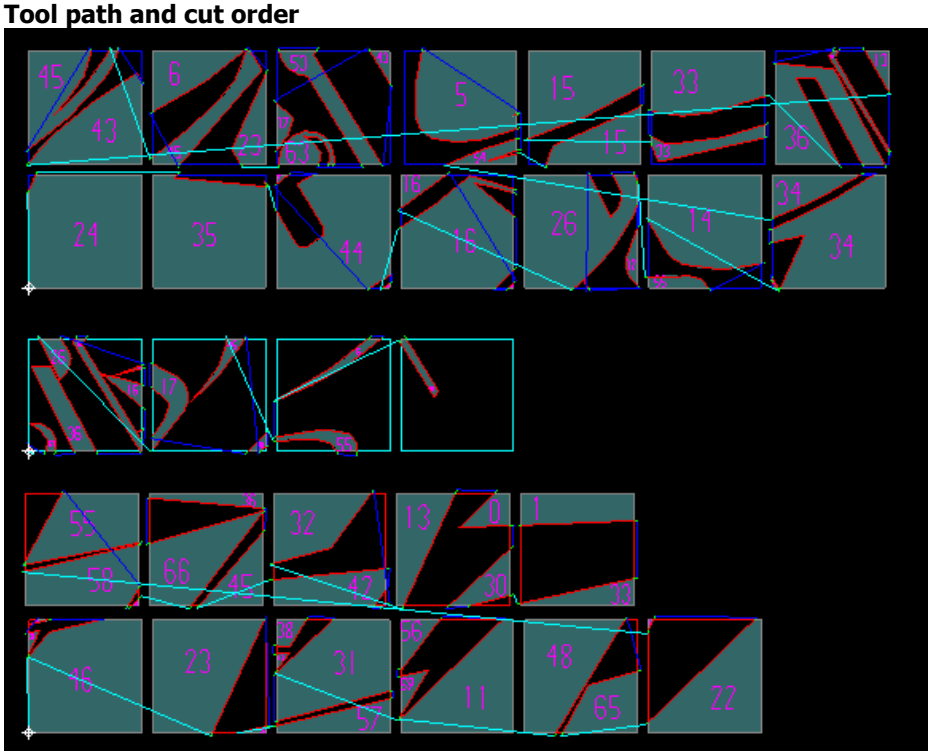
Labels

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



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



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.

Step 4: Create CNC-file

You can use the normal post processing function to generate the CNC-file. The Tile maker option does not require any modification of existing postprocessors.

Chapter 8.
Changes in the nesting modules

One big change is that the nesting routines now are split in two levels.

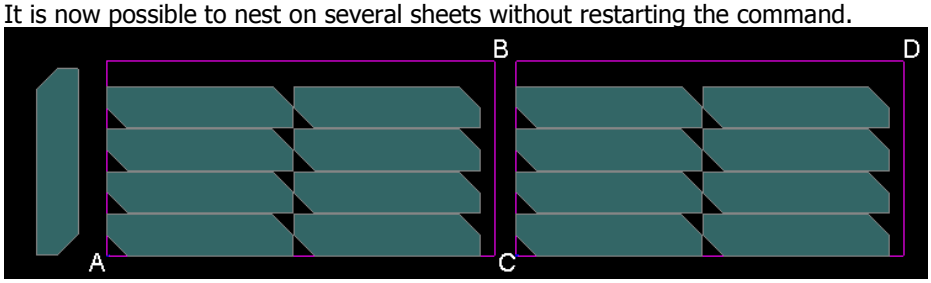
Nest level 1

The nesting commands, Single, Rectangle and Quick nest is now member of the level one option.

Rectangle nest

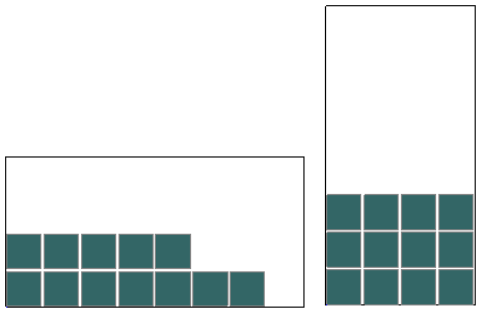
The algorithm for the rectangle nest has been improved.

Multiple sheets
It is now possible to nest on several sheets without restarting the command.

