cncKad
Punching Manual

PunchMan.85.006

Limited Warranty

Metalix CAD/CAM Warrants the software will perform substantially in accordance with the accompanying printed materials. ***Metalix CAD/CAM*** does not warrant that the functions contained in the software will meet your requirements or that the operation of the software will be uninterrupted or error-free.

Limitation of Liabilities

In no event will ***Metalix CAD/CAM*** or its suppliers be liable for direct, indirect, special, incidental, contingent, consequential or similar damages of any kind resulting from any defect in the software or documentation, including without limitation, any lost profits or savings.

Metalix CAD/CAM LTD
Koranit
D.N. Misgav 20181
ISRAEL

Tel: (+972) 4-999 8436
Fax: (+972) 4-999 8519
E-mail: info@metalix.net

Table of Contents

Introduction

cncKad is an NC code generator for CNC Punch and Laser machines. In this manual we will be covering the use of ***cncKad*** by punching machines

The cncKad modules include a complete set of tools to aid you in creating the optimal program for your machine. Some of the functions included are listed below:

- CONTOUR CREATION: The contours of the parts that you want to cut can be created either graphically by the user, with the aid of ***cncKad***, or by importing a file from another drafting package (See the Punch Tutorial for a full description of this module).
- GRAPHIC PROGRAM DEFINITION: Graphic interaction with the user that enables defining the desired CAM process, including the setting of all program options.
- PROGRAM GENERATION (G-Codes Post Processor): Automatic generation of CNC programs. ***cncKad*** supports a very wide range of controllers and CNC languages.
- FILE EXCHANGE: ***cncKad*** supports import of files in DXF, Iges or Cadl format from other drafting packages.
- PROGRAM ARCHIVING: Sorted directories of CAM processed drawings, drafts and CNC programs, for backup and retrieval.

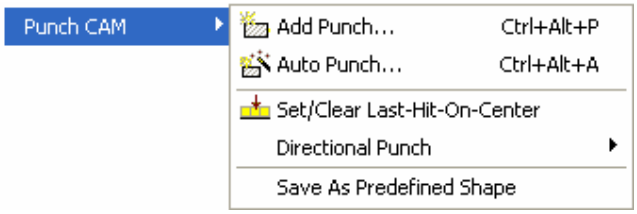
This current version of ***cncKad*** is intended for use on Windows2000, Windows XP and Windows Vista® workstations.

cncKad includes five modules:

- Drafting
- CAM placement (processing)
- Nesting
- Simulation
- DNC (communication to machine)

1 Punch Manual

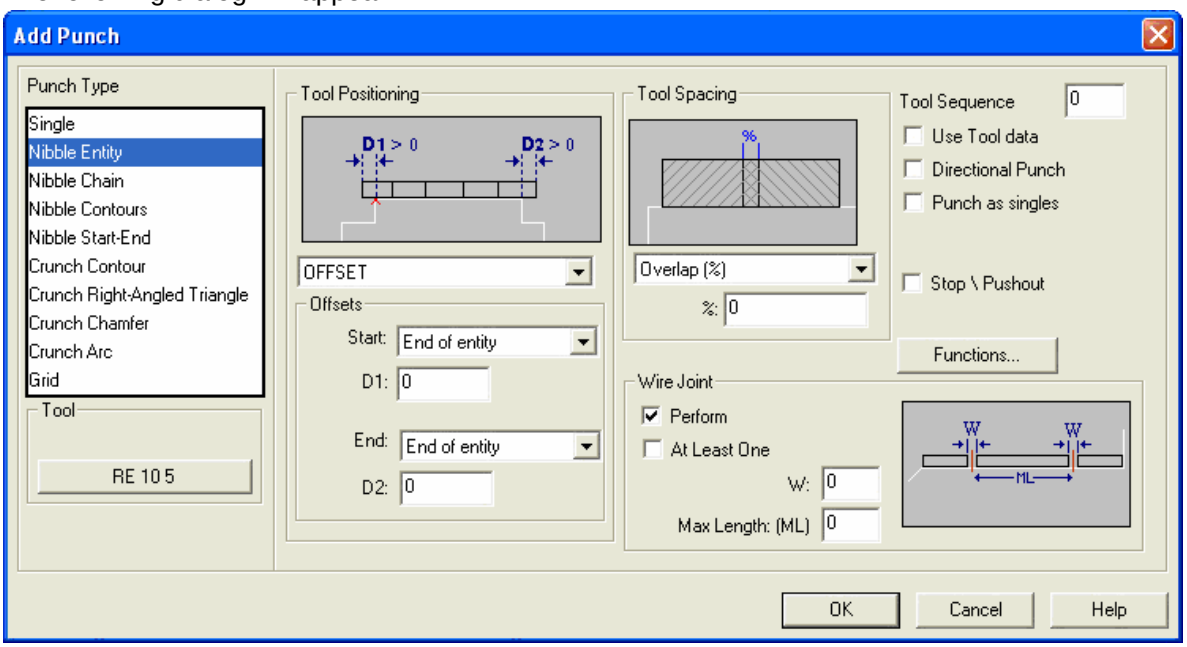
From this menu you can place the actual punches on your part. There are two options for placing punches on your draft, adding a Single Punch, or using Auto Punch, and several options for defining your **CAM**ing:



1.1 Add Punch

You can also click the **Add Punch** button from the Common Toolbar –

The following dialog will appear:

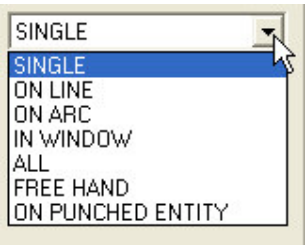


1.1.1 Single Punch

Adding single punches to your part will place one instance of a selected tool at a set position. A single punch can be a hole in the sheet or a punch on a contour.

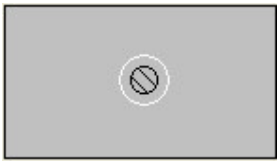
There are several ways to add single punches to your geometry, the one that you choose depends on the cut you need.

Single punches are selected by clicking the **Single** option in the **punch type** area. The precise type of punch can be selected from the **Single Type** dropdown list:

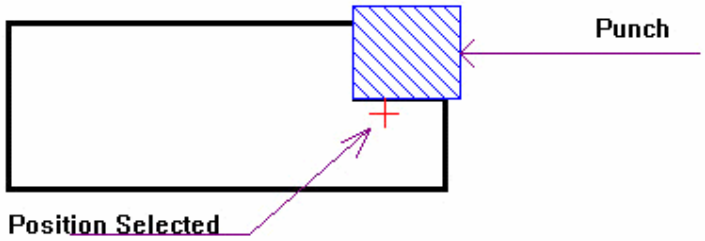


All single punches visible on the drawing are marked with hatched lines.

1.1.1.1 Single



With this option the user will be able to add a single punch to a specified entity. After selecting the single option, click the entity on the drawing that you wish to punch. It will be punched using the currently selected tool.

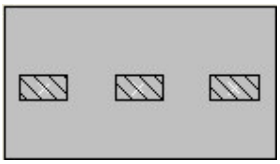


If you are punching a circle place the cursor near the contour of the circle and click the mouse button. *Don't try to fit the punch outline into the circle*. The tool selected does not have to fit exactly to the hole to be punched.

When a single punch is placed on the part, it is defined as the tool shape filled with hatch lines.

If you selected a line entity for punching, the placement of the punch on the line will depend on the option selected from the **Tool Positioning** dropdown list. For more information on this option, see the **Tool Positioning** section of this chapter.

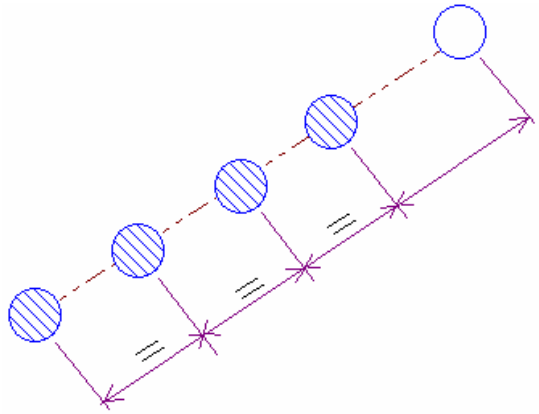
1.1.1.2 On Line



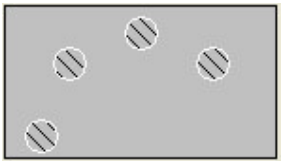
This option will punch all the entities that lie on the same line at equal distances.

The user is required to locate two entities that lie on the line. ***cncKad*** will take the angle and the distance between the two points and will search for more holes which were placed at the same angle and distance. Selected holes will be punched with the current tool. If the **Auto Tool** option is checked, the ***cncKad*** will select the appropriate tool from the Tool library.

Depending on the type of controller that your machine has, the NC code will differ for this command. For instance with ***Fanuc*** controllers, the created code will be G28.



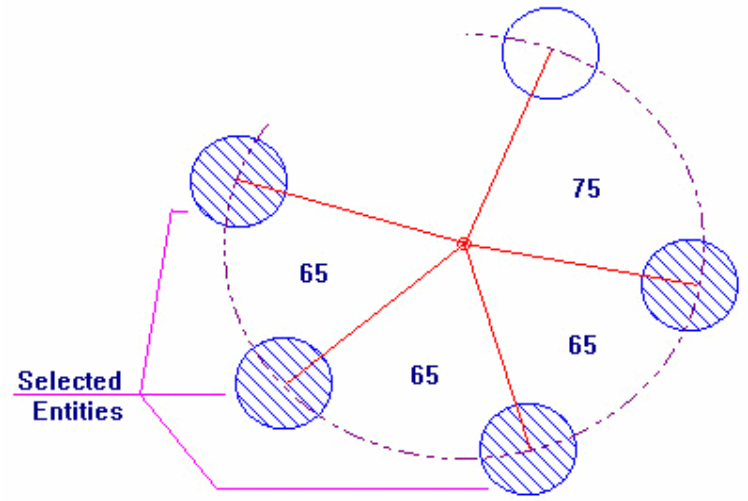
1.1.1.3 On Arc



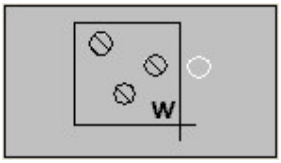
This option will punch all the entities that lie on the same radius, with the same angle between them.

The user is required to locate three points on the circumference of the arc, the ***cncKad*** will then check if there are any more holes on the same radius and the same angle between them. All selected holes will be punched with the current tool. If the **Auto Tool** option is checked, ***cncKad*** will select the appropriate tool from the tool library.

Depending on the type of controller that your machine has, The NC code will differ for this command. For instance with ***Fanuc*** controllers, the code issued will be G29.



1.1.1.4 In Window

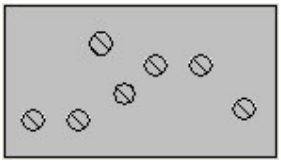


This command will punch the selected closed contours that fall within a user defined window, providing they are either Squares, Circles or Ovals. If the **Auto Tool** option

is *not* checked, the ***cncKad*** will only punch contours that fit the currently selected tool. If checked, all the contours that have suitable tools in the tool library (or the Setup File, whichever is chosen) will be punched.

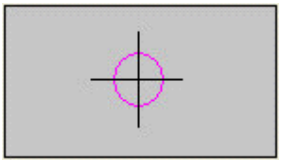
Contours that fall only partially within the boundaries of the window will also be punched (if they conform to the above mentioned shapes).

1.1.1.5 Punch All



This option takes all the closed contours on the part and punches them with the appropriate tool, when the **Auto Tool** option is checked. If it is in the manual mode, only contours that fit the currently selected tool will be punched.

1.1.1.6 Punch Free Hand



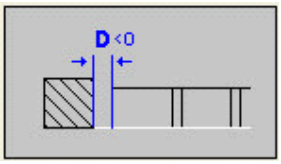
With this option, the user can freely punch with the currently selected tool, anywhere on the part.

When this option is selected, an Angle field is opened: it allows the user to enter the angle for the punch placement.

1.1.1.6.1 Dynamic Angle Change

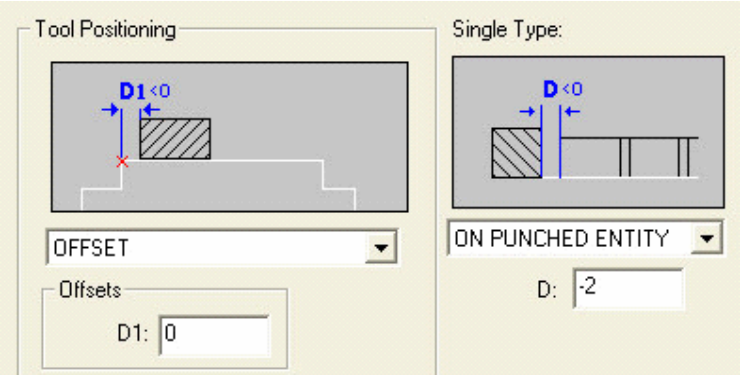
There is a function for single free hand punches, which allows for **dynamical** change – increasing or/and decreasing the angle of the tool we want to punch with. All you have to do is to press on your keyboard the "**+**" button, in order to increase the angle of the tool or "**-**" button, to decrease it.

1.1.1.7 On Punched Entity

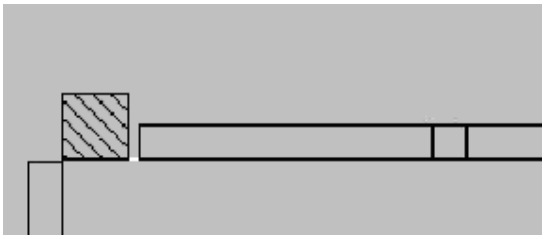


With this option, operate a split on an already **Punched Entity** and place a **Single Punch**. While placing it you can define two values: an Offset from the beginning of the Entity and the Distance separating the new Single Punch from the Existing Punch.

For the following example,

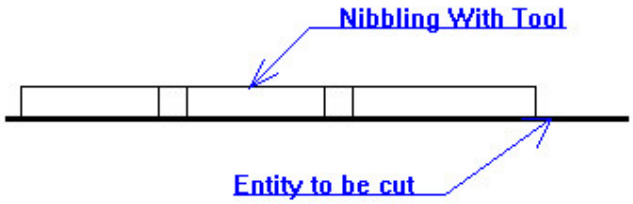


a Single punch on a Punched Entity will look like the following:



1.1.2 Nibble Entity

This command will nibble one selected entity. Select a tool to nibble with, define the positioning of the tool on the entity, define the spacing (if needed) and click the **OK** button to close the punching dialog. Choose a line entity you wish to punch and click the near one of its endpoints. The endpoint that is nearest to your selection point will be defined as the *starting point*. If you have defined offsets, the first offset distance will be from this endpoint.



1.1.3 Nibble Chain

This command will nibble, with a selected tool, a chain of entities. For this command to work, all the entities in a chain must be connected at their endpoints.

When selecting a tool to cut a chain of entities, you must select a tool that can cut at different angles. This means that if you select a rectangular tool, that tool must be in an index station, or at an angle that fits the desired cut.

Select a tool to nibble with, define the positioning of the tool on the entity, define the spacing (if needed) and click the **OK** button to close the punching dialog. Next locate and click with the cursor the first entity in the chain, and then the last one. These two entities and all those in between them will be punched.

1.1.4 Nibble Contour

This command nibbles a closed contour. The contour can be made up of lines and arcs, the end points of the entities must be touching.

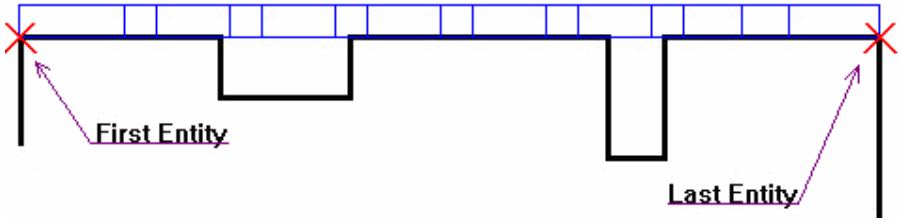
Select a tool to nibble with, define the positioning of the tool on the entity, define the spacing (if needed) and click the **OK** button to close the punching dialog.

Select an entity on the contour and the contour will be punched.

1.1.5 Nibble Start-End

This command will nibble entities that are not connected with each other, lying along the line.

Select a tool to nibble with, define the positioning of the tool on the entity, define the spacing (if needed) and click the **OK** button to close the punching dialog. Select the first entity and then the last one, as shown below. As you can see, all the entities lying on the same line will be punched.



1.1.6 Crunch

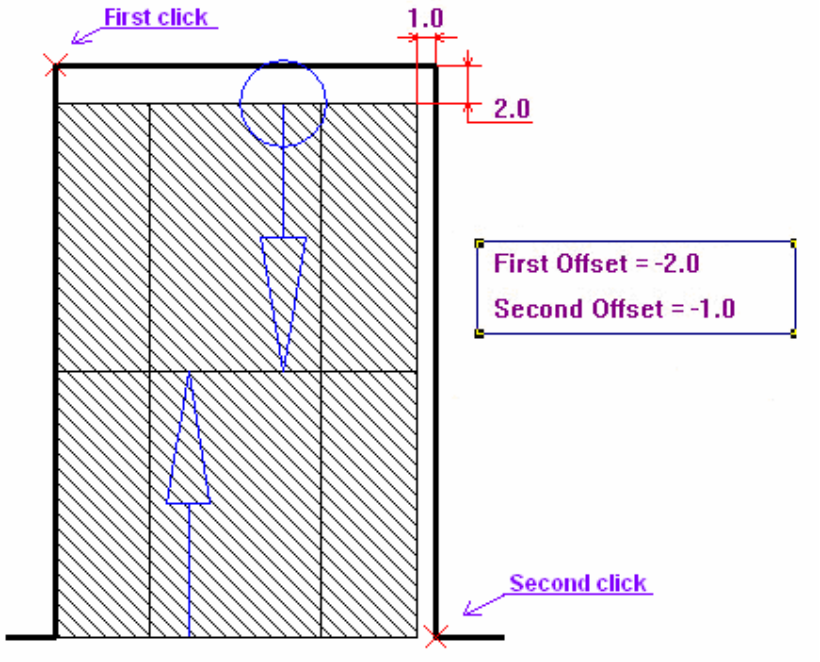
This command works on closed contours of circular areas. On rectangular areas it works also on open contours.

Select an appropriate tool for crunching with. If you are crunching a rectangular area you should use a square or a rectangular tool and if you crunch a circular area you should use round tools.

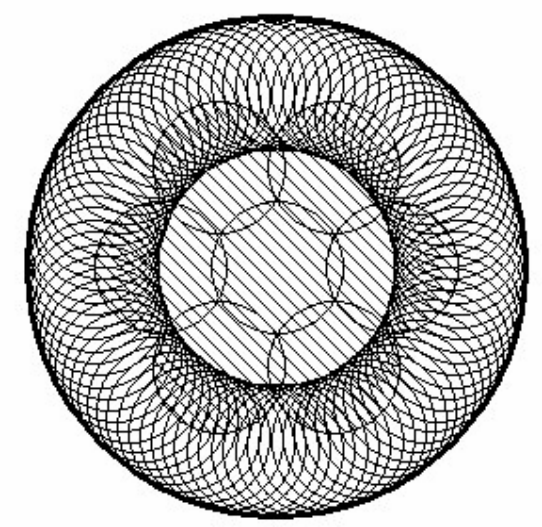
Define the offsets, if any, and click the **OK** button to close the punching dialog.

Click with the mouse cursor next to one of the corners (entity's endpoint) of the rectangular shape to be crunched, e.g. the bottom left, and then select the perpendicularly opposite one, e.g. top right. The area in between will be punched.

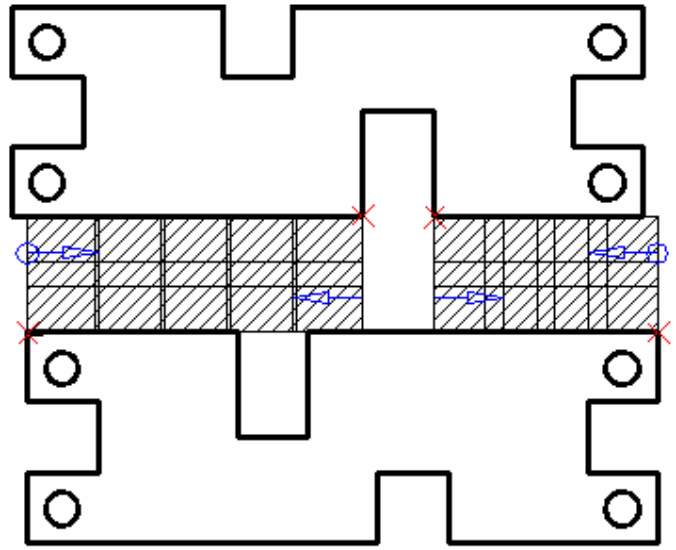
When you want to crunch the circular shape, just choose the proper tool and click the area you want to be punched.



The following shows crunching of a circular hole:



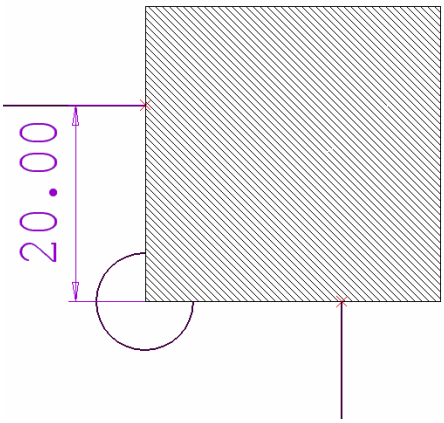
The below example shows crunching of the outer contours between two parts:



The same way this option can be used to crunch material between any two perpendicular points.

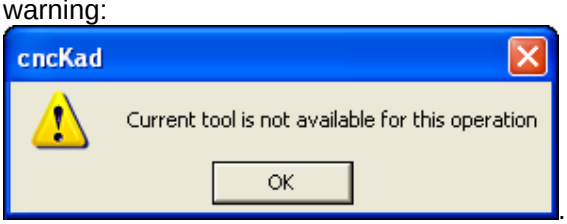
1.1.6.1 Crunching a notch with one tool

This feature makes it possible to punch two adjoining entities of the notch with one tool equal or bigger than a notch. This can be done with the aid of **Crunch** option. On the picture below you can see the **20mm** notch punched this way with **RE 30** tool.



1.1.7 Crunch Right-Angled Triangle

This option only becomes available for *Square* tools. If you will choose and try using any other tool type – you will be presented with the following warning:



This option enables you to crunch a triangular geometry only if there is a 90 degree angle.

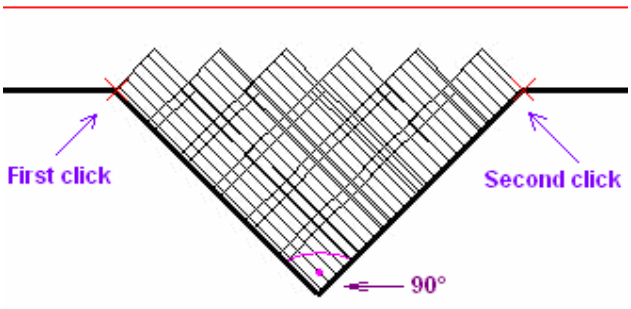
After selecting this option in the dialog, you must choose the starting point for the crunch, (the Status Bar will show the following:

CRUNCH RIGHT-ANGLED TRIANGLE: Select first Corner, Space=Punch parameters, S to shift cutting edge).

Now choose the opposite end point (the Status Bar will show the following:

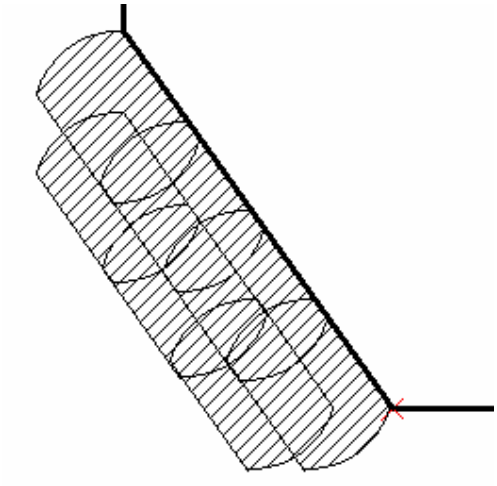
CRUNCH RIGHT-ANGLED TRIANGLE: Select opposite Corner, S to shift cutting edge).

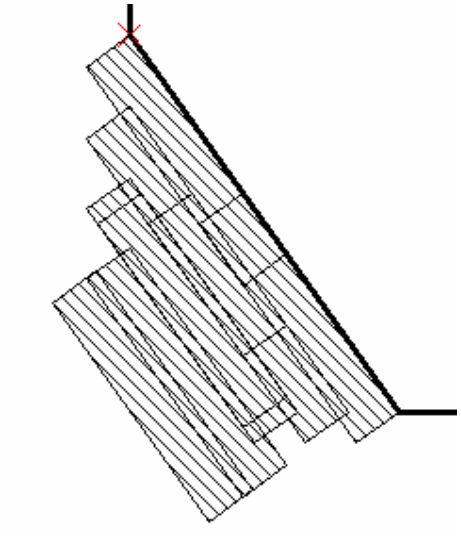
The area between them will be punched.



1.1.8 Crunch Chamfer

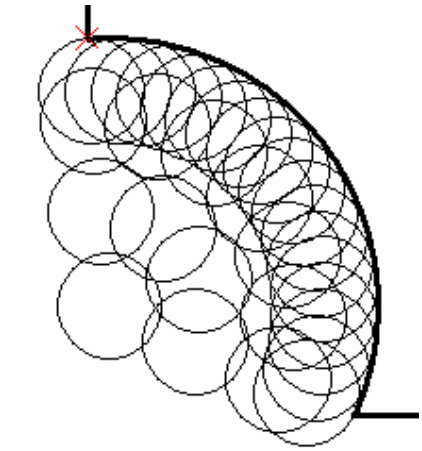
This option enables you to crunch a **Chamfered Entity**, and it can be done with Square, Rectangular, as well as with Double-D tools:





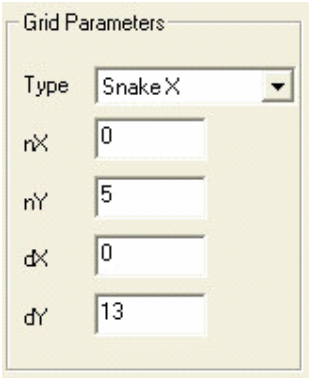
1.1.9 Crunch Arc

This option enables you to crunch an outside **Arc**. This can be done only with round tools:



1.1.10 Punching Grids

This command creates an array or grid of punches. When selecting it, the following area of the punch dialog is activated.



There are four options in the grid dropdown list:

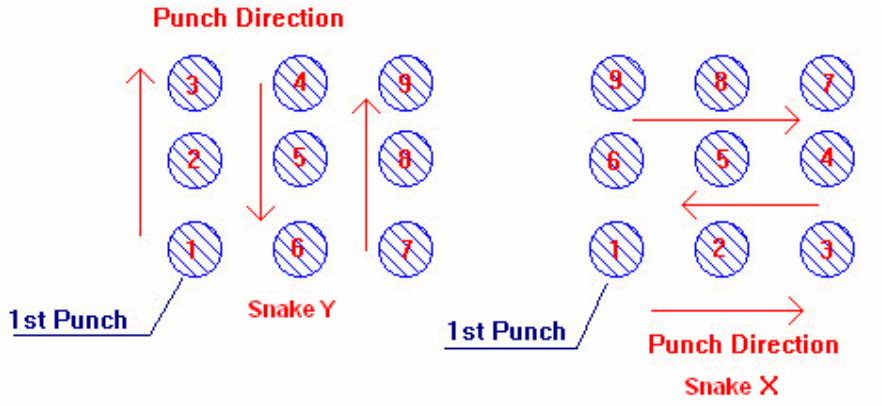
- Snake X

- Snake Y
- Uni-direction X
- Uni-direction Y

The difference between these options is the way (Snake, Uni-direction) and the direction (X, Y) in which the grid will be punched.

Creating a Grid:

- Select the **Grid** option and select the tool that you wish to punch with.
- Define the amount of punches in the **X** and **Y** directions.
- Define the distances between the punches. The distance is defined from the center of each punch to the center of the next punch.
- Click the **OK** button to close the punch dialog.
- Locate the entity on which the first punch of the grid will be placed.
- The grid can be deleted by selecting the first punch in the grid.



1.1.11 Choosing a Tool

Pressing the Tool button, from the **Tool Section** of the Add Punch dialog, you will be redirected to **Tool Selecting** dialog, where from Tool Library and Setup File tabs you will be able to choose desired tool for processing entities with.

1.1.12 Tool Positioning

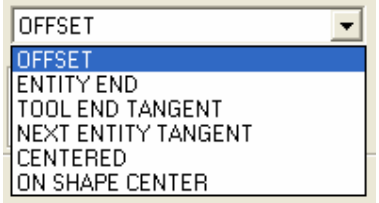
Before positioning the punch, the user must define two parameters:

- The **Side** of the punch:
  The **Side Button** can be found on the **State Bar**, on the left side of the screen; there are three options which define how the tool is placed in relation to the line, arc, or circle selected for punching:
  - **Side** – the edge of the punch will be placed on the side of selected entity.
  - **Tip** – the center of the punch will be placed on top of the line.
  - **Auto** – the punch will be placed automatically on the correct side of the entity. Before activating this option, make sure to run the contour recognition checks on the current part, so that the program would recognize the difference between holes in the part and sheet metal and would know on which side of the contour to place a punch.
- The Index definition:
  The Auto Index button can be found on the **State Bar**, and has two states:
  - **On** – In this state, the currently selected tool will rotate itself to the angle of the line selected. This is only for machines that have Auto Index

stations (rotating stations). There are usually only a few rotating stations on most machines, therefore you should be careful not to use more rotating tools than you have stations for.

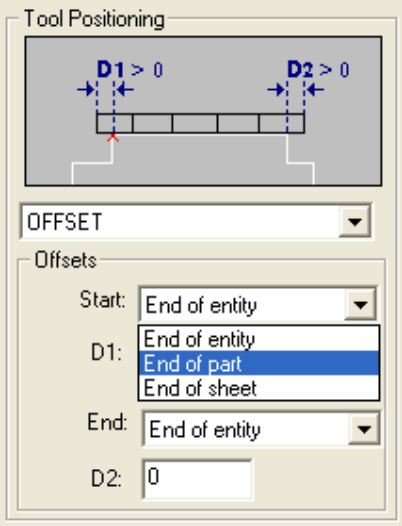
- **Off** – In this state, the tool will be placed on the selected line at the angle that was defined, for the tool within the tool library. If this angle is incorrect for the line selected, a warning will appear giving the user the option to cancel the punching procedure or to continue.

After defining all the above parameters, from the dropdown menu choose the **Positioning** for your punch:



1.1.12.1 Offset

This option gives you an opportunity of offsetting the tool from the **edges** of the **entity**, of the **part** (rectangular block) or of the **sheet**, by a given distance (from **Start** and **End** dropdown lists you can choose one of the options).



You can set the **Start value** of the offset for instance from the end of the **entity** and the **End value** for the end of the **part**.

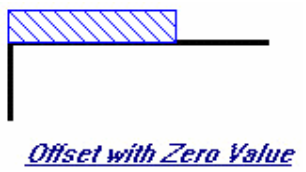
The Start and End dropdown lists are available for **Nibble Entity**, **Nibble Start-End**, **Crunch** and **Crunch Chamfer** options.

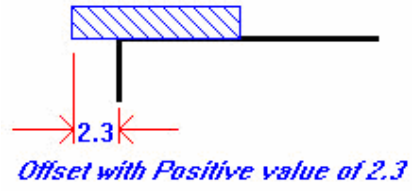
The offset distance values are defined in the offsets **Start** and **End** fields within the Tool Positioning section. They can be positive or negative. If the value is negative, the punch will be placed in from the start of the chosen element, if it is positive it will overhang from line's end.

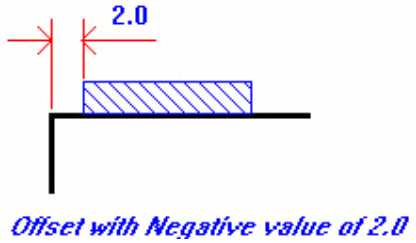
The second offset value (End) is used for nibbling and crunching of non restricted spaces. Therefore for **single** punches, **crunch triangle** and **grid** options there will be only one offset value field available.

Entering a negative offset value can be used for creating Micro Joints between parts.

The following are examples of different offsets:



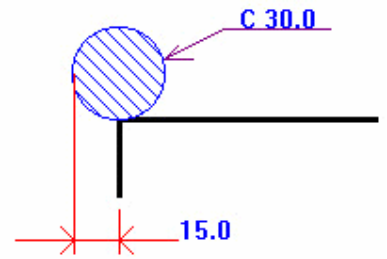




- i.e. "-2", in the offset field

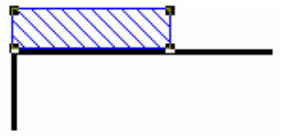
1.1.12.2 Entity's End

With this option, the center of the tool will be placed at the endpoint of the line. This is done automatically with no need for any additional parameter input.



1.1.12.3 Tool's End Tangent

Using this option will place the end of the tool tangent to the end of the line. This is equivalent to **Offset** = **0**.



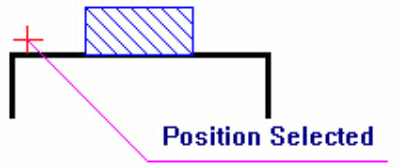
1.1.12.4 Next Entity's Tangent

This option is useful for round and oblong tools.

This option will place the tool tangent to the next entity in a chain of entities. The endpoints of the entities must be connected.

1.1.12.5 Centered

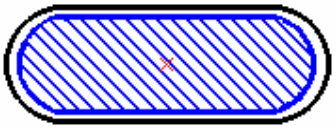
This option will place the punch on the center of the line selected. It doesn't matter where you select the line, the punch will always center itself on the line.



1.1.12.6 On Shape Center

This option will place the shape of the tool exactly in the middle of the shape (**NOT** entity) we want to punch.

This way we will be able to process a hole with a tool that is not an exact match, and the punch will be still placed in the hole's center, e.g. an oval hole of 31 by 10 processed with an **OB 30 10** tool:



This option works only for standard shapes – Rounds, Squares, Rectangles, Ovals, Rounded Rectangles and Single/Double D.

1.1.13 From Base Line

This option is available for **Crunch** and **Crunch Chamfer** Punch Types.

Marking this checkbox allows you for crunching nonstandard notches, of an angle larger than 90°. This means that the offsets set for the crunch will be measured from the Base Line of the notch.

This option is unavailable for Crunch punch type for which **scallop** type of Tool Spacing was selected.

1.1.14 Tool Spacing

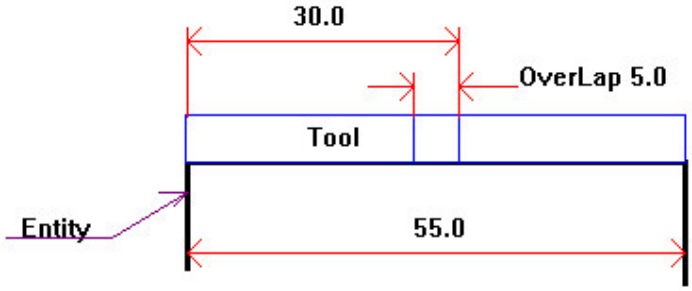
The Tool Spacing options define how much overlap there will be on the tool when nibbling.

1.1.14.1 Overlap Definition

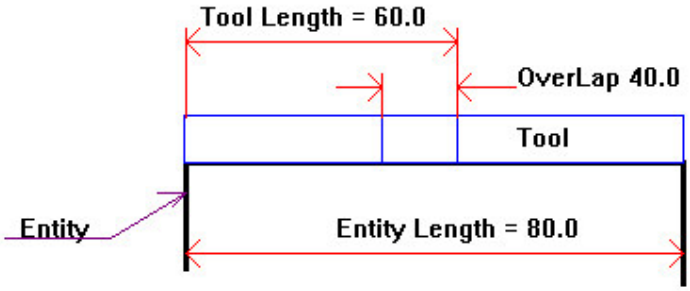
When the overlap is defined as **0%**, the current tool will arrange itself on the contour with the minimal overlapping. If you choose to nibble an entity with a tool that is longer than one and a half times the length of the entity, you will receive a warning (provided you have marked the appropriate option in the **Punch and Cut Warnings** of the **Workspace Settings** dialog in ***Drafting and Nesting Manual***). Clicking **OK** button will nibble the entity, clicking **Cancel** button will annul the operation, and the entity won't be punched.

See the examples presented below. The punches here were made with **0%** overlap:

- The following case produces no warning because it is a correct procedure:

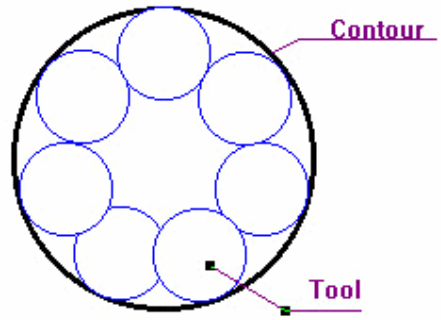


- The following case will produce a warning because more than half the punch is overlapping:

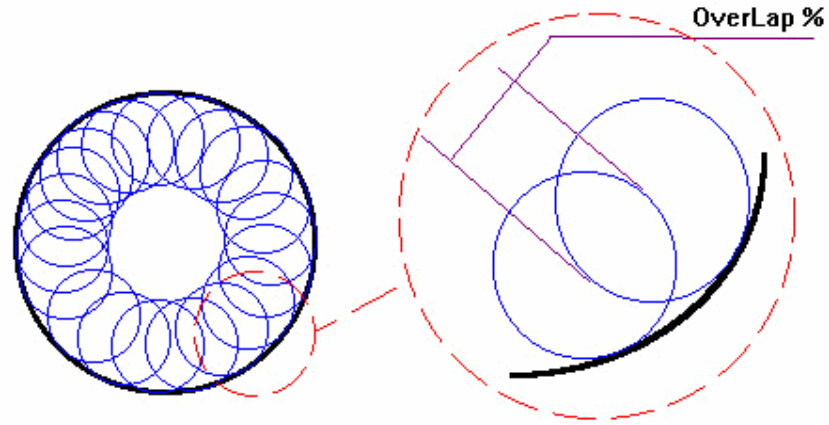


When the overlap is defined with a percentage other than 0, the tool will be overlapped by that percentage. This can be used for cutting arcs and circles. The greater the overlap, the smoother the cut will be. See the following two examples:

- Cut defined with **0%** overlap.



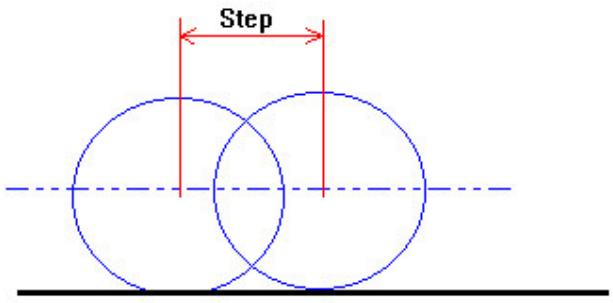
- Cut defined with approximately **75%** overlap.



If you define an overlap that has a step less than the sheet thickness, you will receive a warning (as before, only if you mark this option in **Punch and Cut Warnings**).

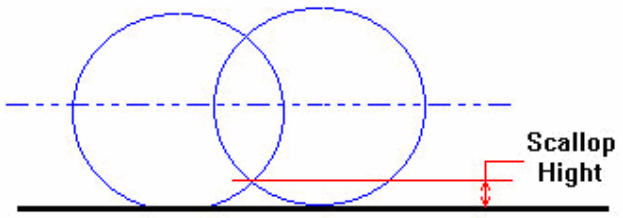
1.1.14.2 Step Definition

With this option the distance between the centers of each punch in a nibbling cycle, is defined. If a distance, that is less than the sheet thickness is defined, a warning will be received.



1.1.14.3 Scallop Definition

This option is accessible only when a circular tool is being selected. The scallop is the height of the tooth that is left between the punches of a circular tool. See the following picture:



1.1.15 Tool Sequence

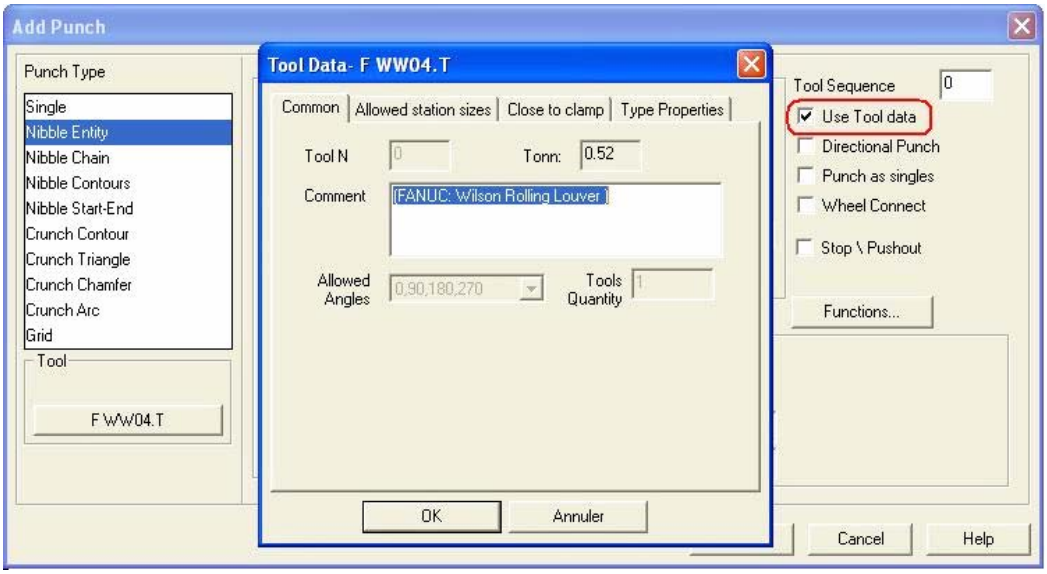
Here the user can set the sequence of the tool.

For more information on this function go to Set Tool Sequence section of the CAM Menu chapter of this manual.

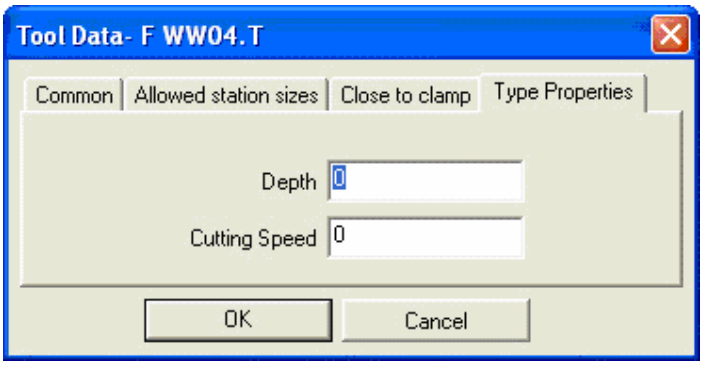
1.1.16 Use Tool Data

With this function you will be able to utilize any special features assigned to a certain tool. Here you can view and edit chosen tool's data – add a comment, choose the station for it and define whether it should be used as **Close to Clamps** tool or not.

In this example we will view the **Tool Data** of a *Wilson Wheel Louver* tool:



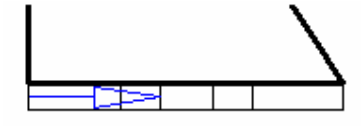
Special tools have one more option – **Type Properties** tab. Clicking it will enable you to define specific parameters for a punch made with the tool you chose:



This means that for every punch you place on your part with this tool you can use different settings.

1.1.17 Directional Punch option

Choosing this option will add a direction to your punching. When you place the punch on a part, clicking the entity endpoint, you will see that a blue arrow (or a red one – depending on the background you are working on) was added to the beginning of your placement, showing a direction the punch was made in.



1.1.18 Punch as Singles

This option concerns **Nibbling** and allows the user to execute the **Nibble** as **Single Punches**. Thereby, each punch will get a separate line in the NC code.

1.1.19 Wheel Connect

This function is available, when the **Wilson Wheel Tool** is currently chosen.

If this box is checked, the Wilson Wheel will process the part's contour continuously (without stops, which it usually does) – it will connect between one punch and the next one.

> By default this option is checked for all nibbling types, except for nibble entity.

1.1.20 Functions button

This option allows you to access the **Functions for Selected CAM** dialog directly from the **Add Punch** dialog.

1.1.21 Stop or Push Out

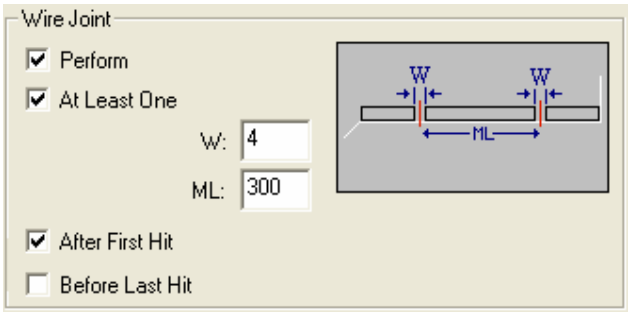
This command is for machines that have a chute on the table. Parts that have been punched/cut out of the sheet can be pushed out via the chute. This prevents from having to use a **Stop Machine** command, in order to remove the part. The position of the chute varies from machine to machine.

See full explanation in the **Push Out (Chute)** section of CAM Menu chapter in this manual.

1.1.22 Wire Joint

Choosing this option (by checking the **Perform** box) you will be allowed to place Wires as an integral part of the Punch.

You can set the **width** of the Wires (**W**) as well as the **maximum distance** between them (**ML**), and they will be integrated into the Punch, without splitting the entity being punched:



Choosing one of the **After First Hit** and **Before Last Hit** options enables creating Wires after the first and before the last punch. They address the issue of rounded corners when punched by round tools.

After placing the punch, you can return and **Edit** it (clicking on the **Edit CAM** icon –  from CAM Toolbar) changing the parameters, without having to edit several entities (as it would happen with regular Micro Joints that have been placed after the entity has been punched).

1.1.22.1 Width

This parameter sets the width of the Wire joint(s) placed for this punch.

1.1.22.2 Max Length

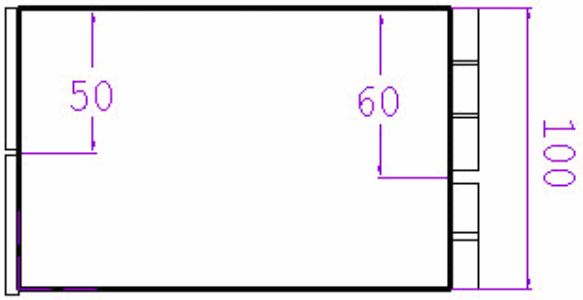
This option is very useful when the precise position of the Wire joint(s) is not important. This parameter sets the maximum distance between two Wires, and NOT the position relative to the entity's end.

If you define a **Max Length** of 80 for an entity with a length of 100, this *does not* mean that there will be a Wire at 80mm from entity's beginning. The exact position will depend on the tool chosen for the punch – the Wire will be placed at tool's *Hit Boundary*.

Below there are two punches defined with a **Max Length** of 80:

The punch on the left was made with an *RE 50 5* tool and the Wire was placed 50mm from the entity's beginning.

The one on the right was made with an *RE 20 10*, and the Wire was placed at 60mm.



(These Wires were made with a **Width** of 5 for demonstration purposes.)

1.1.22.3 At Least One

This option is very useful when you want to place just one Wire more or less in the middle of a punch, and the exact position is not important:

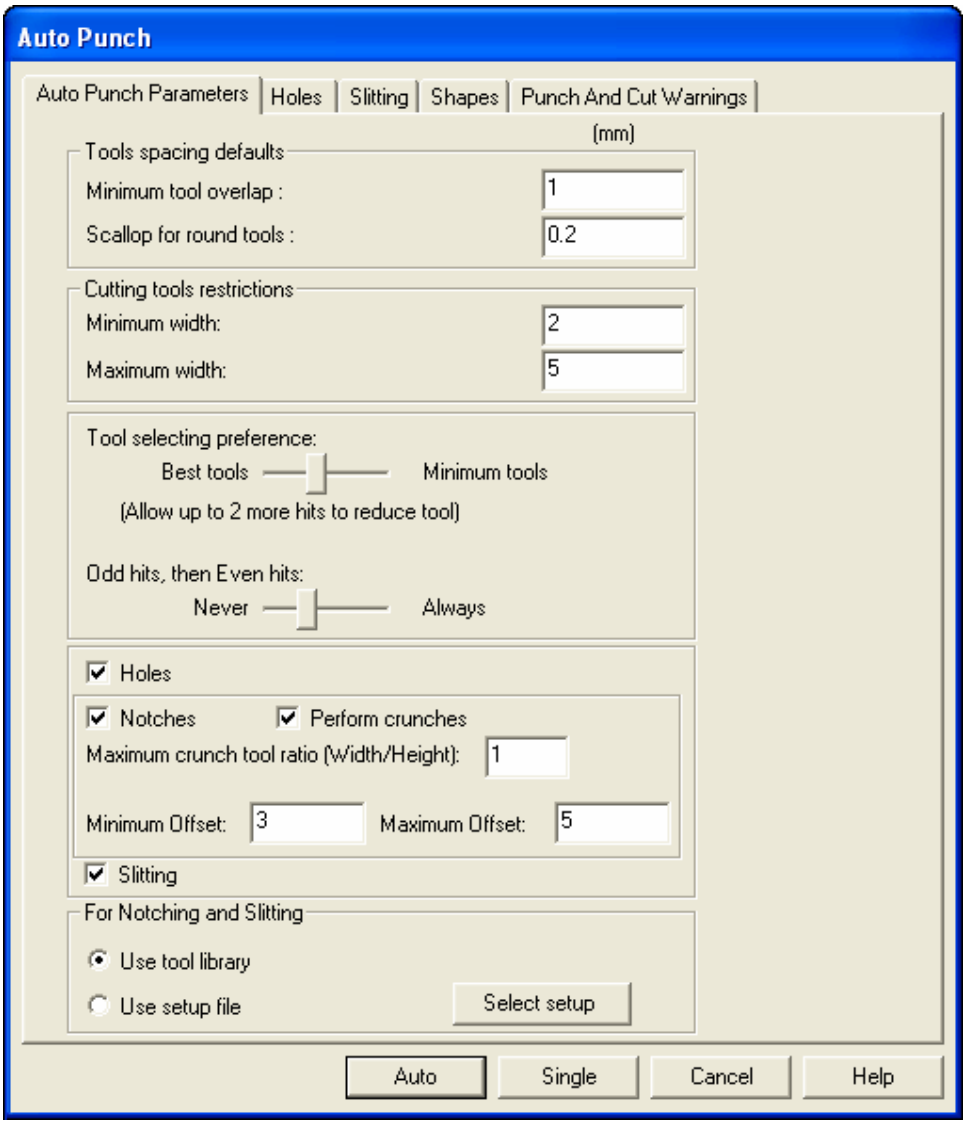
Set the **Width** at the desired value, and the **Max. Length** at a very large value (larger than possible, such as 10000).

The result will be one Wire positioned the tool Hit Boundary nearest the middle of the entity being punched.

1.2 Auto Punch

By using this feature you allow the program to punch the whole part automatically.

You can also click an icon  from the **Common Toolbar** –  . The following dialog will appear:



1.2.1 Auto Punch Parameters tab

On this tab you will be able of defining optional settings for automatic processing of the part.

Auto Punch will only work on closed contours and viable entities, so before using this feature we would recommend to run a **Check**. Even if there are some bad contours, this will not prevent processing the correct ones.

There are two options for running Auto Punch:

- **Auto** will run **Auto Punch** for the whole part.

- **Single** will use **Auto Punch** on a selected entity. This option will punch just one entity at a time. But when you want to punch next entity you don't have to come back to this dialog. All you need to do is just to click the cursor next to it and it will be punched.

See the subsequent sections for explanation on all the features which can be found on this tab.

1.2.1.1 Tool Spacing Defaults

These options address the issues discussed in **Scallop Definition** topic of the **Tool Spacing** section of this chapter.

The value in this section is given in millimeters or inches, depending on your machine and the option you chose from **working units** section of Settings Menu chapter of ***Drafting and Nesting Manual***.

1.2.1.1.1 Minimum Tool Overlap

Select a desired minimum **Overlap** to be used when tool overlap occurs.

1.2.1.1.2 Scallop for Round Tools

Here the user can enter the value of **Scallop** size, which will be implemented for round tools.

1.2.1.2 Cutting Tools Restrictions

In this section some limitations for the tools can be set.

1.2.1.2.1 Minimum Width

When punching the outer contour, use tools of no less than this width.

1.2.1.2.2 Maximum Width

When punching the outer contour, use tools of no more than this width. Use this parameter, to prevent the material waste on the outer side of contour.

1.2.1.3 Tool Selecting Preferences

Choose a strategy for selecting the tools. Move the slider between these two poles and set the proper option, using your judgment based on the situation.

1.2.1.3.1 Best Tools

Use as many tools as it is necessary (up to maximum quantity of tools that can fit the turret), in order to achieve best punching results.

1.2.1.3.2 Minimum Tools

Try, as much as possible, to re-use tools. This reduces efficiency.

1.2.1.4 Odd hits, then Even hits

Select a strategy for the last punch in the middle:

- **Never** – don't ever place the last punch in the middle.
- **Always** – always, in every case, place the last punch in the middle.

Choose an option between these two poles, based on the situation.

1.2.1.5 Punching scope

You can choose which range of entities will be punched.

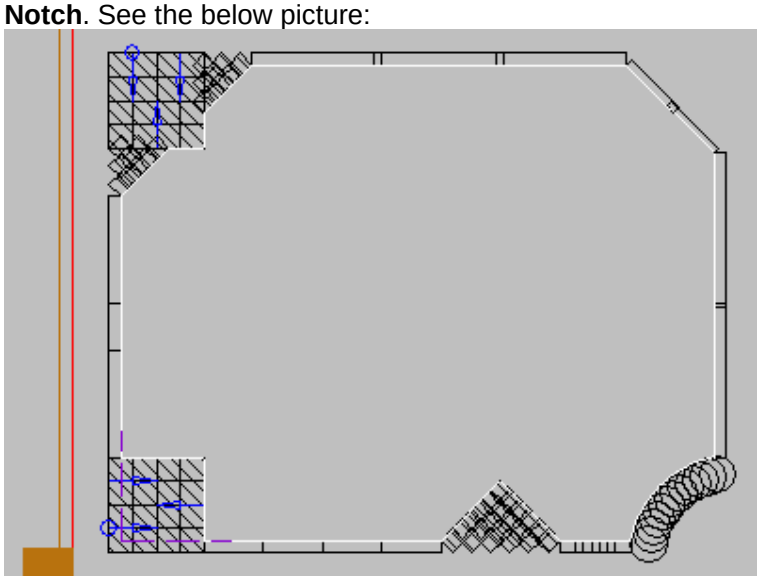
1.2.1.5.1 Holes

If you enable this option, the holes in your part will be processed, in accordance with the settings in the **Holes** tab.

1.2.1.5.2 Notches

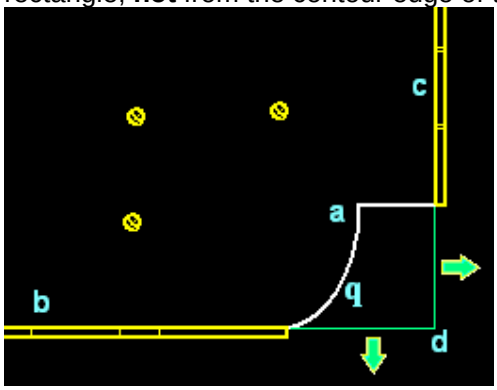
If you want that all the notches in your part would be processed while running Auto Punch, you need to activate this option by marking the checkbox. Otherwise the **Notch** section will not be active and you won't be able to edit any of the following options:

- **Perform crunches** – Selecting this checkbox will allow for automatic **crunching** the notches of the part; otherwise each entity will be processed separately. There are three types of notches that may be processed by Auto Punch: **Corner Notch**, **Corner with Chamfer**, and **Right Angle Triangle Notch**. See the below picture:



Checking this option enables editing of the Maximum crunch tool ratio and Minimum Offset features:

- **Maximum crunch tool ratio** – You can define maximum ratio between *Width* and *Height* of the tool that will perform the crunch. If you don't care much about this option – set large number like "80"; if you want that a hole will only be crunched with a square tool – set "1".
- **Minimum Offset** – Use this parameter to create an offset for the processed notch. The actual offset will depend on the tools and strategy chosen for processing the notch. Its value will be measured from the part's bounding rectangle, **not** from the contour edge of the notch.



  - **a** – the part's outer contour.
  - **b** and **c** – the slitting.

- **d** – the bounding rectangle

  In this case, a **Maximum Offset** of **0** will mean that the arc (q) cannot be processed!

- **Maximum Offset** – This parameter will be used as a maximum – the actual offset will depend on the tools and strategy chosen for process the notch.

1.2.1.5.3 Slitting

If you enable this option, all the outer contours of your part will be processed.

1.2.1.6 Use Tools Library

This option will allow Auto Punch to process the parts on the sheet, using any fitting tool from the **tool library**.

This setting **DOES NOT** apply to processing **Holes**. When you want that the holes would be processed using Tools Library, you need to set this option on the **Holes tab**.

1.2.1.7 Use Setup File

This option will make Auto Punch to use only tools from the selected setup file (any tools you have already added yourself to the part, will remain).

You can select the appropriate setup file with the **Select Setup** button.

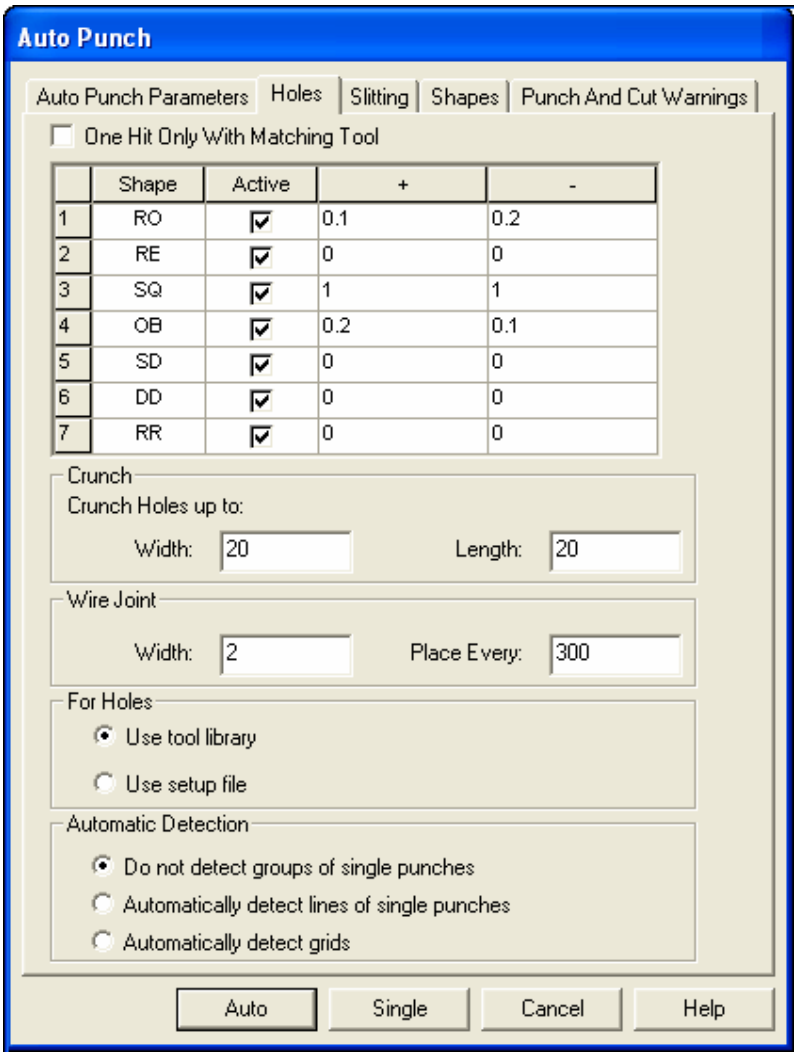
This setting **DOES NOT** apply to processing **Holes**. When you want that the holes would be processed using Setup file, you need to set this option on the **Holes tab**.

1.2.1.8 Select Setup

Clicking this button you will be moved to the **Turret Setups** dialog of the Tools Menu. There you will be able to choose the setup file you want to use for processing the current sheet.

1.2.2 Holes Tab

Use this tab to define how holes will be processed by **Auto Punch**:



1.2.2.1 One Hit Only With matching Tool

Checking this option means that all holes will be processed with single punches. Holes that cannot be punched in this way, will be left unprocessed.

This option disables the **Crunch** option.

1.2.2.2 Tolerance Settings

In the following window it is possible to set tolerance for punching with specific tools. This means that by checking **Active** box and entering a value in either positive or negative column by the particular shape, the user will be able to punch holes whose size does not exactly match the size of the tools he possesses.

	Shape	Active	+	-
1	RO	☑	0.1	0.2
2	RE	☑	0	0
3	SQ	☑	1	1
4	OB	☑	0.2	0.1
5	SD	☑	0	0
6	DD	☑	0	0
7	RR	☑	0	0

For example: If you have a hole with a 3.1 diameter and only a **RO 3** tool, you can define a negative Tolerance (in "**–**" column type positive value) of **0.1**, and when running **Auto Punch**, the RO 3 will be used. If on the other hand you have a hole of 9.5 diameter and have no exact fitting tool, but only a **RO 9.6** tool, set the positive value (in "**+**" column type positive value) of tolerance to 0.1 and when running **Auto Punch** the RO 9.6 tool will be used.

Keep the **Active** row checked for all Shape types, unless you do **NOT** want a specific type to be processed.

1.2.2.3 Crunch

This option is used for setting the maximum size of rectangular holes that will be crunched, by entering their **width** and **length** in the appropriate fields.

1.2.2.4 Wire Joint

These parameters define how to place Wire joints when processing holes.

For an explanation about how to set the size of the **Place Every** parameter, see the **Wire Joint** section in the **Add Punch** dialog.

Regardless of the settings defined here, Auto Punch will place at least one **wire** on the long sides of rectangles, to ensure they remain attached to the sheet.

1.2.2.5 For Holes

Selecting one of these options you define the source from which the tools will be taken for processing holes:

- **Use tools library** – this option will allow Auto Punch to use any tool from the tool library.
- **Use setup file** – this option will force Auto Punch to use only the tools from the setup file chosen in the **Auto Punch Parameters** tab.

1.2.2.6 Automatic Detection

In this section, you can order the Auto Punch process:

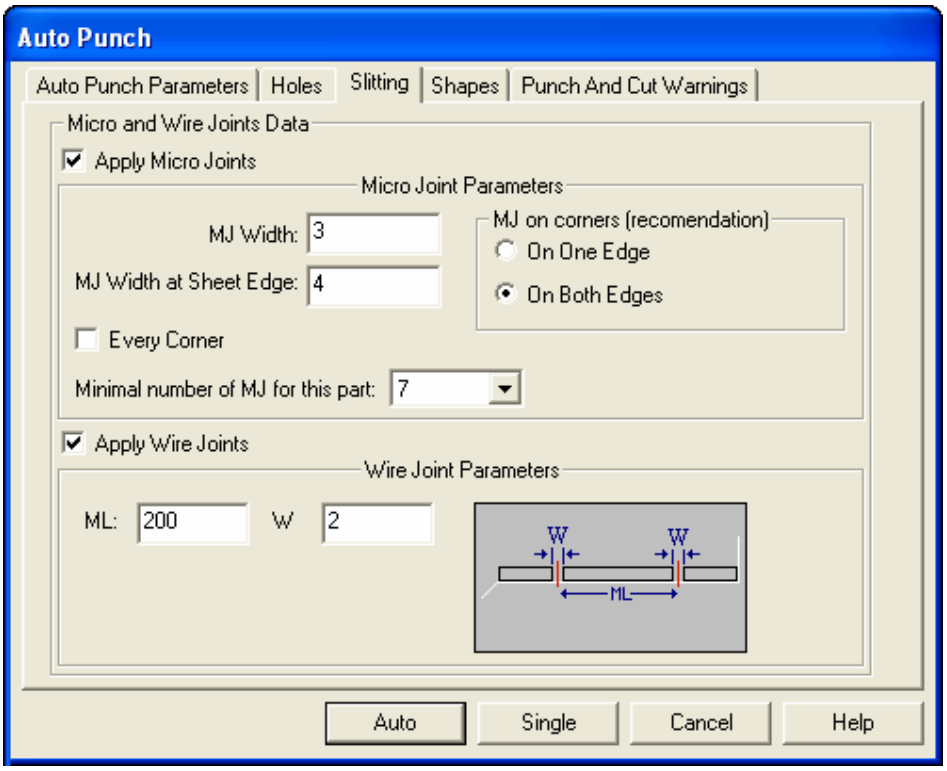
1. **Not** to **detect groups of single punches** – no group detection will be operated.
2. To **automatically detect lines of single punches** – ***cncKad*** will detect automatically a line of single punches and turn their type into a "single; on line".
3. To **automatically detect grids** – ***cncKad*** will recognize grids of singles and will change their punch type into a grid.

The advantages of this option are the shortening of the NC code, but also the widening of edition possibilities for entities within ***cncKad***: for instance, a punched grid will be shortened or extended more easily.

If you are not interested in activating this option while using the Auto Punch feature, you can still define **Automatic Detection** for the NC code via the NC Program Creation **=> Post Processor Options** dialog. In this case, the part will contain singles but the NC will contain "single on line" punches and grids.

1.2.3 Slitting tab

On this tab you can define various aspects of automatic placing of Micro Joints on the processed entity.



1.2.3.1 Micro Joint Parameters Section

Here you have the possibility of setting various parameters for placing Micro Joints. They will be enabled only after you check the **Apply Micro Joints** box.

1.2.3.1.1 MJ Width

In this field you can set the general width of the Micro Joints to be created.

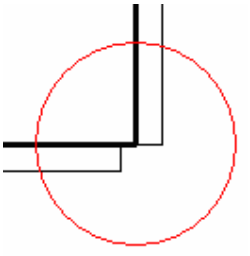
1.2.3.1.2 MJ Width at Sheet Edge

In this field you can set the width of the Micro Joints to be created on the sheet edge(s).

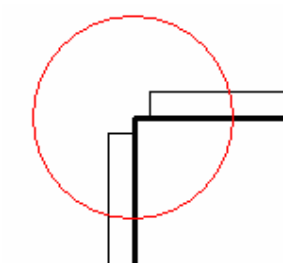
1.2.3.1.3 MJ on corners

There is an option of automatic setting the Micro Joints on the corners. This can be achieved in two ways:

On **One** Edge:



On **Both** Edges:



1.2.3.1.4 Every corner

Marking this checkbox you choose to place the Micro Joints on each corner of the part. But choosing this function disables the usage of **The Minimal number of Micro Joints for this part** option.

1.2.3.1.5 The Minimal number of Micro Joints for this part

As the topic says, from the dropdown list you can select the **minimal** quantity of the Micro Joints to be placed on the current part.

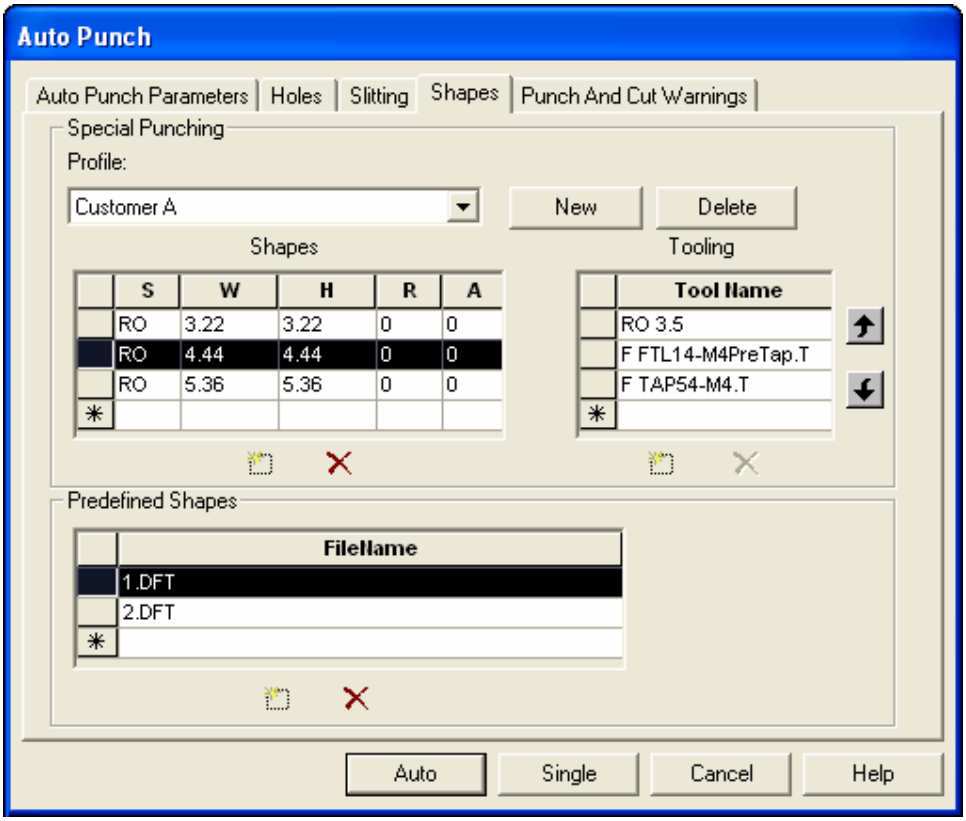
1.2.3.2 Wire Joints Parameters Section

The options of this section will be enabled only after you check the **Apply Wire Joints** box.

Here you can define parameters such as the **width** of the Micro Joints (**W**) as well as the **maximum distance** between them (**ML**).

1.2.4 Shapes Tab

From this dialog you can create special treatment for particular cases by creating **Profiles**, and using **Predefined Shapes**:



1.2.4.1 Special Punching Section

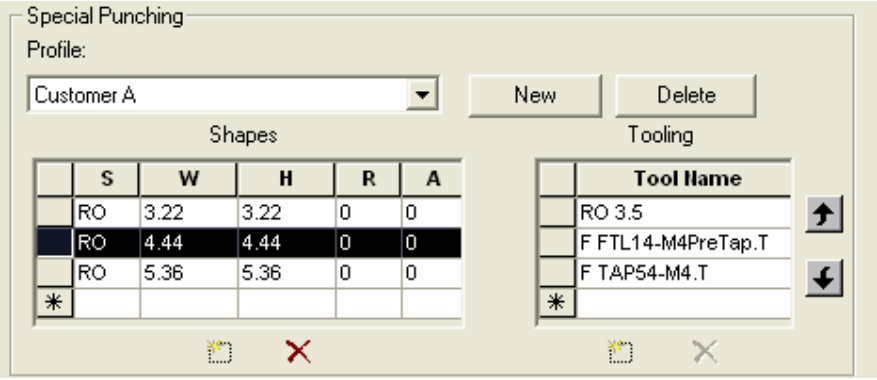
This section presents the options of special punching of untypical sized round holes.

1.2.4.1.1 Profile

A **Profile** is a set of instructions telling how specific holes will be punched.

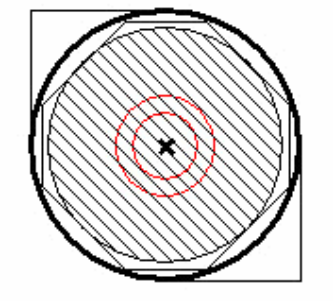
You can use the dropdown list to select which from the available profiles you want to use. If there are none, use the **New** button to create a new one, and the **Delete** button to delete an existing one.

In our example we created a profile called **Customer A**, in which we have three round holes of atypical sizes. From the **Shapes** table let's take a round shape of **4.44** diameter and in the **Tooling** table let's add the tools we want to process this hole with and set them in desired sequence. See the below picture:



After running **Auto Punch** each circle with a **4.44** diameter will be punched with the sequence of tools we just have set – first **RO 3.5**, then a **Pre-tapping** tool, and last a **Tapping** tool.

Our hole processed with this sequence will look this way:



1.2.4.1.2 Shapes

In this table we can see various shapes that are specified in the current **Profile**. When a specific shape is selected, the tooling sequence defined for it appears in the **Tooling** section of this dialog.

You can use the button to add a shape to the profile, and the button to delete a shape from the profile.

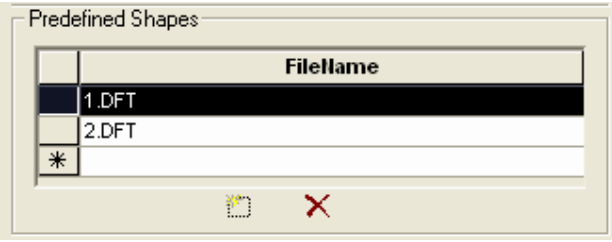
1.2.4.1.3 Tooling

This table shows the specific tooling sequence set for the item selected in the **Shapes** section of this dialog.

You can use the button to add a tool to the list, and the to delete it from the list.

1.2.4.2 Predefined Shapes Section

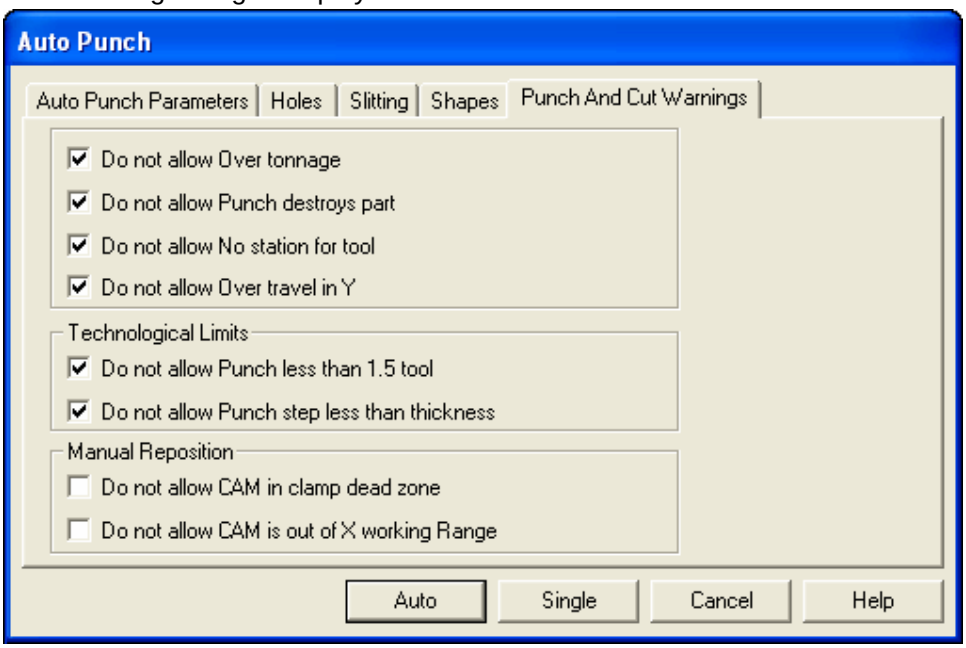
After **saving** some specific shape, together with its process, as predefined one (see the **Save As Predefined Shape** section of this chapter) this section allows you to add such a predefined shape to the list:



After clicking **Auto** button all the **Predefined Shapes** from this table will be used to punch suitable notches and contours of the part. If there are neither suitable notches nor contours, the **Predefined Shapes** won't be used.

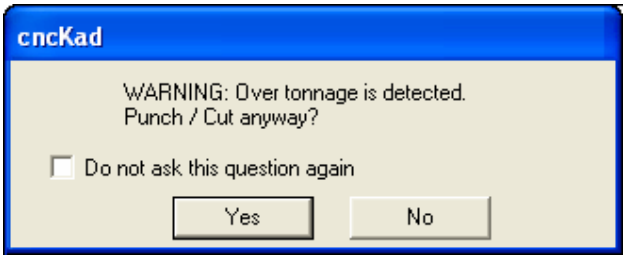
1.2.5 Punch and Cut Warnings Tab

The following dialog is displayed:



1.2.5.1 Do Not Allow Over Tonnage

Will not allow punching if a tool exceeds the **Tonnage Limit** of the machine. If the over tonnage will be detected, the following warning will appear on your screen:



Here, as well as in the subsequently described other processing warnings dialogs, you will be able to decide about the action to be taken – you can choose not only whether continue the CAM or not, but also you will be given the possibility of choosing a default option for the similar situation in the future.

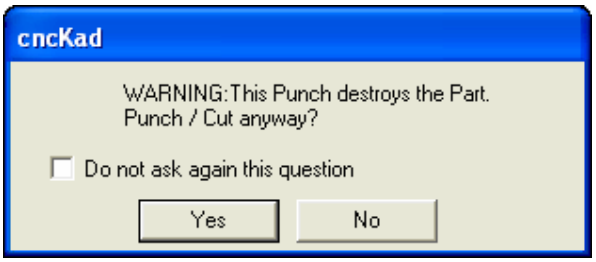
To do so, check the **Do not ask again this question** box, and click one of the buttons. From now on, depending on your answer, ***cncKad*** will or will not process the entity, when there will be a danger of over tonnage will be detected.

> The action undertaken, chosen from the warning dialog, will be the case for the current part only and as long as you do not re-run ***cncKad***.

1.2.5.2 Do Not Allow Punch Destroys Part

Will not allow punching if a punch will destroy the part. ***cncKad*** identifies also when the punch destroys the part for D, double-D, Cross and Special Tools.

If in the **Workspace settings** on the **Punch And Cut Warnings** tab a proper option will be checked, a warning will be issued whenever Punch will destroy the part:



1.2.5.3 Do Not Allow No Station for Tool

The warning appears only during NC creation.

This option will not allow placing a punch on the part if there are not enough stations on the turret needed for placing the tools for this punch.

1.2.5.4 Do Not Allow Over Travel in Y

The warning appears only during NC creation.

This option will not allow placing a punch if the tool cannot reach the point to be processed from where it sits in the turret.

1.2.5.5 Do Not Allow Punch Less than 1.5 Tool

This option will not allow placing a punch if the size of selected nibble or crunch is less than 1.5 times the tool size.

1.2.5.6 Do Not Allow Punch Step Less than Thickness

This option will not allow placing a punch, if the tool's step is less than the material's thickness. It will not allow for punching also when during using **Single** punching the tool is smaller than the thickness.

1.2.5.7 Do Not Allow CAM in Clamp Dead Zone

Will not allow placing a punch in a clamp *Dead Zone*.

1.2.5.8 Do Not Allow CAM is out of X working Range

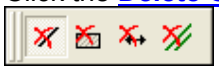
Will not allow placing a punch outside the table limits.

1.2.6 Punching sheet edge

The Auto Punch checks if the **Automatically remove punches on sheet edge** box is selected on the Post Processor Options tab of Workspace Settings. If yes, then in all cases where the part is placed at the sheet edge, the sheet edge won't be punched; if no, the sheet edge will be punched.

1.3 Deleting Punches

There are three methods for deleting punches:

- Press **Alt+F3** keys.
- From the **Edit** menu select the **Delete CAMs** option.
- Click the **Delete CAMs** icon –  from the Delete Toolbar – 

Use the standard **Selection** options to choose punches for deletion, and click **Enter** to confirm the selection; it is recommended you click Redraw icon –  in order to clean the draft after completing the deletion.

1.4 Set or Clear Last Hit on Center

When crunching or nibbling, you can set the order of punches so that the last punch in the nibbling process will be in the middle.

1.5 Directional Punch

A directional punch is a cut that is defined to punch in a specified direction. This option may be used for tool technological reasons as well as for optimization reasons.

1.5.1 Uni Directional

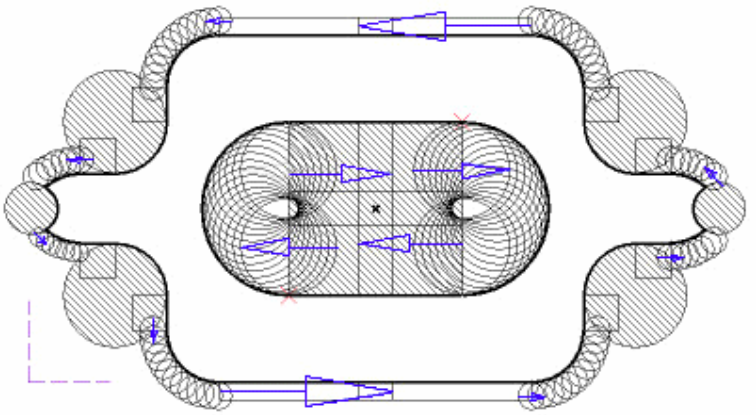
This option is used for making a punch directional:

Click the punch that you want to make directional.

A red **X** will be placed at the starting point of the cut and you will be asked if you wish to reverse the direction of the punch, reply **Yes** or **No** and the dialog will be closed.

A blue arrow (or red one – depending on the background you are working on) will be placed on the punch showing the direction that the cut will be made in. No optimization will change the direction of the cut.

You can continue and select more cuts and make them directional:



1.5.2 Bi Directional

This option converts a directional punch to a punch with no specified direction, meaning that during the post processing stage the direction of this punch will be determined.

Click the punches that you wish to make non-directional.

Press **Enter** when you have selected all the cuts to change to non-directional.

1.6 Save As Predefined Shape

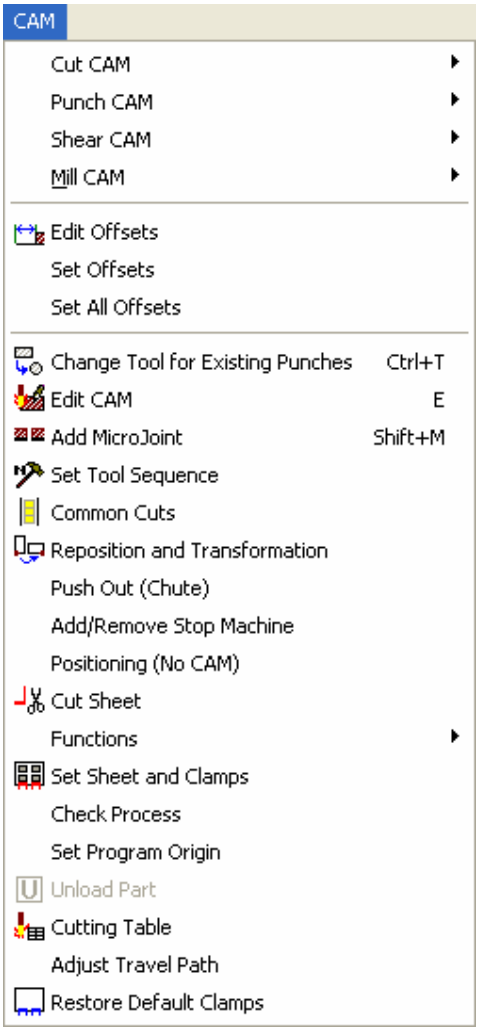
This feature enables you to define a specific shape with its punch(es), which will be recognized by ***cncKad*** and processed automatically with predefined tools assigned to it, after you run **Auto Punch**.

After choosing this option, click on chosen punch, and save it.

Then go to the **Shapes** tab of Auto Punch dialog, create new profile (or choose an already existing one) and in the **Predefined Shapes** section, clicking on the asterisk sign, choose the name of the file(s) you want to use.

2 The CAM Menu

This menu deals with all matters pertaining to the manufacturing process, from placing the CAMs on your draft to various definitions governing the actual machining of the part.



2.1 Setting Offsets

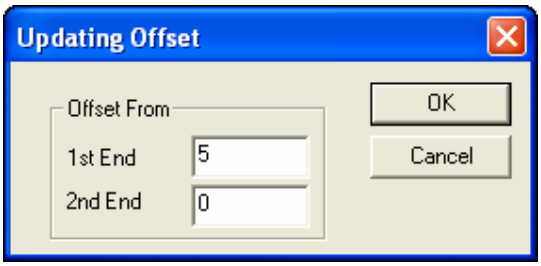
These options allow you to create Offsets or edit existing ones.

Changes may produce technological warnings.

2.1.1 Edit Offsets

After you have drafted your part and placed CAMs on it, this option allows you to choose a specific CAM and define an **Offset** for it.

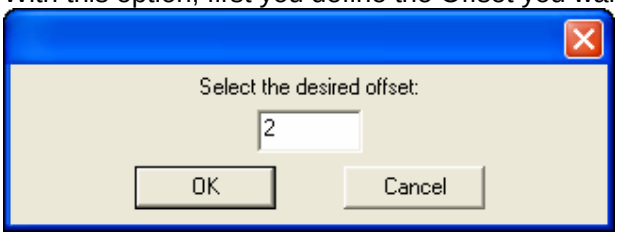
Choose this option from the menu, and click the CAM you wish to offset. The following dialog will open:



Define appropriate Offsets from the ends of the entities, and click **OK** button.

2.1.2 Set Offsets

With this option, first you define the Offset you want to set,



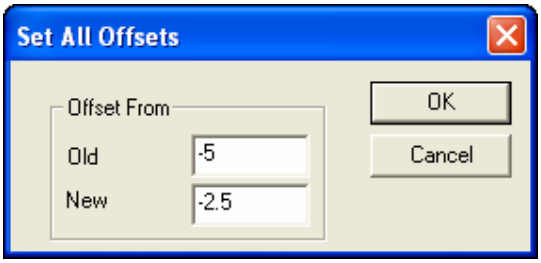
and then click the CAMs you wish to apply it to.

2.1.3 Set All the Offsets

This option allows you to change a previously defined **Offset**.

For example, if you have defined five punches with an **Offset** of "-5", and now you want to change all of these Offsets to "-2.5", this option lets you do this with one command.

After you choose this option, the following dialog will open:

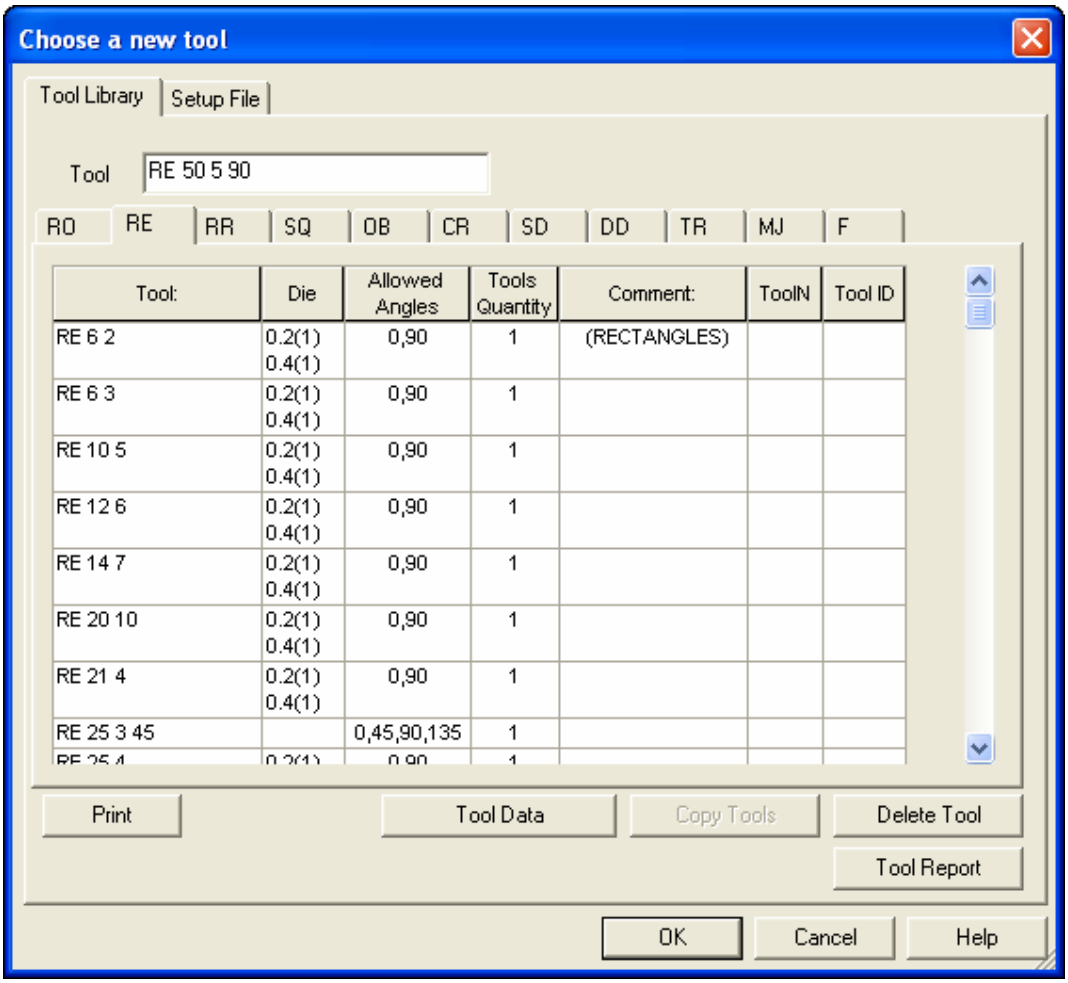


- **Old** – here type the value of existing Offset.
- **New** – here type the value of the offset you wish to change the previous one into.

  If you define **Old** as "**0**", it will change all the punches without any **Offset**.

2.2 Change Tool for Existing Punches

Select this option after choosing a tool in the **Used Tools List** of the State Bar. The following dialog will appear:



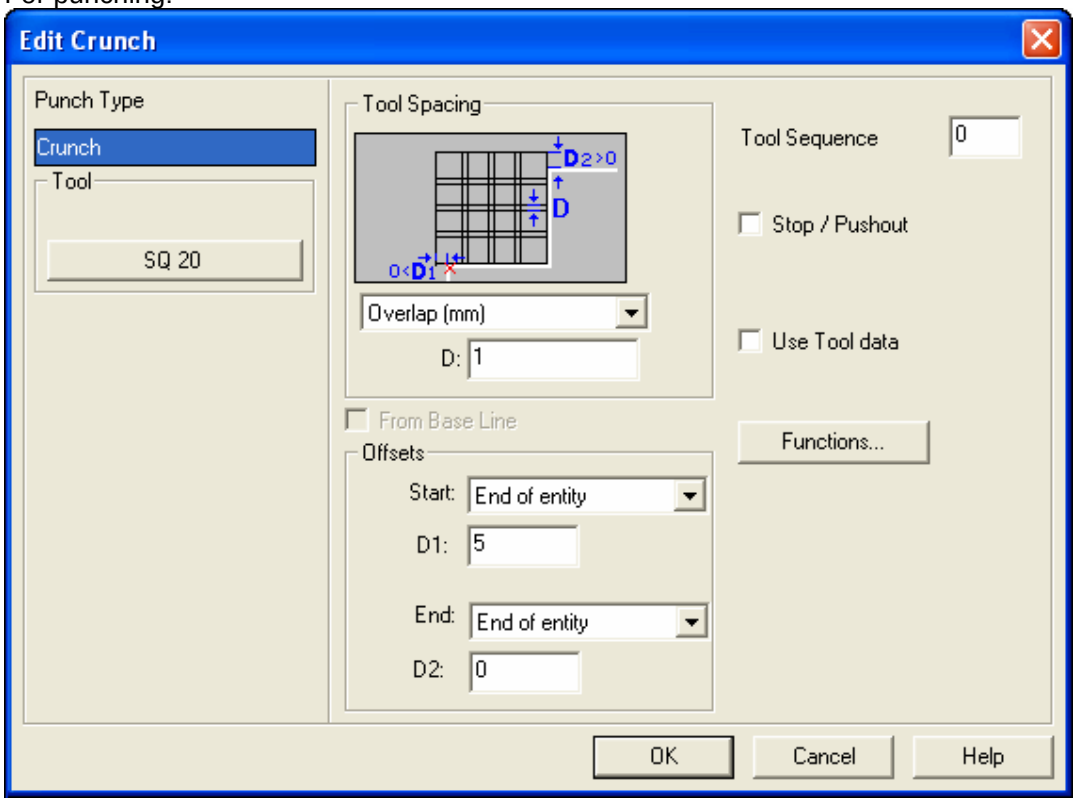
Here the user will have the possibility of assigning one new tool for all the selection. Choose the new tool, and confirm your choice clicking **OK** button. Then, on the part, click all the CAMs you want to apply this tool to.

2.3 Edit CAMs

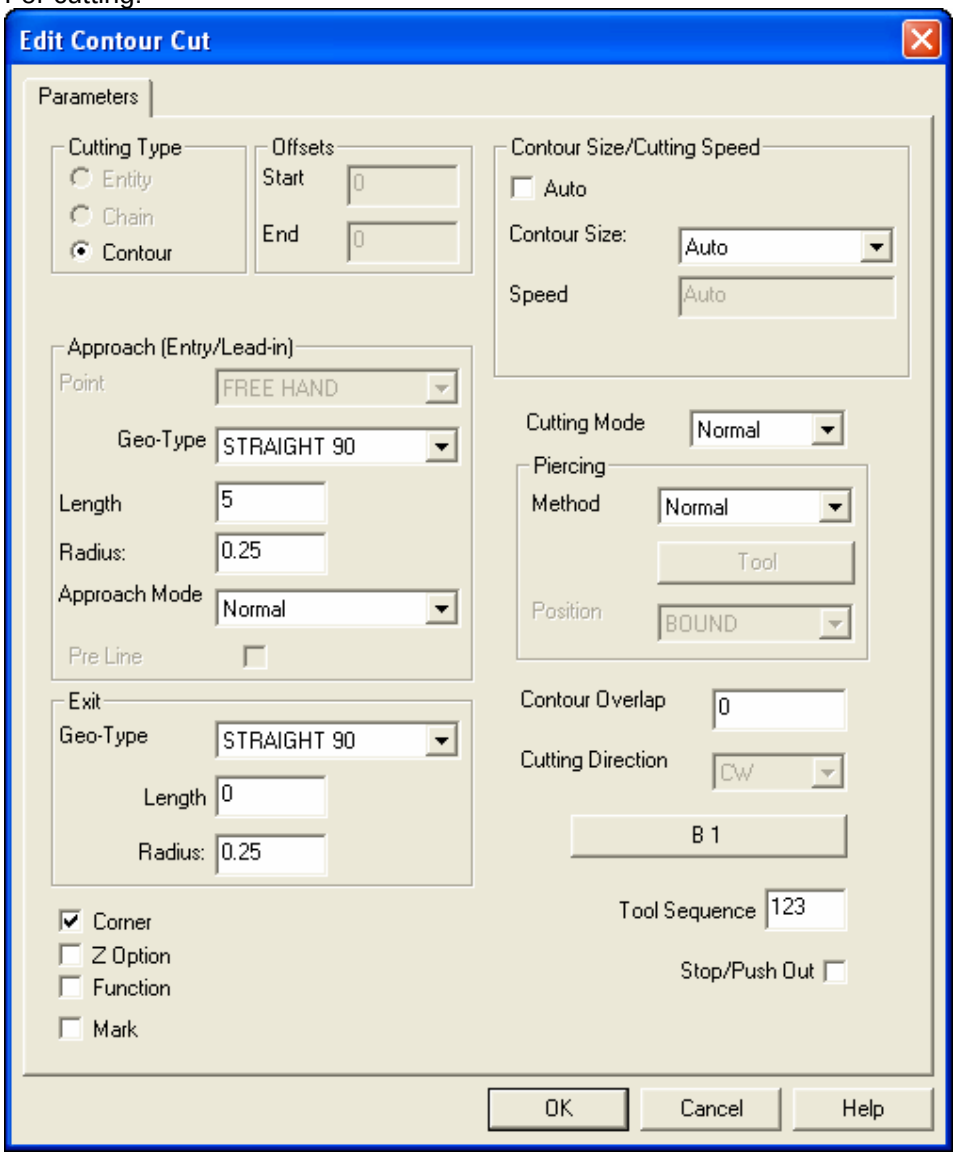
This option allows you to edit the tooling on an already processed entity.

After you choose this feature from the **CAM** menu, click the CAM you wish to edit. A dialog, presenting its settings, will open:

- For punching:



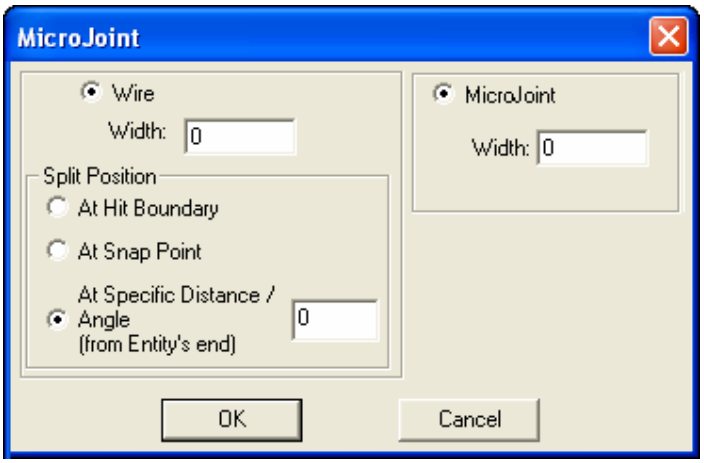
- For cutting:



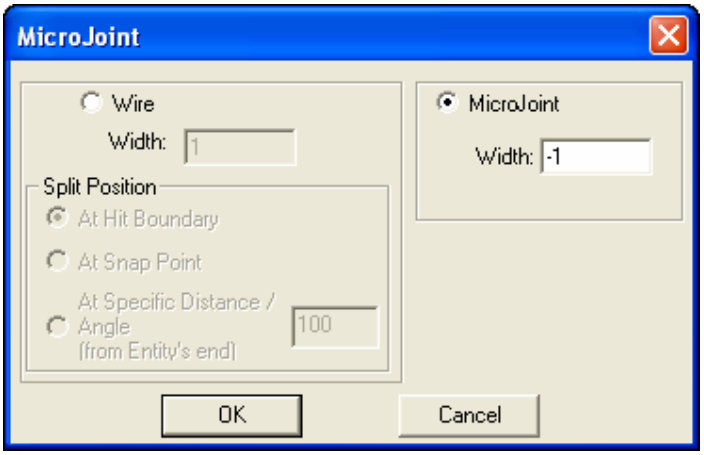
As you can see, not all of the CAM parameters are open for editing – you cannot change, respectively, the **Punch**, the **Cutting Types**, the **Offsets**, **Cutting Direction** and a few more settings.

2.4 Add MicroJoint

This option is also accessible by clicking the  button from the **CAM Toolbar**.
Micro Joints are small bridges of material that are left to make connection between different parts on a sheet, in order to prevent them from falling into the machine.
There are two options for adding a **MicroJoint**:

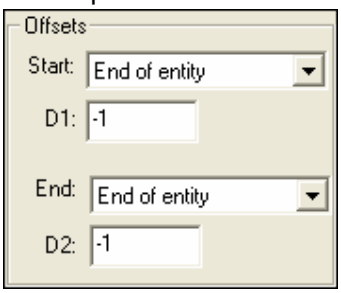


2.4.1 MicroJoint



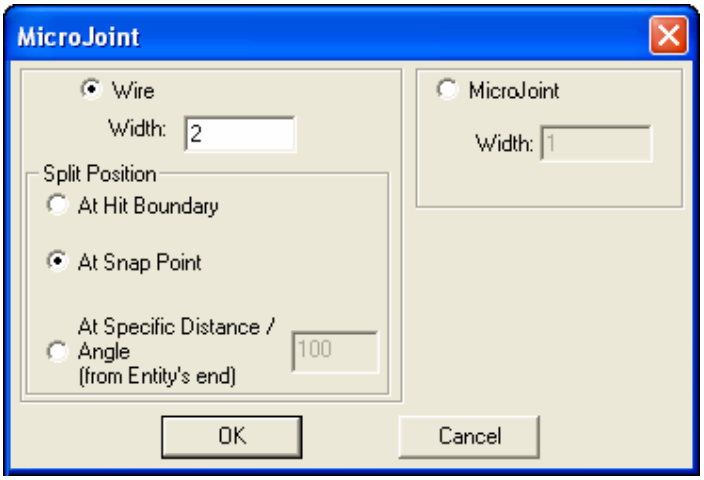
This option allows adding MicroJoints at the ends of entities. The parameter entered here is in fact an equivalent of the **Offsets** field of the **Add Punch** dialog. In other words – using this option is in fact editing the punch's offsets.

The following example shows an entity to which one's end a MicroJoint had been added (with the setting shown above). The **Edit CAM** dialog shows the resulting **Offset** parameters:



As with the Offsets, entering a negative number will create a Joint, while a positive one will result in an overlap.

2.4.2 Wire

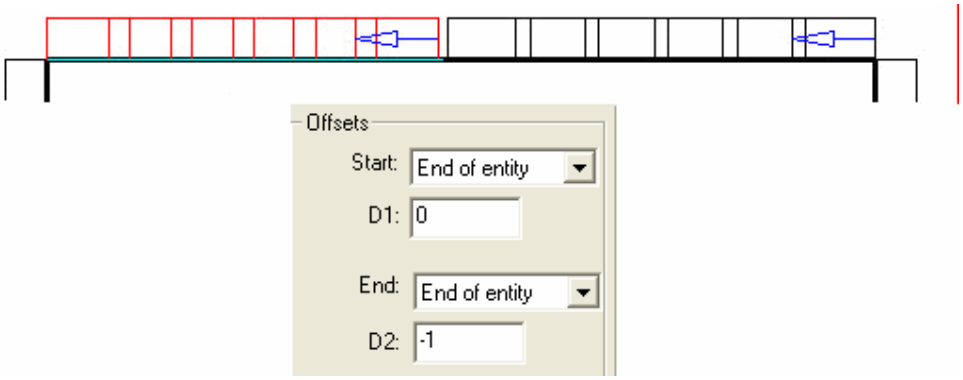


This option also lets you add joints, but here you can create a joint in the middle of an entity. This is done by splitting entity into two parts with a Joint between them, and CAMing them individually.