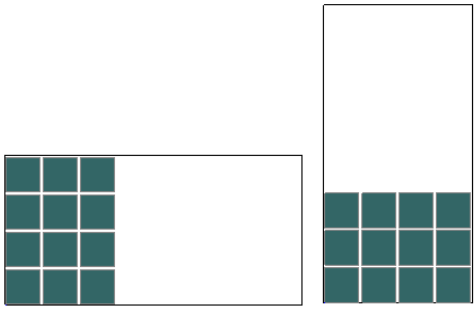


Selecting the part
Select the first and second corner of the first sheet (A and B)
Select the first and second corner of next sheet (C and D) and so on.

Nesting direction
Now also has a more intelligent nesting direction.



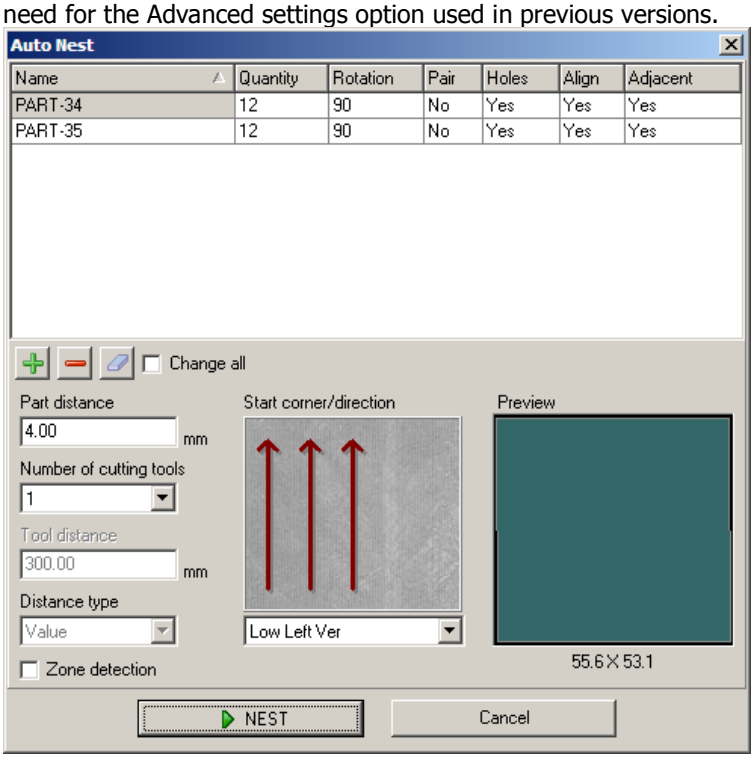
In previous versions the nesting directions was always at the X-axis.



In IGEMS R8 the nesting direction is always at the shortest edge of the sheet.

Nest level 2

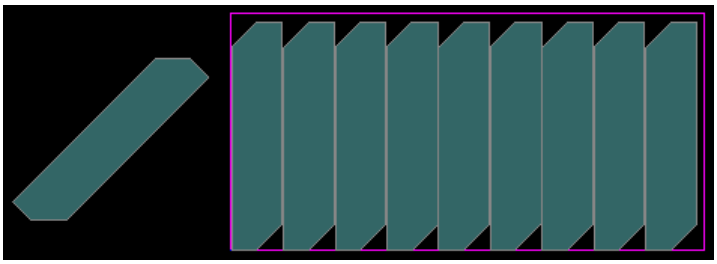
This option has the command for Auto nesting. The layout for this command is now changed. The nesting algorithm is improved and the result is that there is no need for the Advanced settings option used in previous versions.



The former advanced settings are now adjusted automatically by the program. It will result in a more user friendly layout.

Align

The Align setting is now activated automatically for all selected parts.