The **Split Position** setting determines the positioning of the Joint:

- **At Hit Boundary** – will place the Wire at the nearest Hit Boundary to the clicking position.
- **At Snap Point** – will position the Wire according to the current **Snap** setting.
- **At Specific Distance\Angle** –
  - **For Line entities** – will treat this parameter as a distance, and place the Wire accordingly.
  - **For Arc entities** – will treat this parameter as an angle, and place the Wire accordingly.

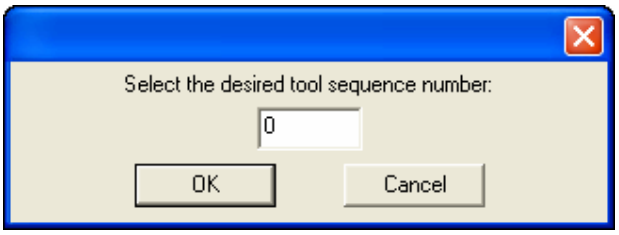
The following Wire example was created with the setting shown above, that is – **Width** value equals 2, **At Snap Point**, with Snap = Center. As you can see, the entity is being punched with two distinct punches. Using **Edit CAM** option, you will be able to see the resulting **Offsets** parameters:



2.5 Set Tool Sequence

This option is very useful when on the part there were placed a few CAMs with the same tool, but you don't want all the CAMs to be made in one sequence (for example, if for each of the CAMs you want to apply different Optimization strategies). In this case for one group of CAMs with the same tool you can set a certain *Sequence* and for the other group you can set another *Sequence*.

When you choose this option, a following dialog will appear:



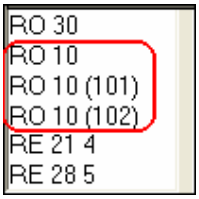
Enter **desired tool sequence number** and confirm your choice clicking **OK** button. When the dialog will close, click on the project the instance of the tool you want to apply this sequence to, or, with the help of **Select Option Toolbar**, assign it to several different tools at a time.

The default sequence number is "**0**", and it denotes the main tool.

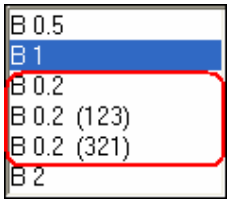
From now on these CAMs will be considered as if they were made with different tools – you can apply to them different strategies, place them in different positions in the general CAMing sequence, etc.

In the list of used tools, which can be found on the panel on the left side of the screen, you will see the tools to which sequence was added, behaving as if they were different tools:

For **punch**:



For **laser**:



You can also add sequence to the tool in **Add Punch** dialog, **Add Cut** dialog and in the **Used Tools** dialog, in the **Sequence** column.

Let's presume that a certain part is being CAMed with three tools. The following picture shows an area of **Used Tools** dialog, containing these tools:

- For punch:

Stations	Lock station	Current Tool	Lock Die	Die	Seq	Grouped	Auto Index	Hits	Tool Opt	Optimize Path	Minimize Rotation	Tool Subroutine	Functions
206	☐	RE 50 5 90	☐	0.2				5	☑				
104	☐	RO 30	☐	0.2				1	☑				
102	☐	SQ 20	☐	0.2				4	☑				

- For laser:

Stations	Lock station	Current Tool	Lock Die	Die	Seq	Grouped	Auto Index	Hits	Tool Opt	Optimize Path	Minimize Rotation	Tool Subroutine	Functions	Tool ID
9999	☐	B 0.5						1						
9999	☐	B 1						5						
9999	☐	B 0.2						9						
9999	☐	B 2						2						

As a result of optimization considerations, the CAMs have been split up into different Sequences. See the below pictures:

- For punch:

Stations	Lock station	Current Tool	Lock Die	Die	Seq	Grouped	Auto Index	Hits	Tool Opt	Optimize Path	Minimize Rotation	Tool Subroutine	Functions
206	☐	RE 50 5 90	☐	0.2				3	☑				
206	☐	RE 50 5 90	☐	0.2	123			2	☑				
104	☐	RO 30	☐	0.2				1	☑				
102	☐	SQ 20	☐	0.2				2	☑				
102	☐	SQ 20	☐	0.2	103			1	☑				
102	☐	SQ 20	☐	0.2	100			1	☑				

As you can see above, the tools with sequence assigned to it have the same station as the tools without it, since machine controllers see them as just one tool.

- For laser:

Stations	Lock station	Current Tool	Lock Die	Die	Seq	Grouped	Auto Index	Hits	Tool Opt	Optimize Path	Minimize Rotation	Tool Subroutine	Functions	Tool ID
9999	☐	B 0.5						1						
9999	☐	B 1						2						
9999	☐	B 0.2						3						
9999	☐	B 0.2			123			4						
9999	☐	B 0.2			321			2						
9999	☐	B 2						2						
9999	☐	B 1			456			1						
9999	☐	B 1			654			2						

2.6 Common Cuts

If you are going to use multiple parts on a large sheet, you will probably wish to save time and utilize the sheet as much as possible. This can be done with the usage of this option. It allows you to process outer contours of two adjoining parts at a time, even when they are processed by special tools.

After choosing this option, the following dialog will appear:

Choose the desired direction of Common Cuts – they can be either **Vertical**, **Horizontal**, **None** or both directions, and confirm your choice clicking **OK** button.

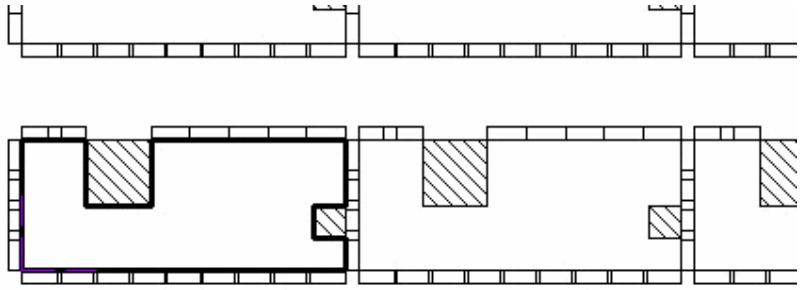
For **Trumf** machines choose **Use Bottom** for horizontal cuts.

For punching (turret) machines it is recommended to choose **Use Top** for horizontal cuts, since if the machine has a chute, the processing of a part should finish on the bottom of the part.

For information on laser common cuts go to: **Laser Common Cuts** section of **Appendix** chapter of current manual.

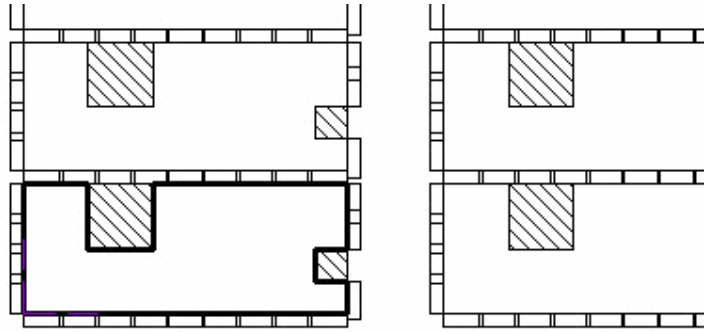
2.6.1 Vertical

Use **Common Cuts** to process together parts adjoining in the **Y** direction. Here you can see an example of utilizing the **Use Left** option:



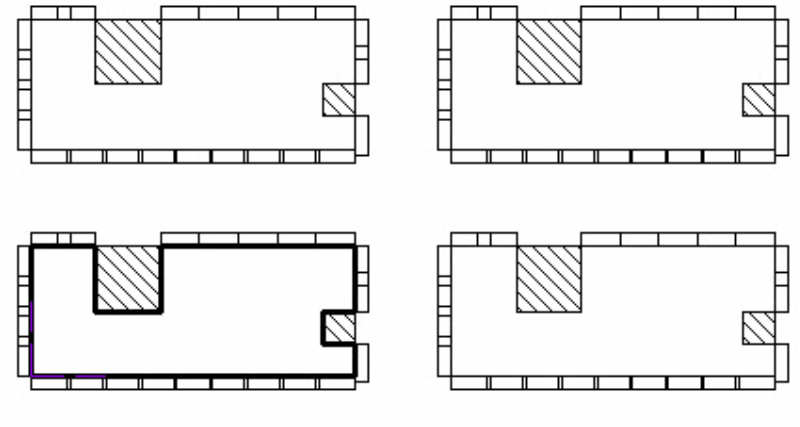
2.6.2 Horizontal

Use **Common Cuts** to process together parts adjoining in the **X** direction. Here you can see an example of utilizing the **Use Bottom** option:

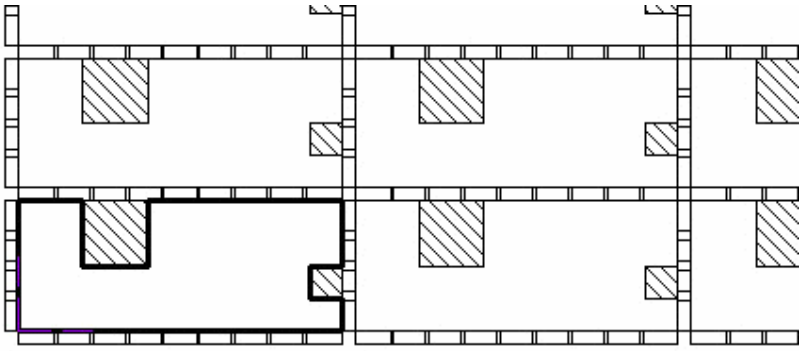


2.6.3 Common Cuts Example

Naturally, you can choose **None** for both directions:

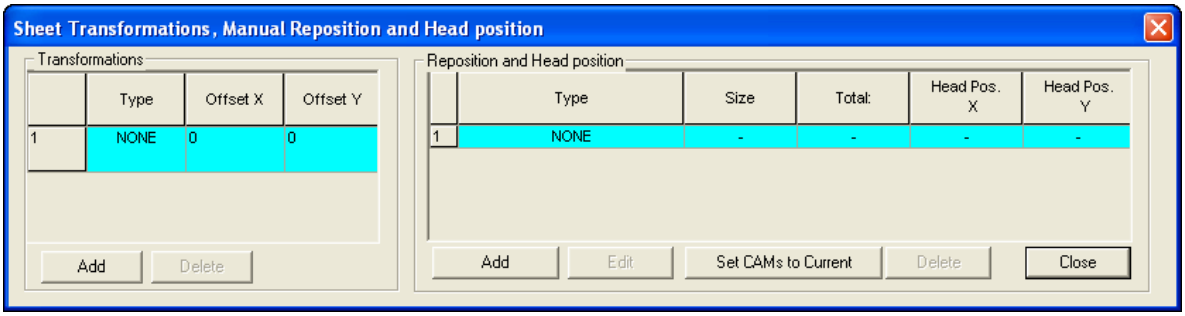


Or you can use **both** directions:



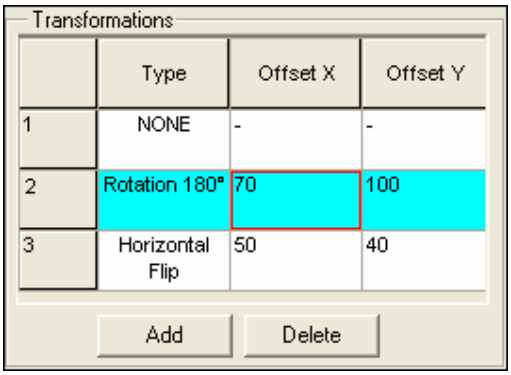
2.7 Reposition and Transformation

After selecting this option, the following dialog box is displayed:

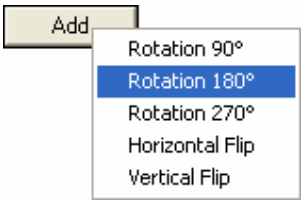


2.7.1 Transformations

In this section you can not only select the **Type** of transformation, but also for each of them you will be able to enter the values of the **Offsets** from the sheets' edges in **X** and **Y** directions.



In this section of the dialog you can **Add** the following options of Sheet Transformations:



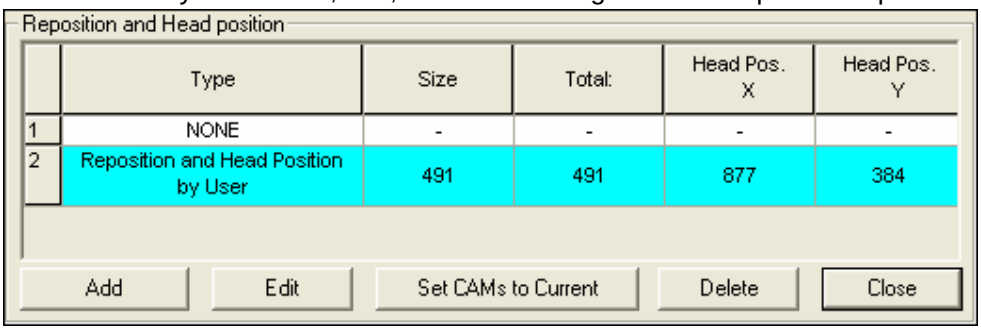
After adding a Transformation type, you can also **Delete** it, however you cannot delete the Sheet's initial state - **Transformation None**.

In order to set the offsets in X and Y directions, click th appropriate field and enter desired value.

For information about using Repositions with Transformations, see the **Sheet Transformation with Reposition** section of Appendices chapter in this manual.

2.7.2 Reposition and Head position

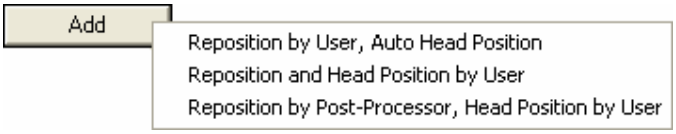
In this section you can add, edit, delete and assign CAMs to specific Repositions:



The window in this section shows which Repositions are defined for the Transformation selected in the **Transformations** section.

2.7.2.1 Add

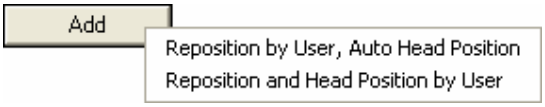
You can add Repositions by clicking the **Add** button; you will be presented with the following menu:



The options for manual determining the Head position are intended for those occasions when prior punches have been placed where the Post Processor might have positioned the Head.

After you have set a **user defined** Reposition, initiating the Repositions by Post-Processor is no longer possible!

This means that when you wish to add more Repositions to your part, you will only be offered the manual options:



2.7.2.2 Edit

This option enables you to edit pre-defined Repositions.

2.7.2.3 Set CAMs to Current

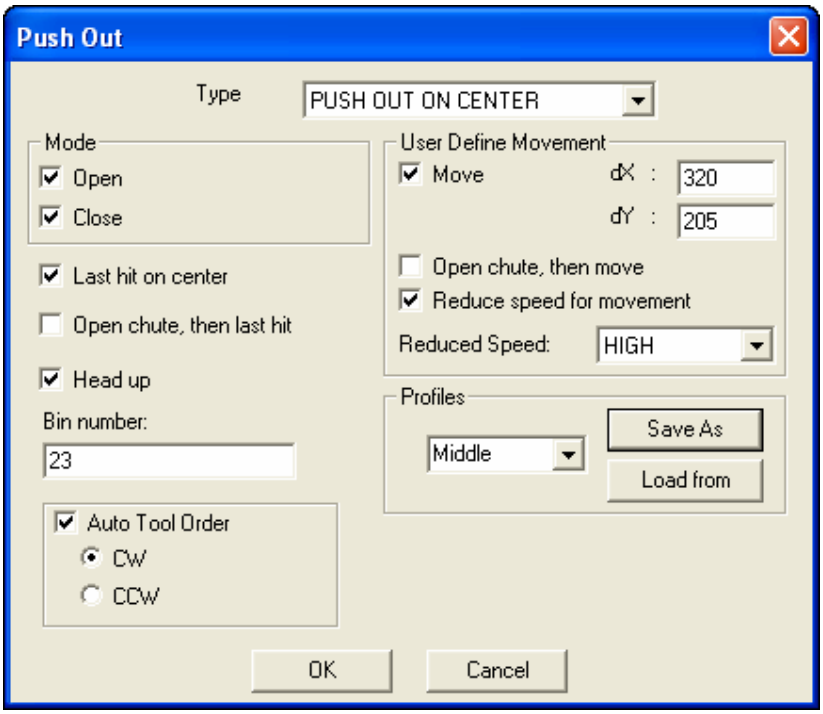
This option enables you to associate specific CAM(s) to a specific Reposition.

After clicking this button and using the **selection** options, from the Selection Toolbar, you can select the CAMs you want to associate with the particular Reposition.

2.8 Push Out (Chute)

This function is designated for machines that have a chute on the table – parts that have been cut out of the sheet can be removed from the table via this chute. This avoids having to use a **Stop Machine** command and remove the part manually. The position of the chute varies from machine to machine.

When you choose this option, the following dialog appears:



After setting desired parameters, confirm your choice clicking **OK** button. The dialog will close and you will be able to add this attribute to an existing CAM. All you have to do is click the punch you want to **Push Out** through a Chute. It will be marked with green color.

You can also use this option to place a **Stop Machine** command.

2.8.1 Chute Type

From this dropdown list you will be able to select the position of the chute in relation to the punching/cutting head on the machine:

- **Stop** – choosing this command equals applying the **Add or Remove Stop Machine** option from **CAM menu**.
- **Push Out on Center** – if the machine has a chute in the middle of the table, choosing this option will allow you to push out the processed part or the remnant (depending on your preferences) through this chute to the bin.
- **Push Out to Left** – if your machine has a chute on the left side of the table, choosing this option will give machine a command to use it.
- **Push Out to Right** – if your machine has a chute on the right side of the table, choosing this option will give machine a command to use it.
- **No Movement Push Out** – selecting this option cause only the part which is above the chute to fall through it to the bin. Other parts won't be moved by machine towards the chute.
- **Fixed Push Out** – this option is designated for machines that have a fixed chute, with no possibility of opening or closing the chute door.
- **Remove Push Out** – choosing this option you resign from using an already set **Push Out** function.

2.8.2 Push Out Modes

The chute has two modes: **Open** and **Close**. They are designated especially for **Trumpf** machines, which have an option of opening the chute at the beginning of the process and closing it afterwards.

2.8.3 Last Hit On Center

Choosing this option you decide to place the last hit on center of processed entity.

2.8.4 Open Chute, then Last Hit

Marking this checkbox tells the machine to open the chute first and then place the last hit on the processed entity.

2.8.5 Head up

If you want the turret head to be raised during pushing the part out, select this option.

2.8.6 Bin Number

Here you enter the number of the bin/palette to which the pushed out part will fall into.

2.8.7 Auto Tool Order

Marking this checkbox allows you setting automatic tool order according or opposite to the hands of the clock (**CW** or **CCW**), by selecting the appropriate option.

2.8.8 User Defined Movement

In this section you will be able of defining how the machine should move the part in order to push it out through a chute to the bin.

2.8.8.1 Move

If you mark this checkbox, you will be able to enter values, in the **dX** and **dY** fields, of the point to which the machine should move.

2.8.8.2 Open Chute, then Move

Checking this option you make machine open the chute first and then move the part.

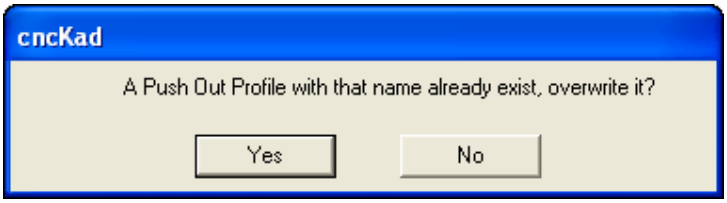
2.8.8.3 Reduce Speed for Movement

When you mark this checkbox the **Reduced Speed** option becomes available for editing. From the **Reduced Speed** dropdown list you will be able to choose the most fitting speed for your movement: **Low**, **Medium** or **High**.

2.8.9 Push Out Profiles

This option is useful for machines with a few chutes. You can create and store the profile with settings for each chute and load it, when needed, without having to assign all the definitions form the beginning every time you make a Push Out.

After configuring definitions for specific push out type, go to **Profile** section. In the dropdown list field type the name for the profile you want to create and click **Save As** button. If such a name already exists, you will be presented with following warning:



If you want to overwrite (update) the existing profile, click **Yes** button; otherwise click **No**.

Profiles are stored separately for each machine you have. You can't use one machine's profiles on another one (you have to create a new profile), but you **can** give profiles of different machines the same name.

When you assign a **Push Out** to a CAM, this action will be executed after the CAM is performed.

A CAM defined as a **Push Out** will be colored green.

For loading an existing profile configuration, from the dropdown menu choose the name and click the **Load from** button.

2.9 Add or Remove Stop Machine

This function adds or removes a **Stop Machine** from a specific CAM. To do this, after choosing this option simply click the CAM you want to apply the **Stop** to. To remove the **Stop** from it, click again on the CAM.

Using this option is the same as choosing a **Stop** type of the **Push Out**.

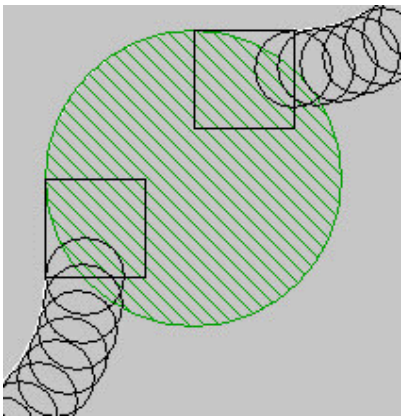
A punch with assigned **Stop Machine** function to it will be colored red.

2.10 Positioning (No CAM)

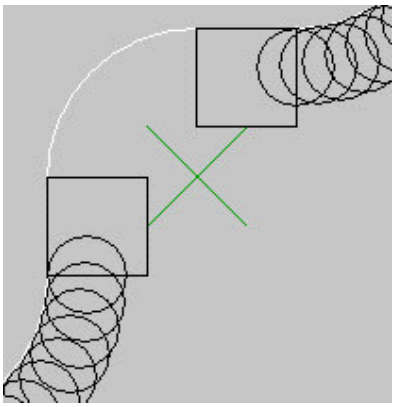
A **No CAM** means that the machine will go to the coordinates of the chosen CAM, but the selected position will **not** be processed, when the program is run.

To apply this feature click the CAM you want to change. The processing will disappear and an **X** will be located in its place (the color of the **X** will depend on the nature of the CAM it replaced, e.g. a **Push Out** CAM colored green will become a green mark, a **Stop Machine** CAM colored red will become a red mark, etc.):

Before:



After:

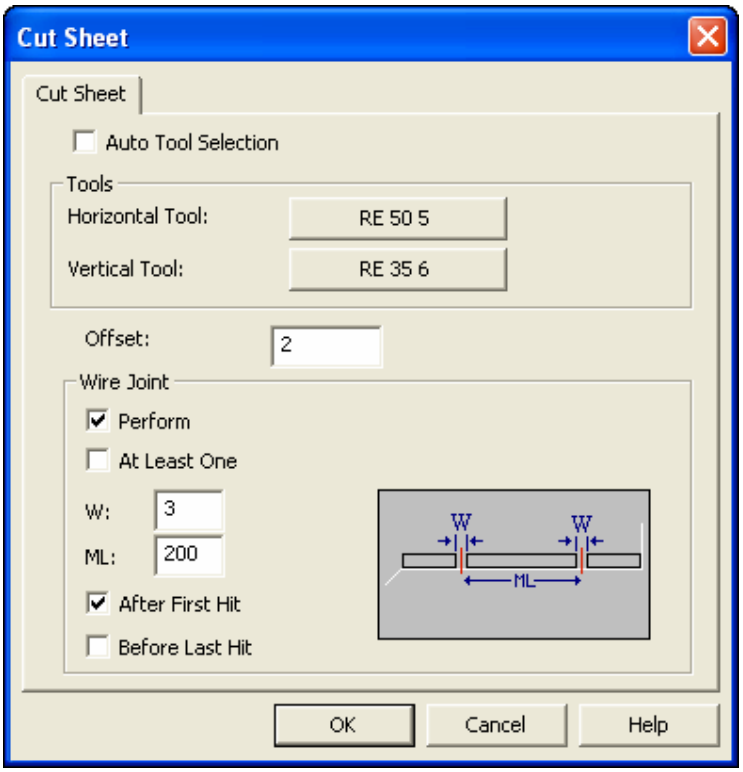


This command is useful for forcing the machine to traverse a selected path even when there are no CAMs in that path, for example when you want to avoid a clamp.

A **No CAM** can be converted back into a CAM by clicking the same position again.

2.11 Cut Sheet

This feature enables you to cut your sheet to a predefined size. After selecting this option, the following dialog will appear:



After setting the **Tools** and **Wire Joint** click **OK** button to close the dialog. Click one side of the sheet, and then click another one. As you now move the cursor, you will see a line under its crosshairs – this is the *Cutting Line*, and it will be drawn perpendicular to both sides chosen. When you click the mouse, the sheet will be cut in the defined position:

- If you have chosen *opposing* sides of the sheet, the cutting line will be drawn up-down *or* left-right. (see example below).
- If you have chosen *un-opposing* sides, e.g. left and top, two intersecting lines will be drawn (see example below):
  - The first one, left-right or top-bottom (depending where you clicked first), will cut the sheet from side to side.
  - The second one will be placed from the side of your second click to the first line.

Apart from the direction of the cut, advanced programmers can set a value for the offset from the sheet's edge.

2.11.1 Tools

Marking the **Auto Tool Selection** checkbox you can let ***cncKad*** select appropriate tools for **Cut Sheet** operation or you can decide about it yourself by clicking on **horizontal** and **vertical** tool buttons. The tool library will open allowing you to choose the tools.

2.11.2 Offset

The Cut Sheet **Offset** can be defined according to your processing needs – by entering the value in the **Offset** field or by entering the value in the machine's MDL file, in the `[Sheet Cutting]` section, e.g.:

```
Cutting Offset = 5.0 // To be used for CutSheet command
```

The value can be:

- **Positive** – the CAM (punch or laser cut) will be offset beyond the edge of the sheet. In this case a **Stop Machine** feature will be added after this CAM.
- **Negative** – the CAM (punch or laser cut) will be offset into the sheet, leaving a **MicroJoint**. In this case a Stop Machine feature will **NOT** be added.

2.11.3 Wire Joint

This option is available only for punching machines.

If in this section you will check **Perform** box you will be allowed to place Micro Joints (*Wires*, actually) as an integral part of the punch. You can set the **width** of the Micro Joints (**W**) as well as the **maximum distance** between them (**ML**), and they will be integrated into the punch, without splitting the entity being punched.

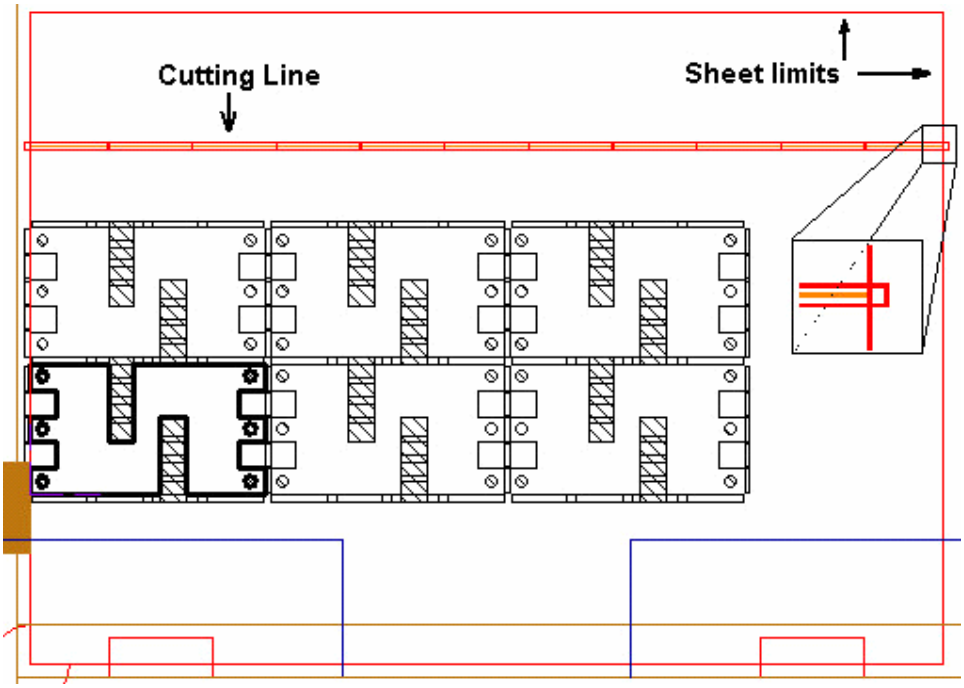
Choosing one of the **After First Hit** and **Before Last Hit** options enables creating Micro Joints after the first and before the last punch. They address the issue of rounded corners when punched by round tools.

More information on this option you will find in the **Wire Joint** section of the Add Punch dialog of Punch Manual chapter of ***Punch Manual***.

2.11.4 Cut Sheet Examples

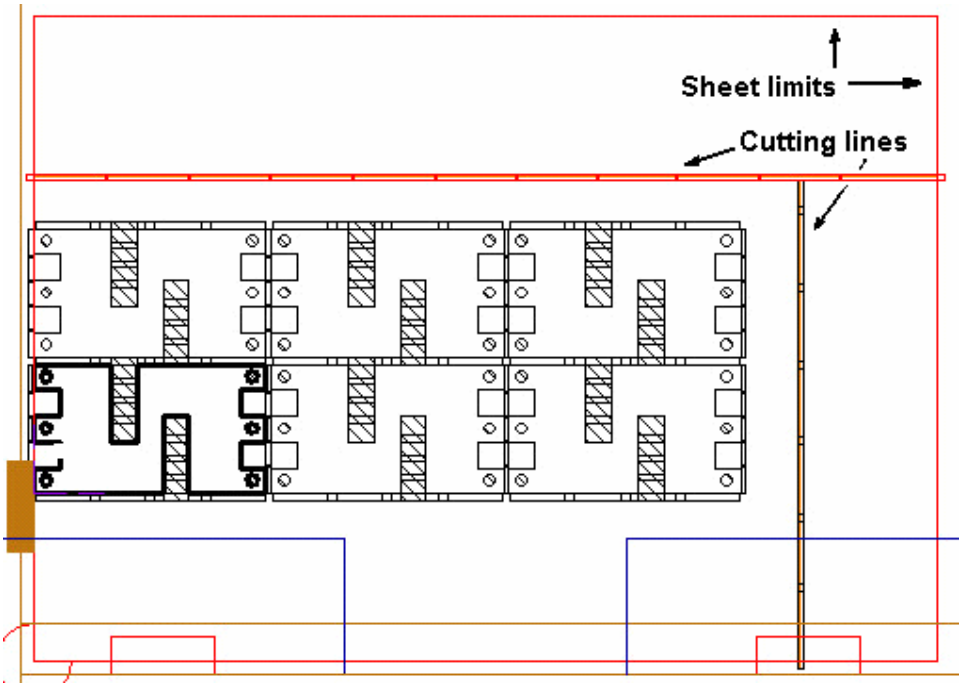
See below:

2.11.4.1 Cut Sheet Left Right



In this example the punching is colored red to denote a **Stop Machine**, as the **Offset** is 5mm over the sheet's edge.

2.11.4.2 Cut Sheet Left Down

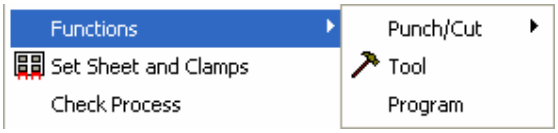


In this example the horizontal punching is colored red to denote a **Stop Machine**, as the **Offset** is 5mm over the sheet's edge.
However, **Stop Machine** option wasn't assigned to vertical punching and at this time there is no need doing this.

2.12 Defining Functions

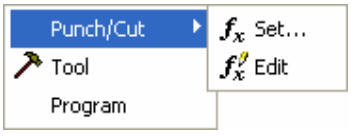
Functions are special machine related commands that can be inserted into NC codes at specific positions. The codes and the quantity of functions available vary from machine to machine. For instance, an older mechanical punching machine will have fewer abilities than a newer hydraulic machine.

These special commands can be associated with specific CAMs, tools or programs:



2.12.1 Punch or Cut Functions

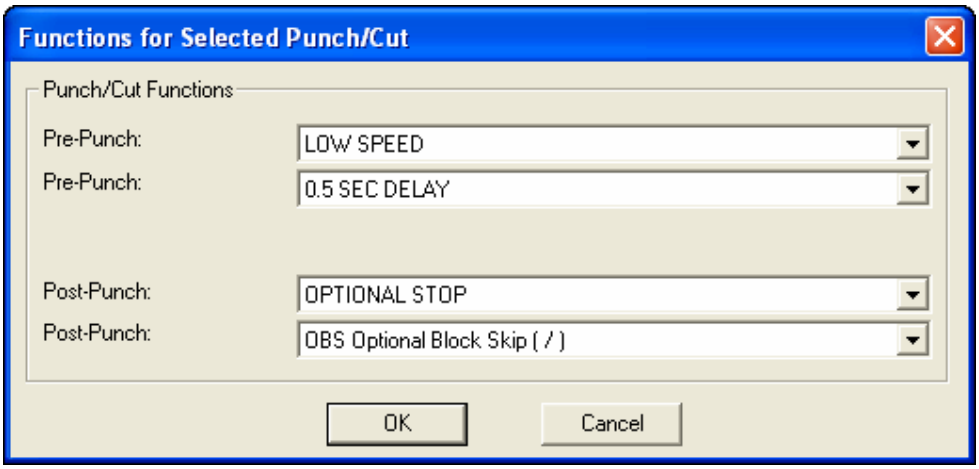
These options allow for adding functions immediately before or after a CAM:



2.12.1.1 Set CAM Functions

This option allows you to set function/s for various CAMs.

After choosing this option, the following dialog will open:



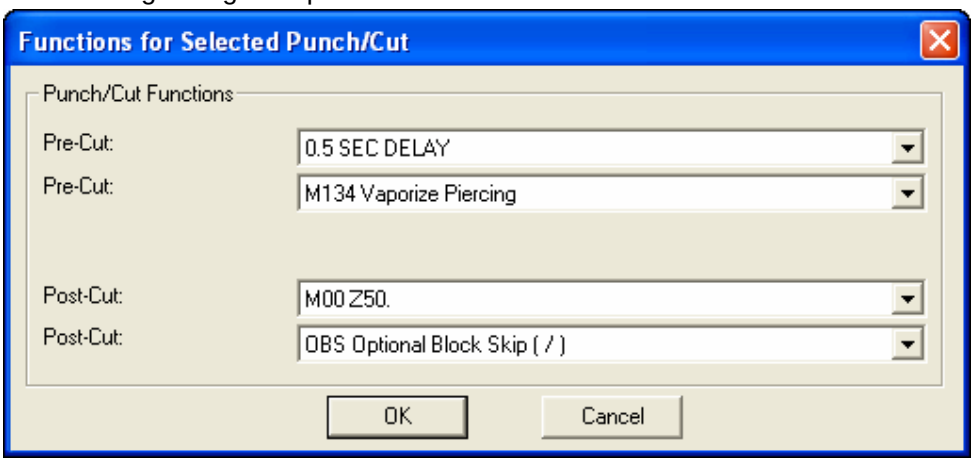
Define the Pre- or Post-CAM functions you desire, and click **OK** button.

You will be returned to the drafting window, where you can click all the CAMs you want these functions to apply to.

2.12.1.2 Edit CAM Functions

This option allows you to edit already existing functions for a specific CAM, or set new ones if none are defined.

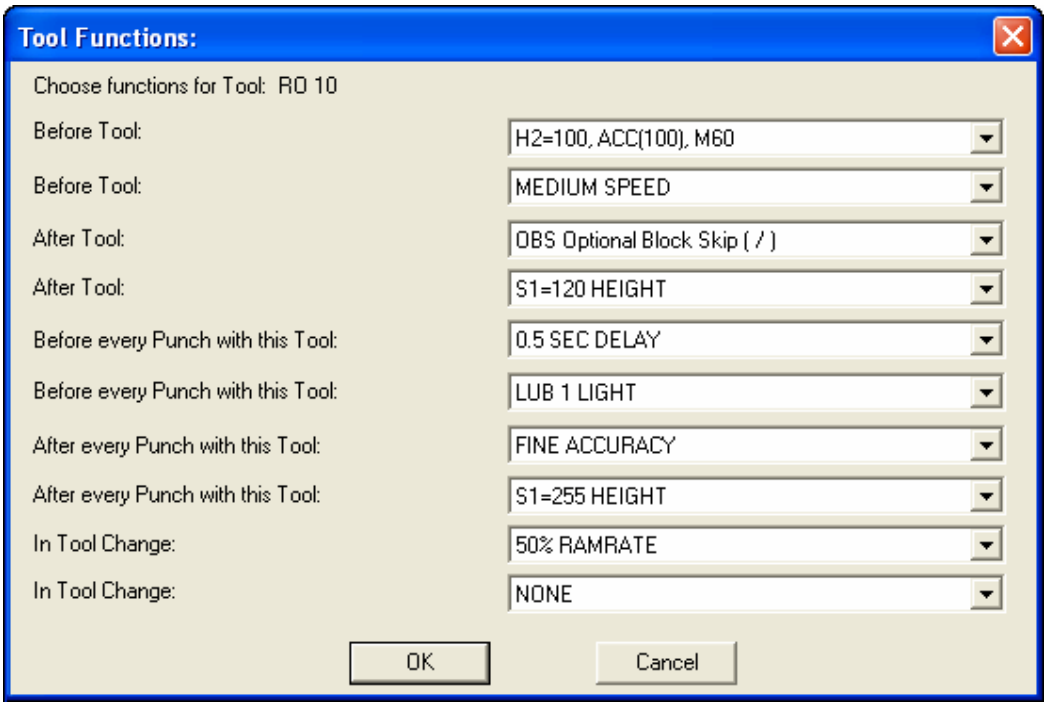
After clicking this option, you will be moved to the drafting window, where you can click the CAM you want to assign the selected functions to. After choosing a CAM, the following dialog will open:



Define the Pre- or Post-CAM functions you desire, and click **OK** button.

2.12.2 Defining Tool Functions

This option is accessible also by clicking the  icon from the Common Toolbar and loads the **Used Tools** dialog, allowing you for adding functions to a specific tool. In the dialog, from the tools list, select the tool that you wish to add the functions to, and click the **Tool Functions...** button; the following dialog will be displayed:



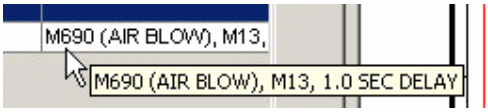
Clicking the dropdown lists displays the functions that can be added to the program (they vary for machines). Select the ones you want to add – you can add two functions for each of the following options:

- **Before Tool** – Before loading the tool.
- **After Tool** – After putting the tool back.
- **Before every Punch with this Tool**
- **After every Punch with this Tool**
- **In Tool Change** – When changing one tool for another one.

After defining the functions confirm your choice clicking **OK** button.

If you don't want to assign any function to any of above mentioned fields, select from the dropdown list the option **NONE**.

The selected functions will be shown in the Used Tools dialog, in given tool's line:

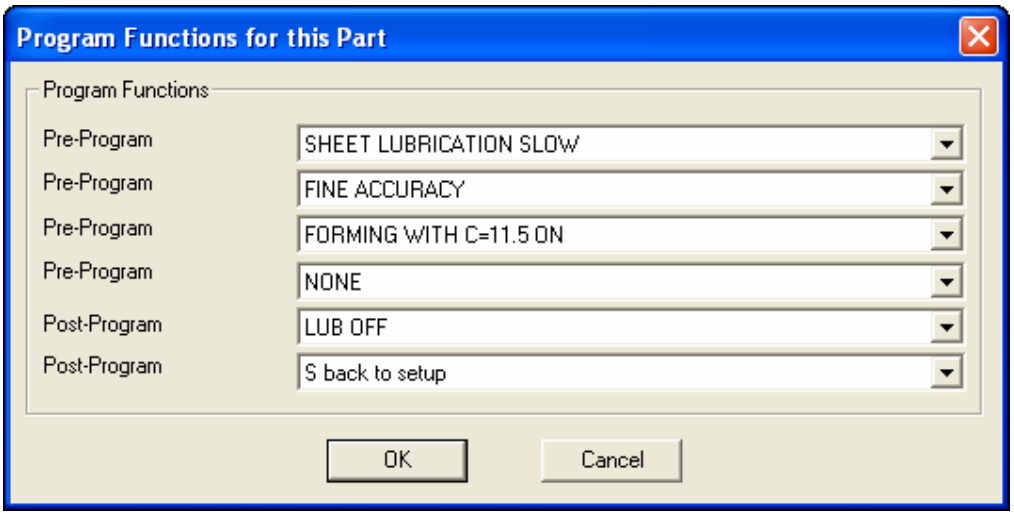


It is possible to add default functions to a tool so that every time that specific tool is used, in any program, it will always come with the defined functions.

2.12.3 Defining Program Functions

This command gives the user the option to place special machine commands at the beginning or at the end of the program.

After choosing this option you will be presented with the following dialog:



From this dialog you can define functions that will be activated before or after running NC program.

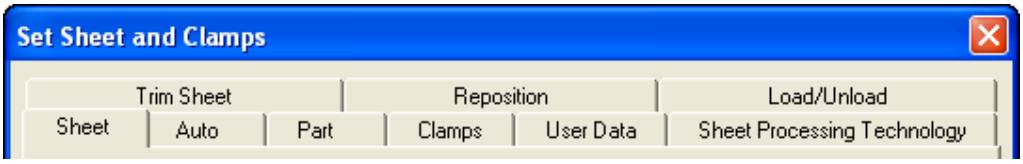
2.13 Set Sheet and Clamps

This dialog can also be accessed by pressing the **Set Sheet and Clamps** button –  from the **Common Toolbar** –  .

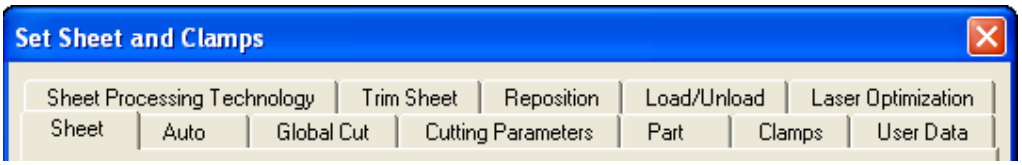
The parameters you define through this dialog determine how the part you created will be processed.

Click the tab you wish to view:

- For punching machines:



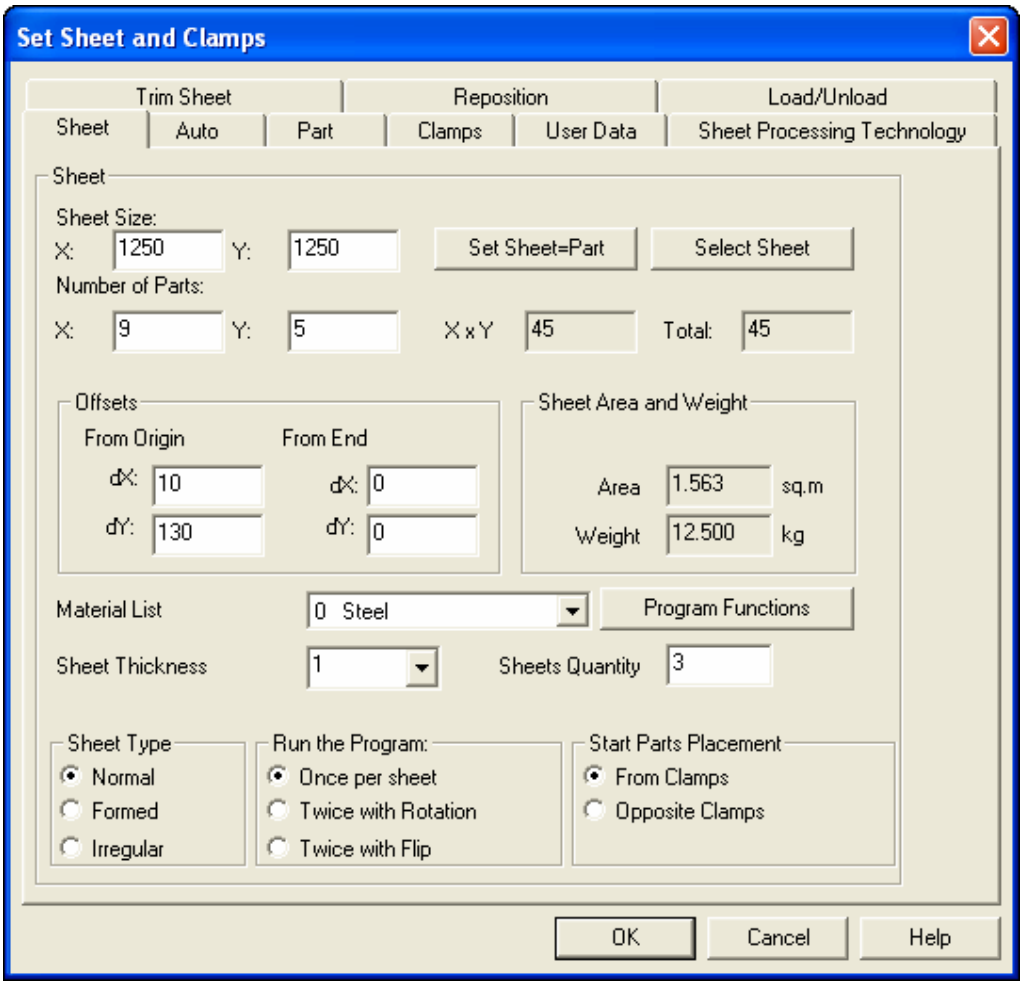
- For laser machines:



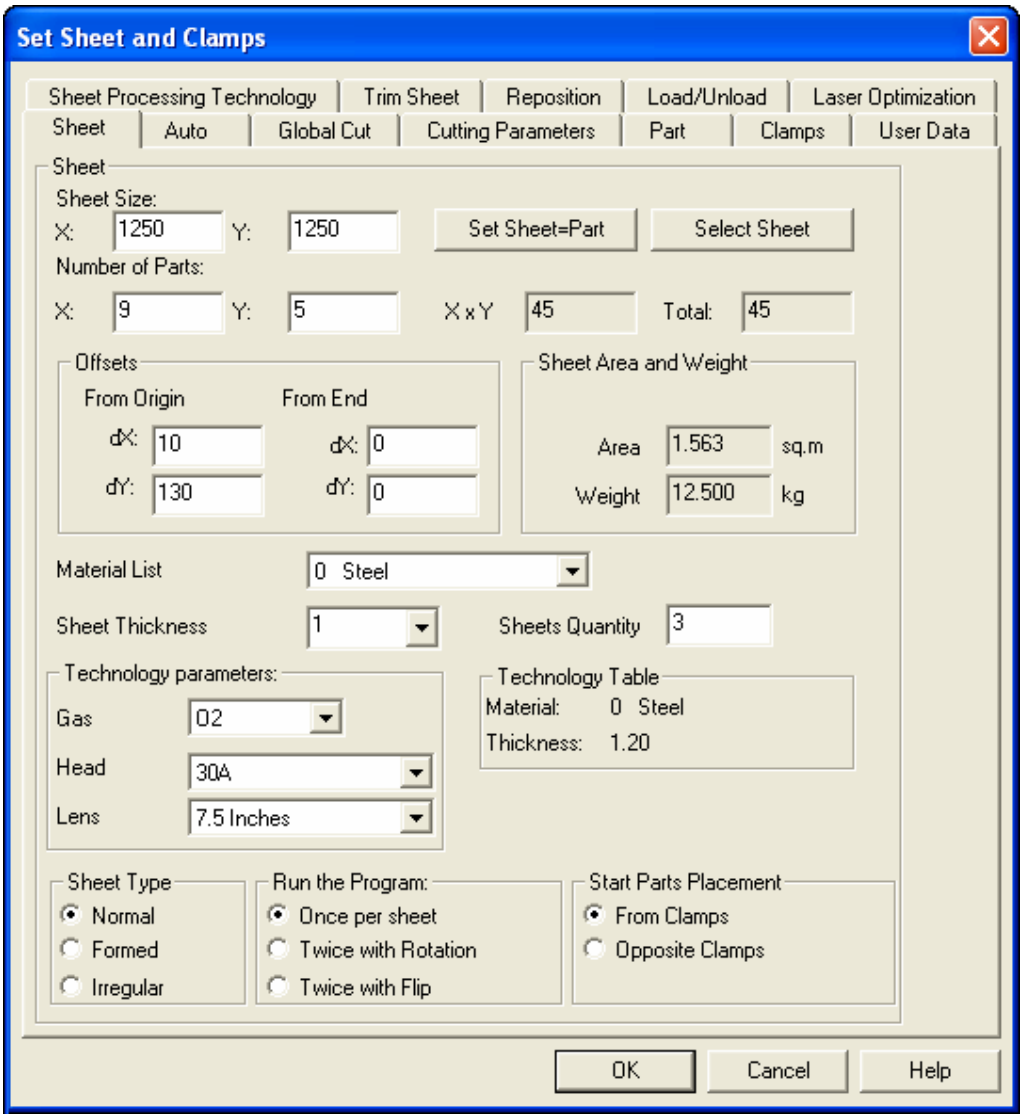
2.13.1 Sheet Tab

This tab varies for punch and laser machines. The parameters in this tab pertain to the Sheet you will be placing your part on.

For punch machines it looks like this:



and for laser machines it looks like this:



2.13.1.1 Sheet Size

In the **Sheet Size** fields you can view and manually define the **X** and **Y** sizes of the sheet.

2.13.1.2 Set Sheet = Part

Clicking the **Set Sheet = Part** button will cause the **Sheet** to become the same size as the **Part**. If you use this option, you should pay attention to the clamps' status.

2.13.1.3 Select Sheet

By clicking this button the following dialog will appear from which you will be able to select standard sized sheet:

The sheet sizes presented here are being taken from **Material Database**, which can be accessed from Settings menu **=>** Workspace Settings **=>** Material tab **=>** **Standard Sizes** button.

In the **Quantity** column you will see the number of specific sized sheets which are stored in Material Database.

2.13.1.4 Number of Parts

In the **X** and **Y** fields you will see the maximum number of parts (or actually – geometries) fitting each direction. These copies of the part are called **multiple parts**. Changing the sheet dimensions will automatically update this data.

In the **X x Y** fields, which are not available for editing, you will see the amount of the geometries (which sometimes consist of one and sometimes of more parts) fitting the currently chosen sheet size. Changing the sheet dimensions will automatically update this data.

In the Total field, also not available for editing, you will see the entire amount of the parts (**NOT** geometries, but all the parts in geometries) fitting the currently chosen sheet size.

Changing the sheet dimensions as well as manually updating the number of **Parts in Geometry** field on Part tab will automatically update data in this field.

You can set manually the number of parts on the sheet in two ways:

- Enter the desired number in the **X** and **Y** fields.
- Change the size of the sheet.

2.13.1.5 Offsets

The parameters entered in these fields define the distance by which the parts will be moved from the edges of the sheet.

2.13.1.6 Sheet Area and Weight

This section, not available for editing, presents the **area** of the sheet (in square meters) and its **weight** (in kilograms). This data is calculated on the basis of the sheet size, it's thickness, material type it is made of. The values presented here will be automatically updated any time you will change the **material**, **thickness** or the **size** of the sheet.

2.13.1.7 Material Choice

From the dropdown **Material List** you can choose or change the material for the sheet you are working on. The materials in this inventory are being taken from the **Sheet Database** of Material tab of Workspace Settings.

For more information on this subject go to the appropriate section of Settings Menu chapter of ***Drafting and Nesting Manual***.

2.13.1.8 Program Functions

This option is available just for punch machines.

Using this option you can add functions for current part that will be included to your NC program.
For a full explanation of this feature go to the Defining Program Functions section of this chapter.

2.13.1.9 Sheet Thickness Choice

From the dropdown **Sheet Thickness** list you can choose or change the thickness for the sheet you are working on. The thicknesses in this inventory is being taken from the **Sheet Database** of Material tab of Workspace Settings.
For more information on this subject go to the appropriate section of Settings Menu chapter of ***Drafting and Nesting Manual***.

2.13.1.10 Choosing Sheets Quantity

In this field you can enter the number of the sheets you want to process with this part.

2.13.1.11 Technology Parameters

This option is available just for laser machines.

The definitions of the options seen in **Technology Parameters** section, like **Gas**, **Head** and **Lens** type, are being taken from the **Cutting Technology Table** described in Settings Menu chapter of ***Drafting and Nesting Manual***.

2.13.1.12 Technology Table

This option is available just for laser machines.

The non-editable parameters presented in **Technology Table** section are taken from the **Cutting Technology Table** described in Settings Menu chapter of Drafting and Nesting Manual. Sometimes the thickness here may differ a bit from the one you chose in Thickness dropdown list. This means that your machine's Cutting Technology Table doesn't contain exactly the same the thickness your sheet has, and that's why it shows the value closest to it.

2.13.1.13 Sheet Type

Here the user can define a type of the sheet to be used. This parameter is important for your machine, that she would know when to increase the height of the head, if necessary.
The types of sheets are:

- **Normal** – this type designates regular, plain sheets.
- **Formed** – sheets that have been formed beforehand.
- **Irregular** – sheets with rough surface.

2.13.1.14 Run the program

This option defines how your sheet will be utilized. Some machine operators tend to process a half of the sheet, turn it over or rotate it and then continue processing the other half.
Select the most suitable option for your machine:

- **Once per Sheet** – the program will be run once, on the entire sheet.
- **Twice with Rotation** – the sheet will be split and rotated, therefore the program will be run twice.

- **Twice with Flip** – the sheet will be split and flipped, therefore the program will be run twice.

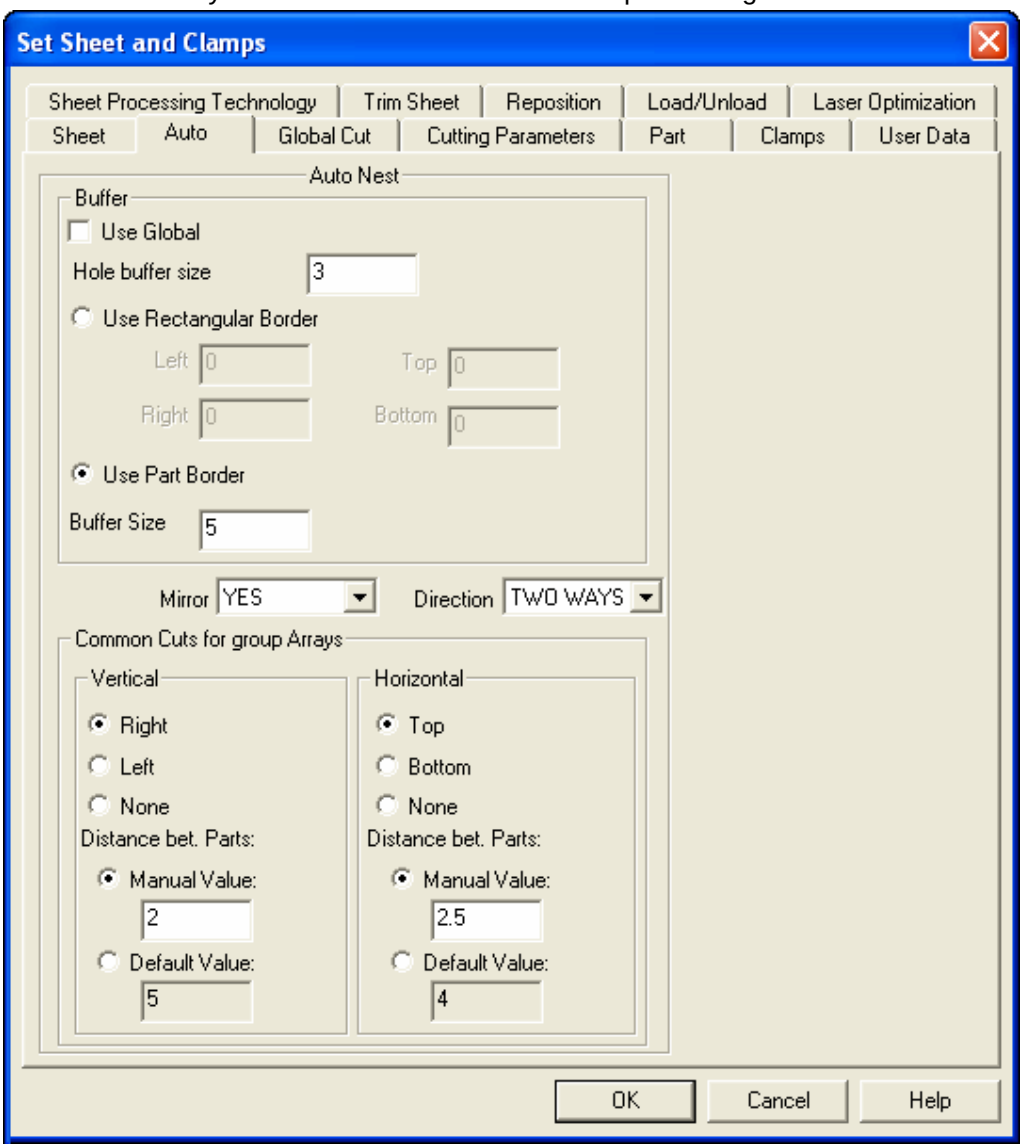
Your choice may be changed at any time during the NC program creation process.

2.13.1.15 Start Parts Placement

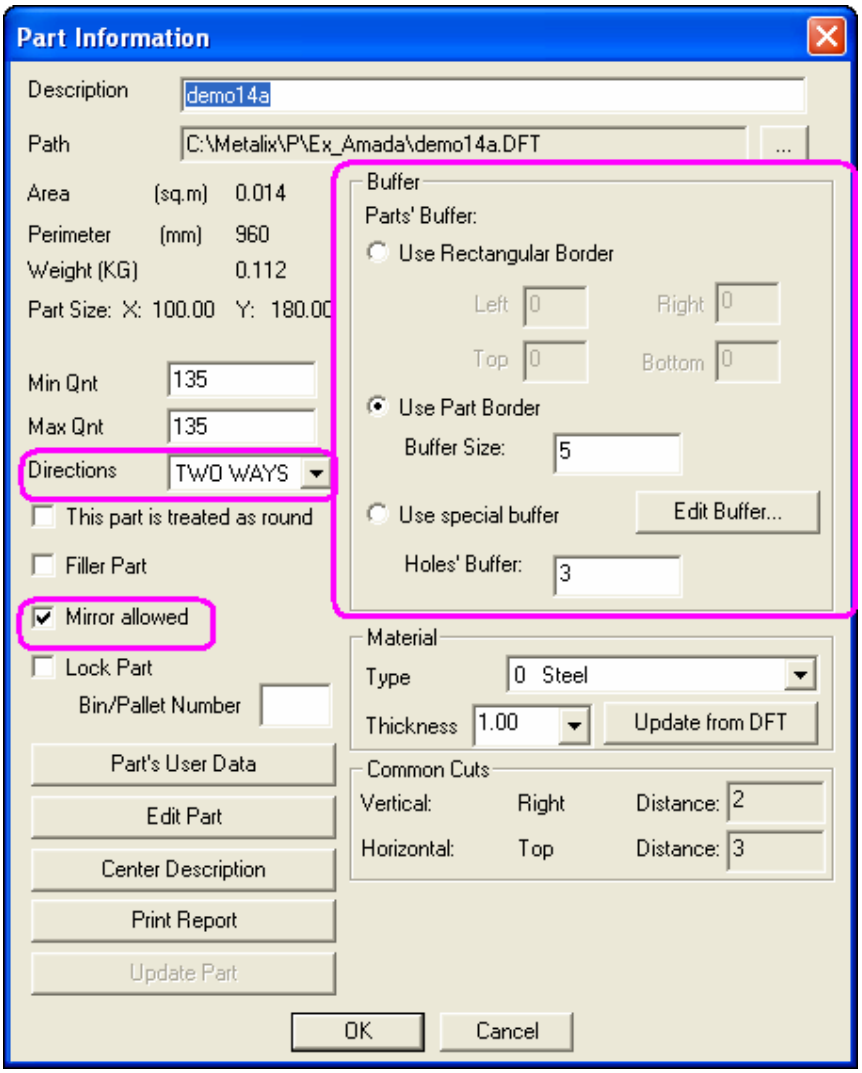
This section allows the user to determine from which side of the sheet the parts' placement will be initiated. You can choose one of two options, as their names suggest: **From Clamps** or **Opposite Clamps**.

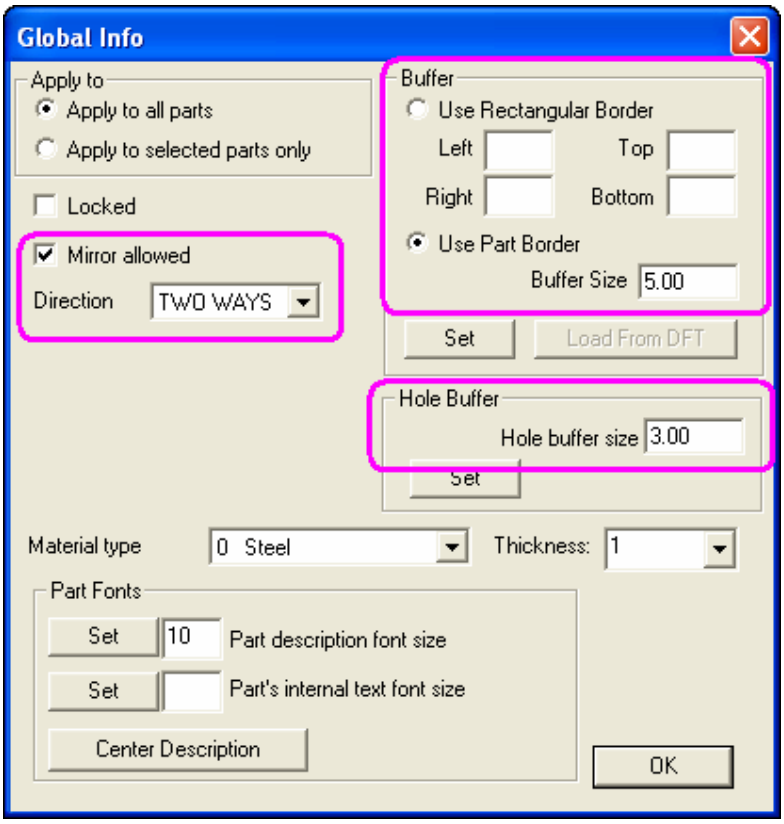
2.13.2 Auto Tab

All the options presented in this tab relate to the export of files from ***cncKad*** to the Automatic Nesting module – ***Auto Nest***. After setting here proper parameters ***cncKad*** will analyze contours and allow automatic processing to occur.



All the parameters set in this dialog are being defined for the current part you are working on. After exporting the part to ***Auto Nest*** you will be able to view and edit them in **Part Information** and in **Global Info** dialogs:


Part Information
Description
demo14a
Path
C:\Metalix\P\Ex_Amada\demo14a.DFT
...
Area (sq.m) 0.014
Perimeter (mm) 960
Weight (KG) 0.112
Part Size: X: 100.00 Y: 180.00
Min Qnt 135
Max Qnt 135
Directions TWO WAYS
This part is treated as round
Filler Part
Mirror allowed
Lock Part
Bin/Pallet Number
Part's User Data
Edit Part
Center Description
Print Report
Update Part
Buffer
Parts' Buffer:
Use Rectangular Border
Left 0
Right 0
Top 0
Bottom 0
Use Part Border
Buffer Size: 5
Use special buffer
Edit Buffer...
Holes' Buffer: 3
Material
Type 0 Steel
Thickness 1.00
Update from DFT
Common Cuts
Vertical: Right Distance: 2
Horizontal: Top Distance: 3
OK
Cancel



2.13.2.1 Buffer

Buffer is the area around a part that separates it from the other parts placed in the nest. There are a few options for setting parameters for buffer:

2.13.2.1.1 Use Global

If this option is checked in ***cncKad***, before exporting parts into ***Auto Nest***, the default buffering parameter will be taken; else the parameters you defined on the **Auto** tab will be used.

When importing a part, while being in ***Auto Nest***, you will have the option to set a default buffering parameters in the **Import Setup** dialog.

2.13.2.1.2 Hole buffer size

This parameter defines the size of the area around a hole inside the part.

2.13.2.1.3 Use Rectangular Border

If you check this option, the buffers will be calculated according to the part's bounding rectangle. Use the **Left**, **Right**, **Top** and **Bottom** parameters' fields to set the desired buffering values for each side of this rectangle.

2.13.2.1.4 Use Part Border

When you check this option and define the **Buffer Size** value, the buffering will follow the part's contours.

2.13.2.2 Mirror Allowed

Here the user can define whether the part can be mirrored on the sheet during automatic nesting or not. When the **Global** option will be chosen, the mirroring settings will be set to default, determined during the importing to ***Auto Nest***.

2.13.2.3 Direction

This option allows user to define the number of directions the part can be rotated in, while being nested. When the **Global** option will be chosen, the mirroring direction will be determined during the importing to ***Auto Nest***.

2.13.2.4 Common Cuts for Grouped Arrays

In this section the user will be able to determine whether, and how, the part will be nested utilizing common cuts option for arrayed group.

This is done by choosing desired direction of the CAMs and selecting the side of the part they will be connected to:

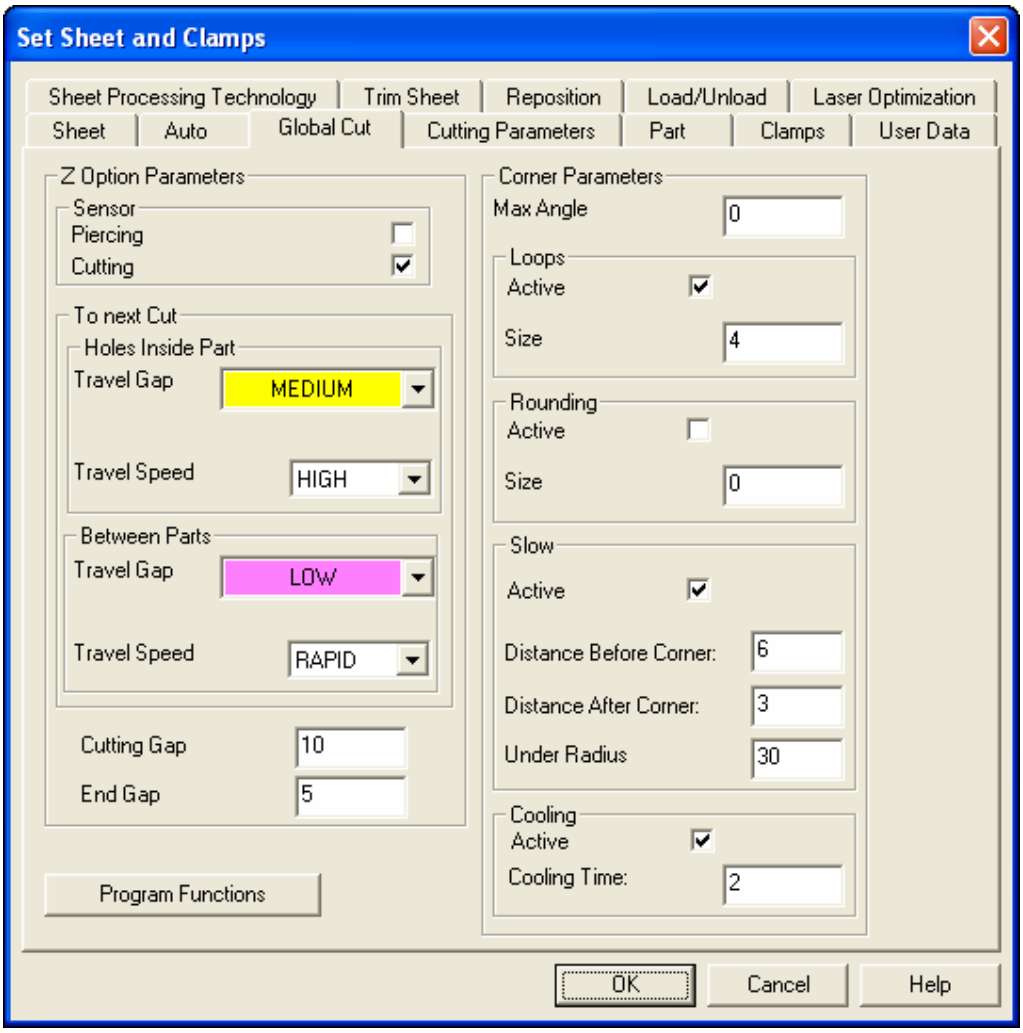
- **Vertical** – use **Common Cuts** to nest parts adjoining in the **Y** direction; the available side options are: **Right**, **Left** or **None**.
- **Horizontal** – use **Common Cuts** to nest parts adjoining in the **X** direction; the available side options are: **Top**, **Bottom** or **None**.

It is also important to define the **Distance between Parts** for common cuts. You can choose a **Default Value**, which is being taken from ***Auto Nest*** module or by entering some other, **Manual Value**.

2.13.3 Global Cut tab

This tab is available for laser machines only.

The options in this tab allow you to define **Z option** and **Corner** parameters for the current part:



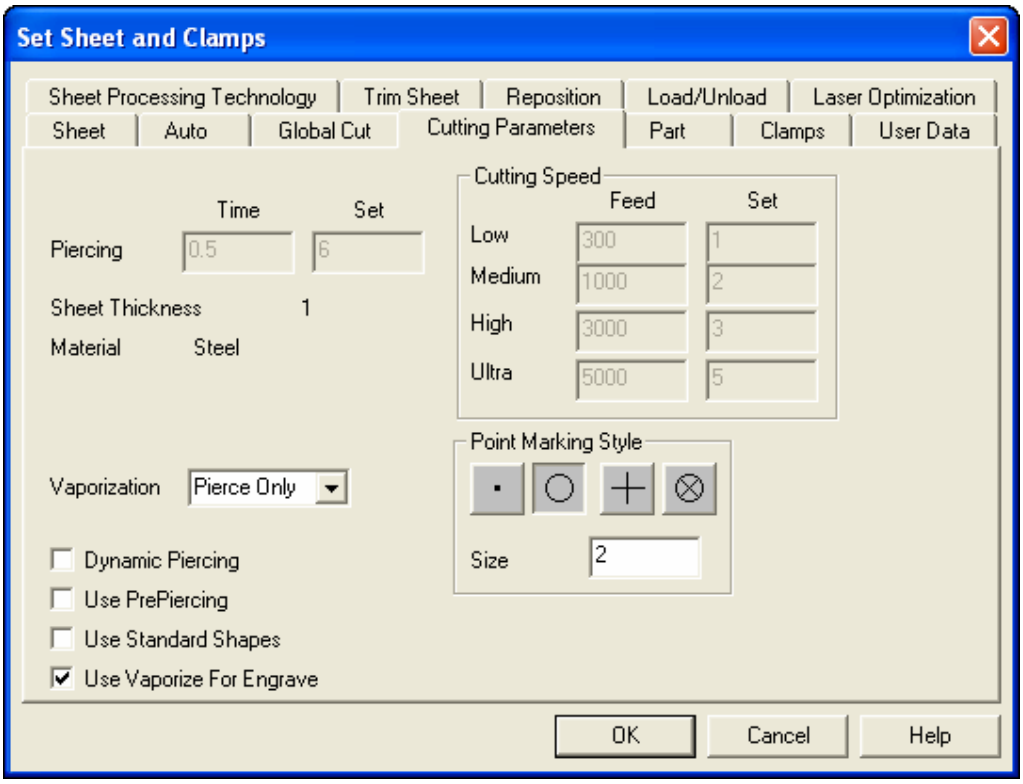
These definitions will be active for all the entities and corners that are set to **GLOBAL**.

The full explanation on the options presented here, you will find in proper sections of **Laser Manual** chapter of ***Laser Manual***. For explanation on specification of the colors we use to mark cutting speeds with go to View menu **=> Colored Tool Path by Speed** section of ***Drafting and Nesting Manual***.

2.13.4 Cutting Parameters Tab

This tab is available for laser machines only.

In this tab you will be able to set the **cutting parameters** for the current sheet:



2.13.4.1 Piercing

The parameters presented in the **Time** and **Set** fields, not available for editing here, give an information on the current sheet's piercing properties. This information is taken from the **Piercing** tab of **Cutting Technology Table** of the current laser machine.

See the appropriate section of Settings Menu chapter of ***Drafting and Nesting Manual***.

2.13.4.2 Sheet Thickness and Material

The data presented here for **Sheet Thickness** and **Material** type, not available for editing from here, is being taken from the **Sheet tab**.

2.13.4.3 Vaporization

This command is useful when the sheet is covered with nylon film: the laser beam will pass a first time, with low power, over the parts in order to burn the nylon, before it passes a second time for actual cutting.

From the dropdown list the user can choose how he wants his **Vaporization** to be set: for **Whole Cut**, for **Pierce Only**, or **None**. If you choose **None**, the **Use Vaporize for Engrave** function will be deactivated.

2.13.4.4 Dynamic Piercing

Checking this box activates the **Dynamic Piercing** option.

2.13.4.5 Use PrePiercing

If you check this option, the "**Pierce Only**" command will be generated a for the entire sheet – once the piercing will have been executed, the sheet will be cut without piercing.

2.13.4.6 Use Standard Shapes

Activate this option to define **Laser Standard Shapes** in the NC code as one-line codes (if the machine supports this option; otherwise it will be unavailable for editing).

2.13.4.7 Use Vaporize For Engrave

The user can choose here whether he wants to use vaporization for engraving.

This option becomes available only when the **Vaporization** is set for **Whole Cut** or for **Pierce Only**.

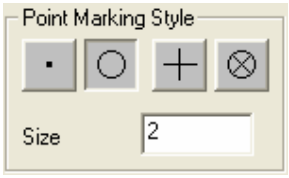
2.13.4.8 Cutting Speed

The parameters presented in this section, not available for editing here, give information on the current sheet's cutting speed properties. This information is being taken from the **Cutting** tab of **Cutting Technology Table** of the current laser machine.

See the appropriate section of Settings Menu chapter of Drafting and Nesting Manual.

2.13.4.9 Point Marking Style

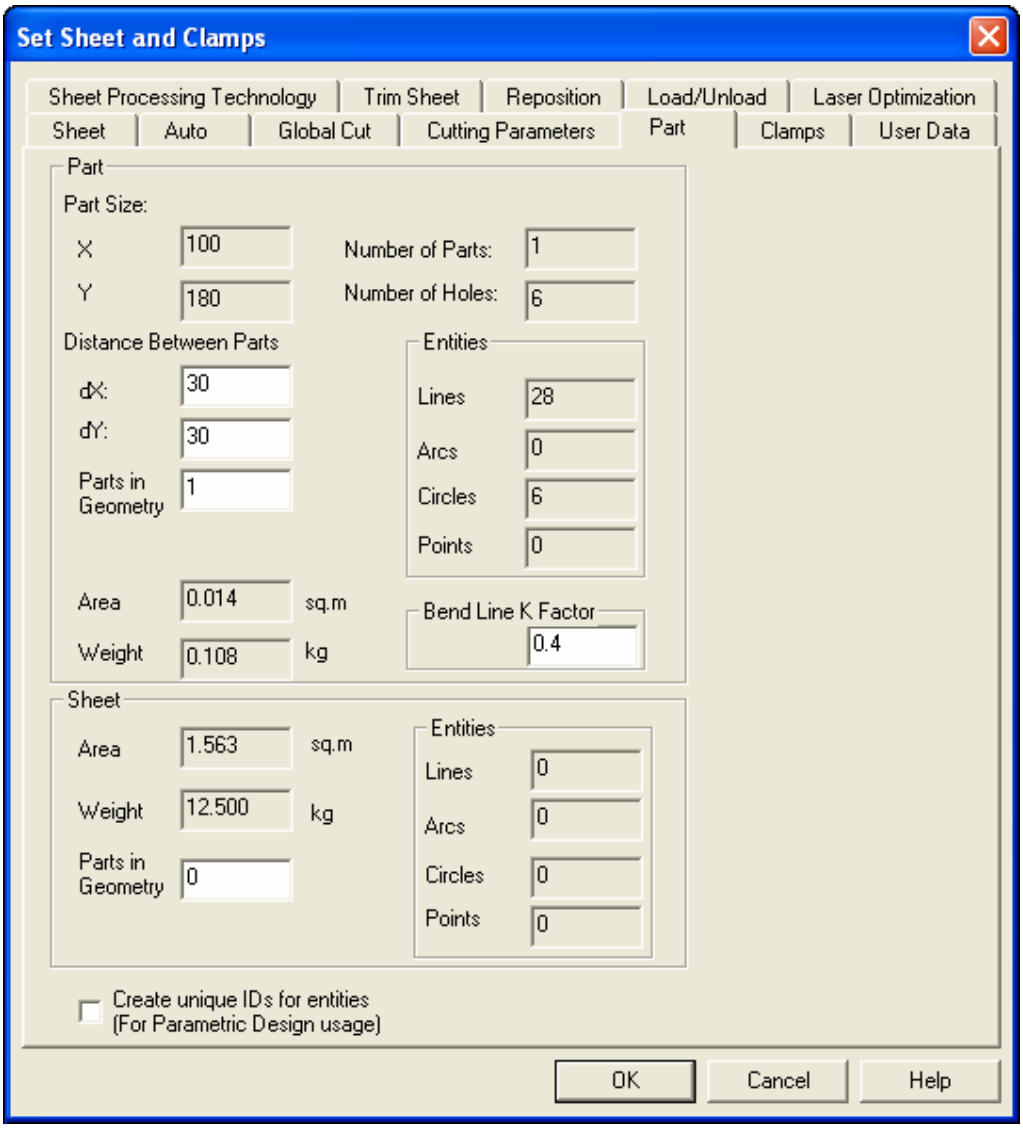
In this section the user can select the style and define the size of **Point Marking Styles**, which will indicate the location of the hole(s) you are not interested in cutting. These holes will be engraved instead.



Usually this function is being used for marking the undersized holes. By default the style of point marking is defined as a spot. For this option there is no possibility of setting the point size, but if you will choose one of the other marking signs, you will be able to define its dimensions.

2.13.5 Part Tab

The attributes of this tab are concerning various aspects of the current part:



2.13.5.1 Part section

This section gives you various information about the part in your draft. Most of these options, but some, are unavailable for editing.

2.13.5.1.1 Part Size View

In the **X** and **Y** fields you can see the dimensions of the current geometry. Any number of parts copied and placed on the draft will be always treated by ***cncKad*** as one geometry, and not as multiple parts. Therefore in these fields you will see the size of the whole geometry area, and not an area of a single part (unless your geometry contains just one part).

2.13.5.1.2 Number of Parts

This field reveals you how many parts does your geometry consist of. If there is more than one copy of the part placed in your geometry, here you will see how many.

2.13.5.1.3 Number of Holes

In this field you will be able to see how many holes are in your geometry. If your geometry consists of a few copies of a part, the number of holes will be a total number of all of them.

2.13.5.1.4 Entities Section

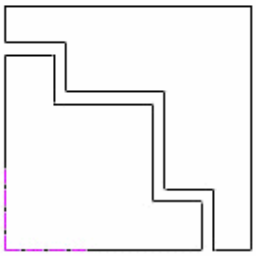
In this section you will see the specification of all the **Lines**, **Arcs**, **Circles** and **Points** placed on your geometry. If your geometry consists of a few copies of a part, the number of entities will be a total number of all of them.

2.13.5.1.5 Distance Between Parts

In the **dX** and **dY** fields of this sections you will be able to enter the distance you wish to be between the geometries placed on your draft, if your geometry consists of a few copies of the part. If a geometry actually equals a part, the value entered here will be in fact a distance between the parts.

2.13.5.1.6 Parts in Geometry

cncKad has an ability to maximize sheet usage by placing on a draft a number of parts that behave as a single part.



cncKad sees the two (or more) contours as a single part, and this way when you enlarge the sheet, make multiple-parts and use common cuts the upshot of this procedure is, in fact, a very efficient "Nest".

When you use such a procedure, you must also manually update the **Parts in Geometry** field, otherwise the program will not be able to give you a correct **total number of parts** for your sheet.

2.13.5.1.7 Part Area

This field, not available for editing, presents the **area** of geometry (in square meters). This data is calculated on the basis of the geometry size, sheet thickness and the material type it is made of. The values presented here will be automatically updated any time you will change the **material**, **thickness** or the **number of parts** the geometry consists of.

If the part is an open contour, the program will give an approximation of the values, indicating this with the sign ~.

2.13.5.1.8 Part Weight

This field, not available for editing, presents the **weight** of geometry (in kilograms). This data is calculated on the basis of the geometry size, sheet thickness and the material type it is made of. The values presented here will be automatically updated any time you will change the **material**, **thickness** or the **number of parts** the geometry consists of.

If the part is an open contour, the program will give an approximation of the values, indicating this with the sign ~.

2.13.5.1.9 Bend Line K Factor

In this field you will be presented with the default **Bend Line K Factor**. Here you have also a chance of changing it for current part you are working on.

More on **K Factor** you will find in appropriate section of Drawing Entities chapter, in ***Drafting and Nesting Manual***.

2.13.5.2 Sheet section

cncKad gives you the ability to place parts in **Sheet Mode**, as distinct from the parts that are placed in **Part Mode**.

2.13.5.2.1 Sheet Area

This field, not available for editing, presents the **area** of the sheet (in square meters). This data is calculated on the basis of the sheet size, it's thickness, material type it is made of. The values presented here will be automatically updated any time you will change the **material**, **thickness** or the **size** of the sheet.

2.13.5.2.2 Sheet Weight

This field, not available for editing, presents the **weight** of the sheet (in kilograms). This data is calculated on the basis of the sheet size, it's thickness, material type it is made of. The values presented here will be automatically updated any time you will change the **material**, **thickness** or the **size** of the sheet.

2.13.5.2.3 Entities on Sheet

In this section you will see the specification of all the **Lines**, **Arcs**, **Circles** and **Points** of which the shapes placed on your sheet consist of.

2.13.5.2.4 Parts in Geometry on Sheet

You should fill in this field, in order to allow ***cncKad*** for a correct counting of overall parts on the sheet.

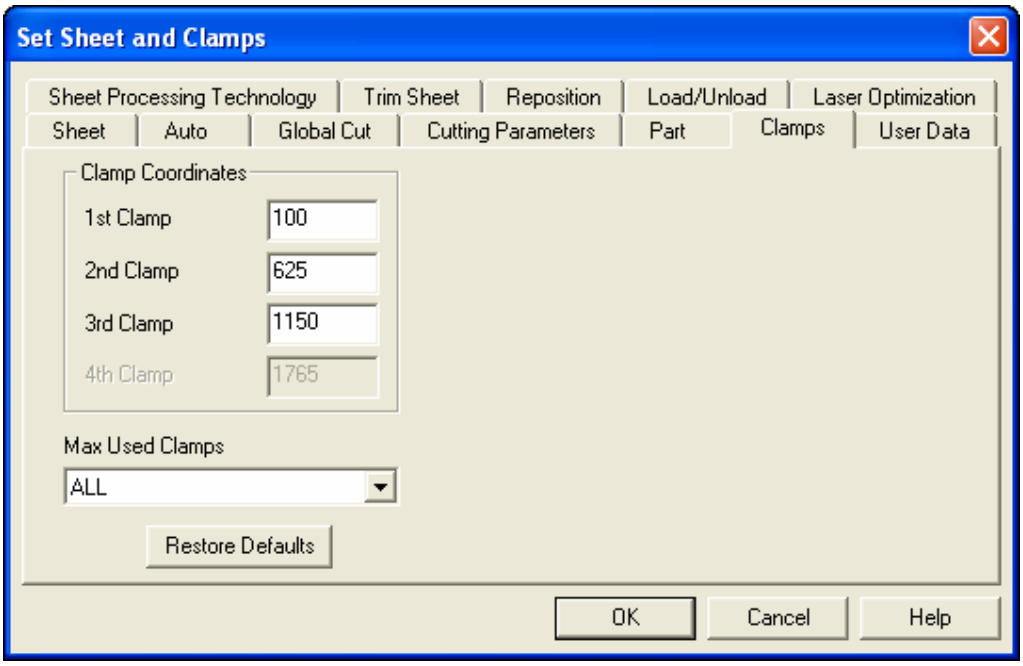
2.13.5.2.5 Create unique IDs for entities

This option is designated for the **parametric design**. It gives the user control over every entity in the draft file (.DFT) by activating the allocation of unique IDs for each entity, generating a **unique ID** for all the existing entities in the drafting and for every new entity that will be created from now on in this part.

These IDs can be used for methods dealing with Entities, Drawing, Punching, Cutting and adding Dimensions.

2.13.6 Clamps Tab

If your machine has movable clamps, you can define their settings here:



If your machine doesn't have movable clamps, you won't be able to edit their positions from here.

2.13.6.1 Clamp Coordinates

Normally, when you set a new Sheet Size in **X**, the Clamps are automatically being set at recommended values, according to the current machine's options.

You can change these values manually, either by dragging the clamps with the mouse or by entering values in the appropriate fields of the Clamps tab dialog.

cncKad will NOT allow you to place clamps in positions which cannot be set physically on the machine, but you will be able to choose any other legitimate position you wish.

These positions will be changed automatically in the following situations:

- Working in the Dead Zone
- Changing the Sheet Size in X
- Changing the machine

2.13.6.1.1 Working in the Dead-Zone

In this case, ***cncKad*** will try to generate an NC program based on the current positions of the Clamps, and if successful, will NOT change their positions. However, if unsuccessful, ***cncKad*** will set new positions for the clamps. This means that the positions will be changed (if necessary) during the **Post Processing stage** – the change will not appear during the drafting and processing stages.

2.13.6.1.2 Changing the Sheet Size in X

After you change the sheet size in X, ***cncKad*** will recalculate the recommended positions for the new size, and will automatically set the clamps accordingly.

2.13.6.1.3 Changing the machine

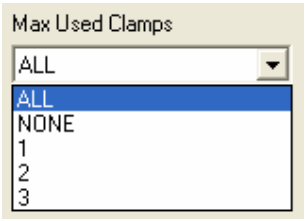
After you change the machine, the clamps positions will be corrected only if needed (e.g. when a clamps positions are off the new machine's table)

If you want to set the clamps to the recommended positions for the new machine, you can change the sheet size in **X**, and their positions will be recalculated.

2.13.6.2 Max Used Clamps

You can choose the number of clamps you want to use, if your machine allows it. If your machine doesn't support clamps, this option will not be accessible for editing.

If you have a machine with four clamps (such as Trumf 6000L), you can choose, from dropdown menu, one of the below listed options:



If you choose **ALL**, the maximum number of clamps will be used – in this case 4. If you choose NONE, no clamps will be holding your sheet. If you want your sheet to be holed by one to three clamps, select the appropriate number from the dropdown list.

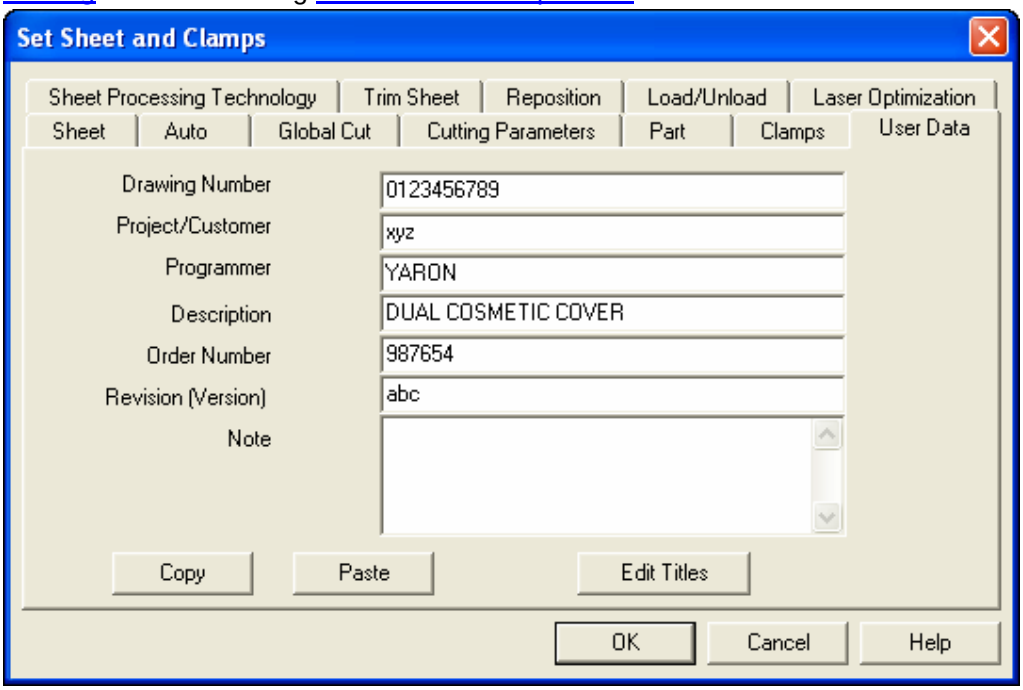
2.13.6.3 Restore defaults

Clicking this button will return to the primary settings of the clamps for your current machine.

2.13.7 User Data Tab

On this tab you can include the information about the customer who ordered the part to be created, the programmer who designed it, and about the project itself – its number, description, as well as other additional information. This data can be saved together with the part file.

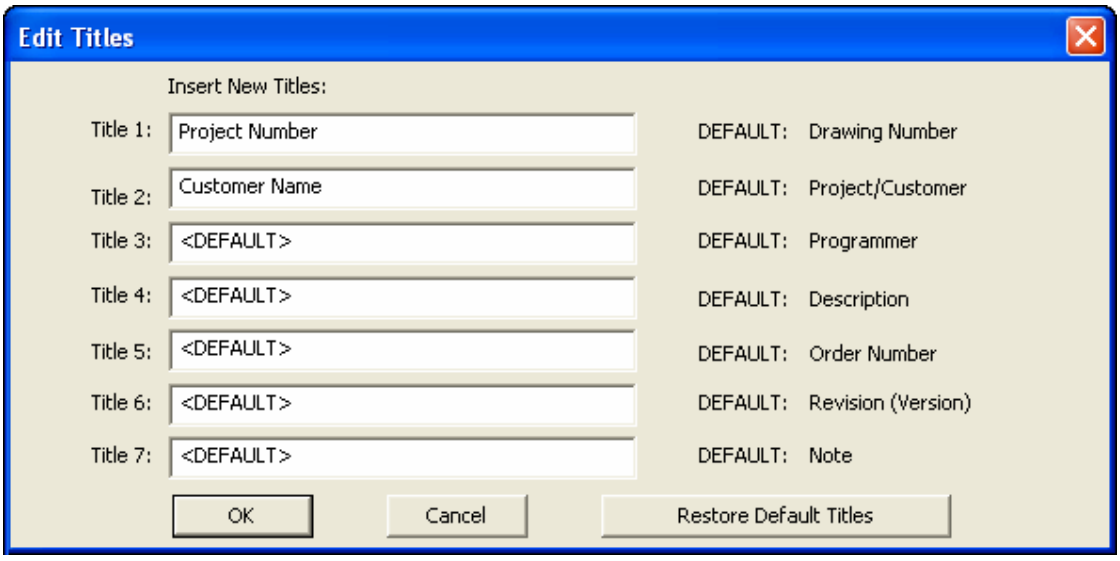
Entering it into these fields is not compulsory and can be added at any stage of part drafting as well as during NC code creation process.



You can **copy** the information entered here and later on **paste** it to different part, as well as **edit** the default titles of these fields.

2.13.7.1 Edit Titles

The default titles, which will appear later on in the Report File, of **User Data** dialog, can be edited and customized according to your needs. To do so, click on the **Edit Titles** button. The following dialog will appear:



The numbered titles here refer to the titles of the fields in the User Data tab and they are set to their default (for reminder see the right side of this dialog). You can change them just by clicking on the field and inserting the new title (as shown on the picture above). It will be then updated in the User Data, and eventually also in the corresponding fields of **Report File**.

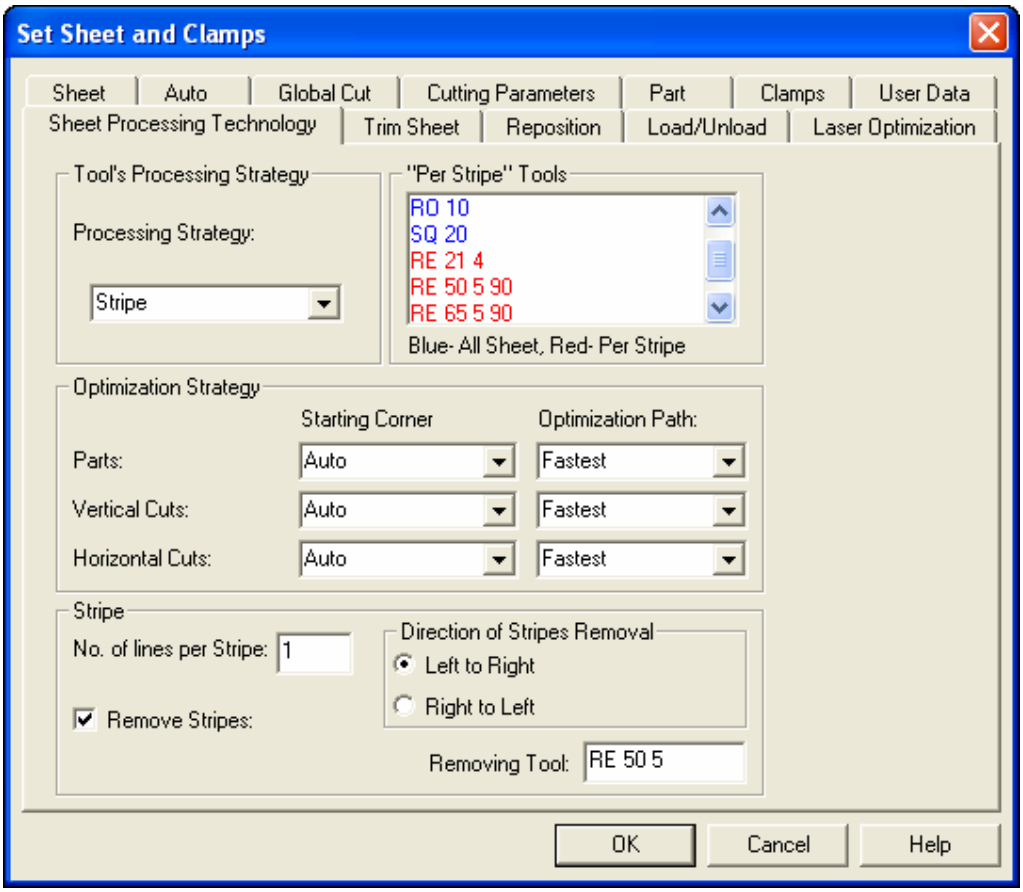
If you wish to restore a title's default, just leave its field empty. If you want all the titles to be restored, click the **Restore Default Titles** button.

2.13.8 Sheet Processing Technology Tab

This tab deals with overall strategies to be used while creating your part.

More explanation on the reasons for using the **Sheet Processing Technology** and other important issues concerning this topic you will find in the Sheet Processing Technology Tab dialog of Machine Settings dialog in Settings Menu chapter.

The difference between that tab and this one is that the settings defined here influence only the current part, and the ones defined there are general defaults which will be used only after you reload the current part, create a new one or the next time you open ***cncKad***.



2.13.8.1 Tools' Processing Strategy

From this section of the dialog you can set the strategy ***cncKad*** will use when creating your NC code. Here you can decide how to distribute the CAMs of various tools.

2.13.8.1.1 All Sheet Processing

First the machine will perform all the CAMs on the sheet with a one tool, and only then it will move on to the next tool.

2.13.8.1.2 Part Processing

Perform all CAMs on one part with ALL the tools (actually, all the **Blue** tools), and then go to the next part.

This option is useful when working with thick material or with old machines, where there is a high risk of the machine getting "stuck" – in such case, at least some of the parts are completed. Mark the "safe" tools as **Blue** – they will work all over the sheet, the **Red** tools will work **Per Part**.

2.13.8.1.3 Stripe Processing

Perform all CAMs with all the tools (actually, all the **Blue** tools) on current stripe, and then go to next stripe. Use this option for parts with many holes, where the sheet may easily get bent. After completing processing a stripe, you can remove it.

This option will **not** work on nestings.

2.13.8.2 Tools Section

This section shows the tools used in the processing of your part. If you have selected the strategy for **All Sheet**, this field will be grayed out and thus unavailable for editing. If your strategy is either **Part** or **Stripe**, you will be able to mark here "safe" tool as **Blue** ones and the tools, which are allowed to work only per selected strategy, as **Red** ones.

2.13.8.3 Optimization Strategy

The options presented in this section govern the way your CAMs will be optimized.

For processing of **Parts**, **Vertical** and **Horizontal Cuts** you can define here the Starting Corner and the Optimization Path:

- **Starting Corner** – sets the point where the optimization path begins.
- **Optimization Path** – sets the optimization type; Snake X, One Way Y, User Defined, etc.

More explanation on this subject you will find in Starting Corner and Optimization Path in the Settings Menu chapter of ***Drafting and Nesting Manual***.

2.13.8.4 Stripe section

These options provide settings for the **Stripe Processing** strategy:

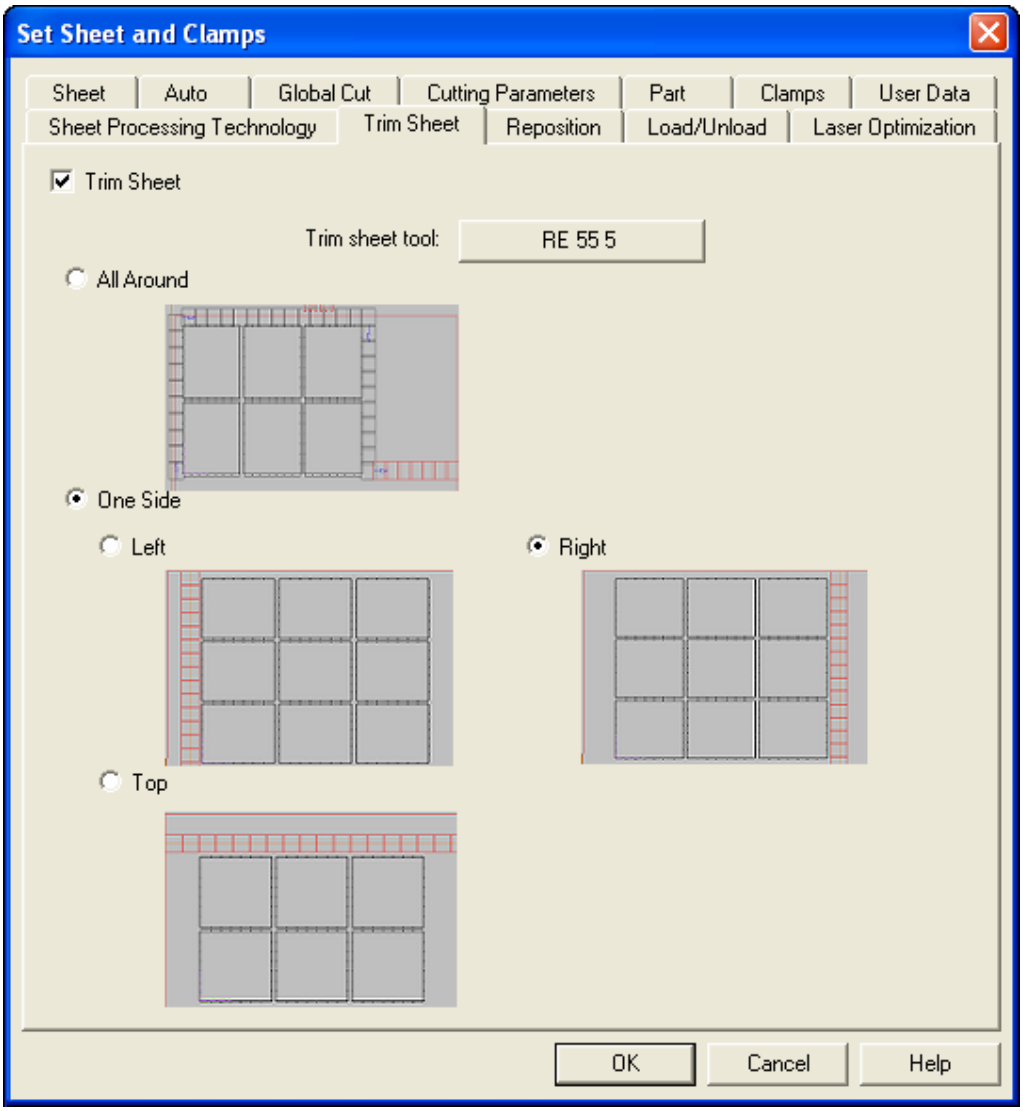
- **No. of lines per Stripe** – here you need to enter the quantity of parts per rows, which will be processed in one stripe, at a time.
- **Remove Stripes** – with this option you select whether or not to remove the stripe when the machine finishes processing it. If this option is marked, a trimming cut is created.
- **Direction of Stripes Removal** – when you choose to remove the stripe, you can select the direction of the punch (or cut) that will disconnect the stripe from the rest of the sheet.
- **Removing Tool** – when you choose to remove the stripe, you must choose the tool that will be used for the cut removing the stripes.

2.13.9 Trim Sheet Tab

After you have finished processing all the parts on your sheet, you will be left with the outer remainder of the sheet. The options of this tab let you dispose of this remainder.

After selecting the trimming tool, choose the way you want the sheet to be trimmed.