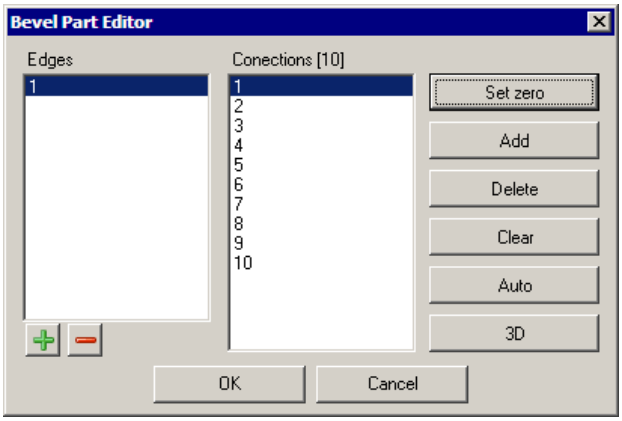
The align option automatically rotates the part so the longest direction becomes parallel to the X-axis. In the example above the parts are nested with 90 degree rotation and the align option activated.

Chapter 9.
Bevel cut

Following changes has been done in the Bevel cut command.

Define bevel cut

In previous version of IGEMS there was a separate window to add contours. This has now been moved in to the Bevel Part Editor. However, when you start the command you must define the external geometry of the part.
The command asks for following:
Select top contour:
Select bottom contour:



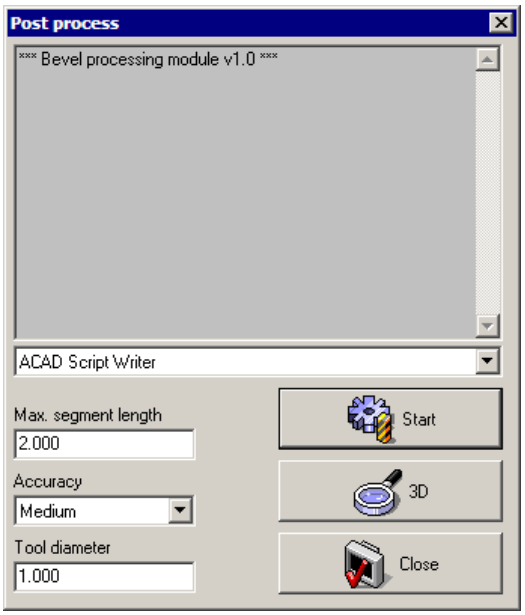
Add edges

If you have some holes in the parts, then this can be defined by using the Plus button. The command asks for:
Select top contour:
Select bottom contour:
Be sure that all selected contours are closed and have no cross over.

Remove edges

If you need to change the geometry of the hole, then you can remove the edges, modify them by normal CAD command and add them again.

Post processing

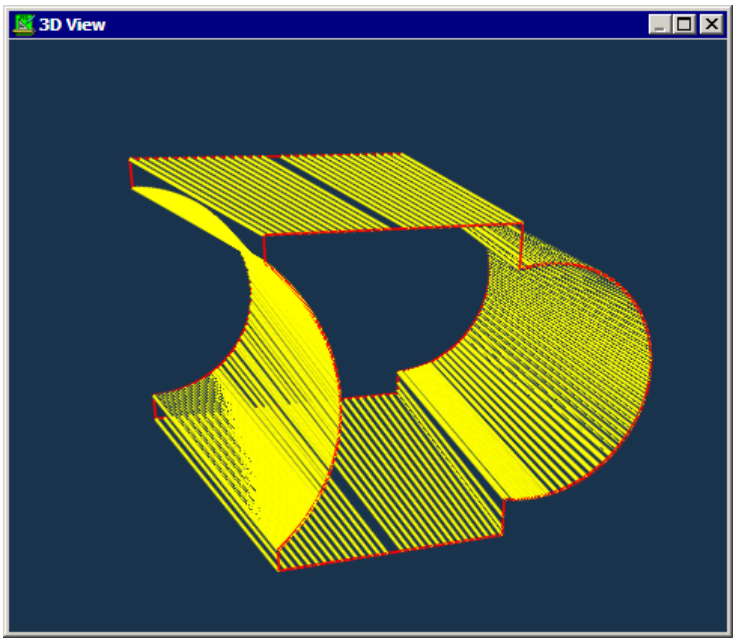


The post process command has been redesigned and following features has been added.

Tool diameter

In previous versions of IGEMS this command had no support for tool radius compensation. To make a correct part with correct measurement, then you had to make the external geometry smaller and internal geometry larger. IGEMS R8 has now a built in 5-axis tool radius compensation. This means that you no longer need to pre modify the geometry.

3D View



The new 3D-View function shows the result of the tool radius compensation as cutting vectors. It is a good idea to always check the vectors before creating the CNC-file and produce in the cutting machine.