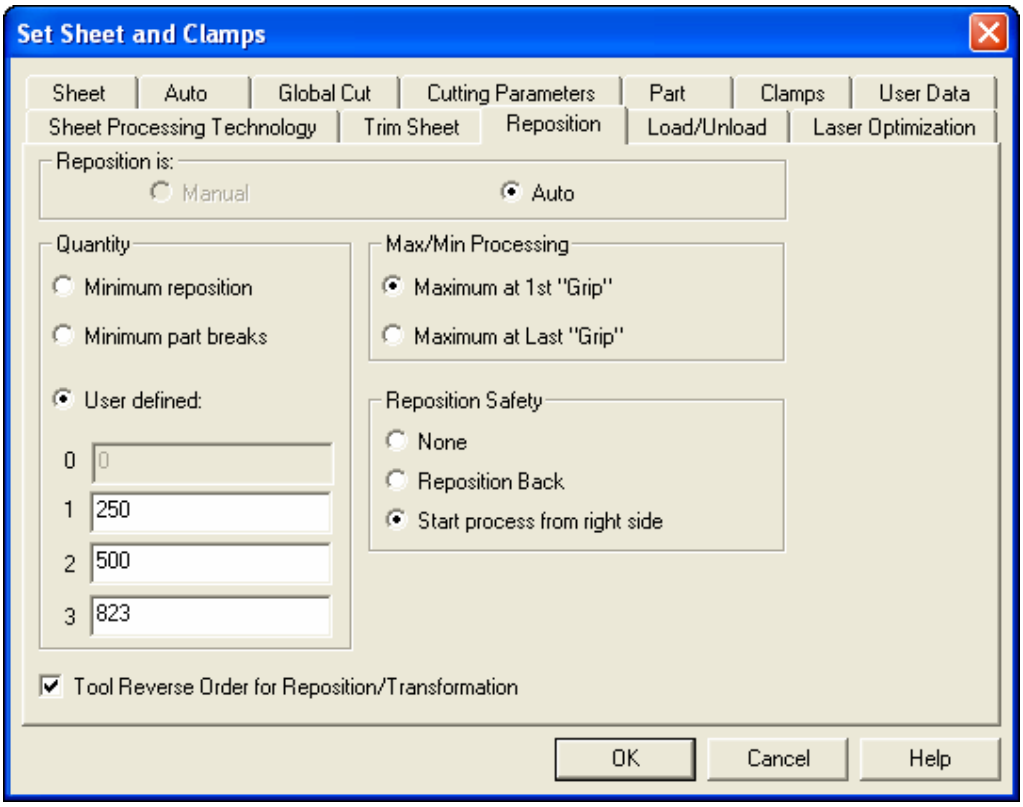
For more information on trimming the sheet go to **Creating a Trimmed Sheet** section of **Appendices** chapter of this manual.



2.13.10 Reposition Tab

This tab gives you an opportunity of defining the reposition parameters that will be used during the Post Processing stage, during which the NC code is created for the part. Repositions are typically performed by moving the clamps and placing ‘pins’ on the sheet to hold it in place when the clamps change their position.

Here you can see various repositioning options:



2.13.10.1 Manual Reposition

This option is valid only when you have already defined repositions manually, with the aid of the **Reposition and Transformation** function described in current chapter. This will give priority to repositions you have already set and will make no changes to them.

2.13.10.2 Auto Reposition

When selected, ***cncKad*** will set the repositions as you specify in the dialog, overriding any previous reposition definitions you may have set.

2.13.10.2.1 Quantity of Repositions

In this section the you will be able to define the number of repositions you want to be made for your sheet. Choose the option that fits you best.

2.13.10.2.1.1 Minimum repositions

Choosing this option causes the machine to create as few repositions as possible in order to perform the needed CAMs. If necessary, it will split the processing of a part.

2.13.10.2.1.2 Minimum part breaks

With the aid of this options the machine tries to set the minimal number of repositions needed, so as not to split the processing of any part. This may result in having more than the minimal number of repositions possible, but ***cncKad*** will make sure that each part is processed in a single reposition (unless the part itself is bigger than the reach of a single reposition).
For example, when your parts are placed on a sheet that extends beyond the table, and therefore requires repositioning, ***cncKad*** will try to process the entire part, make the reposition, and then process the next part. In this way there will be less of a chance for inaccuracies resulting from repositioning.

Another reason for using this function is when your machine has a slow repositioning mechanism and you want to make as few repositions as possible.

2.13.10.2.1.3 User defined

This option lets the user define the repositions by himself. If he selects illegal positions, or the Post Processor cannot reach the whole sheet due to the defined repositions, an error message will be issued and abort the NC creation. In any case, the repositions set by the user will not get changed.

When defining repositions, the user chooses the position relative to the sheet's left edge, where the machine's **Working Range** will begin.

For example, for a machine with a **Working Range** of 1000, processing a sheet 2000x1000 the following repositions have been defined – 600 and 1000. The processing will proceed as follows: The machine will first process the CAMs in the 0-600 range, then the ones in the 600-1000 range, and lastly the ones in the 1000-2000 range

There are a few things to be noted when entering reposition values:

- No negative values are allowed.
- Each reposition value must be bigger than the previous one (it doesn't apply to the range).
- Make sure that in the last reposition, the machine reaches the end of the sheet.
- The "zero" position is always the sheet origin, therefore it is unavailable for editing.

2.13.10.2.2 Max/Min Processing

The options presented in this section allow you to choose the minimal or maximal range of the sheet to be processed during each reposition.

- **Max at 1st grip** – at each grip (reposition) the machine will try to process as much of the sheet as possible.
- **Max at last grip** – whenever two grips (repositions) overlap, the machine will let the overlapped section to be processed in the next grip.

When a Forming Tool is used, it is often placed in a turret as the last tool, so that the tools following it (especially punching tools like a long **RE** tool) will not "flatten" the form. However, if the sheet you want to process is longer then the working range, and Reposition is required, the cutting tool used after the Reposition (in the 2nd "grip") can flatten the forms that were created in the 1st "grip". To solve this issue, we can define the **Min/Max Reposition per tool**.

For example, suppose that we are processing a 2000mm sheet on a 1250mm machine. We will set in **Set Sheet and Clamps** **=>** **Reposition** tab to have:

- [x] Max at 1"st Grip

However, in **Used Tools** **=>** **Tool Data** **=>** **Reposition Types** we will set for the Forming Tool:

- [x] Max at last Grip

Now, all the tools will process up to 1250mm, but the Forming Tool will process only up to 750mm.

After the Reposition, all the tools will process between 1250 - 2000, but that will not be a problem for the forms – they were made only up to 750. Finally the Forming Tool will complete the forms from 750 - 2000.

2.13.10.2.3 Reposition Safety

These options deal with the repositioning of the Sheet before and after it has been processed:

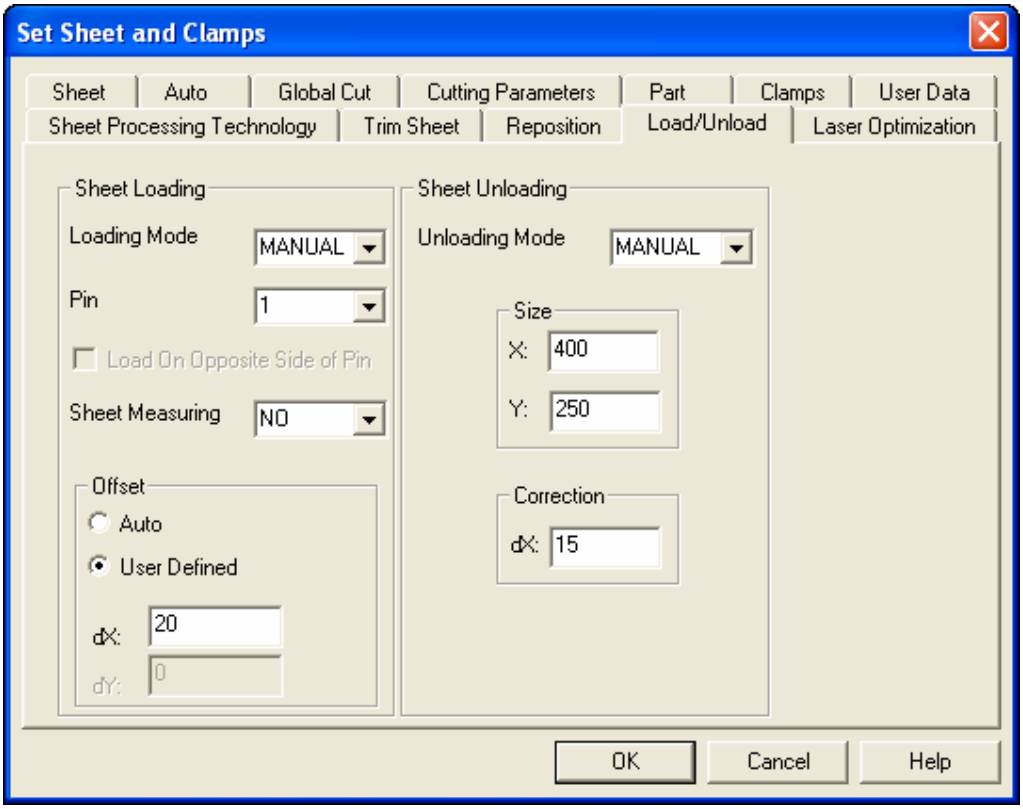
- **None** – will not add any reposition before or after the sheet processing.
- **Reposition Back** – after the sheet have been processed it will be brought back to its original position by repositioning the clamps.
- **Start process from right side** – the processing of the sheet will start at the opposite side of the sheet than the one usually in use (left corner at 0,0 location). Therefore, the sheet will be repositioned before the processing will start and in the end it will have the same position as when it was loaded. This option is useful for sheets that contain few micro-joints, for instance, in order to avoid loss or damaging of parts.

2.13.10.3 Tool Reverse Order for Reposition/Transformation

This function is being used after the reposition/transformation in order to save the tool change. Choosing this option we reverse the order of tools set in Used Tools dialog of Tools Menu chapter, for processing the parts on transformed or repositioned sheet.

2.13.11 Load/Unload Tab

This tab presents options intended only for machines with an ability of automatic sheet loading or unloading. Their attributes are as follows:



2.13.11.1 Sheet Loading

This section allows the user to define parameters for loading a sheet.

2.13.11.1.1 Loading Mode

The mode of loading the sheet can be manual or automatic, depending on machine you have.

2.13.11.1.2 Pin

Pin refers to those machines that have a second pin, for pinning smaller sheets.

2.13.11.1.3 Load On Opposite Side of Pin

This option is available for machines who have two pins. Choosing it your sheet will be loaded from the opposite side of the pins.

2.13.11.1.4 Sheet Measuring

Some machines know how to make measuring of the sheet loaded. If they don't, the default option "**No**" will be set.

2.13.11.1.5 Offset

Here the user can define an offset for the loaded sheet. If you will select **Auto** option the offset will be defined automatically by ***cncKad***. If you will choose the **User Defined** option, you will be able to enter offset value in **dX** field.

> **dX** is the distance between the 0.0 point (the endpoint of the **table**) and the endpoint of the **sheet**.

2.13.11.2 Sheet Unloading

In this section you can define the parameters for the sheet unloader.

2.13.11.2.1 Unloading Mode

The mode of unloading the sheet can be manual or automatic, depending on machine you have.

2.13.11.2.2 Size of Processed Sheet

After the sheet has been processed, it is often smaller than it used to be to begin with. The automatic unloader has to know this new size, therefore you should enter its new dimensions in **X** and **Y** fields.

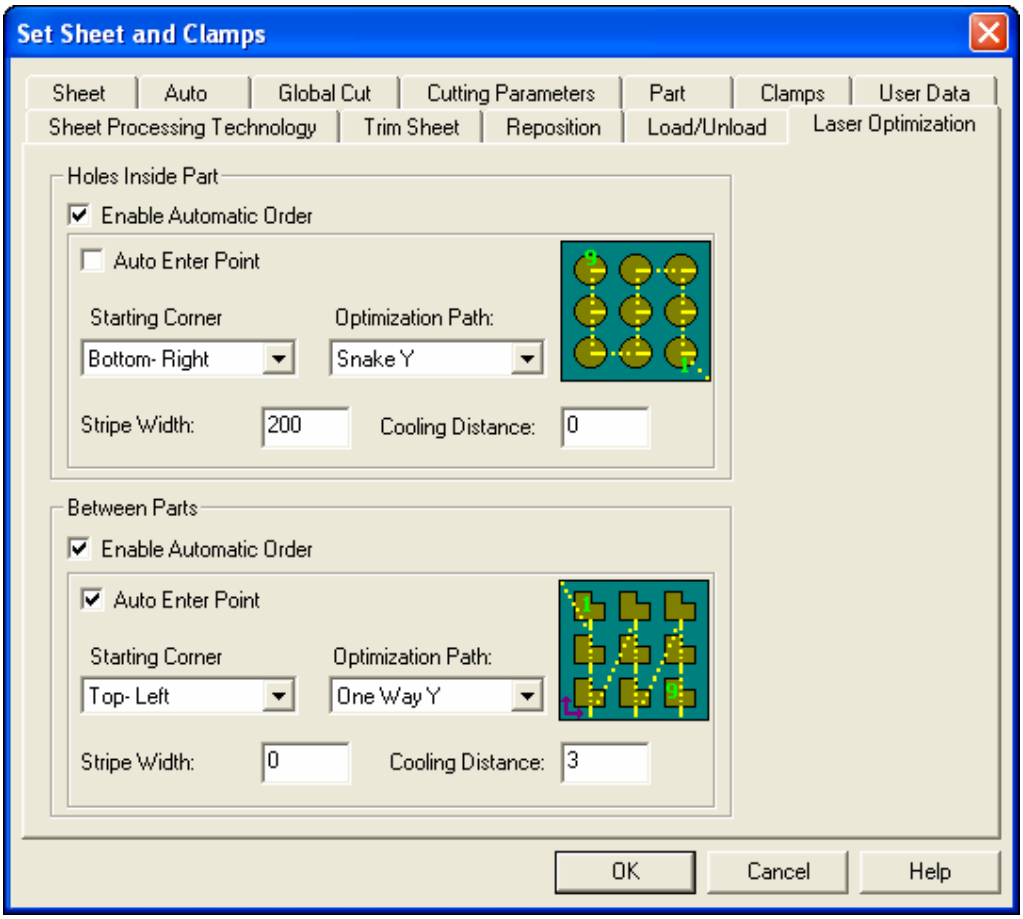
2.13.11.2.3 Correction for Processed Sheet

If the sheet's size has changed during processing, it may appear that the sheet's edge is no longer flush against the table's edge. In such a case the unloader will have to compensate for this difference.

2.13.12 Laser Optimization tab

> This tab is available for laser machines only.

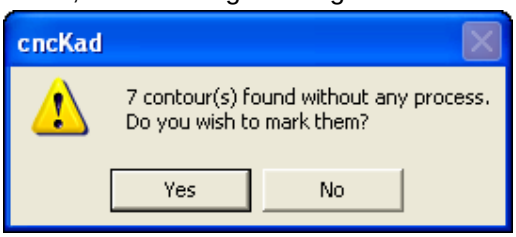
This tab is similar to the **Laser Optimization** tab of the **Auto Cut** dialog of Laser Manual chapter, with one important distinction – the parameters set here are the default settings for a part, offered when the Auto Cut dialog is opened for a new cut.



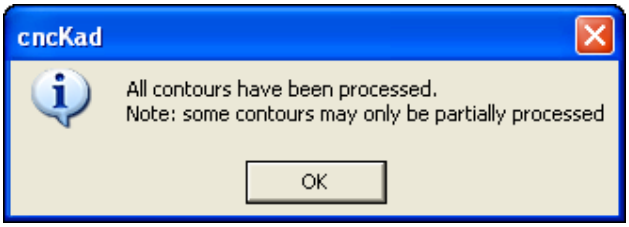
For more information about the parameters in this tab, please go to the **appropriate** section of Laser Manual chapter of ***Laser Manual***.

2.14 Check Process

This option examines the part for unprocessed contours. If any such contours are found, the following message will be shown:



If you select **Yes**, the unprocessed contour(s) will be highlighted in blue. If there are no unprocessed contours found, you will be presented with the following message, instead:



The Check Process feature is run automatically after **Auto Punch** and **Auto Cut**, allowing you to make sure all the desired contours have been processed. If you want

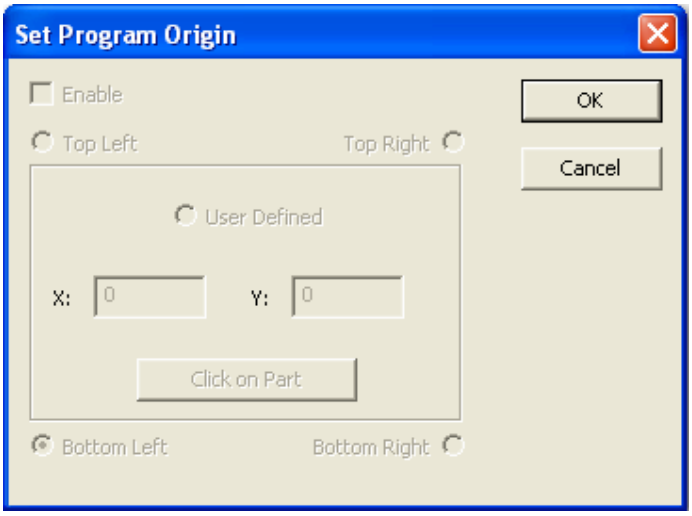
to set this option to be run automatically as a default, select the **Check For Not Processed Contours** option of the Post-Processor Options tab in the Workspace Settings dialog.

2.15 Set Program Origin

Program Origin refers to position, which is considered to be a "zero position" (**0,0**), or a reference point, from which you start running the program.

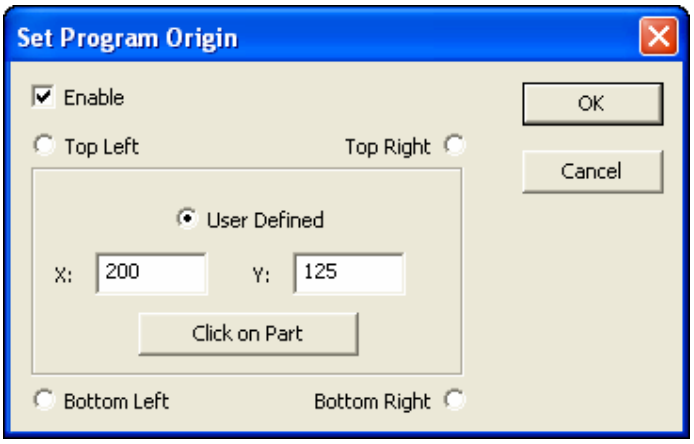
For many machines, this position is fixed, and defined as for example: ***bottom-left corner of the sheet***. In some cases, usually in cutting machines such as Laser/Plasma/OxyCut, the operator can bring the machine head (cutting head) to any position on the sheet, and start the program from there, thus establishing that position to be (**0,0**) or **Program Origin**.

Normally, the operator will ALWAYS bring the head to the same position, which is one of the sheet corners. In this case, the **Set Program Origin** dialog will be grayed and cannot be used.

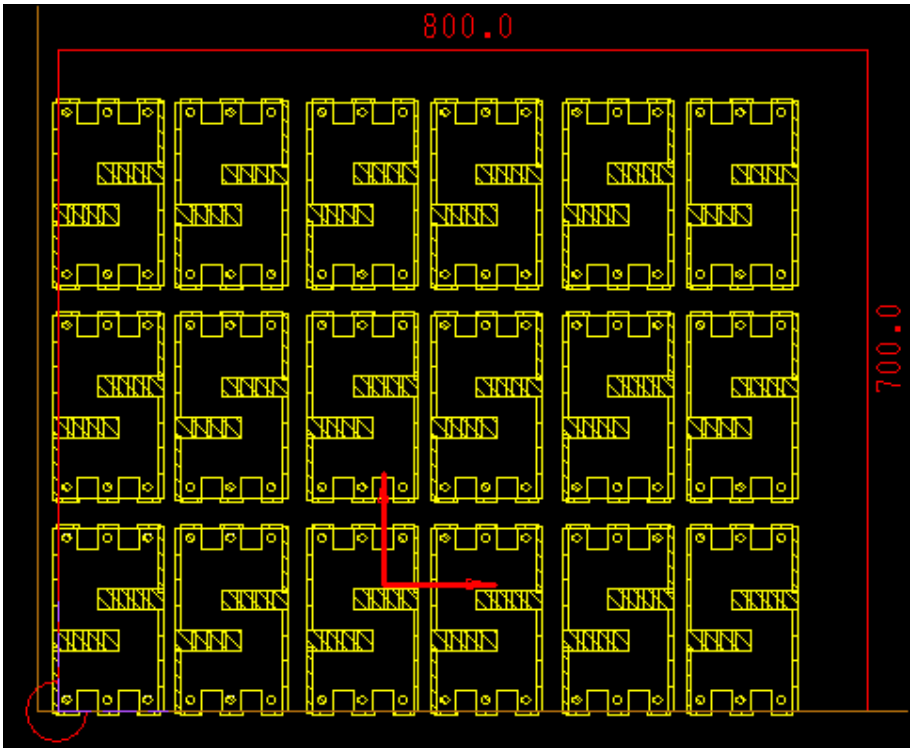


If the machine is able support this mode of operation AND you wish to control the settings of the **Program Origin** (for each Part/Sheet), then this dialog can be enabled in the MDL file (see **Support for Program Origin** section in the Appendices chapter of current manual). This means that, for example, you want to start some programs at bottom-left, others at top-left, and yet another one, at some specific point – maybe at the center of a teeth-wheel for which you have prepared some fixture on the machine.

When this dialog is enabled in MDL file, you should enable it also within the dialog itself. Only then the other features of this table will be available for editing, and you will be able to set the **Program Origin** position by choosing one the four sheet corners, defining it by yourself, entering its values in **X** and **Y** fields or by pressing the **Click on Part** button to snap to any position on the Part.

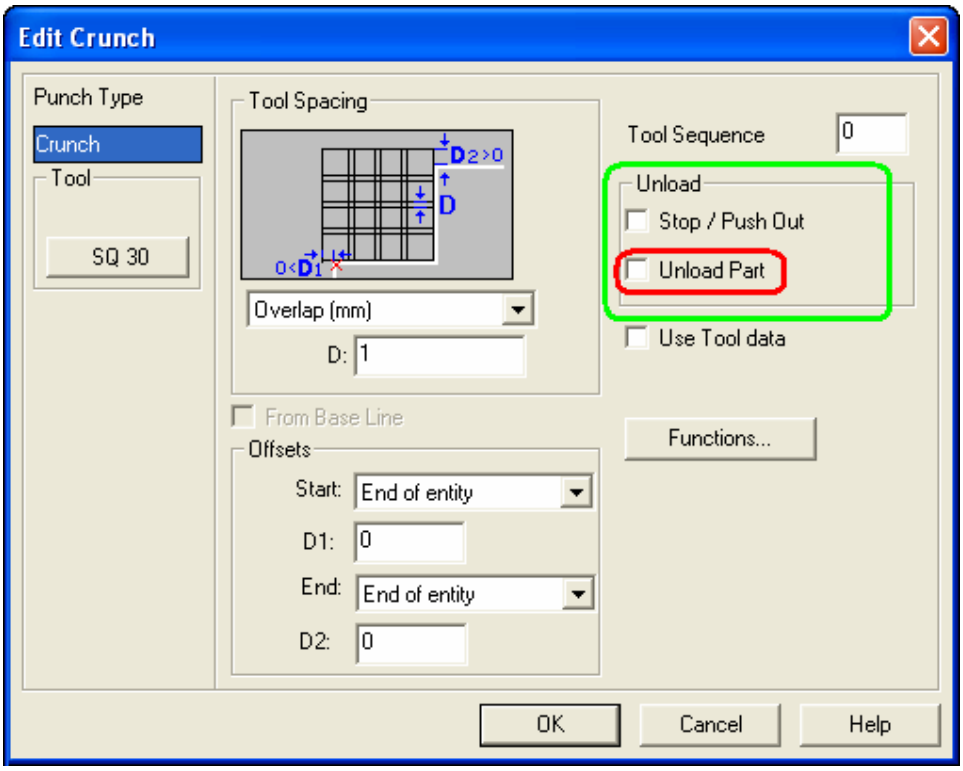


The **Program Origin** is displayed as a **bold**, **red** coordinates origin, with arrows pointing to **positive** direction of **X** and **Y**, as shown on below presented picture:

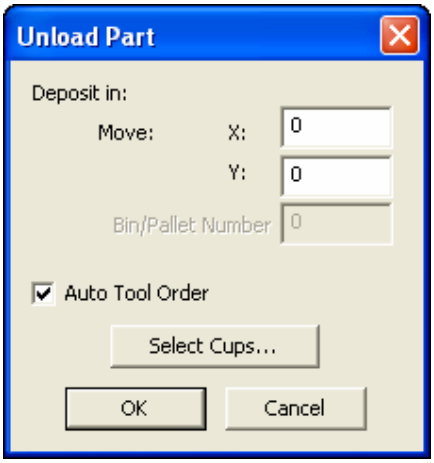


2.16 Unload Part

This feature is intended for machines that have a part removing device. It can be accessed from the CAM Menu or by choosing the **Edit CAM** icon –  from CAM Toolbar. When you click the latter one you should click this part's entity you want to process as the last one before unloading the part. After you do so the following dialog will appear and on its right side, in **Unload** section, you will be able to choose the **Unload Part** option:



Once you have selected it, the dialog will be closed and you will be moved to the drafting dialog, where the following **Unload Part** dialog will appear:



In this dialog, you will be able to determine where to move the unloaded part, defining the position of the bin, by entering its coordinates on the **X** axis (and **Y** axis, if applicable).

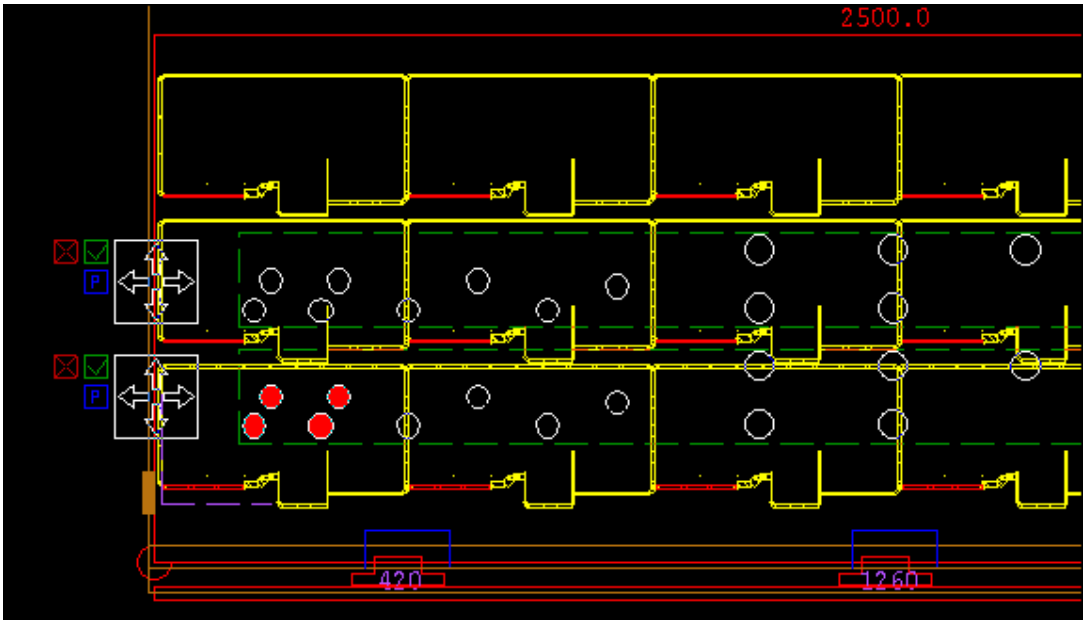
Checking the **Auto Tool Order** box you allow your machine for automatic arranging the part processing order.

Clicking **Select cups…** button you will be moved back to drafting dialog and there you will see an interface that facilitates the cups selection. The rectangles represent the unloader's arms and the circles – the vacuum cups. To move the arms, drag and drop the arrows alongside.

Click on the cups you wish to use. They will be marked with red color to indicate they are active.

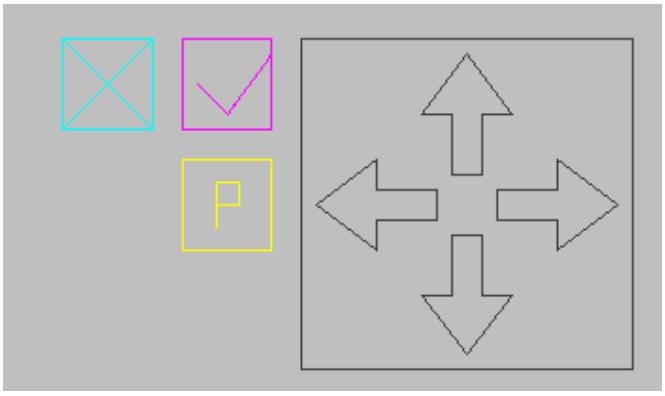
On some machines the cups that must be activated together are joined by a line.

If you want to select all of the cups, press the **Alt** key while clicking any cup. If you want to select just cups on one arm, press the **Ctrl** key while clicking any cup of this arm. If you wish that the machine to mark all the cups that are on parts, click on either one of the **P** boxes next to the arrows. And if you wish to mark each cup separately, click it with the right mouse cursor.

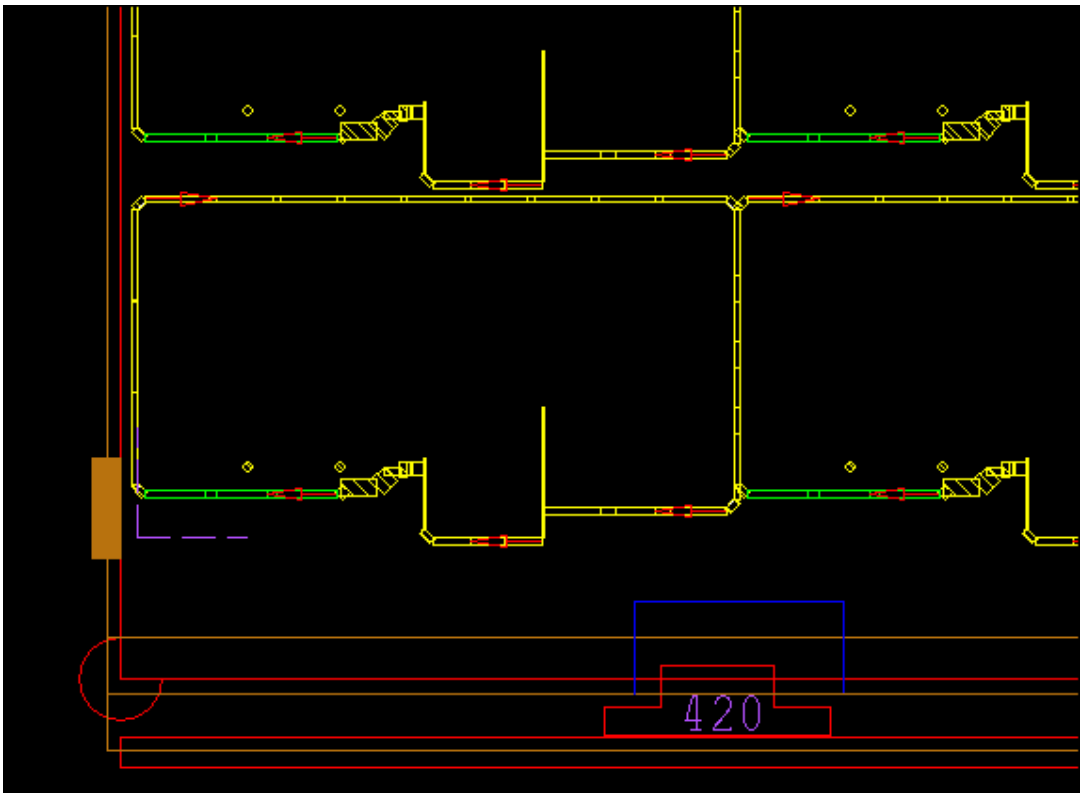


To confirm the position, click on either one of the tick (**V**) boxes next to the arrows. The **Unload Part** dialog will reopen, and there you should press **OK** button to confirm your choice.

If you wish to discard the position, click on one of the cross (**X**) boxes.



After adding **Part Unload** function to your part, it will be presented on your sheet this way:



The green marked entity is the one, after processing which, the part will be unloaded from the sheet. The red arrows show the direction of processing of the part.

To define the type of cups available, go to Settings Menu **=>** Machine Settings **=>** Machine tab **=>** **Machine Settings** dialog and make changes to the appropriate machine file in Metalix Editor.

2.17 Cutting Table

This option is a shortcut to the **Cutting Technology Table** accessed through the Settings Menu **=>** Machine Settings **=>** Cutting Parameters **=>** **Cutting Table** button.

For more information on this feature, go to appropriate section of ***Drafting and Nesting Manual***.

2.18 Adjust Travel Path

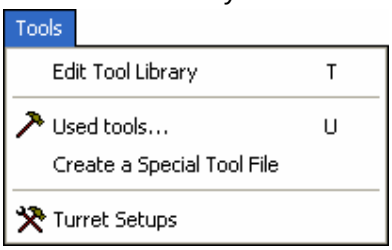
This option gives you an opportunity to correct machine's travel path. After selecting this option you will be able to select new machine path, by clicking the places on the part you want the machine to process in the new order.

2.19 Restore Default Clamps

Choosing this option will restore the clamps to their default positions.

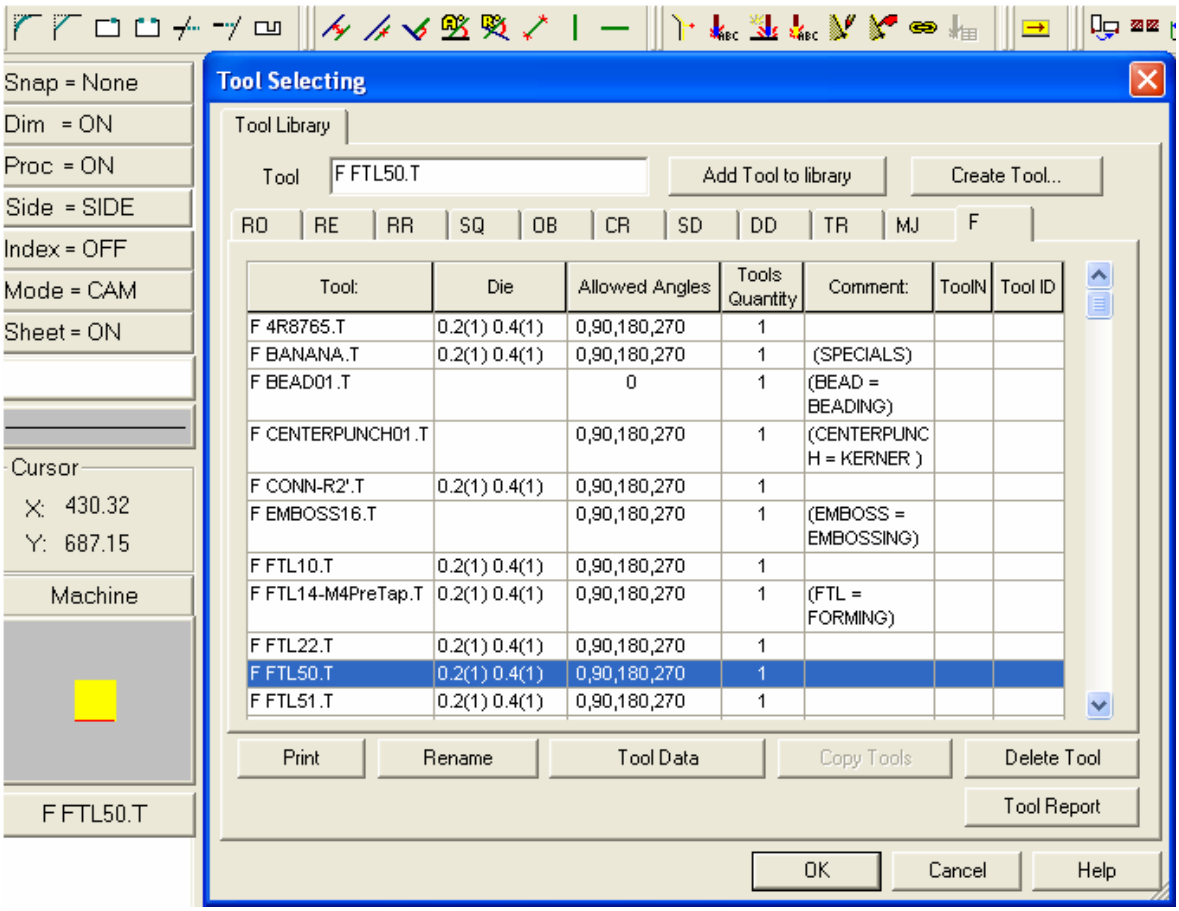
3 Tools Menu

This menu allows you to edit various tool-related setups:



3.1 Edit Tools Library

With this option you can access and edit the **Tools' Library**:



Accessing this dialog from few other places of ***cncKad*** you will be presented also with **Setup File** tab.

3.1.1 Tool Library tab

On this tab you will find large choice of various tools and options of manipulating them.

3.1.1.1 Tool Field

When from the **Tool Library** you select a tool for processing your part its **definition** will appear in this field. You can also enter here the tool's definition manually. If the

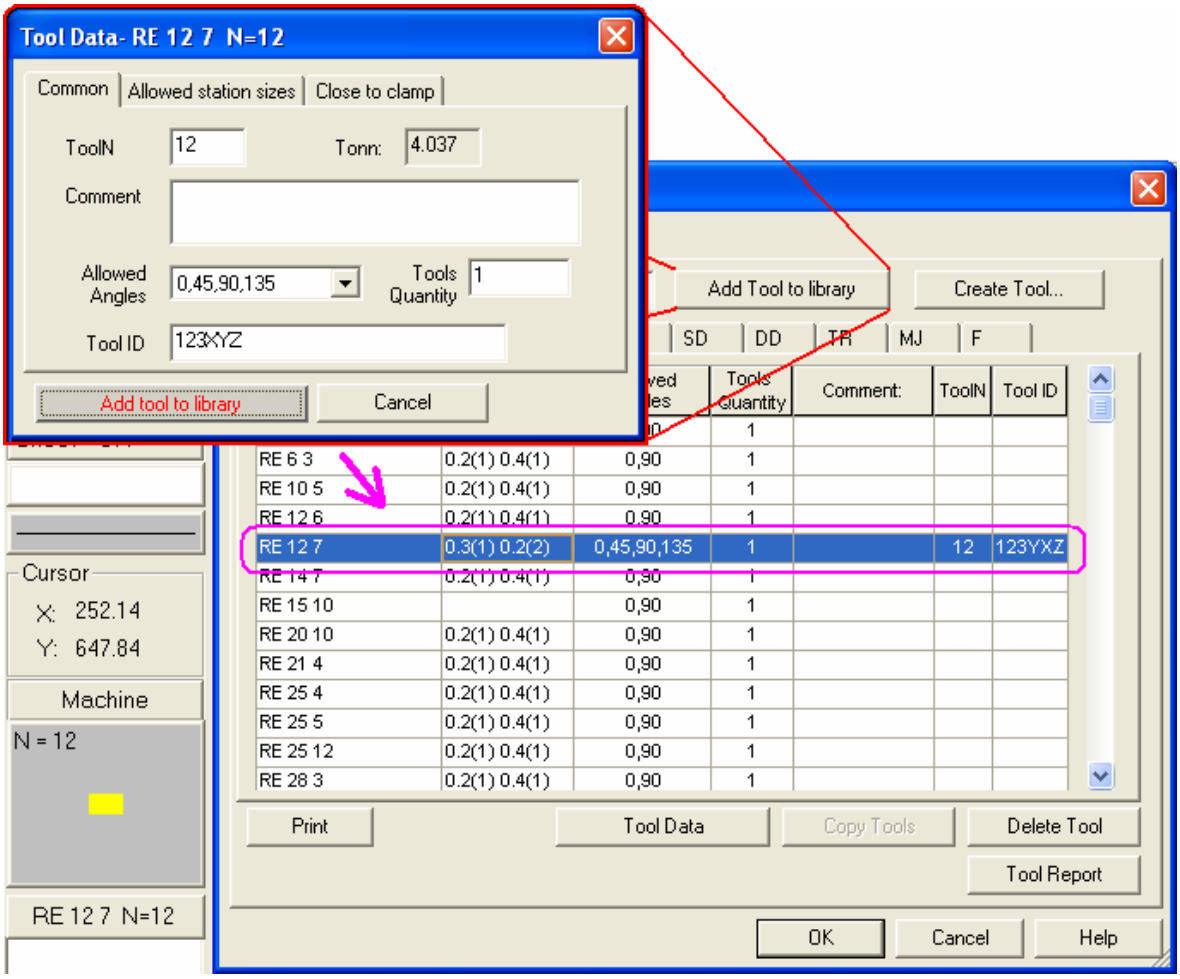
tool you typed in doesn't exist in your tool library you will be asked if you want to **add it to library**. In order for the tool to be used for processing it is **not** necessary to add it to the library.

3.1.1.2 Add Tool to Library

The **Tool Data** dialog can be opened either when **adding a new tool**, or by clicking the **Tool Data** button at the bottom of the **Tools Library** dialog.

If you wish to **add a tool** (for **CTC tools**, see the appropriate section of this manual), just enter the tool's name in the **Tool Field** and either press **Enter** key or click the **Add tool to library** button.

You will be presented with the following dialog:

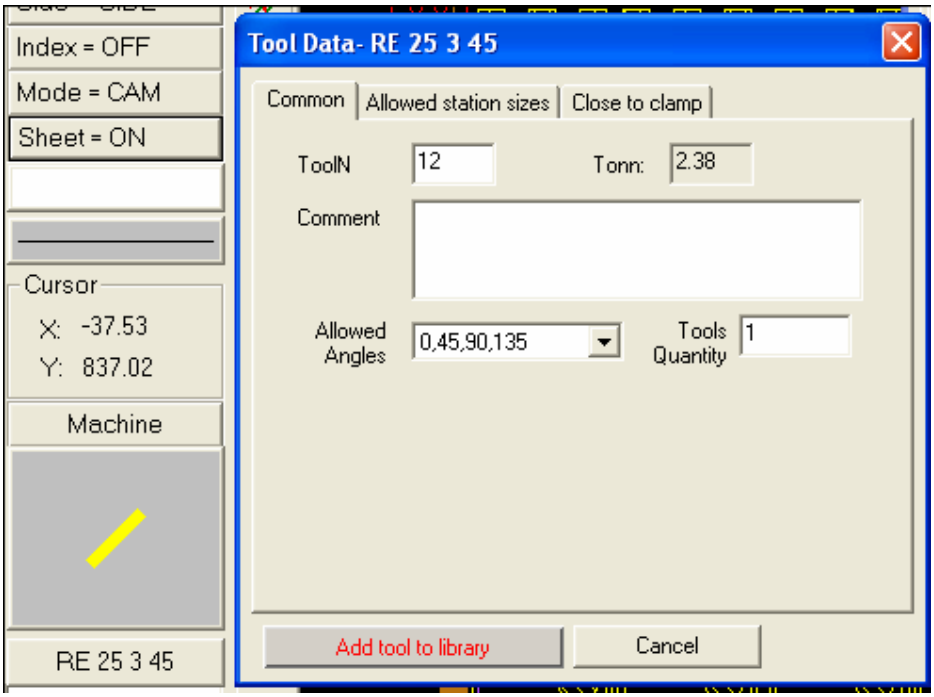


3.1.1.2.1 Common tab

In this tab, user will be able to add detailed features to the new tool, such as: choose **Allowed angles**, enter **Tool Number** (ToolN) as well as **Tool Quantity** and write a **Comment**. They all will be later seen in **Tool Table** of the Tool Library tab.

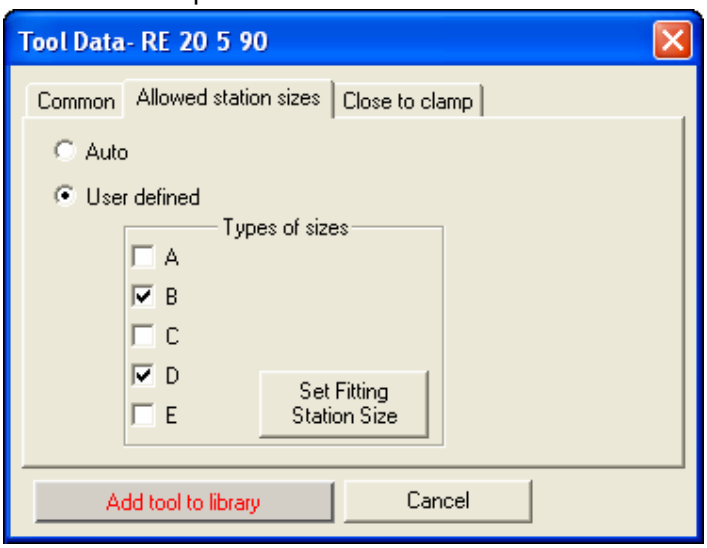
From here you can also see the **Tonnage** of the tool, but you can't enter here nor change already existing parameters. The value displayed here is being calculated according to the material tonnage, taken from **Material List** (Settings **=>** Workspace Settings **=>** Material tab **=>** Edit Material button), and tool's circumference.

Note that the new tool is being shown on the display, on the left of the screen:



3.1.1.2.2 Allowed station sizes

In this tab, you can define whether the station size for a tool is selected **automatically** according to the tool size, or manually. In the latter case, select the **User defined** option and check the box next to desired standard station size:



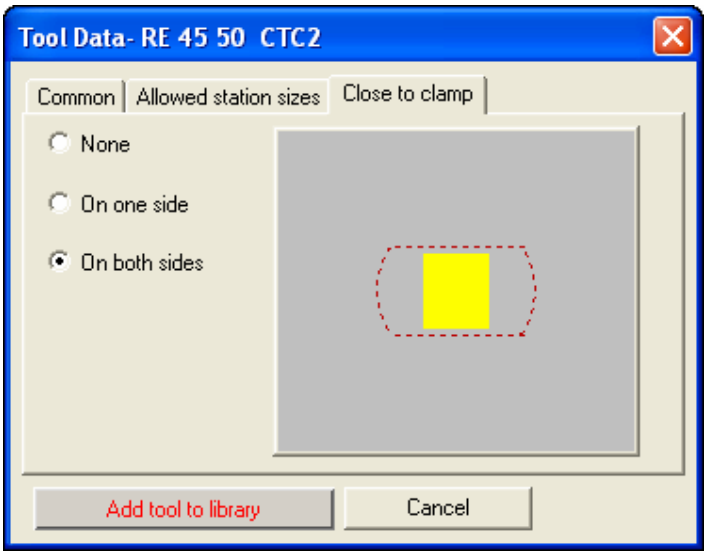
Clicking on the **Set Fitting Station Size** button will make ***cncKad*** choose the most fitting station for the tool you are adding to your library.

3.1.1.2.3 Close to clamp

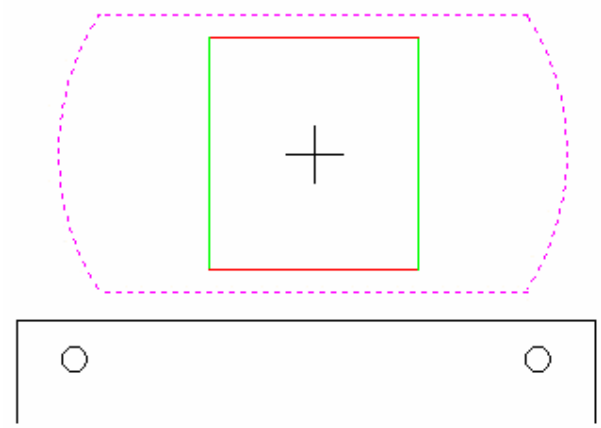
When assigned to a tool, the **Close to clamp** option allows performing punches closer to clamps while avoiding clamp/die collision. This option is available for machines of which dies are not entirely round, but have one or two straight side(s). It is per tool, and it can be defined for the following tools only: **RO**, **RE**, **SQ**, **OB** and **MJ**.

When creating/adding a new tool to the **Tool Library**, define a tool as **CTC** by entering the tool type followed by the abbreviation "**CTC**" or "**CTC2**" (if you wish to

use a die with two straight sides). Else, you can define the **CTC tool** through the **Close to clamp** tab. **RE 45 50 CTC2** will look like the following:



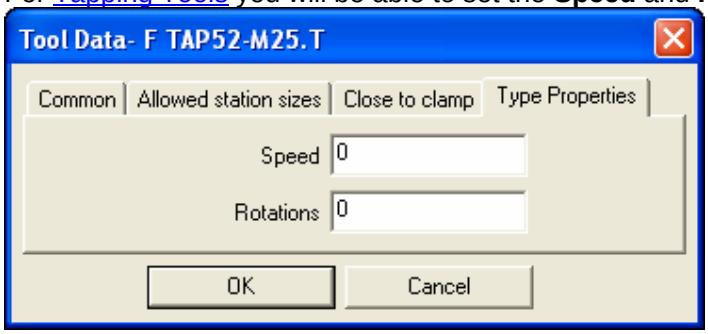
And the punch:



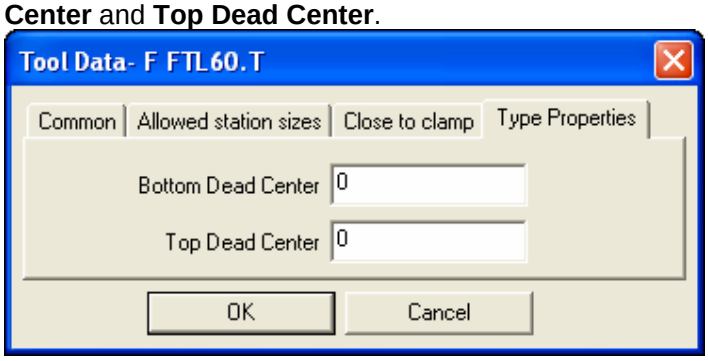
3.1.1.2.4 Type Properties

For some of the **Special Tools** you can set an additional information in **Tool Data**, which is **Type Properties**. They vary according to the type of the tool:

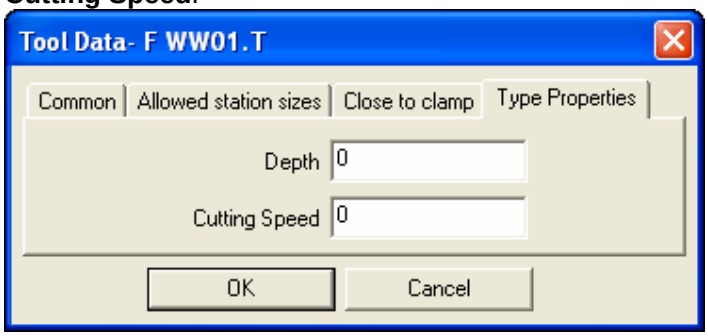
- For Tapping Tools you will be able to set the **Speed** and **Rotation**.



- For Forming Tools you will be able to define the values of **Bottom Dead Center** and **Top Dead Center**.



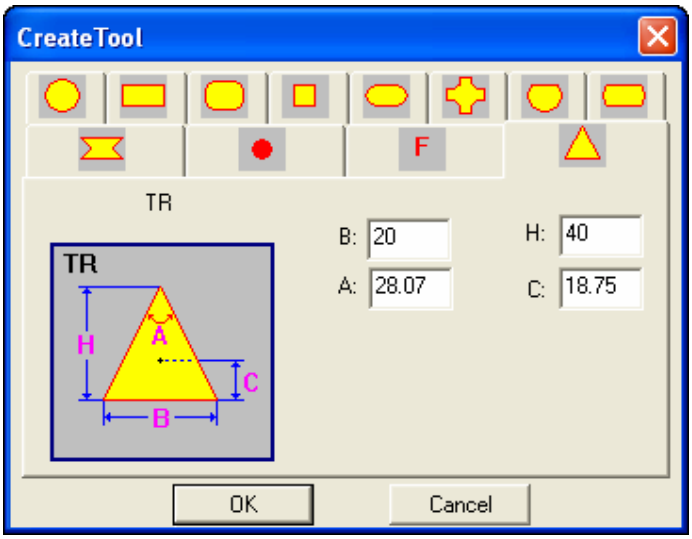
- For Wilson Wheel Tools you will be able to set the **Depth** of the CAM and the **Cutting Speed**.



3.1.1.3 Create Tool

After clicking the **Create Tool…** button you will be presented with a dialog where you can define the exact tool you need.

On the example presented below, you can see how easy it is to create a **Triangle** tool:



All you need to do is enter the desired values to proper fields of **base** (**B**) and **height** (**H**). When you do this, the **angle** (**A**) and the triangle's **center** (**C**) will be calculated automatically by ***cncKad***.

The **center** value here is the center of the triangle's bounding circle.

If you are using Milling machine, you will be able to see also **Drill** and **Mill** tabs.

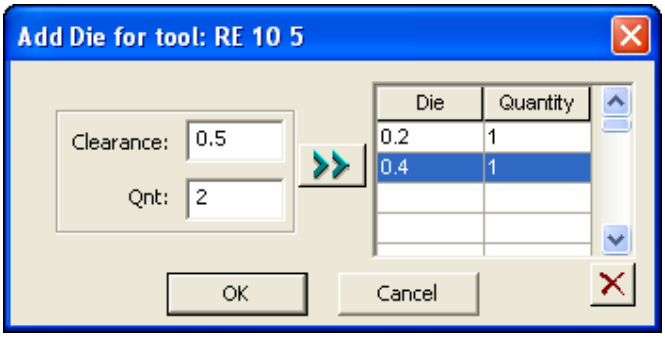
3.1.1.4 Tool Table

In this chart you will find eleven tabs containing various tool types. Go to **Tools Definitions** section of this chapter, to see what kind of tools you can find on each of these tabs.

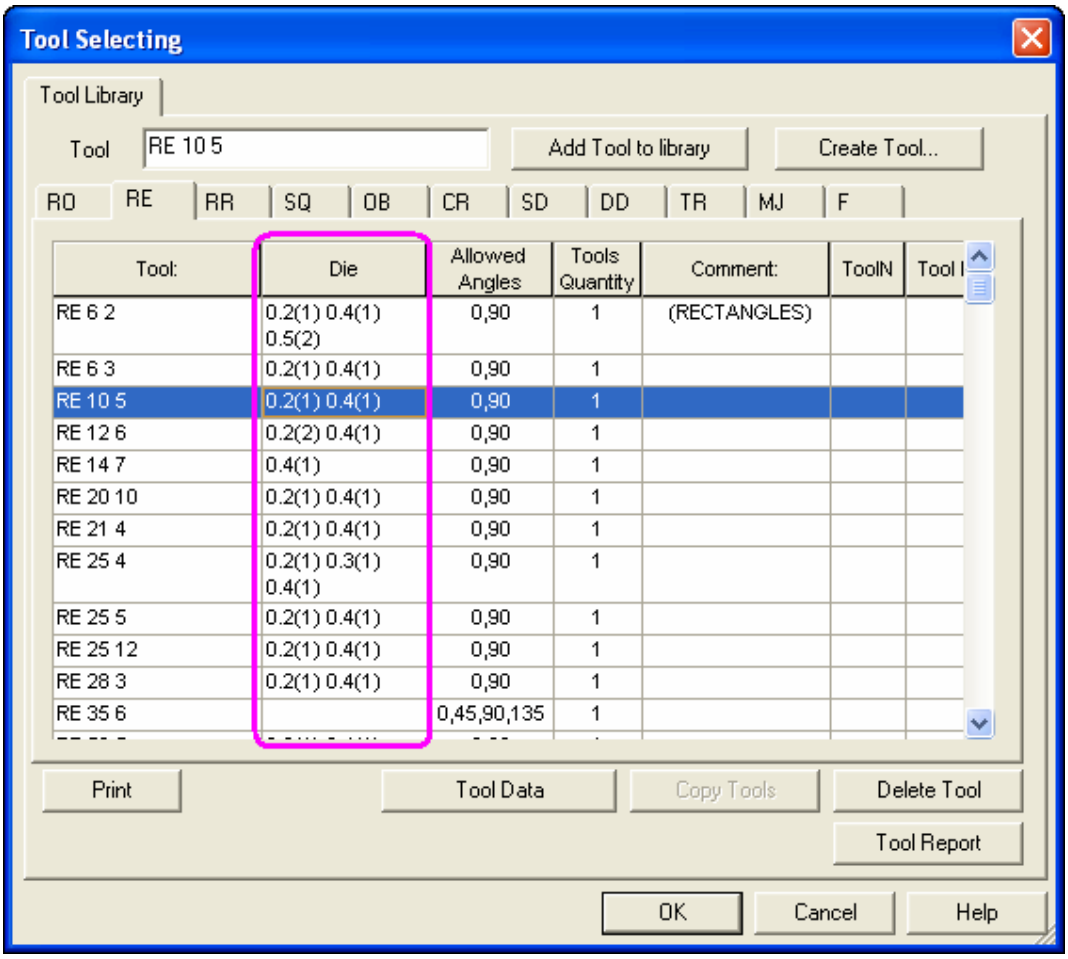
In order to choose the tool you desire to use for your part's processing, click on it in **tool column**, or type its name in the **tool field**, at the top of the window, and then confirm your choice clicking **OK** button.

3.1.1.4.1 Die Selection

The user has a possibility to define **Dies** for tools. In order to do so, click on the row with the tool you are interested in, in the **Die** column. The following dialog will open:

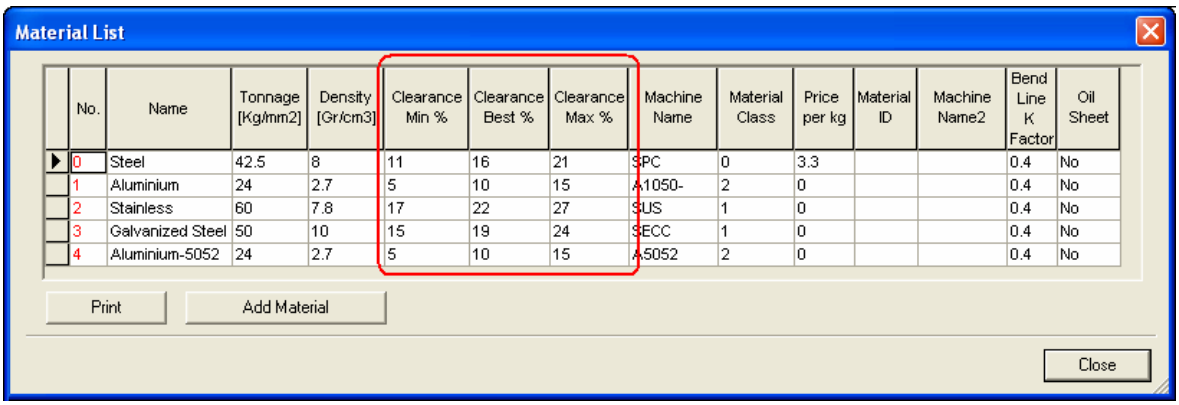


Here you will be able to define the die's clearance and quantity. You also will be able to edit the already existing ones. Click the  button, to add your definitions to the list, or the  button to delete the existing ones. Then click **OK** button to return to the Tool Library. There you will be able to view the dies' list in the appropriate section:

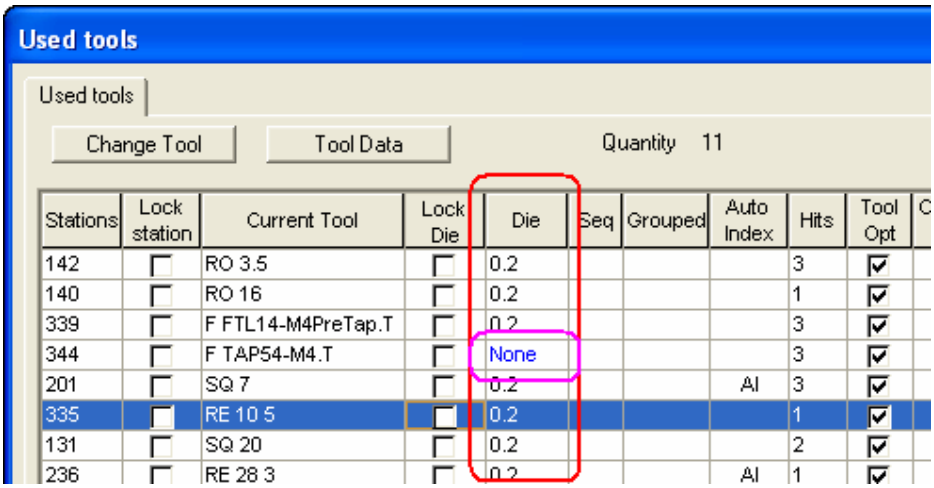


As you can see, each tool has its own die list.

cncKad checks what material you are working with, and what is its thickness. Then according to these parameters, and after verifying in **Material List** (see **Settings =>** Workspace settings **=> Material tab => Edit material**), what are the **Minimal**, **Best** and **Maximal Clearances** for that material ***cncKad*** will decide which of the dies will be used for current tool.



The results of which die had been chosen for which tool, you see in Die column of Used Tools dialog:



Die's clearance will not be below or above the minimum and maximum percent clearances of that sheet's thickness, and it will be as close as possible to the **Best** clearance defined. If the program will not find the proper die for some tool, in a **Die** column a word **None**, highlighted in blue, will be displayed.

3.1.1.4.2 Allowed Angles

In this column you will be able to see the angles in which tool will be allowed to work. Clicking on this column next to chosen tool, you will be able to change, choosing one of the dropdown list options, the angles for this tool.

3.1.1.4.3 Tools Quantity

In this column you will be able to see the quantity of instances of specific tool, which exist in your library. Clicking on this column next to specific tool, allows you to change this number.

3.1.1.4.4 Comment column

The comments you entered on **Common tab** of Tool Data dialog will be visible here. They can be edited by clicking on this column next to desired tool row.

3.1.1.4.5 ToolN column

If in the **Common tab** of Tool Data dialog you assigned a specific number to any of your tools, it (Tool Number) will be visible in this column. You can edit and change it from here, simply clicking on the chart.

3.1.1.4.6 Tool ID column

This option is designated for Trumpf machines.

cncKad creates automatically the Tools ID for the tools which don't have ones. From here you can enter the Tool ID's manually, edit and change them, by simply clicking this column next to desired tool row.

If on the **Common tab** of Tool Data dialog you entered a Tool ID, it will be seen in this column.

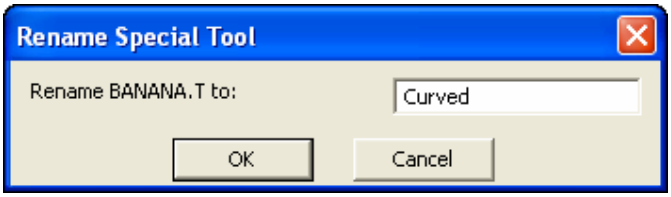
3.1.1.5 Printing the Library

When you click the **Print** button the printers settings will appear. Set printing definitions according to your wish, and the whole Library will be sent to your printer.

3.1.1.6 Renaming a Tool

This button is available only for the special tools tab.

Clicking the **Rename** button you will be presented with the following dialog:



Enter the name you want the selected tool to be called by and click **OK** button. The following warning will appear:

That means – if you have used this special tool on your part, before renaming it, it will still contain the old name as long as you will be using it on the current part. When you will use it on a new part – it will be called by its new name.

3.1.1.7 Tool Data

Clicking this button will bring you to the same dialog, which appears while you are adding a new tool to the library. Here you will be able to add or/and edit tool data.

For more details on this dialog, go to: **Add tool to library** section of current chapter.

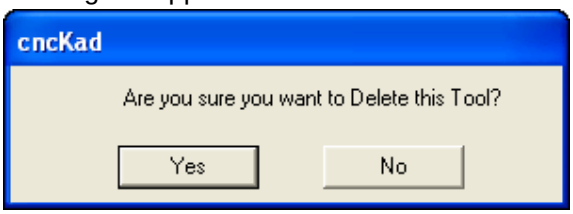
3.1.1.8 Copying Tools from one Library to another

Tools can be **copied** from one tool library to another. In order to do that, select the Tool(s) you wish to copy to another Library, by the help of regular Windows selection conventions (hold down **Shift** key while selecting different tools located one next to the other, hold down **Ctrl** key while selecting tools which are far from one another), and then click the **Copy Tools** button.

A dialog will appear, offering you to copy selected **Tools** into one of the existing **Private Libraries**. Select the Target Library and confirm your choice, clicking **OK** button. The tools will be copied to the designed Target Library.

3.1.1.9 Deleting a Tool

Select the tool you wish to delete, and click the **Delete Tool** button. The following warning will appear:

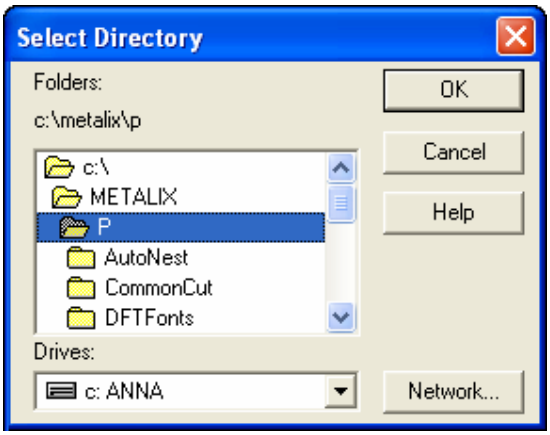


After confirming the action, the tool will be removed from the library.

3.1.1.10 Tool Report

This feature enables you to find all the DFT files on which a specific tool has been used for processing. In our example we will look for **RO 10** tool.

After pressing the **Tool Report** button the following dialog opens:



Choose the directory in which you want to search and confirm your choice clicking **OK** button.

The following **Tool Report** will display all the instances of the tool found in the directory we selected:

Tool: RO 10

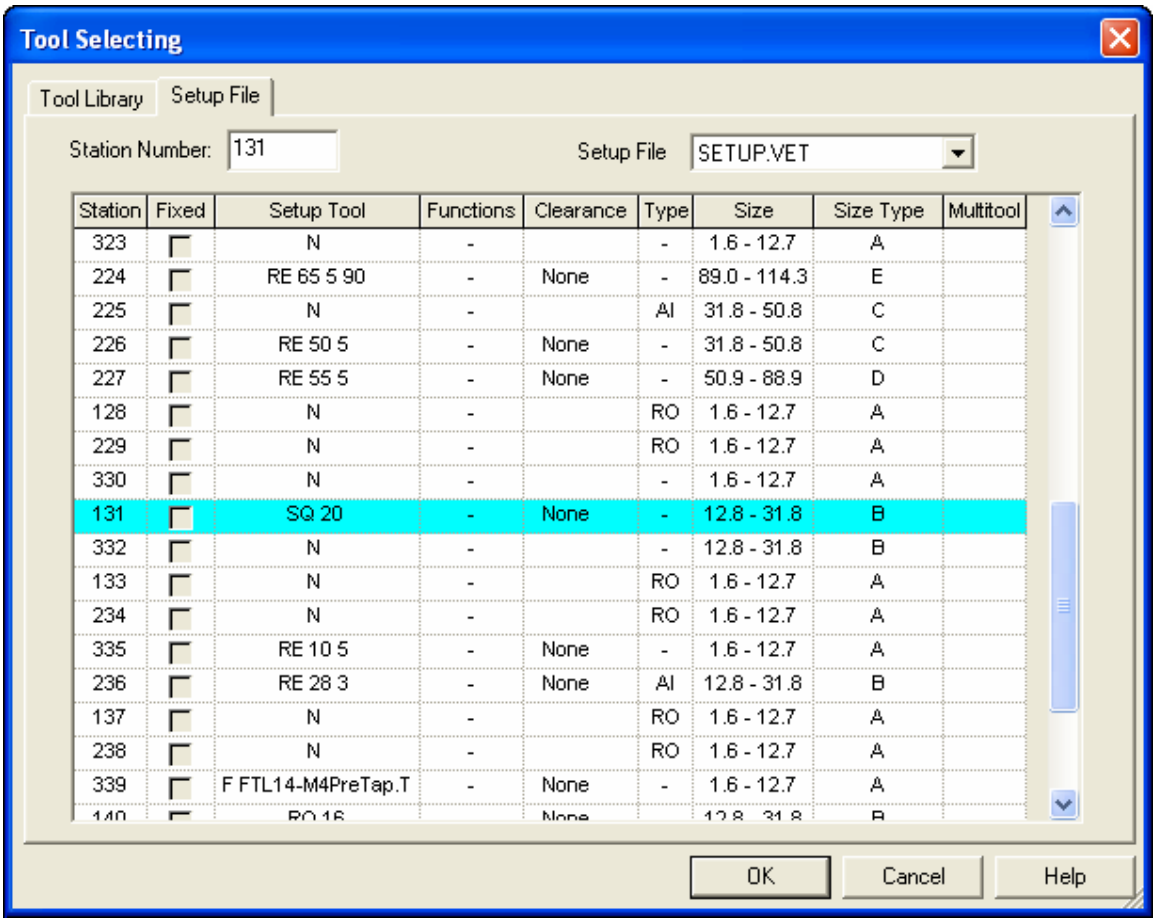
Customer	Path	Ver.	DrawNum
	C:\Metalix\P\Ex_Amada\demoWheel1.dft		
	C:\Metalix\P\Ex_Amada\demoWheel2.dft		
	C:\Metalix\P\Ex_Amada\demo11.DFT		
Green Frog	C:\Metalix\P\Ex_Amada\demo13.DFT	abc	123-456-789
Green Frog	C:\Metalix\P\Ex_Amada\demo16.DFT	321	000-123-456
	C:\Metalix\P\Ex_Amada\Demo8.DFT		
	C:\Metalix\P\Ex_Amada\demo9.DFT		
	C:\Metalix\P\Ex_Amada\demo14a.DFT		
	C:\Metalix\P\Ex_Amada\ppp.DFT		

If no DFT files with such tools were found, no Tool Report will be displayed.

3.1.2 Setup file tab

This tab is visible from CAM Menu **=> Punch CAM => Add Punch**, when you click the tool button, intending to choose a tool to process your part with. It is also accessible from **Tool Selection Button** of the State Bar (see the Introduction chapter of ***Drafting and Nesting Manual***).

The tools displayed here are being taken from the Turret Setups database.

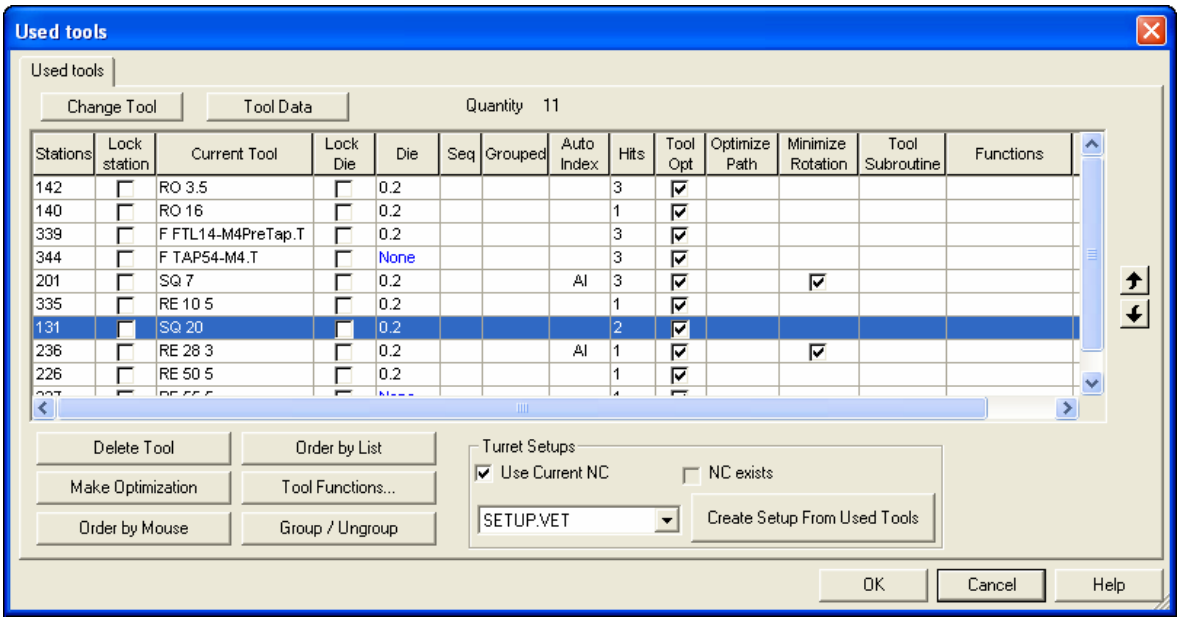


In the **Station Number** field you will be shown the number of the station of the tool you chose from the table. The **Setup File** dropdown list allows you for selecting desired setup file from available ones for currently used machine.

More information on other issues presented in this tab you will find in Edit Existing Setup section of this chapter.

3.2 Used Tools...

This option opens a dialog, which shows all the tools currently used for processing your part:



3.2.1 Setting Stations for Tools

There are two ways to set a specific station for a tool:

- If you know the station number you desire, click the **Stations** field of a specific tool, type-in the number and move on to the next tool.
- If you wish to choose from the stations' list, click the tool's **Stations** field. You will be presented with a dropdown list of the stations and the tools assigned to them:

Stations	Lock station	Current Tool	Lock Die	Die	Seq	Grouped	Auto Index	Hits	Tool Opt	Optimize Path	Minimize Rotation
201	☐	SQ 7	☐	0.2			AI	3	☑		☑
142	☐	RO 3.5	☐	0.2				3	☑		
344	☐	F TAP54-M4.T	☐	None				3	☑		
140	☐	RO 16	☐	0.2				1	☑		
339	☐	F FTL14-M4PreTap.T	☐	0.2				3	☑		

Station	Setup Tool	Used Tool	Fixed
335	RE 10 5	RE 10 5	☐
236	RE 28 3	RE 28 3	☐
137	N	-	☐
238	N	-	☐
339	F FTL14-M4PreTap.T	F FTL14-M4PreTap.T	☐
140	RO 16	RO 16	☐

Choose the station you prefer and click elsewhere in the dialog.

3.2.2 Change Tool

With this option the user will be able to replace the current tool with another one. When you click this button, you will be redirected to **Edit Tools Library** dialog and there you will be able to choose a new tool.

3.2.3 Tool Data option

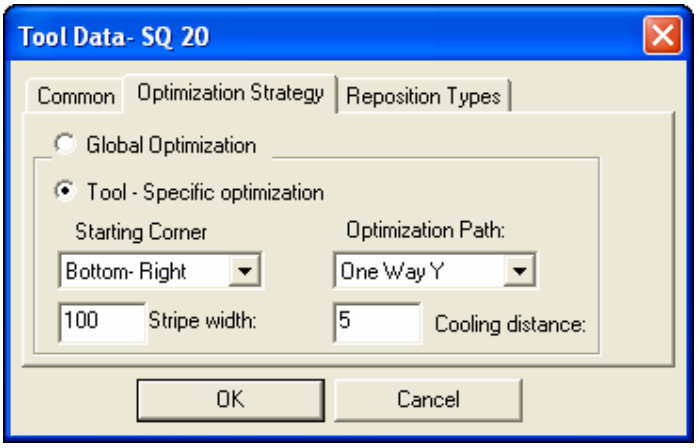
Here you can also add or/and edit **Tool Data**. But this dialog varies from the one in Edit Tools Library window.

3.2.3.1 Common

This tab differs from the **Common tab** described in **Add Tool to Library** section of this manual. In this tab user can only edit Tool Number and Tool ID and can add his Comment.

3.2.3.2 Optimization Strategy

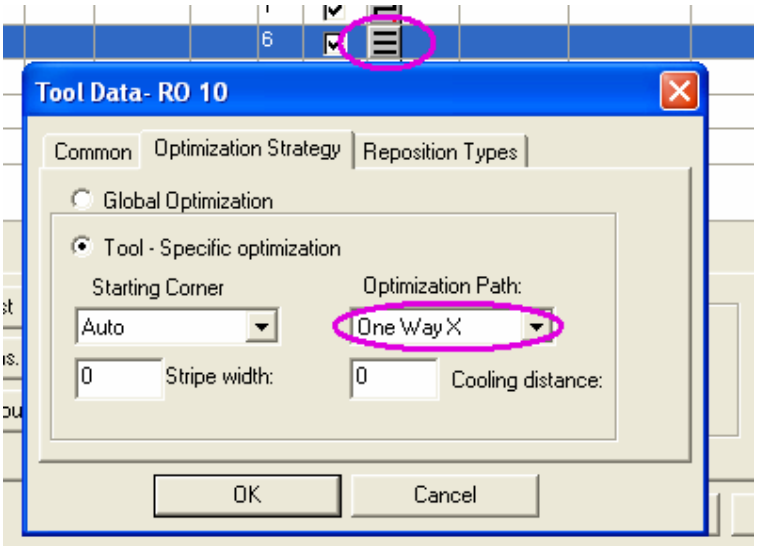
In this tab you have the option to determine the optimization strategy for each tool individually. You can get to this dialog also by clicking chosen tool's field of **Optimize Path** column.

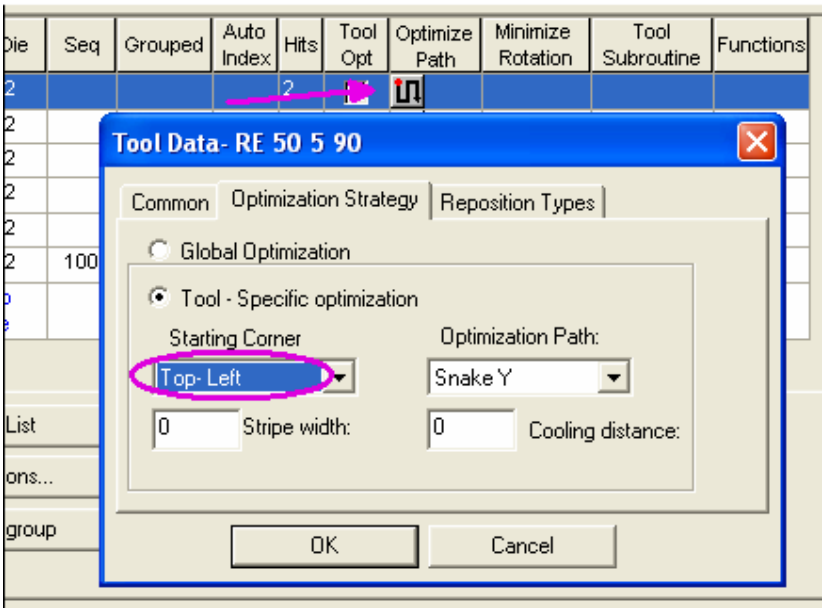


You can choose one of two options:

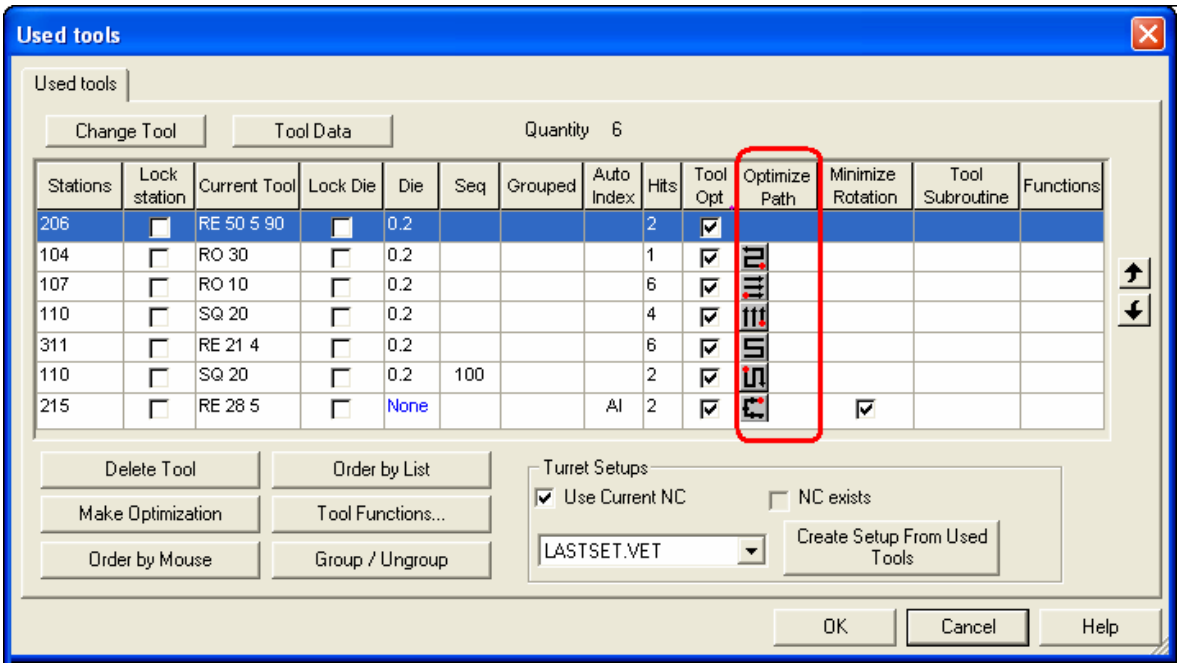
- **Global Optimization**, in which case the parameters will be those defined for the part.
- **Tool–Specific optimization**, where you have two parameters with which you will be able to determine your strategy:
  - **Starting Corner** – in this dropdown list there are five options of setting the initial processing corner: **Auto**, **Bottom-Left**, **Bottom-Right**, **Top-Left** and **Top-Right**. Selecting one of them will determine the place of cut entrance onto the contour.
  - **Optimization Path** – in this dropdown list you will find six variations of optimization paths: **Fastest**, **Snake Y**, **Snake X**, **One Way Y**, **One Way X** and **User defined**. Choosing one of them determines the way the cut will be made.
  - **Stripe width** – in this field you can set the width of the Stripes, that will be optimized, before the machine will move to the next stripe.
  - **Cooling distance** – here you can define the minimum **Cooling Distance** between the future cuts.

For each choice you make, you will be shown a corresponding icon in the tool's record on the **Used Tool** dialog:





After you finish defining all the tools, you will see the corresponding icon for each tool's strategy:



3.2.3.3 Reposition Types

There are three options of reposition types:

- Global
- Max at 1st grip
- Max at Last grip

For more information on these options, go to **Max/Min Processing** section of **CAM** Menu chapter of this manual.

3.2.4 Delete Tool

Clicking this button removes selected tool from the current processing.

3.2.5 Make Optimization

This option activates the optimization algorithm, which recalculates the processing sequence making any changes, e.g. defining tool-specific optimizations.

3.2.6 Ordering the CAMs

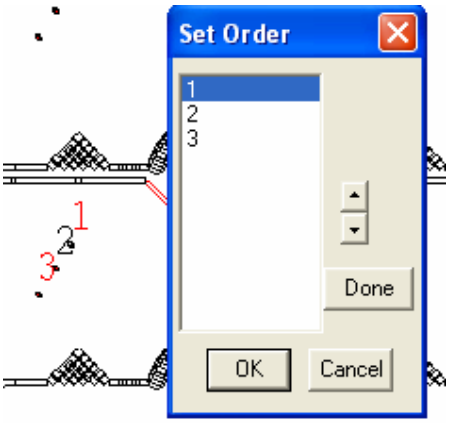
After placing your CAMs, you can rearrange the order in which the entities will be processed.

Mark one of the tools in the dialog, and then click one of the following buttons:

- **Order by List** – to rearrange the processing according to a list of the entities processed by the tool.
- **Order by Mouse** – to graphically rearrange the processing on the part itself.

3.2.6.1 Ordering by List

You will be presented with your part and a list of the entities processed with selected tool (the number of entities is shown in the **Hits** field of **Used Tools** dialog):



As you move the cursor inside the list, each corresponding entity will be highlighted.

To change the order of the processing, mark an entity's number and with the arrows move it up or down inside the list. When you move to the desired position, click **Done** button to see the change. After you finish setting the order for all the instances of the tool, confirm changes clicking **OK** button. If you want to resign from changes, click the **Cancel** button.

3.2.6.2 Ordering by Mouse

After clicking this button you will be presented with your part and with marked and numbered entities processed by this tool.

To determine the ordering, click the entity you wish to be processed *first*, then the *second*, and so on. After arranging the processing sequence of entities, you will be returned to the previous dialog.

3.2.7 Sorting by Stations

You can sort the tools, used on the part, by their stations, clicking **Stations** column heading.

The tools will be organized in a sequence designated to minimize turret rotation – the trimming tool will be placed last, then the tool nearest to it, then the tool nearest to that one, and so on for the rest of the tools.

3.2.8 Sorting by Type

You can also sort the tools by their types, clicking **Current Tool** column heading. The tools will be grouped by round, oblong, rectangular (which includes also square tools) shapes etc. Within each group the tools will be arranged by their size.

3.2.9 Locking Stations

If you want the tool to be assigned to the station no matter what, you can lock/un-lock all the stations with a click on the **Lock Station** column heading.

3.2.10 Die Selection in Used Tools

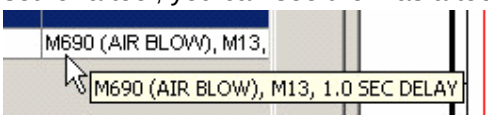
In the **Die** section of the **Used Tools** dialog, the most appropriate die (if such exists) out of the dies' list entered in the **Tool Library** is displayed for each tool.

The considerations taken into account for the choice of the suitable die depend on the **sheet's thickness** and on the definitions entered in the **Material list** for the current Sheet type (see **Workspace Settings** dialog **=>** Material tab **=>** **Edit material** button): the **Die**'s clearance cannot be below or above the **minimum** and **maximum percent** clearances of that sheet's thickness, and it will be as close as possible to the **Best Clearance** defined.

However, if you wish to change this decision, or to enter a die for a tool directly into the **Used Tools** dialog, just click on the **Die** column of selected tool, ignore the dropdown list that appears and simply type-in the desired clearance. As you do, the **Lock Die** checkbox (in the nearby column) will be automatically checked, meaning that the value just entered will be kept.

3.2.11 Tool Functions…

This option allows you to set functions for a specific tool. They will be enumerated in the **Functions** column; when there is not enough room to preview all the functions set for a tool, you can see them as a tool-tip when you lower the cursor over the cell:



For more information on this subject, please refer to the **Defining Tool Functions** section of CAM Menu chapter of this ***Manual***.

3.2.12 Group/Ungroup

This option allows you to group together a number of tools, making them subject to the same optimization strategy. This feature is especially practical in those instances where it is important to use a number of tools in the same sequence, **for example:** when you are using two (or more) forming tools, where it is advisable to punch from the clamps outwards and not make several passes. In such a case you can group these forming tools together, and set an **Optimization Strategy** of *Snake X* and **Starting Corner** of *Bottom Left*, both tools will be optimized together.

3.2.13 Turret Setups section

This section of the **Used Tools** dialog deals with the usage of **Turret Setups** and NC files:

- **Use Current NC** – checking or unchecking this option determines if an existing NC file will be used instead of creating a new one.
- **NC exists** – this check-box in not available for editing. It indicates if there is an existing NC file to use for current DFT.
- **Setup Window** – this dropdown menu allows you to choose which setup file, from existing ones, to use.
- **Create Setup From Used Tools** – clicking this button allows user to save current list of used tools as a new tools setup.

3.3 Create a Special Tool File

When you choose this option you are in fact attempting to define a new tool.
Before doing this you need to follow a few steps:

1. View the Tool Definitions.
2. Follow the directions outlined in Creating User Defined Tools.
3. Make sure to view the rules for naming tool files.

3.3.1 Tool Definitions

There are several types of tools, each defined by the shape of the tool and the parameters that define it.
The following describes the parameters for defining the various tools:

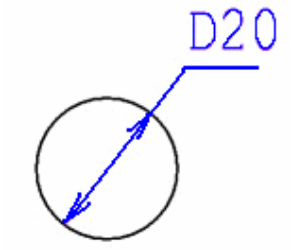
- **RO** – Circular tools
- **RE** – Rectangular tools

- **RR** – Rounded Rectangle
- **SQ** – Square tools
- **OB** – Oval or Oblong tools
- **TR** – Triangular tools
- **SD** – D shaped tools (Single D)
- **DD** – Double-D shaped tools
- **CR** – Corner Radius tools
- **MJ** – Micro Joint tools
- **F** – Special tools, user defined

3.3.1.1 Circular Tool

Definition: **RO 20**

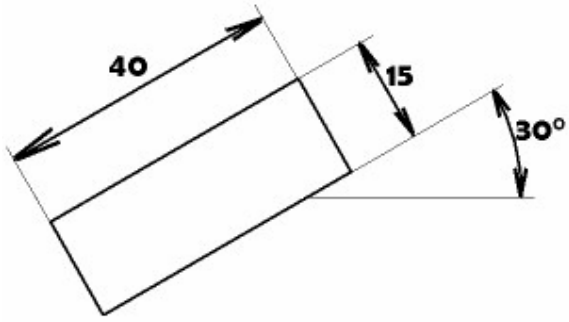
Where 20 is the circle diameter.



3.3.1.2 Rectangular Tool

Definition: **RE 40 15 30**

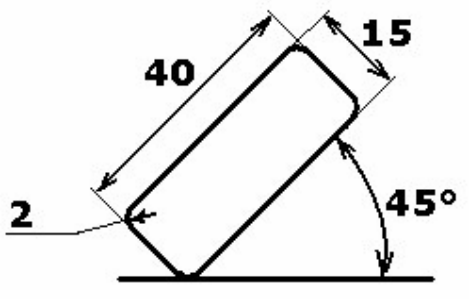
Where 40 is the length, 15 the width and 30° the declination of the Rectangle (if you do not define an angle, it will be set as 0°).



3.3.1.3 Rounded Rectangle

Definition: **RR 40 15 2 45**

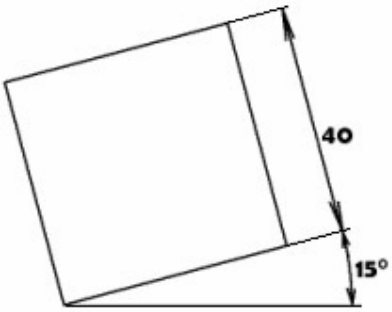
Where 40 is the length, 15 the width, 2 the radius of the corners and 45° the declination of the Rectangle (if you do not define an angle, it will be set as 0°).



3.3.1.4 Square Tool

Definition: **SQ 40 15**

Where 40 is the side and 15° the declination of the Square (if you do not define an angle, it will be set as 0°).



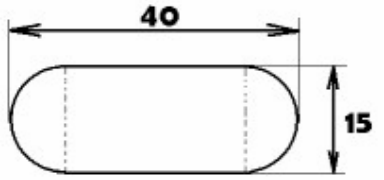
Usually, the angle is left at 0°, or defined as 45°.

3.3.1.5 Oval Tool

Ovals are rectangles, with half-circles as two of the sides.

Definition: **OB 40 15**

Where 40 is the length of the oval 15 is the width (the diameter of the arcs).

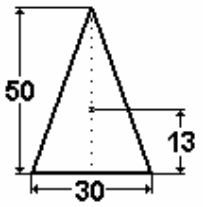


You can also give this tool a declination, e.g. **OB 40 15 <u>45</u>**.

3.3.1.6 Triangular Tool

Definition: **TR 30 50 13**

Where 30 is the basis, 50 is height and 13 is the center of the triangle's bounding circle.

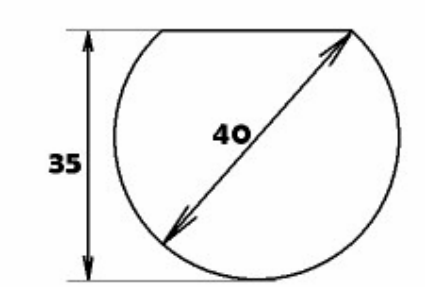


3.3.1.7 D Shaped Tool

D shapes are cut circles, and the D-shape will be located on the part by its straight side, not the circular one!

Definition: **SD 40 35**

Where 40 is the diameter and 35 is the width, measured from the straight side to the opposite rim of the circle.



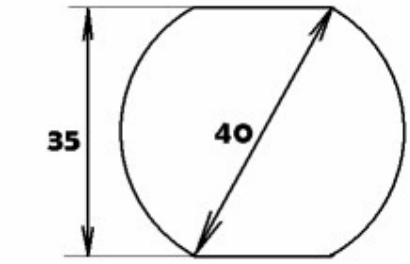
You can also give this tool a declination, e.g. **SD 40 35 <u>45</u>**.

3.3.1.8 Double D Shaped Tool

Double-D shapes are circles cut on opposite sides, and the shape will be located on the part by its straight side, not the circular one!

Definition: **DD 40 35**

Where 40 is the diameter and 35 is the width, measured from one straight side to the opposite one.



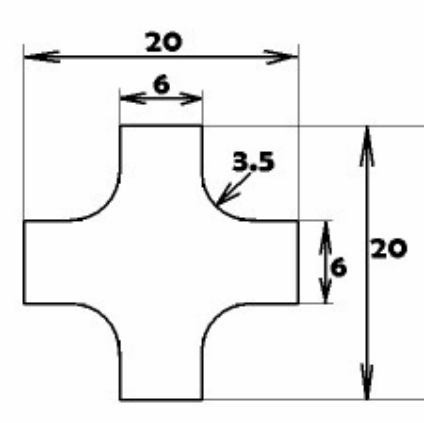
You can also give this tool a declination, e.g. **DD 40 35 <u>45</u>**.

3.3.1.9 Corner Radius Tool

The main use for these tools is to punch Fillet outlines.

Definition: CR 3.5 20 20 6 6

Where 3.5 is the fillet diameter, 20 horizontal length, 20 vertical length, 6 horizontal cutting width and 6 vertical cutting width.



3.3.1.10 Special Tools

While adding a new special tool to the library it is important to pay attention to naming them. It is substantial, since various machines "understand" only the precise naming and if the tools won't be given proper names, the machine will not be able to find them in the library.

The standard tool library supplied with ***cncKad*** contains a variety of special tools like:

- Beading Tools
- Embossing Tools
- Slitting Tools
- Forming Tools
- Tapping Tools
- Wilson Wheel Tools and others.

For more information on the special tools see the **Names for Special Tool Files** section of this chapter.

3.3.2 Creating User Defined Tools

Here you will learn how to define a **special tool** whose parameters do not fit the ones available in the **Tool Library**. In this example we will be creating a new radius tool. All you have to do is to follow procedure described below.

This procedure will be divided into three stages:

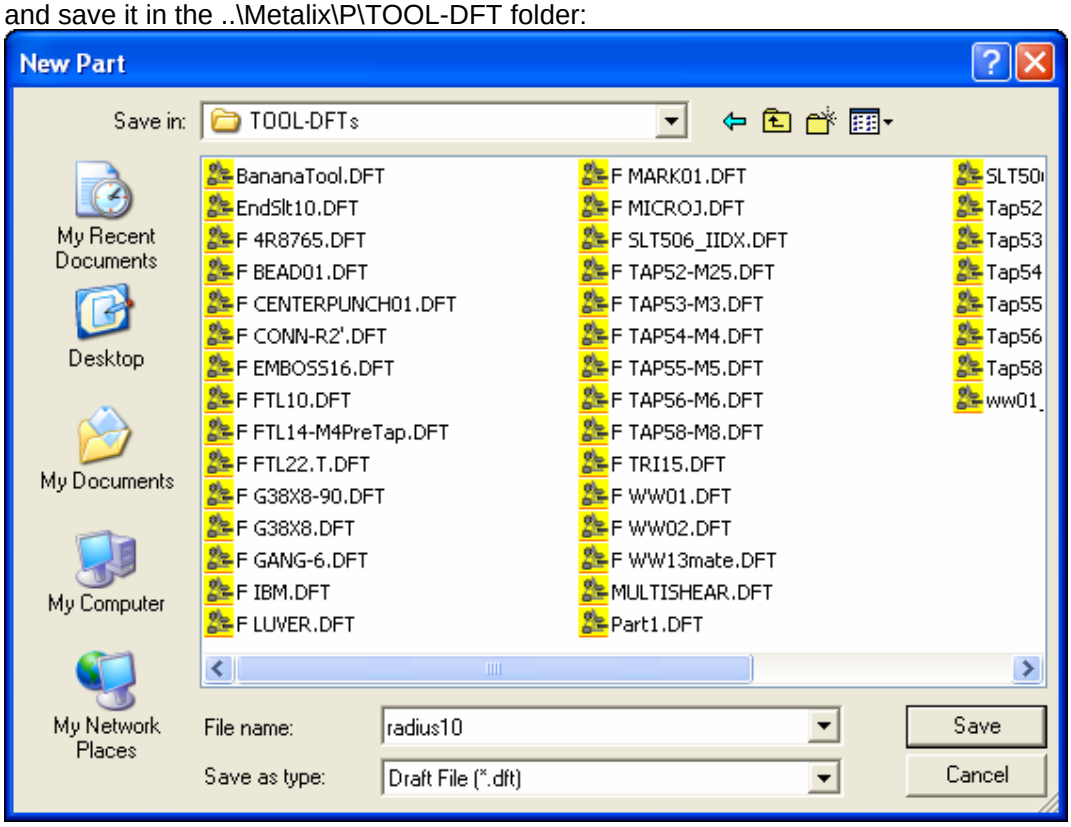
1. Creating the outline of the tool
2. Adding the new tool to the Tool Library
3. The Tool Reference Line

And at the end we will Test the New Special Tool we just created.

3.3.2.1 Creating the outline of the tool

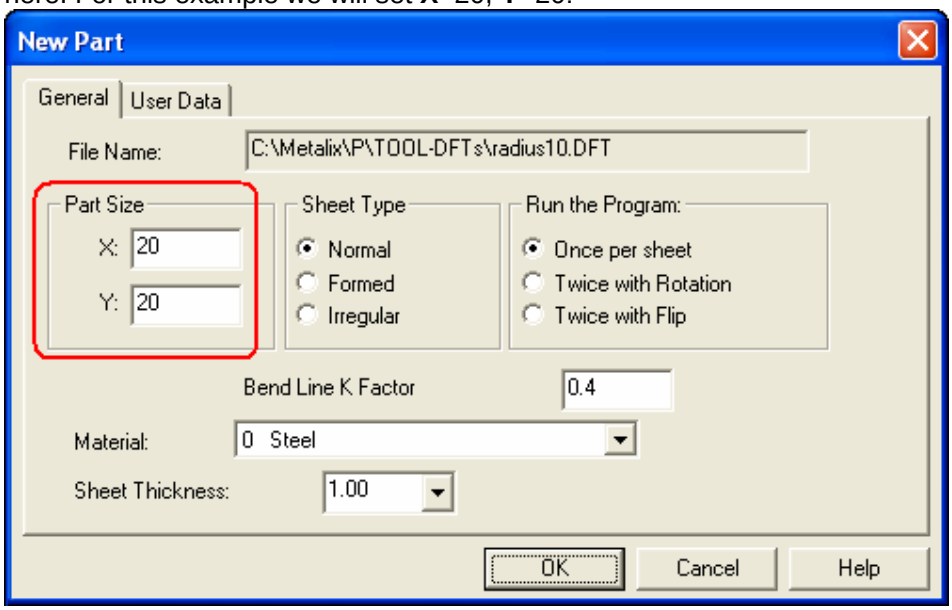
The first stage of creating a new tool is to draw its outline of the tool as a regular DFT part.:

1. Open a new part. Give it the name of the intended tool, e.g. "radius10.dft", and save it in the ..\Metalix\P\TOOL-DFT folder:

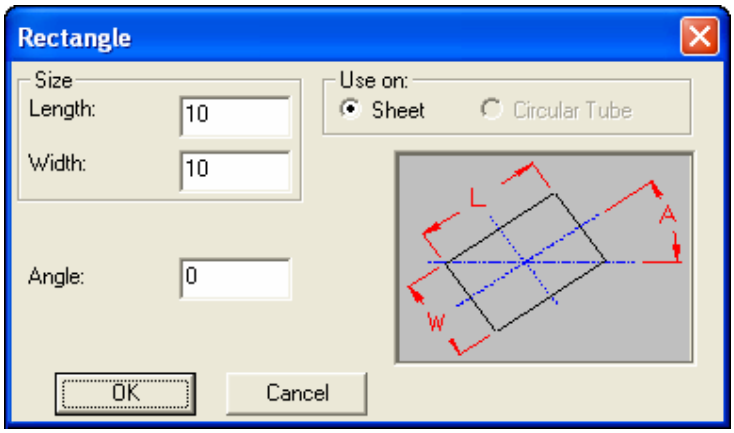


Important: The name **must not** contain spaces.

2. The size you define here is not really important, so you can enter any values here. For this example we will set **X**=20, **Y**=20:

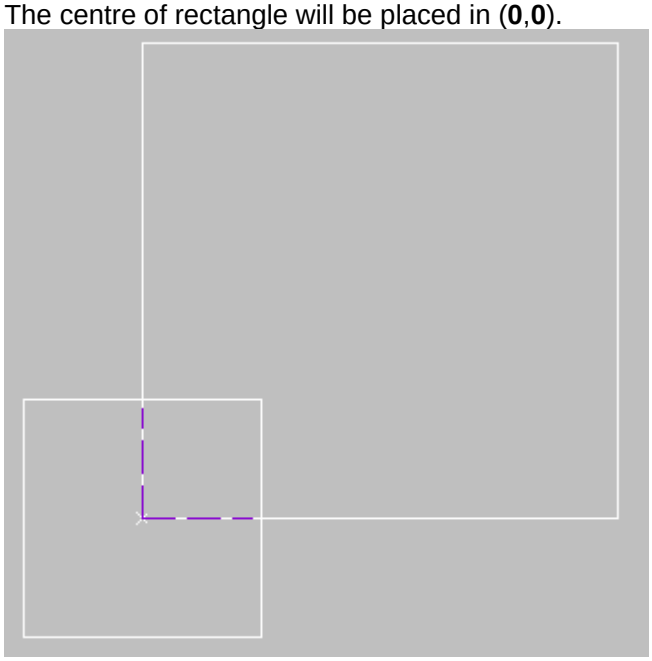


3. Now from **Shapes Toolbar** we will choose the **Draw Rectangle** option. The following dialog will open:

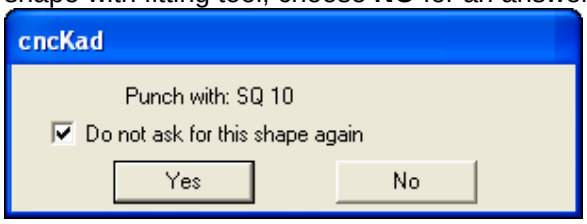


Type the values as presented in the dialog, and click **OK** button.

4. The center of highlighted, with blue color, rectangle will appear on the cursor crosshairs. Click the cursor at the beginning of the origin coordinates (**0**,**0**). The centre of rectangle will be placed in (**0**,**0**).

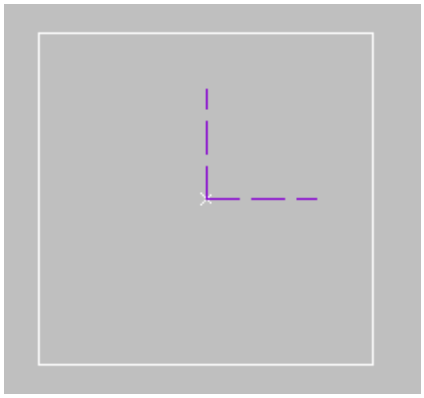


5. If the following dialog will appear asking you whether you want to punch the shape with fitting tool, choose **NO** for an answer.

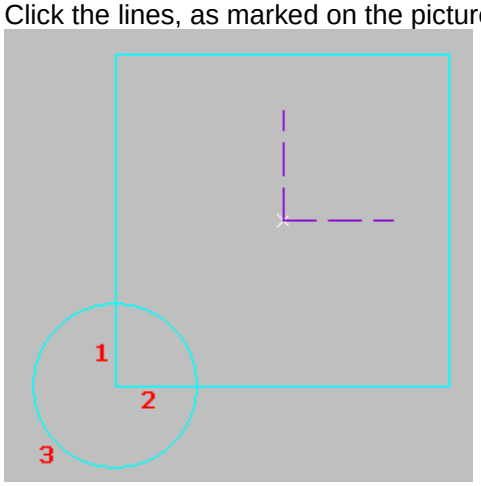


6. Press **Esc** key, to leave the **Draw Rectangle** command.
7. Now delete the bigger contour, click the  button, and then refresh the drawing, clicking the  button. This is the picture you should see on your

screen now:



8. Now from the **Shapes Toolbar** select **Draw Circle** button. Type the diameter value as 5 and click **OK** button.
9. Now with the circle's middle point click the right bottom corner of the rectangle. And then press **Esc** key, to leave the **Draw Rectangle** command.
10. Refresh the drawing, clicking the  button. Now from **Edit Toolbar** select the **Trim** button. The drawing will be highlighted in blue color.
11. Click the lines, as marked on the picture below:

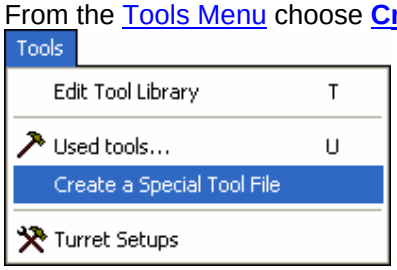


Now press **Esc** key to escape the command, and **Save** your drawing.

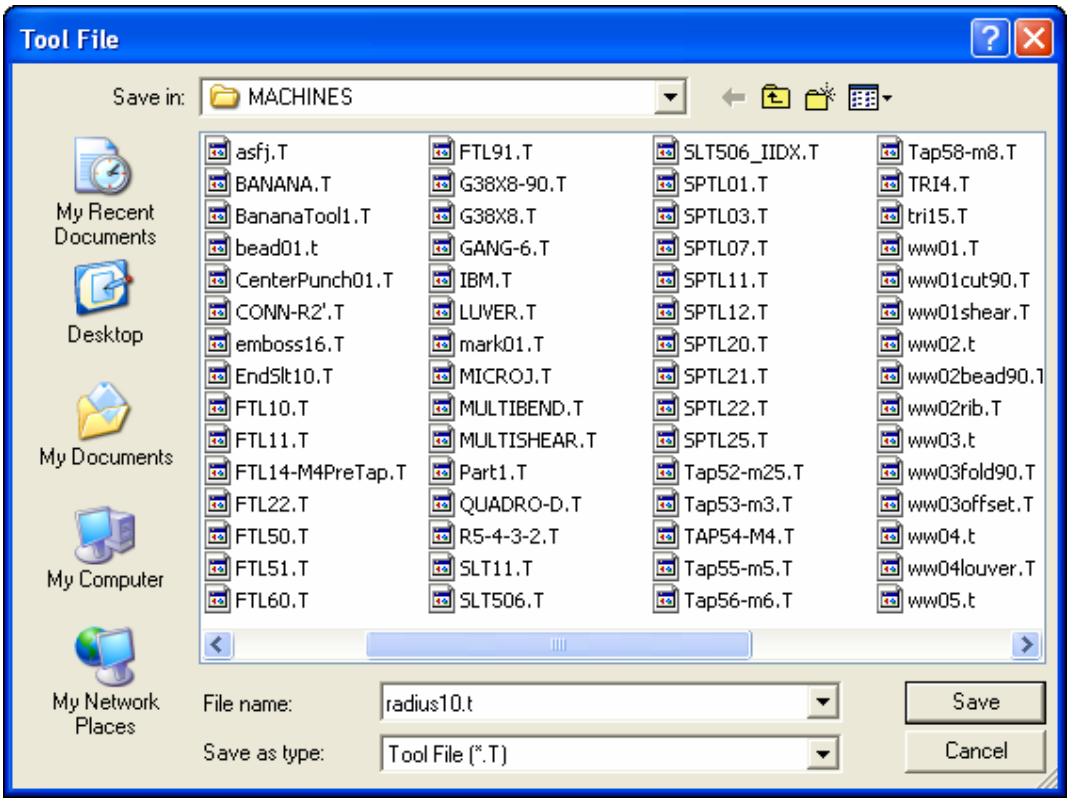
3.3.2.2 Adding the new tool to the Tool Library

Now we will add the new tool to the **Tool Library**:

1. From the Tools Menu choose **Create Special Tool File**:



2. The following dialog will open:



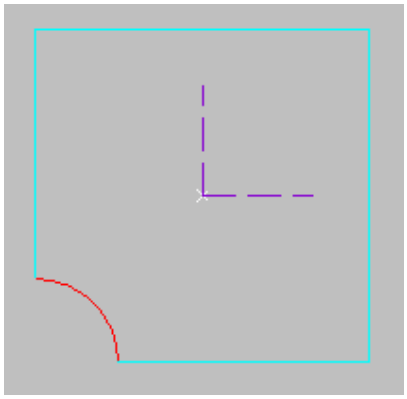
Note that the name of the new tool is the same as the one you defined for the DFT part, and that it will be saved in the ..\Machines folder.

3. Don't forget to **Save** the new tool.

3.3.2.3 The Tool Reference Line

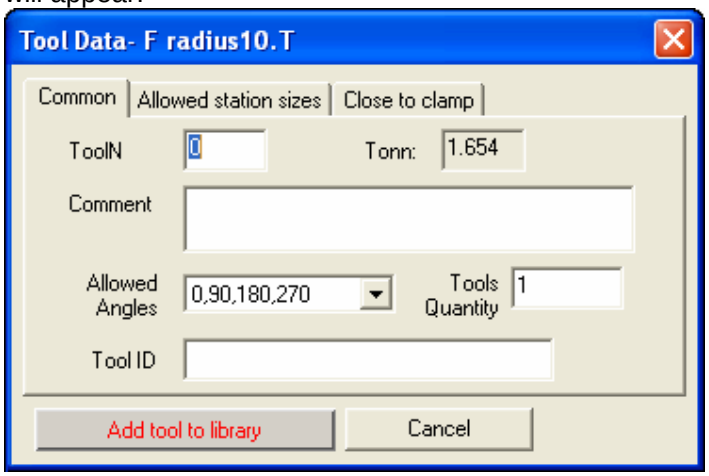
Now you must choose the entity of the tool – the **Tool Reference Line**, according to which the punch with this new tool will be placed on the part.

1. Click on the radius and it will be marked in red, and the rest of the tool will be marked in blue.

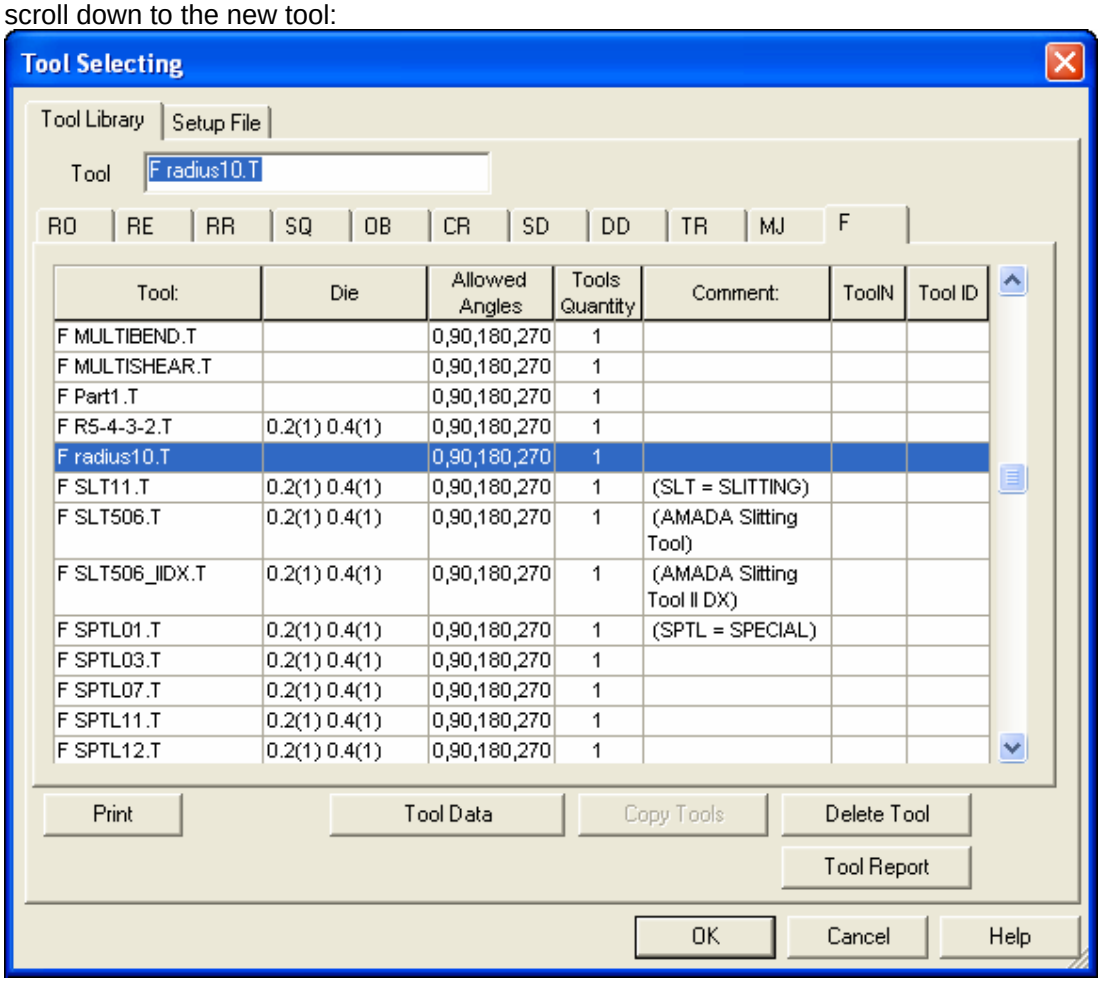


Important: If the tool includes more than one contour (such as a *Cluster Tool*), click all of them.

2. Press **Enter** key to confirm the chosen contour(s), and the following dialog will appear:

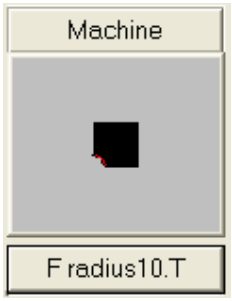


3. Click **Add to tool library** button, and the tool will be added to the **Tool Library**.
4. Check that you have indeed added a new tool to the library: Click the **No Tool** button on the State Bar, and the Tool Library will open. Go to the **F** tab and scroll down to the new tool:



Tool:	Die	Allowed Angles	Tools Quantity	Comment:	ToolN	Tool ID
F MULTIBEND.T		0,90,180,270	1			
F MULTISHEAR.T		0,90,180,270	1			
F Part1.T		0,90,180,270	1			
F R5-4-3-2.T	0.2(1) 0.4(1)	0,90,180,270	1			
F radius10.T		0,90,180,270	1			
F SLT11.T	0.2(1) 0.4(1)	0,90,180,270	1	(SLT = SLITTING)		
F SLT506.T	0.2(1) 0.4(1)	0,90,180,270	1	(AMADA Slitting Tool)		
F SLT506_IIDX.T	0.2(1) 0.4(1)	0,90,180,270	1	(AMADA Slitting Tool II DX)		
F SPTL01.T	0.2(1) 0.4(1)	0,90,180,270	1	(SPTL = SPECIAL)		
F SPTL03.T	0.2(1) 0.4(1)	0,90,180,270	1			
F SPTL07.T	0.2(1) 0.4(1)	0,90,180,270	1			
F SPTL11.T	0.2(1) 0.4(1)	0,90,180,270	1			
F SPTL12.T	0.2(1) 0.4(1)	0,90,180,270	1			

Whenever you choose a special tool to process with, its representation will appear on the **Tool Display**, with the **Tool Reference Line** highlighted in red:

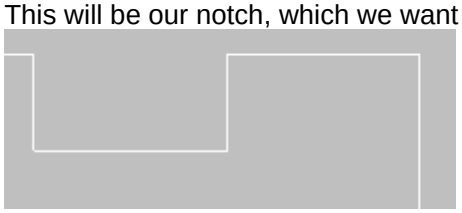


There also is an option of changing the **Tool Reference Line** location: toggle *on* the **Index** button on the **Info bar** and then press the **S** key: the red line on the tool indicating the **Tool Reference Line** will move to the next side of the tool. Press the **S** key until you are satisfied with the cutting line's location on the tool and then proceed with the punch. If needed, the tool will be indexed.

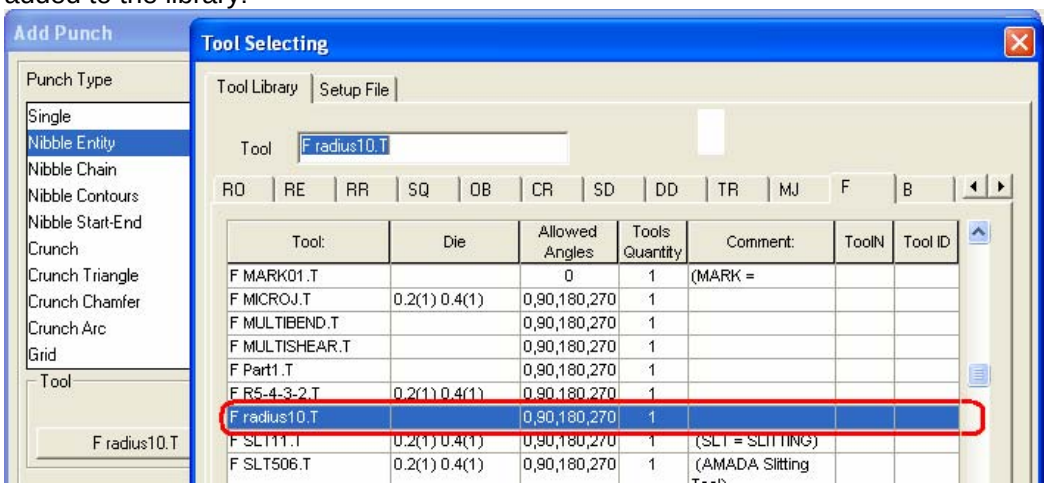
Notice that you can also select two parallel lines on the tool as **Reference lines**: the program will then choose which one is best, with major concern of avoiding using the Auto Index function.

3.3.2.4 Testing the New Special Tool

1. Open the part you want to process.
   This will be our notch, which we want to process with the new special tool:



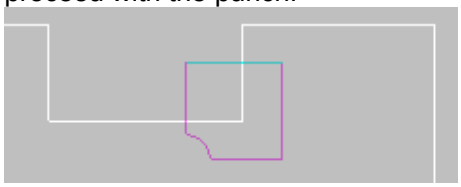
2. From the Common Toolbar click the **Add Punch** button. In the dialog which will open, click the **Tool** button. In the Tool Selecting dialog, which will appear, go to the **Special Tools** tab (**F**) and choose the tool you have just added to the library:



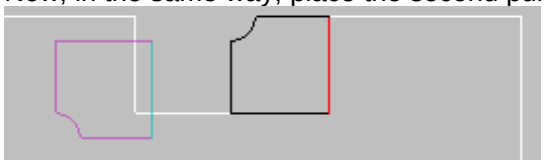
   Click **OK** button to confirm your selection.

3. Turn the **Index Button** **On**, and press the **S** key to toggle the **Tool Reference Line**, highlighted in red (on the gray background this line will be highlighted in blue). When you will are satisfied with the cutting line's location on the tool,

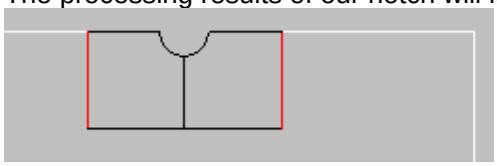
proceed with the punch:



4. Now, in the same way, place the second punch:



5. The processing results of our notch will look this way:



3.3.3 Names for Special Tool Files

Different tool files have different names and there are several categories of tools. Therefore it is important to give special tools proper names, since various machines "understand" only the precise naming.

Below you will see find more information on Forming Tools, Tapping Tools and Wilson Wheel Tools.

3.3.3.1 Forming Tools

Use the format **FTL<dd>**, where each "**d**" is a digit.

For example – FTL27.T

After the number, you can include some relevant info, e.g. a forming tool that is used before an M4 Tapping tool will be named this way:

FTL54-PreM4.T

3.3.3.2 Tapping Tools

The names for these tools are *pre-defined*; there are two types of Tapping tools:

- MM Tapping tools
- Inch Tapping tools

3.3.3.2.1 MM Tapping Tools

The following tools are available for mm tapping:

- TAP52-M25
- TAP53-M3
- TAP54-M4
- TAP55-M5
- TAP56-M6
- TAP58-M8

3.3.3.2.2 Inch Tapping Tools

The following tools are available for inches tapping:

- TAP256
- TAP348
- TAP440
- TAP540
- TAP632
- TAP832
- TAP1032

3.3.3.3 Wilson Wheel Tools

In ***cncKad*** the following Wilson Wheel tools are available:

- **`WW01.T`** - Wilson Rolling Shear
- **`WW02.T`** - Wilson Rolling Rib
- **`WW03.T`** - Wilson Rolling Offset
- **`WW04.T`** - Wilson Rolling Louver
- **`WW05.T`** - Wilson Rolling Pincher

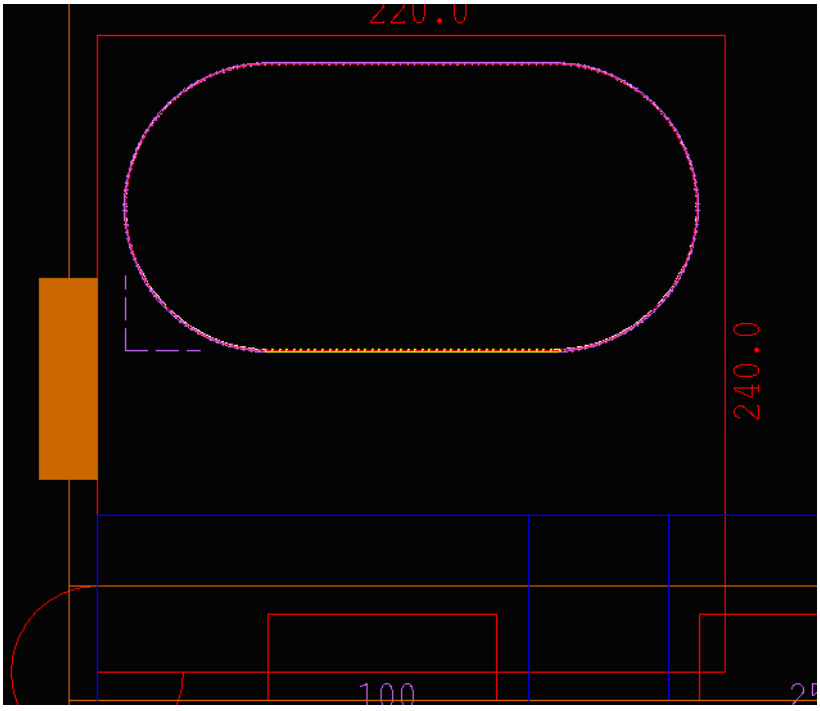
For use with Fanuc controls they are named this way:

- **`WW01`**
- **`WW02`**
- **`WW03`** and so on.

For use with Siemens and Nisshnibo, they look this way:

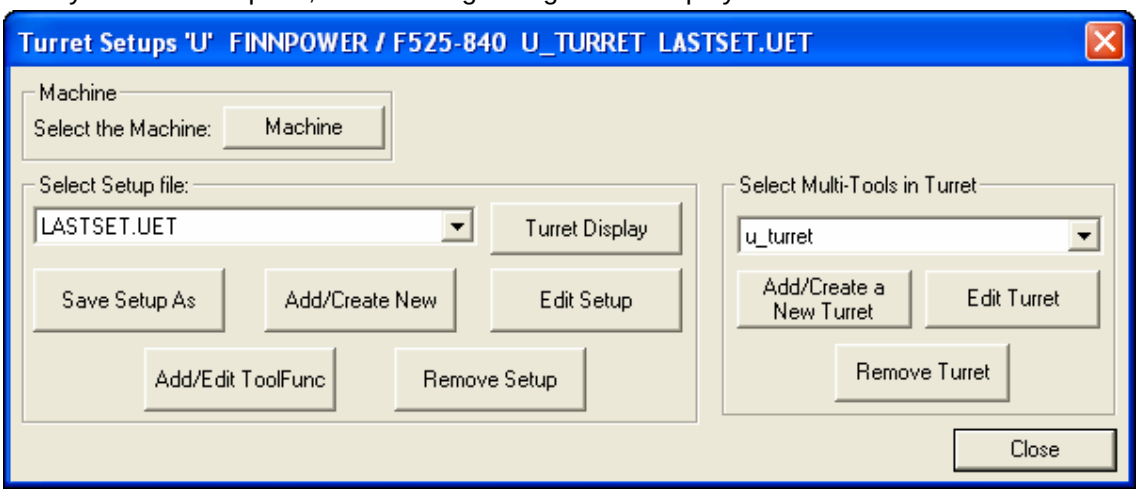
- **`WW11`**
- **`WW12`**
- **`WW13`** and so on.

Below you can see an example of Rounded Rectangle (200mm x 100mm, R=50mm) processed with WW12:



3.4 Turret Setups

After you click this option, the following dialog will be displayed:



3.4.1 Machine section

Options of this section allow you to choose, from the list of installed machines, a **Machine** for which you would like to create new Turret Setups or edit already existing ones.

3.4.2 Setup File section

Here the user can determine settings for his turret's setup file, a file that describes how tools are arranged in the machine's turret.

3.4.2.1 Select Setup File

From this dropdown list you can choose one of the existing setup files.

3.4.2.2 Creating a New Setup

Click on the **Add/Create New** button.

You will be asked to provide a name for the new turret setup file. The file name can be any name you choose; no extension is needed, since one will be set by default, as the machine ID character followed by **ET**. For instance if you machine **ID character** is **V**, then the extension for the setup file will be ".**V**ET". The user can create as many Turret Setup files as needed.

After doing so, choose the new setup from the dropdown list and click **Edit Setup** button. A dialog will open, giving you possibilities of defining various parameters for your turret.

3.4.2.3 The LASTSET File

This setup differs from others, so it is important to explain it more thoroughly.

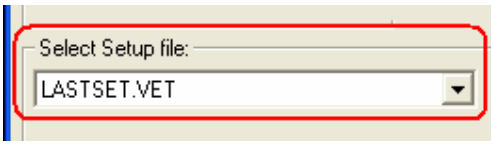
What distinguishes this file from other turret setups is that it always is a representation of the exact tool arrangement in the machine and reflects the turret setup of the last NC (there is one exception to this rule, as we will later see).

> The LASTSET file is an alternative to the "whiteboard" often found next to machines, which machine operators use to update the current tools in the turret.

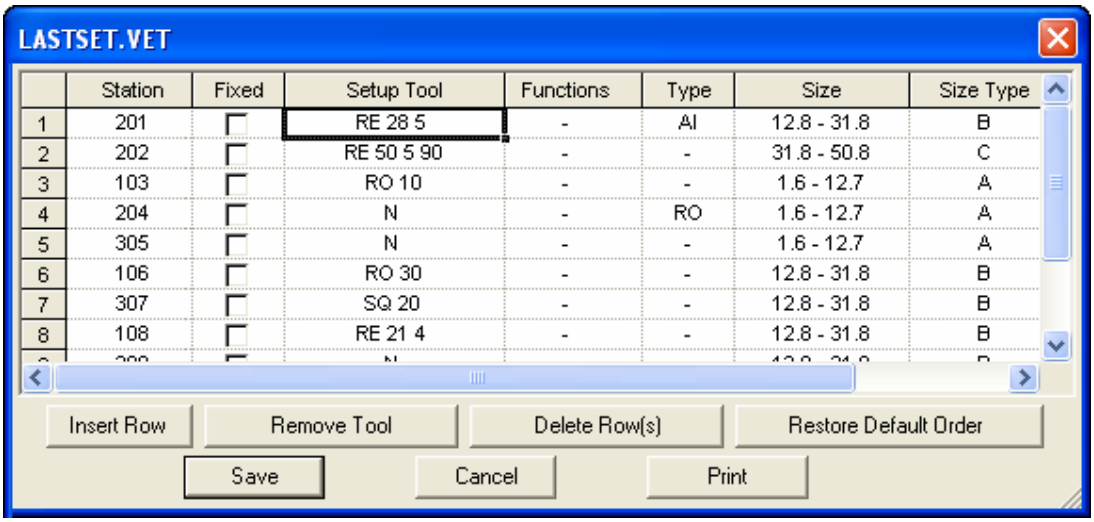
Since it shows which tools are currently available in the machine, the LASTSET feature is useful in two important ways:

- it automatically calculates which changes the machine operators must make in the tooling in order to produce the next part.
- it helps the engineers in tool selection when processing parts – different tools may often be used for the same job, and intelligent selection helps minimize tool changes (i.e. time).

3.4.2.3.1 Creating the LASTSET file



To create the file, select the **LASTSET** option from the Select Setup file dropdown list (shown above) and click the **Edit Setup** button. This will open the file and you will be able to edit it now:



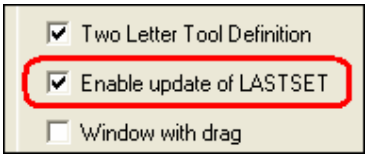
The initial values will be displayed according to the turret file selected for the machine.

3.4.2.3.2 Using the LASTSET file

In order to see how using the LASTSET file helps machine operators with their work, we will use the following example:

We have created a LASTSET file according to our turret tooling, and now we will process a part that requires using a **RO 5** tool that currently is **NOT** in the turret.

To use the LASTSET option, you must first enable it on the **Working Defaults** tab of the **Workspace Settings** dialog:

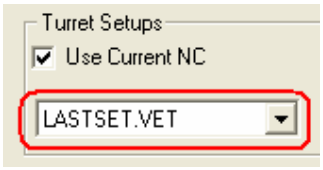


Important:

Only workstations that are on the actual production floor and are used for sending NC to the machine may use this option!

If you wish to generate NC simply to view the Graphic Simulation, **do not** enable this option (concerns workstations e.g. in engineering or design departments).

After enabling the above mentioned option, we can start the Post Processing procedure. In one of its steps, we come to the **Used Tools** dialog; here we must select LASTSET as the Turret Setup:



This will ensure that if you processed your part with a tool that is already in your machine (and therefore assigned to a station in LASTSET), the correct station will be chosen for this tool in the NC, and thus this will minimize changes to the turret's tooling.

Next step is the **Post Processor Options** dialog; here we must enable the following option:

This will cause ***cncKad*** to examine the part you want to Post Process, and check if NC code can be generated using the selected turret setup (i.e. the LASTSET file).

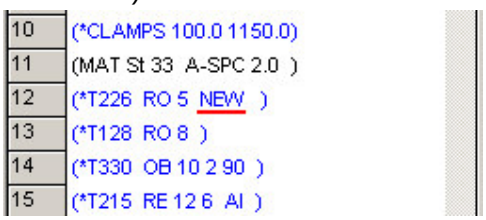
If this is impossible (in this case because there is a tool missing from the setup), ***cncKad*** will update LASTSET to include the new tool arrangement needed to generate the NC, and then it will issue a warning:

Important:

This is not an idle warning – it means that the machine operator **must** change the tool arrangement in the turret, so that it corresponds to the new setup created.

This is the only time that LASTSET does not match the turret, and thus must be corrected as soon as possible!

To help in making the necessary adjustment, ***cncKad*** will include instructions inside the NC file – the word **NEW** will be added for stations that require it (with changed or added tool):



In this case, we must put an **RO 5** tool into station number **226**.

In addition, an asterisk (*****) sign will be inserted into the **Report File** next to any station that needs correction:

ST NO.	Type	Length
226 *	RO	5.00
128	RO	8.00
330	OB	10.00

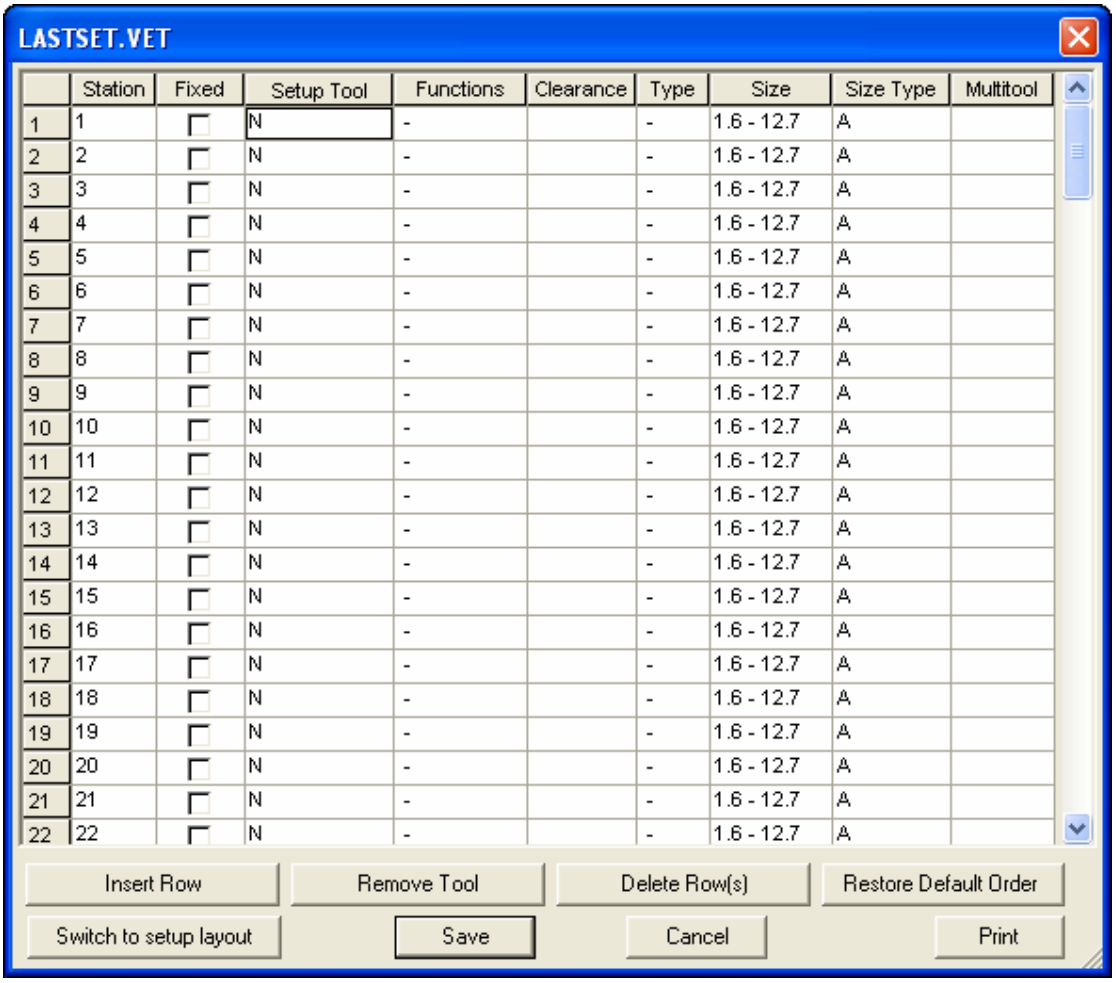
Only **after** we make the specified change, and LASTSET is once again a true representation of the turret, we can produce the part we designed.

3.4.2.4 Save Setup As

Choosing this option gives you an opportunity of saving current setup file under a new name.

3.4.2.5 Editing an Existing Setup

To edit an existing **Setup File**, use the dropdown menu to find the one you want to use, and click **Edit Setup** button. The following dialog will open:



- The **Station** column reflects the number of the turret station.
- The **Fixed** column allows you to choose the station which all the time would be assigned and locked for a specific tool.
- To place a specific tool in a station, click the **Setup Tool** column field for this station; the **Tool Selecting** dialog will open and you will be able to choose the desired tool.

  Important: You must set the tools in such way, so that the file will describe the exact arrangement of tools sitting currently in your machine's turret.

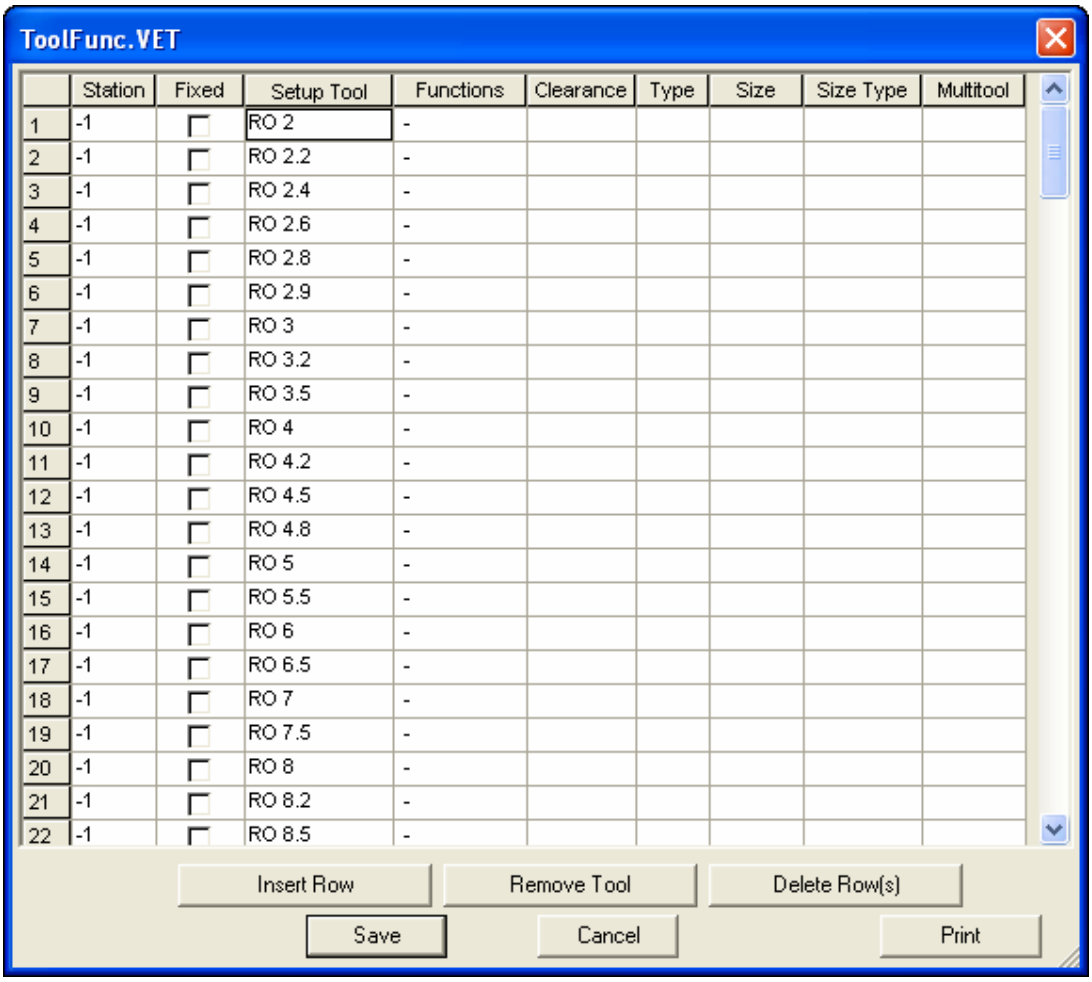
- After you have assigned a tool to a station, you can ascribe Functions to it – click the **Functions** field, and you will be presented with the **Tool Functions** selection dialog.

- In **Clearance** column you will see if any clearance was ascribed for current tool.
- The **Type** field displays information on the type of the station. E.g. looking at this field you can see whether the station is specified for **round tools only** and if it is an **Auto Index** station or a normal one.
- The **Size** and **Size Type** columns give information about the size of the station accordingly – the tool size for this station (how big tools can fit in) and the **Standard annotation** for the corresponding tool size (denotes the station type by a letter).
- **Multi-tool** column displays the number of a Multi-Tool station, if you have such.
- Notice that the data in this dialog can be **sorted** per column: just click on one of the columns' headings and the entire file will be sorted in the ascending or descending order according to that column.
- After you have finished defining the tools, do not forget to click **Save** button to preserve your selections and **Print** it, if you want.
- There are few more options in this dialog – the user can **Insert** or **Delete row(s)**, **Remove Tool**, **Restore Default Order** of the tools, or **Cancel** the changes he made.
- Clicking the **Switch To Setup Layout** button you will be moved to the **Turret Display** dialog.

It is important to note that each machine has its own turret setups, so that if you have created a new setup for machine **P**, machine **V** will not have access to it.

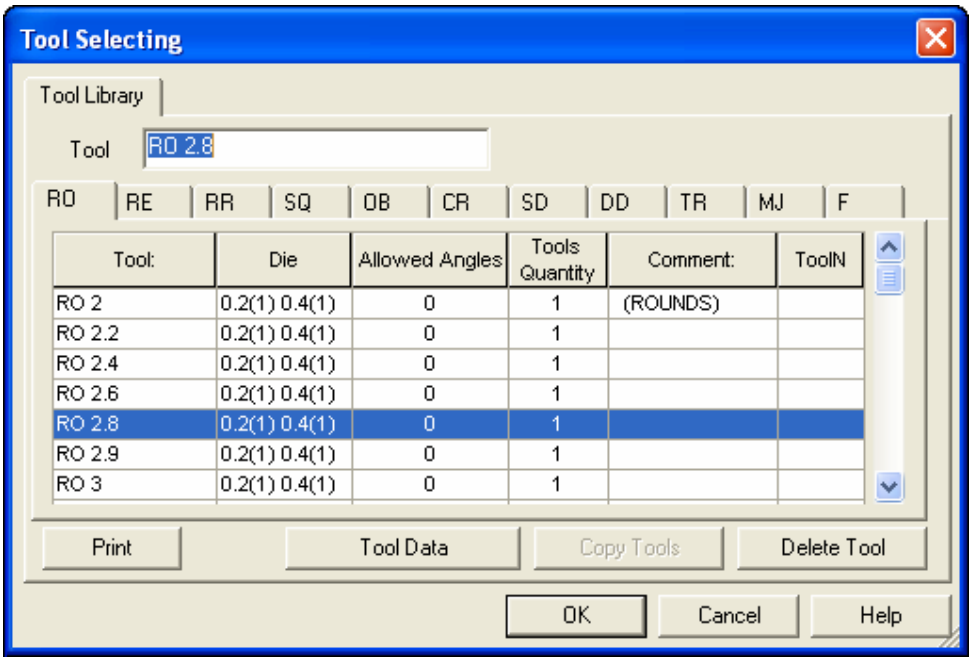
3.4.2.6 Editing the ToolFunc File

You can edit the ToolFunc.**?**ET (where **?** stands for the machine's ID character), by clicking the **Add/Edit ToolFunc** button. The following dialog will open:

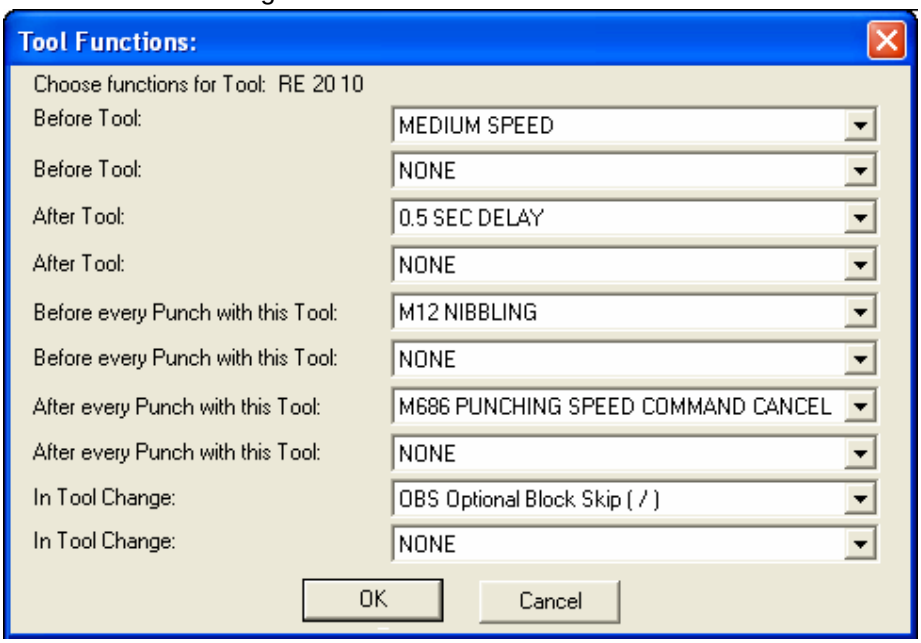


This table consists of the following fields:

- The **Station** column should be permanently set to **-1**. This means that the tool can be placed in any station of the machine turret.
- The **Fixed** column allows you to choose the station which all the time would be assigned and locked for a specific tool.
- The **Setup tool** column displays the tool name, e.g.: *F LUVER.T*. You can fill this field in by clicking it. The **Tool Selecting** dialog will appear and you will be able to choose the desired tool:



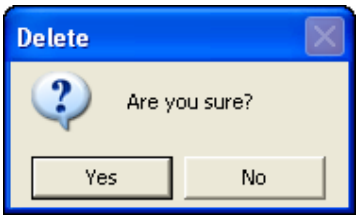
- In the **Functions** column you can choose the desired function the same way as you selected the tool. On the beneath picture you will find an example of functions assigned to tool **RE 20 10**.



After you have finished defining the settings, click **Save** button to preserve your selections. From now on, whenever the defined tool is used, the chosen functions will be added to NC program.

3.4.2.7 Removing an Existing Setup

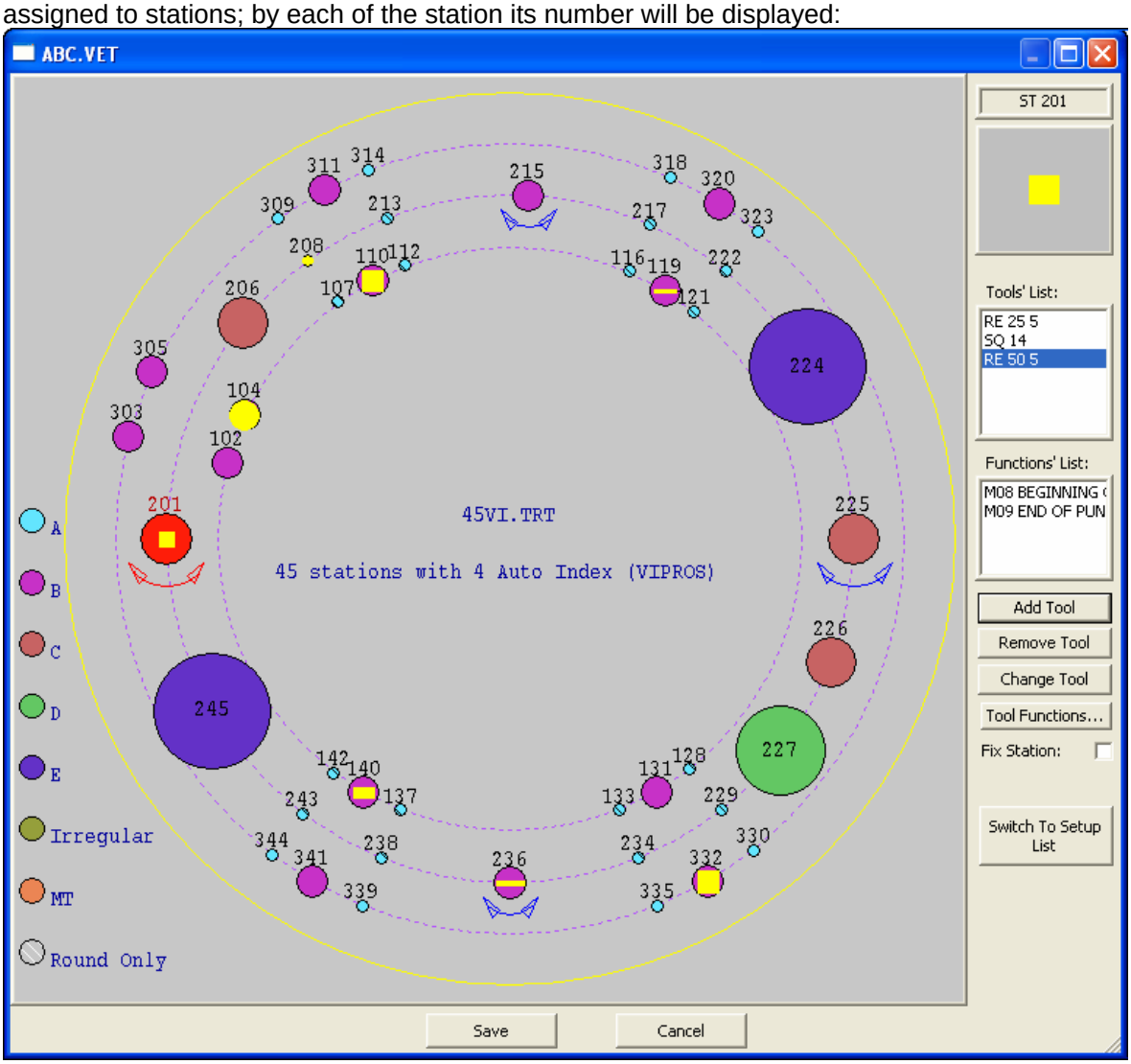
The user can delete selected setup file by clicking the **Remove Setup** button. A following warning message will pop-up, asking you if you are sure of your decision:



Confirm your action clicking **OK** button and the currently chosen setup will be removed.

3.4.2.8 Turret Display

Pressing this button will open the following dialog showing the turret with tools assigned to stations; by each of the station its number will be displayed:



On the panel situated on the right side of this dialog you will find options with which you can manipulate the turret's stations.

3.4.2.8.1 Station number field

When you click any station instance with the cursor, it will be marked with red color and its number will appear here.

3.4.2.8.2 Tool Preview

In this window you will be able to preview the tool selected on the Tools' List.

3.4.2.8.3 Tools' List

Here you will see a list of tools assigned to selected station.

3.4.2.8.4 Functions' List

Here you can see a list of functions assigned to the currently marked tool in the Tool's List.

3.4.2.8.5 Adding Tool

Clicking the **Add Tool** button you will be redirected to **tools' library dialog**, where you will be able to choose the tool you wish to add to certain station.

3.4.2.8.6 Removing Tool

Clicking the **Remove Tool** button causes the tool to disappear from selected station.

3.4.2.8.7 Changing Tool

Clicking the **Change Tool** button redirects you to **tools' library dialog**, where you can select different tool for the current station.

3.4.2.8.8 Tool Functions

Choosing this button opens the **Tool Functions** dialog, from which you will be able to assign new functions for tool or edit the existing ones.

More on this option you will find in **Defining Tool Functions** section of CAM Menu chapter of this ***Manual***.

3.4.2.8.9 Fix Station

Marking this checkbox, you decide that the tool ascribed to the currently selected station will be used only with it.

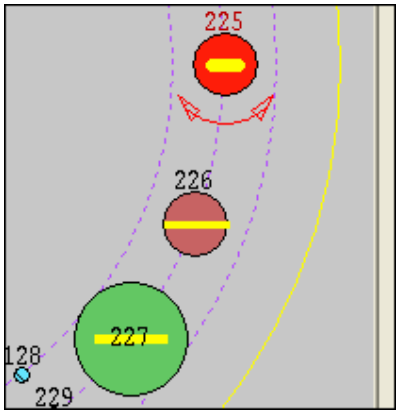
3.4.2.8.10 Switch To Setup List

Clicking this button you will be transferred to the **Setup File** dialog.

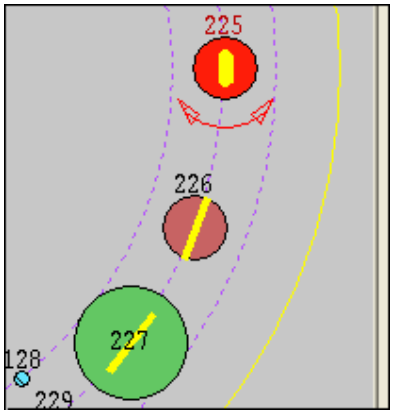
3.4.2.8.11 Radial View

Checking this box you change the turret display. Now all the tools will be viewed as placed in the stations along the turret tracks.

Normal View



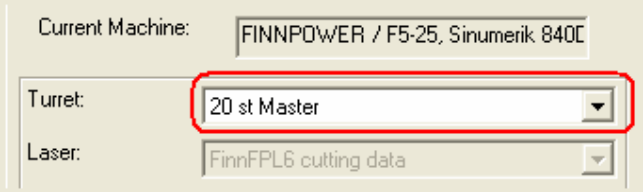
Radial View



3.4.3 Multi-Tools section

This section is available for machines which have Master turrets with Multi-Tools option, e.g. Finnpower, LVD, Strippit, Rainer and Muratec machines.

You should chose Master Turret for your machine in **Settings =>** Machine Settings **=>** Machine tab **=>** **Machine Settings** dialog:



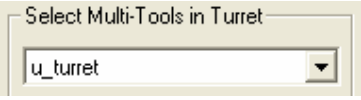
Your choice done there will be available in **Multi-Tools section**, where you will be able to select and manipulate it, with the usage of following buttons:

- Add/Create a New Turret
- Edit Turret
- Remove Turret

3.4.3.1 Add/Create a New Turret

When you click this button a **Save As** window will appear. Type the new name for the multi-tool turret settings. It is recommended to type in the new name with "U_" in order not to override an existing turret and also to differentiate it from regular turrets. For example: **U_CosFinn**.

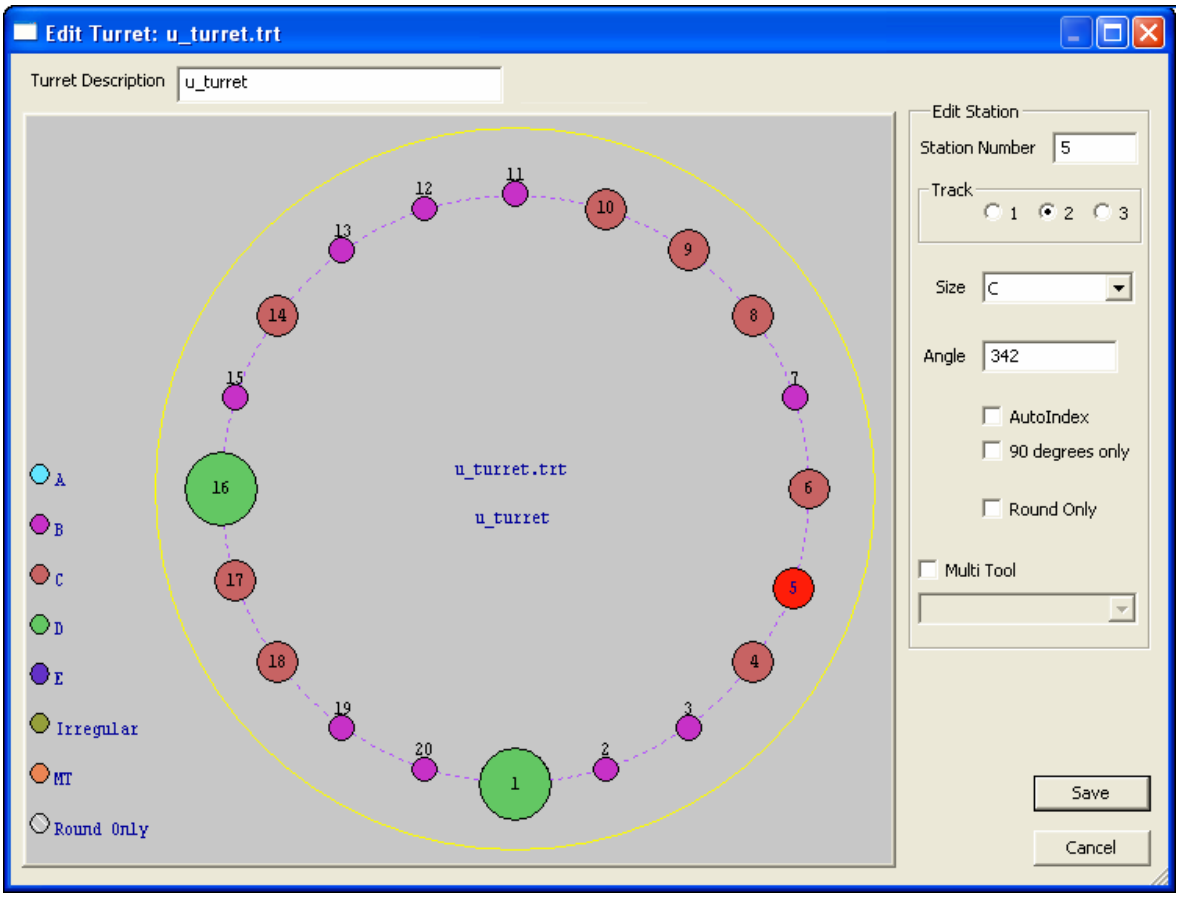
After doing so, click **Open** button. You will be returned back to the **Turret Setups** dialog, and the new multi-tool turret will be shown in the dropdown list as a default:



In order to enter the desired changes, click the **Edit Turret** button.

3.4.3.2 Edit Turret

After clicking the **Edit Turret** button, you will be presented with the following dialog, allowing you to personalize the mutli-tool turret:



The active station, we are defining settings for, is marked with red color.

The legend on the left side of the dialog specifies the colors of the stations according to their sizes, as well as denotes the **Irregular**, **Multi-Tool** and **Round Only** stations.

On the right side of the window you can see various settings options for the selected station:

3.4.3.2.1 Station Number

This field displays the number of the presently chosen station. The multi-tool stations' numbering varies for machines.

More information on the Multi-tool Numbering you will find in **Changing Turret Numbering** section of Appendix chapter of this manual.

3.4.3.2.2 Track

Here you can view and change the track for the current station. It is by default set on 2.

3.4.3.2.3 Station Size

From the dropdown list you can choose the best definition of your station size.

3.4.3.2.4 Station Angle

In this field you can see, as well as edit, the station's angle in turret – from 0° to 360°.

3.4.3.2.5 Auto Index

Checking this box we denote the station as rotating one. Choosing it makes the 90 Degrees Only option available.

3.4.3.2.6 90 Degrees Only

This option is valid only for **Auto Index** station and allows rotating the index station only at 90 degrees.

3.4.3.2.7 Round Only

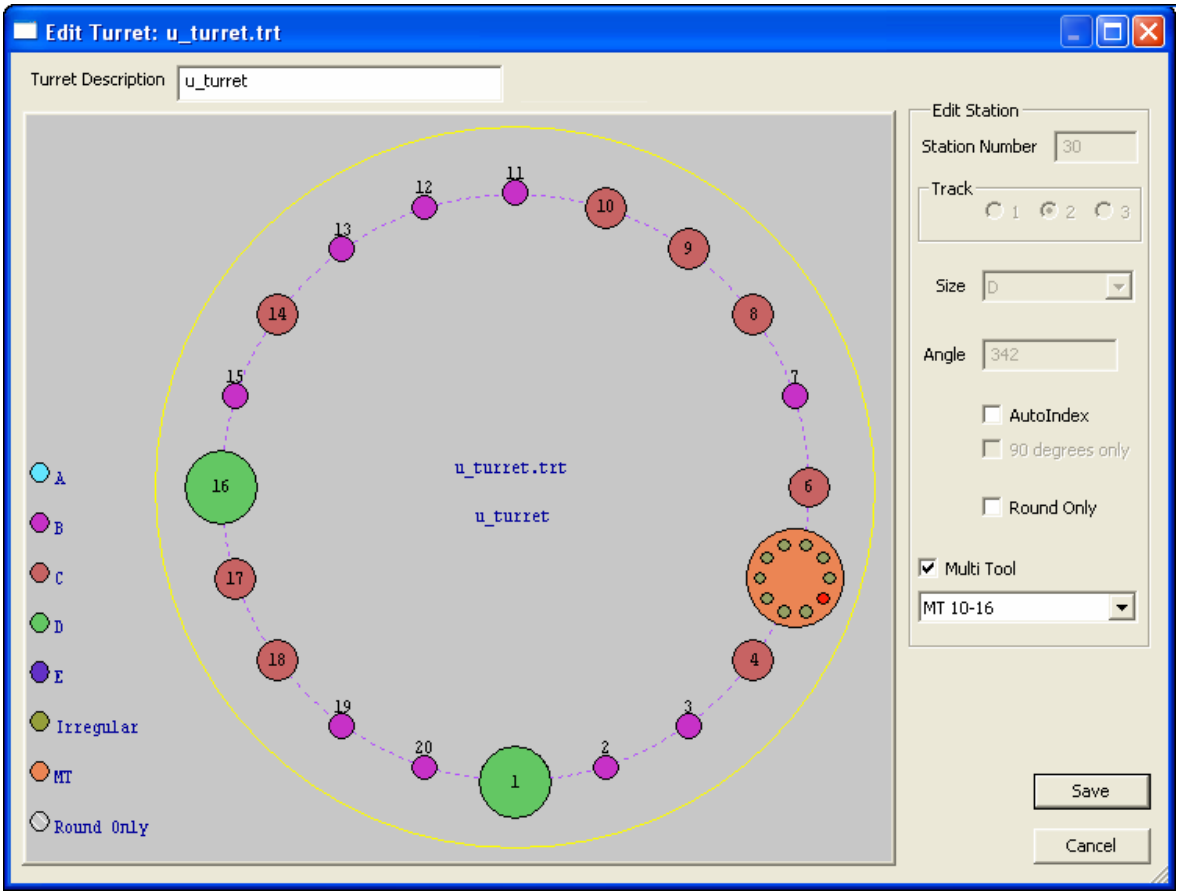
Marking this checkbox the user denotes the station available only for round tools.

3.4.3.2.8 Multi-Tool

Notice (on the picture shown in **Edit Turret** section) that in Master Turrets there are no Multi-Tools as a default; therefore they should be added. Check the **Multi-Tool** box and from the dropdown list detailing Multi-Tools available for this turret select the desired one.

In most turrets not all of the stations can be changed into Multi-Tool stations. It is possible though to add Multi-Tools to more than one station. To see what options can be edited for some station, just click on it and see on the right section of the dialog what options are available.

After choosing the desired Multi-Tool, as you can see on the picture presented below, it is not possible now to change the track and the size of Multi-Tool stations. Only **Auto Index** option can be chosen and if you choose it the **Round Only** option will be also activated.



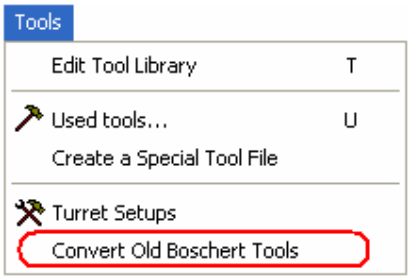
If you would like to change other station properties, like track, size or angle please do so before adding the Multi-Tool to it.

3.4.3.3 Remove Turret

Clicking this button deletes currently selected turret.

3.5 Convert Old Boschert Tools

This option is available for Boschert machines only.



Before, in older versions of ***cncKad***, the tools didn't have ToolN and their number was being taken from the station. So if you were using the old tools and recently have updated your ***cncKad*** to the new version, this converter is for you.

For more information on converting old Boschert Tools and adjusting them for work with ***cncKad***, go to **Release Notes for BOSCHERT users** section of Appendix chapter of this manual.

4 NC Program Creation

There are several stages and options you must go through, leading up to the post-processing stage, where the NC program is being created out of a part.

4.1 Creating the Program

Once you have finished assigning punches to your part and all the settings have been defined, you are ready to start the final phase of program development. In this stage ***cncKad*** will read the graphics, which you have processed, and convert it into a program in a language that your machine controller can recognize, and which can be simulated or/and sent to a machine.

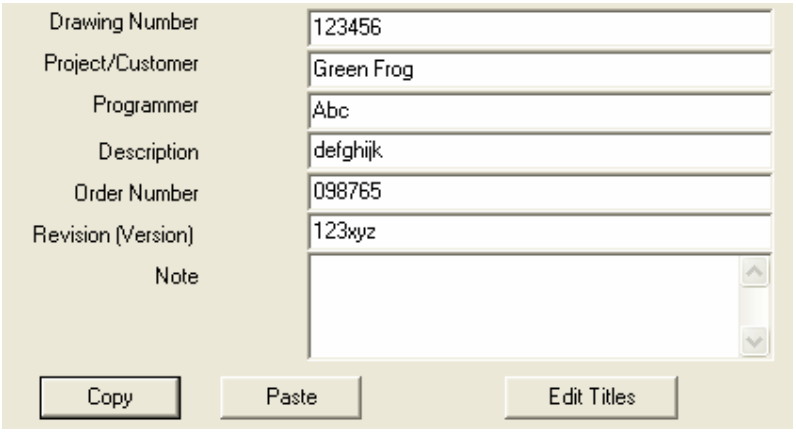
Post Processing is a multi-part procedure. This chapter takes you through it step by step.

4.2 NC Generation Stage 1

From the **File Menu**, select the Post-Process option or on the Common Toolbar click the Post Processing button – 

.

If you hold down the **CTRL** key, as you click the **NC** button, the postprocessor will skip stages 2-4, and jump straight to Stage 5. Otherwise, the following dialog will be displayed:



The **User Data** that you fill in will be used in the Report File. Entering this data here is not compulsory, unless you wish to produce a report file.

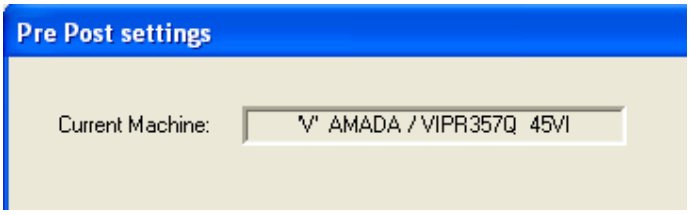
You can use the **Copy** and **Paste** buttons to transfer the user's details between parts. There is also a possibility to **Edit Titles**.

After you are done with filling in the details click the **Next** button.

For more information on the features of this dialog refer to **appropriate** section of **CAM Menu** chapter.

4.3 NC Generation Stage 2

In this stage, you can see for which machine the program is being created.

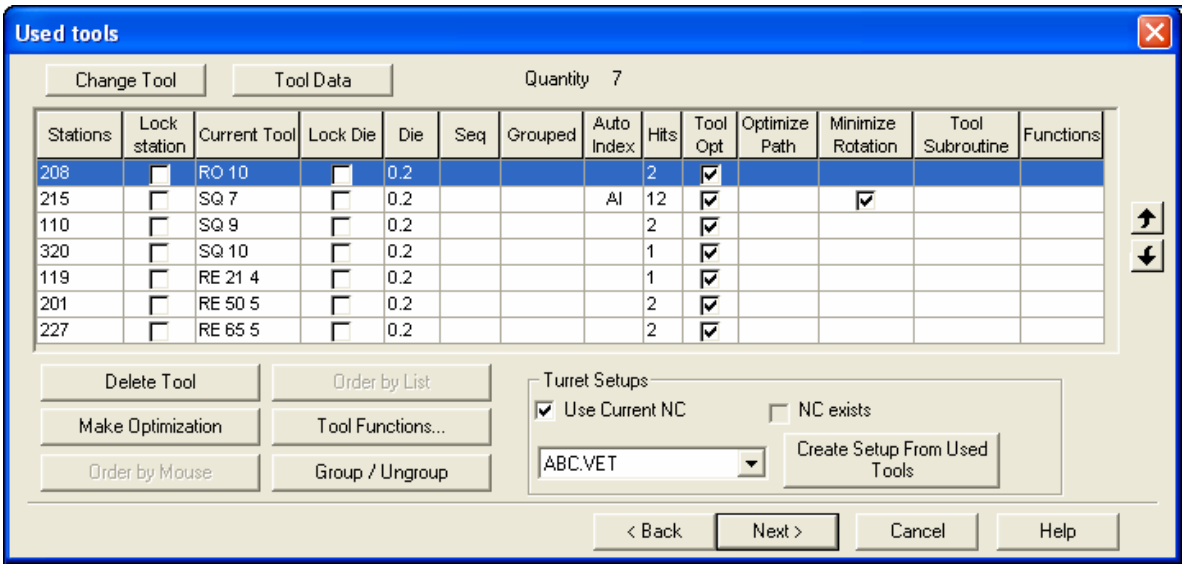


If you have more than one machine installed on your workstation, this enables you make sure you have selected the correct one. If everything is all right, click the **Next** button.

4.4 NC Generation Stage 3

In this stage the **Used Tools** dialog is being displayed, allowing you to set the order of the tools as well as defining other settings for them.

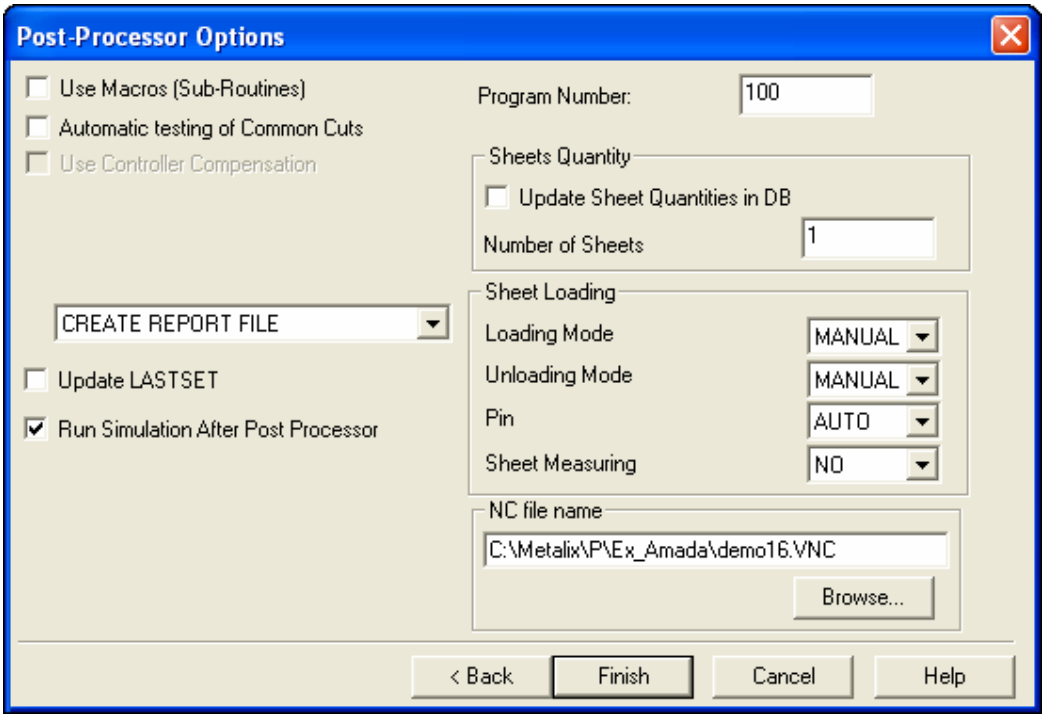
It is advisable to make all major changes to the cutting at this stage.



To go to the succeeding stage, click the **Next** button.

4.5 NC Generation Stage 4

This is the final dialog before the generation of the code. Here, you can set various options pertaining to the code generation:



4.5.1 Use Macros (Sub-Routines)

This option is available only for machines which support it.

It's very useful when using multiple parts on a single sheet, because it allows the user to shorten the program by using macros (or in other terminology – Sub-Routines). Selecting this checkbox will enable them.

If you want to set the Macros as a **default** for your machine, go to **Post-Processor Options** tab of the Workspace Settings menu in the Settings chapter of ***Drafting and Nesting Manual***.

4.5.2 Automatic Testing of Common Cuts

If **Common Cuts** option is chosen, this function will check for duplicates – it will make sure that between two adjoining parts no double CAMs have been placed.

4.5.3 Use Controller Compensation

This option is available only for laser and combined (punch-laser or punch-plazma) machines.

It is useful in case you want to get **CRS** (Cut Radius Compensation) in NC command.

4.5.4 Report File Creation Options

From this dropdown list the user can define the default setting for creating a Report File. The options are as follows:

- Do Not Create Report File
- Create Report File
- Print Report File
- Create and Print Report File

More information on creating **Report File** and the types of **Report Templates** you can find in Appendix chapter of this manual.

4.5.5 Update LASTSET

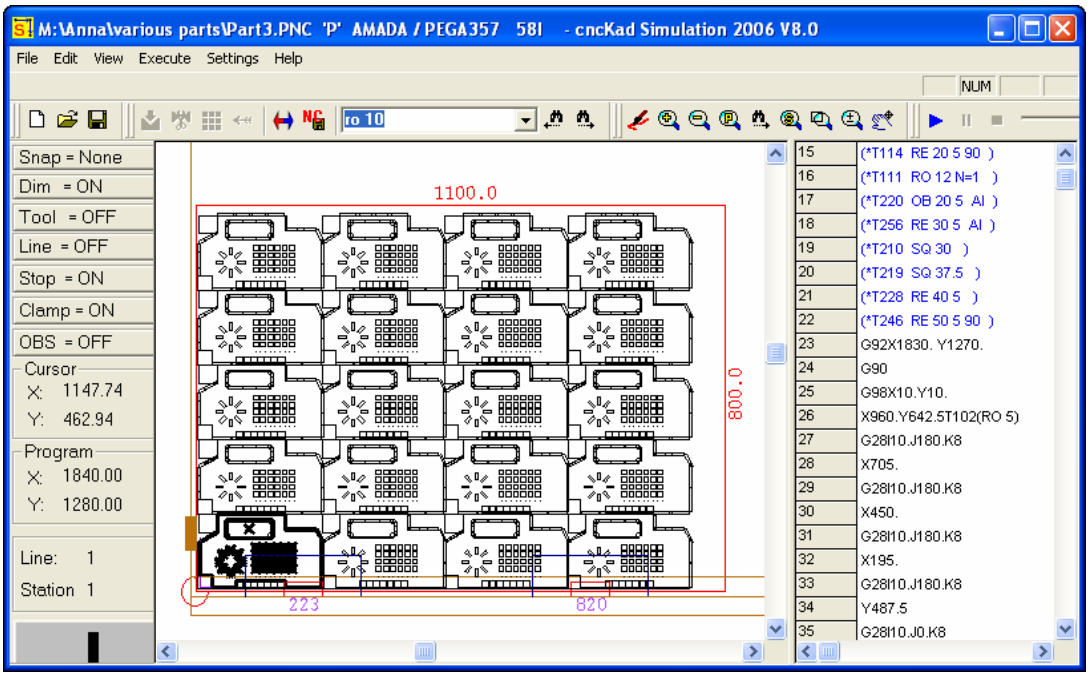
This option is available only when the **Enable update of LASTSET** checkbox is selected in Working Defaults tab of Workspace Settings dialog.

If you check this option, the current Turret Setup will be saved as a Setup File called LASTSET.**?**et (where **?** stands for the machine's ID character). If you are working with such a file right now, and you made a few changes you would like to keep for processing other parts, checking this option will update it.

You can use this setup for the next part you create, being certain that in both cases parts use exactly the same setup.

4.5.6 Run Simulation After Post Processor

After the Post Processing stage the graphic creation will remain open, and the **simulation** of created program will be loaded in a separate module, where the user will be able to test the program and make sure it runs with no problems.



4.5.7 Automatic Detection

This function is available for punching machines only.

Here you can choose one of the few options for automatic parts' processing:

- **Do not detect groups of single punches**
- **Automatically detect lines of single punches**
- **Automatically detect grids**

More information on these features you will find in **Automatic Detection** section of Settings chapter of ***Drafting and Nesting Manual***.

The only difference between the parameters set on there and here is that the settings selected here concern only the currently created **NC Program**.

4.5.8 Program Number

Some machine controllers demand, that there be a program number, others do not. That's why here the user can define a default program number.

4.5.9 Sheets Quantity

The user can set here the number of sheets he wants to process this time.

If the checkbox **Update Sheet Quantities in DB** will be selected, the **Number of Sheets** entered here will be deducted from the quantity of sheets entered in **Select Sheet** dialog which is accessed from **Sheet** tab of the Set Sheets and Clamps window.

4.5.10 Sheet Loading

In this section of the dialog, the setting definitions are being loaded from **Load/Unload** tab of the Set Sheets and Clamps window.

This is the last moment and your last chance for entering changes for sheet loading, before the NC code will be generated.

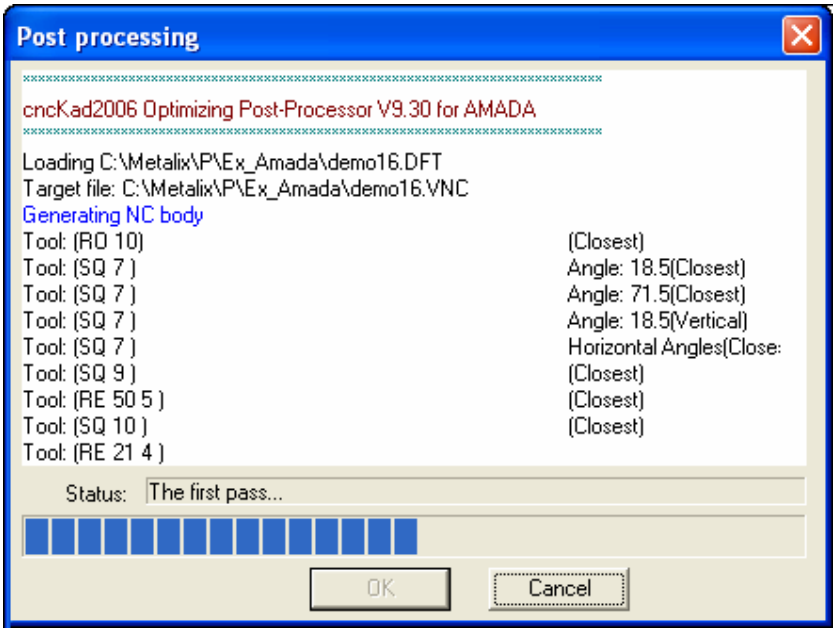
4.5.11 Name and Path of NC file

From this dialog the user can set the name and path for the NC file. The NC file can be given a different name than the drawing file and placed in a different directory. To select a directory, click the **Browse…** button.

When you have defined all the settings, click the **Finish** button.

4.6 NC Generation Stage 5

The following dialog will be opened:



This is the code generation stage, during which you might be asked different questions concerning your machine. Clicking **Enter** button usually attains the default values.

If the Post-Processor could not create the program or there are problems with the clamps or the tools, you will also be notified about them by an appropriate message.

If you placed CAMs within the clamps dead zone and the Clamp Avoidance option is activated, you will be able to decide whether to attempt avoiding clamps or to continue without any automatic clamps avoidance.

If you requested to print the report file, you will have to prepare the printer and, when requested, press a key to print it.

When the Post Processing stage is over, click the **OK** button to open the simulation module.

5 NC Code Simulation and Editing

This chapter deals with ***cncKad***'s simulation program.

The **Simulation** program is a completely individual module of ***cncKad*** and is run separately from the main program – the simulation is loaded into a separate window allowing ***cncKad*** to remain open in the original window.

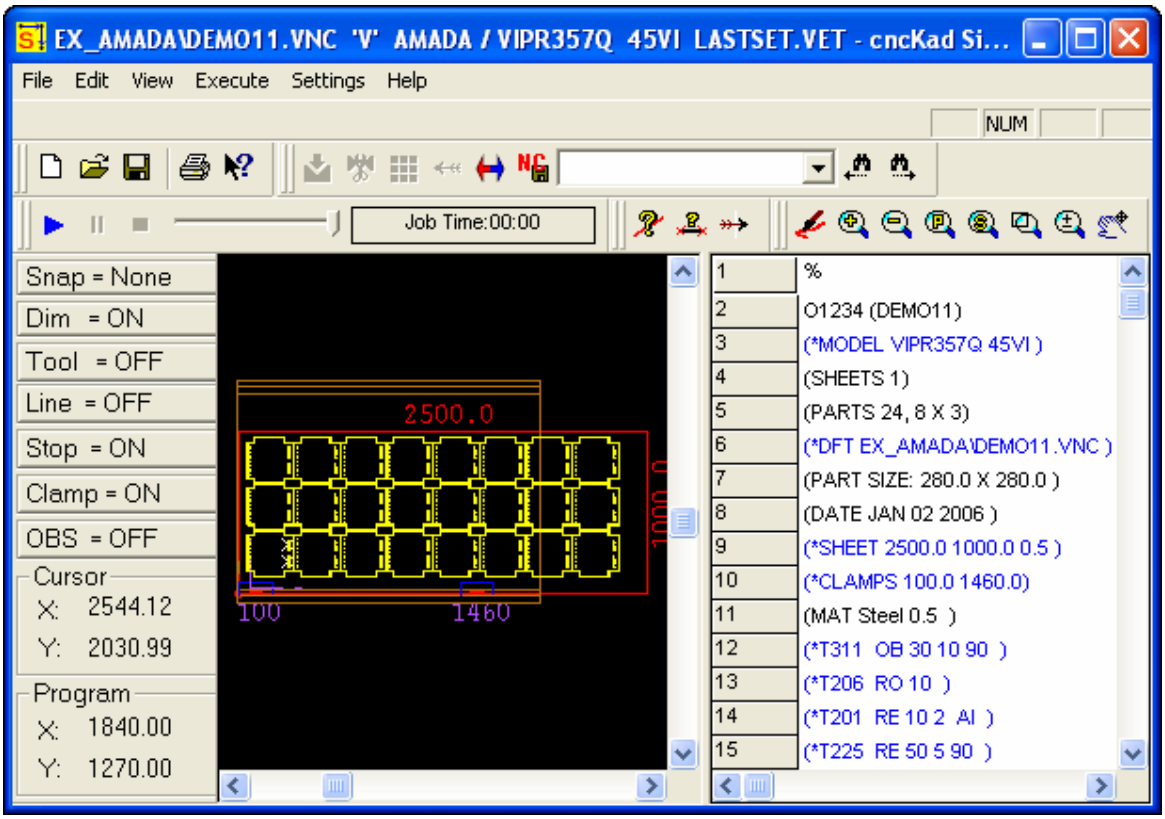
The following pages explain the options that can be used here, and how to run the simulation.

5.1 Opening the Simulation

There are three possibilities of opening ***SimW***:

- After Post Processing
- When Simulation is selected from the **File** Menu within ***cncKad***.
- From the **Start => Programs => Metalix =>Simulation** menu

In any case, the following window is loaded:



The graphic results of the code are displayed on the left side of the screen.

Any change in the code displayed on the right side of the screen will be updated on its graphic representation.

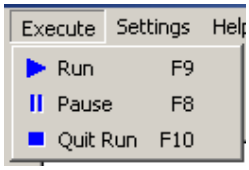
5.2 Running the Simulation

Once an NC code and graphics are visible on the screen, that code can be simulated.

If there was no file loaded when you entered the simulation module, from **File** Menu you can **Open** the file you want to simulate.

The progress of the Simulation is controlled in two ways:

- From the **Execute** menu:



- From the **Execute** toolbar:

5.2.1 Starting the Simulation

Click the  button to begin the Simulation, or to resume it after it has been paused.
If you want to run the program line by line, click the **Line = ON** button on the left side of the screen and then click the **Run** button repeatedly.

5.2.2 Pausing the Simulation

Click the  button to temporarily suspend the Simulation.

If you now click the  button, the Simulation will resume from its current position.

5.2.3 Stopping the Simulation

Click the  button to stop the Simulation.

If you **Stop** the Simulation, it cannot be resumed from the same position, but will start from the beginning, when you **Run** it again.

5.2.4 Simulation Speed

The user can define the simulation velocity that suits his needs, before the simulation is run, as well as during it.
To do this – move the mouse on the speed slider to the left for slower speed, and to the right for faster speed:

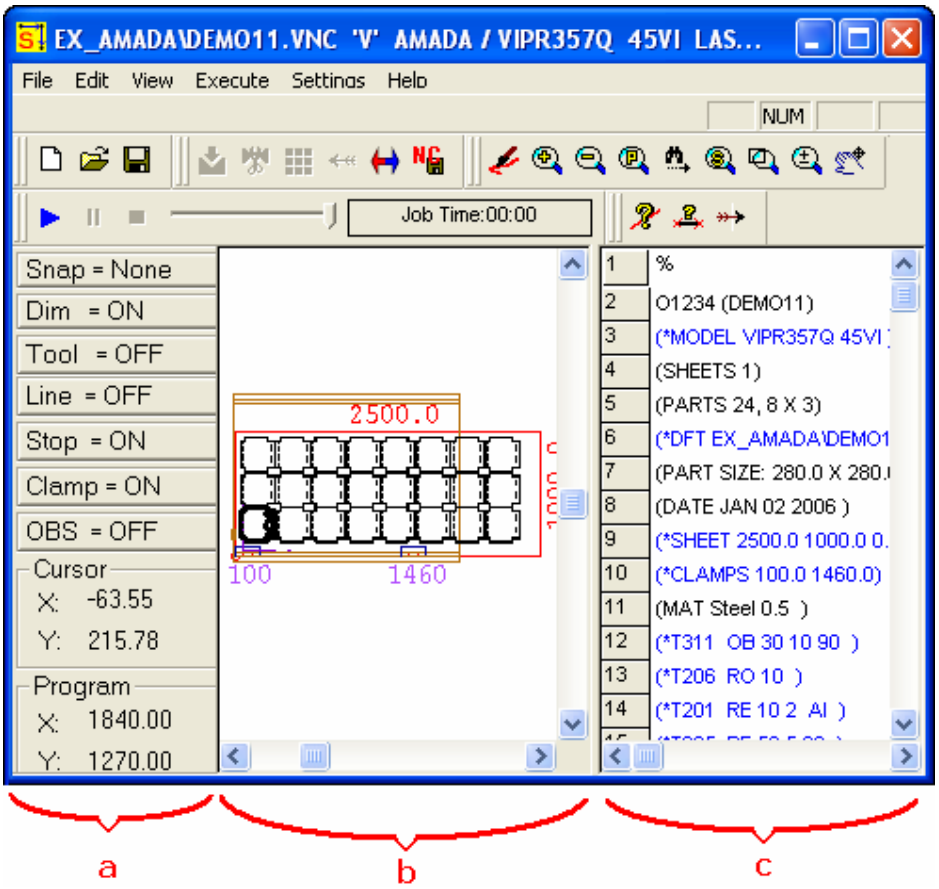
The speed will be updated in effect only after the current line has been executed.

5.2.5 Simulation Time

This window shows the time it will take to run this NC program **on your machine**.
The time displayed here varies for different machines. In the time factor there are various elements taken into account, such as X and Y travel time, turret rotation speed, tool placement within the turret, repositioning and tool change time.

5.3 Simulation Screen Elements

There are three elements in the Simulation screen:
a – The State Bar
b – The Graphic Simulation area
c – The NC Code Editing area



5.3.1 The State Bar

This is the area on the left side of the screen, containing different settings and information pertaining to the Simulation.

5.3.1.1 Snap Button

This button gives a variety of Snap options, giving the user a choice of choosing the most desired one.

More information concerning this feature you will find in **Snapping on an Entity** section of **Introduction** chapter in Drafting and Nesting Manual.

5.3.1.2 Dim Button

This button enables or disables viewing **Dimensions** and **Text** on the part.

Dimensions that have been placed on the drawing during the drafting stage can be toggled **ON** and **OFF** by clicking the Dim = ON button on the **State Bar**.

5.3.1.3 Tool Button

If this button is in the **ON** state, the program will pause every time there is a change tool command in the code. Each tool's preview is being displayed in the tool port.

To continue running of the program, click the appropriate button in the **Execute** toolbar.

5.3.1.4 Stop Button

If there is a **Stop Machine** command defined in the program, this button enables and disables that **Stop** (for the Simulation's run):

When it is in the **OFF** state, the Simulation will not halt for any **stop** commands, (on a lot of machines the stop command is M00), however when it is in the **ON** state, the simulation will pause every time a stop command is encountered. To continue with the simulation, click the appropriate button.

5.3.1.5 Clamp Button

This button enables or disables the warnings that are issued when the Clamps' Dead Zone is entered:

In the **OFF** state, the Simulation will run without any pauses, even if the tool enters a Dead Zone.

> **Note:** If the tool damages the Clamps, a warning will be issued and the program will pause running.

However in the **ON** state, the Simulation will pause every time the tool enters the Clamps' Dead Zone. The tool might not be damaging the Clamps, but the warning is issued, so that you would be aware that the tool is now working in the area of the Clamps.

5.3.1.6 OBS Button

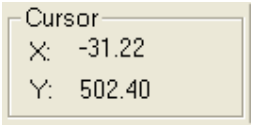
This option is a button of *Optional Block Skip* feature and appears on most of the machines. Its function is to allow the program to ignore or execute lines that have double slashes before them, e.g.:

```
Y527.
X78.
//G72X61.5Y564.5
//G66I8.J180.P-5.K8.T217 (R 5. 5.)
//G72X253.5Y564.5
G66I8.J0.P5.K8.
```

When this button is in the **ON** state, the lines with double slashes before them will be ignored (not executed), but when this button is in the **OFF** state, these lines will be executed.

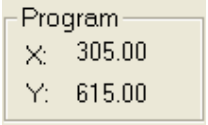
5.3.1.7 Cursor Coordinates

This section shows the coordinates of the cursor's current position, when it is being moved inside the graphics area of the Simulation screen:



5.3.1.8 Program Information

These coordinates show the current position of the turret in the graphics area of the Simulation screen, when the simulation is being run:

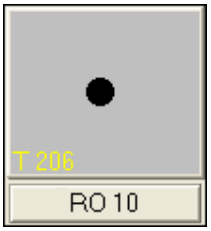


5.3.1.9 Line section

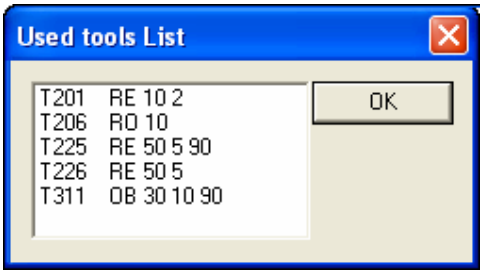
In this section of the State Bar you can see the station number the current tool is using, and a number of the line it is presently processing:

5.3.1.10 Tool information

The square port displays a graphic representation of the tool that is currently being used for sheet processing, as well as the station number it is placed in and the button displays its name:



If you click the button, a list of ***all*** the tools that are used in the program will be displayed, together with the station numbers there are placed in:

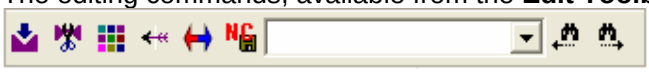


5.3.2 NC Code Editing

The NC code is displayed on the right side of the screen, and can be edited from this area. If the user makes changes to the code, the graphic display will be updated automatically to reflect these changes.

In order to edit the code select a line you want to edit, modify it and then from the keyboard presses **Enter** key, so that the changes made to the current line would be updated on the graphic display.

The editing commands, available from the **Edit Toolbar**



, are:

- Inserting a new line 
- Deleting a line 
- Inserting a G Code (Pattern) 
- Marking a CAM 
- Send to machine (DNC) 

  Note: The **Edit Toolbar** is activated only when the user clicks the cursor in the NC code side of the screen.

- Send NC to Diskette 
- Searching List 

- Find Previous 

- Find Next 

5.3.2.1 Insert a New Line

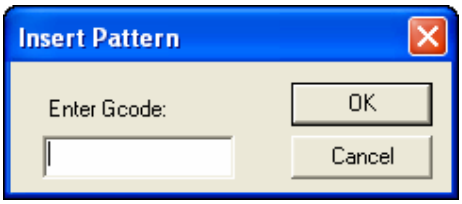
Place the cursor on the line of code before which you want to insert the new line and click the **Insert Line** button –  .

5.3.2.2 Delete a Line

Place the cursor on the line you wish to delete and click the **Delete Line** button –  .

5.3.2.3 Insert a G Code (Pattern)

Place the cursor at the position at which you wish to insert a G-Code and click the pattern button –  . A following dialog will appear, where you will be able to enter the desired G-Code:



When you have finished, click the **OK** button.

5.3.2.4 Mark a CAM

If you want to find out how a specific line of code is being represented in the graphic simulation click on a line of code and then click the  button. Simulation marker (a large cross) will jump to the corresponding CAM.

Note: To see an additional CAM, repeat the preceding procedure.

5.3.2.5 Send to machine (DNC)

The  button sends the NC code to DNC program, so that it can be remotely transmitted to the machine.

5.3.2.6 Send NC to Diskette

Choosing the  button you can send the NC code to a diskette, so that it can be transferred to the machine.

5.3.2.7 Searching List

To this field 

you can enter the value you are looking for in the NC code. When you press **Enter** key, you will be transferred to the line of the code containing this value. By the use of below described buttons, you can look for further instances of this value.

5.3.2.8 Find Previous

When you click the  button, the last found instance of the value typed in Searching List will be marked in the NC code

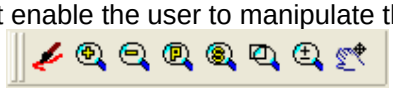
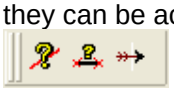
5.3.2.9 Find Next

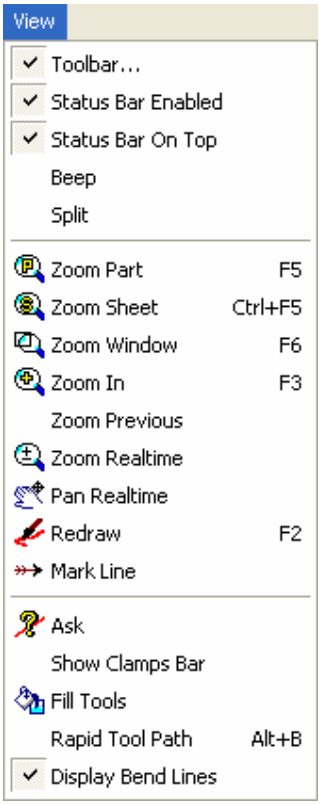
When you click the  button, the next found instance of the value typed in Searching List will be marked in the NC code.

5.3.3 Graphic Simulation

The **Graphic Simulation** area allows the user to preview how the NC code will be run on the machine.

When the Simulation is run, the Simulation marker (a large cross) shows the graphic position of each line of code, as it is performed.

There are several various options that enable the user to manipulate the display, and they can be accessed from the **Zoom**  and **Graf**  toolbars, as well as from the **View** menu:



5.3.3.1 Toolbar...

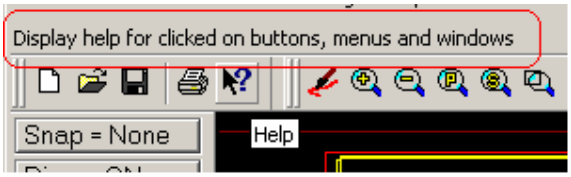
This option is used to customize the toolbars.

More information on this subject you will find in **Using the Toolbars** section of ***Drafting and Nesting Manual***, with this difference that in each ***cncKad*** module the toolbars list and their content vary, but the general rules are the same.

5.3.3.2 Status Bar Enabled

Choosing this option makes Status Bar appear on the screen.

The Status Bar enables or disables viewing tips about the option you are selecting (outlined below):



5.3.3.3 Status Bar On Top

Selecting this option positions the Status Bar above the toolbars, and unchecking it – places it below the Graphic Simulation area.

5.3.3.4 Beep

This option enables or disables a beeping sound, which is used to mark each CAM during the Simulation's run.

5.3.3.5 Split

After selecting this option, the line separating the Graphic Simulation area from the Code Editing area will follow the cursor. In order to set chosen position of the line, click the mouse.

5.3.3.6 Zoom Part – F5

 – This command fills the view port of the Graphic Simulation with the complete extents of the drawing.

5.3.3.7 Zoom Sheet – Ctrl+F5

 – This command zooms the full extents of the sheet; all instances of the part on the sheet will be displayed.

5.3.3.8 Zoom Window – F6

 – This command zooms into an area defined by the two opposite corners of a window.

In order to use this command – place the cursor in the first corner of the area that you wish to zoom into, click the mouse and the window will open. Now drag the opened window to the opposite corner of the area you wish to zoom into and click the mouse again. The area selected within the window borders will fill the viewing port of Graphic Simulation.

5.3.3.9 Zoom In – F3

 – Each clicking on this command enlarges the display by twice the current magnification.

5.3.3.10 Zoom Previous

This command allows the user to return to the previous view of the Simulation.

5.3.3.11 Zoom Out – F4

 – Each clicking on this command decreases the magnification by half the current display.

5.3.3.12 Zoom Realtime

 – This option enables you to zoom the display incrementally: after clicking this icon, click on the draft, and then move the mouse wheel up to zoom in, or down to zoom out.

5.3.3.13 Pan Realtime

 – With this option you can "grab" and move the draft on your screen to view different sections in it – click the right mouse button anywhere on the draft and move the mouse on the screen. As long as the button is clicked, the draft will follow the mouse's movement.

5.3.3.14 Redraw – F2

 – This command allows for refreshing the screen and the drawing on it, cleaning any temporary markers that have been placed on the draft.

5.3.3.15 Mark Line

If you want to find out how a specific processing (punch or cut) is implemented in the NC code, click the  button, and then click a CAM. Its corresponding line of code will be highlighted in the NC code area.

If you want to see an NC code representation of an additional CAM, repeat the preceding procedure.

5.3.3.16 Ask

 – Choose this option and click an Entity to view information about it.

> You can only select Entities in the original part, and not in the multiplied copies.

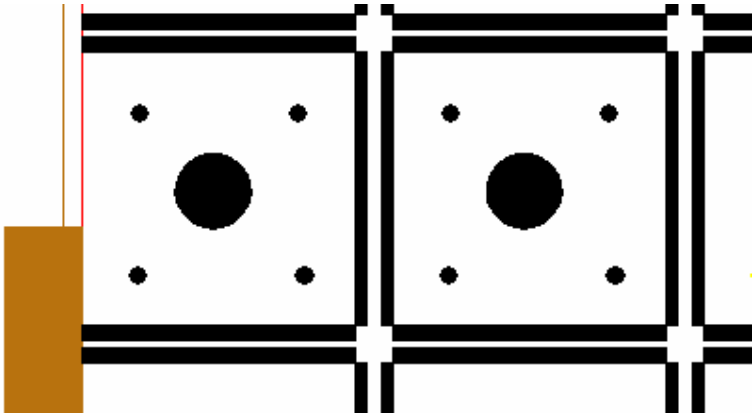
5.3.3.17 Show Clamps Bar

You should choose this option when you want to see the full extension of the Clamps holding the sheet.

More information on this option you will find in the **Clamp Bar** section of **Introduction** chapter in ***Drafting and Nesting Manual***.

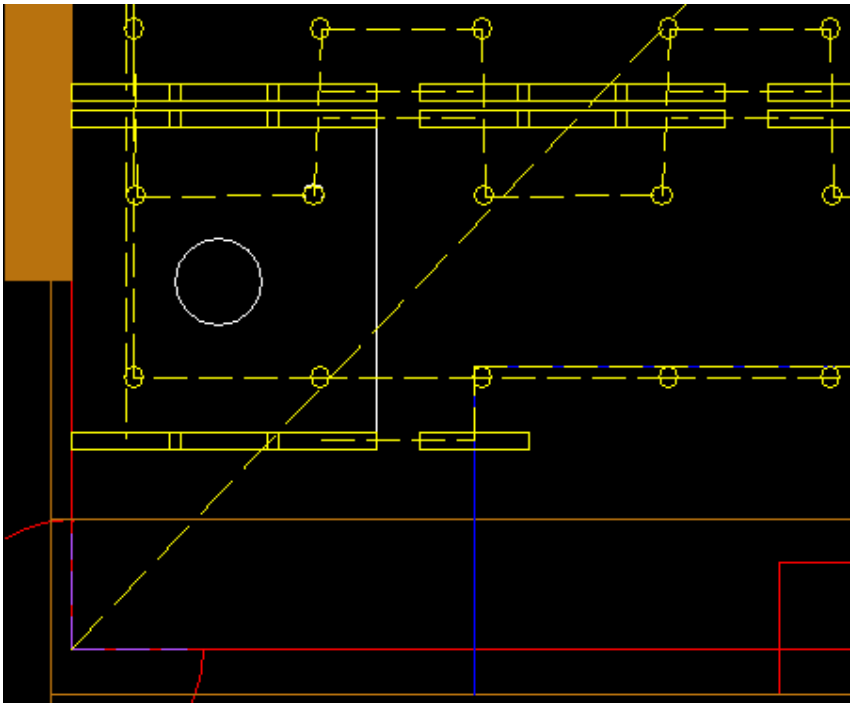
5.3.3.18 Fill Tools

Choosing this option the tools will be filled on the display. The result will be as shown on the below picture:



5.3.3.19 Rapid Tool Path

Choosing this option you will be able to see the path which the tool makes while processing the sheet.



5.3.3.20 Display Bend Lines

Enabling this option will allow you viewing the bend lines – if the currently processed part was bent. If it wasn't bent – no bend lines will be shown.

5.3.4 Measure

Use this option to measure distances on the part:

5.3.4.1 From Base Point

Use this option to measure distances from one point to multiple points on the part:

The first position you click the mouse sets the base point.

The second click measures the distance to the clicked position.

Thereafter, the next click clears the selection and the one after that measures new distance.

5.3.4.2 Point to Point

Use this option to measure distances from one point on the part to another point.

5.4 The Tools Menu

These options allow you to manage and check the tool in the Simulation setup and in incoming NC files.

5.4.1 Edit Tools Library

Here you can add or remove tools from the **Tools Library**.

For a complete explanation see the appropriate section in this manual.

5.4.2 Edit Setup

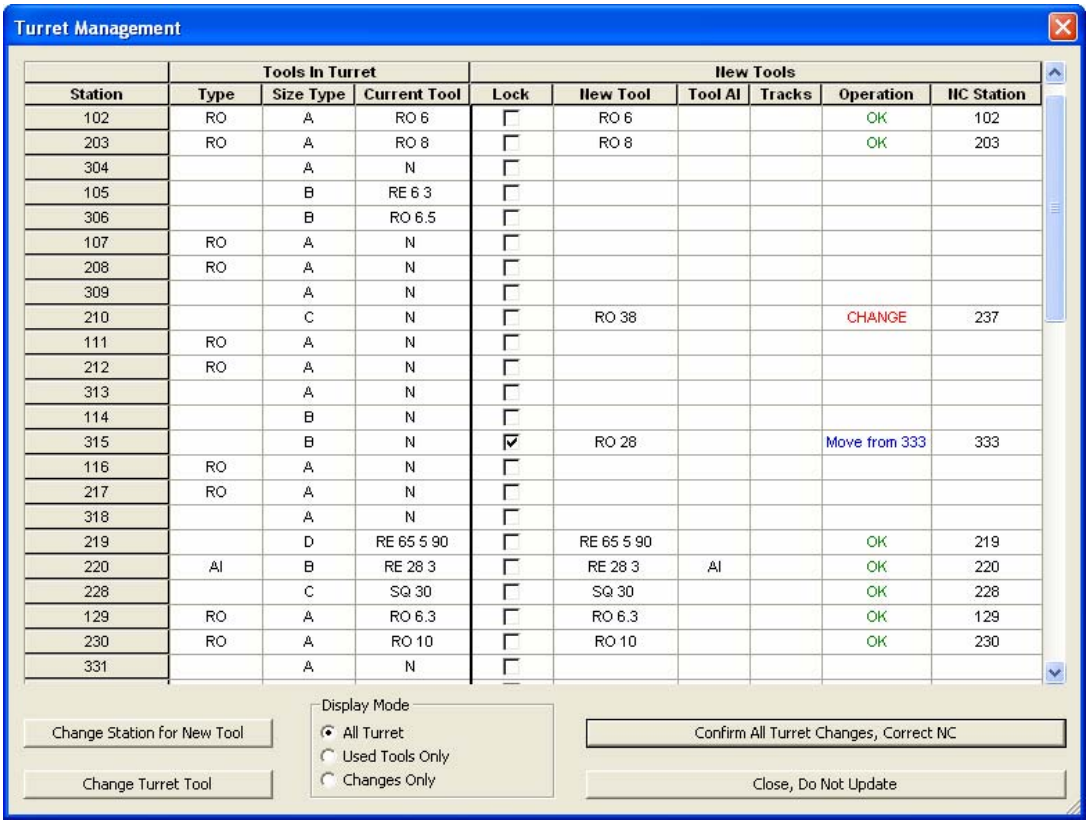
Here you can edit the Simulation's setup file.

Use this option to create a turret setup that corresponds to the machine's tooling.
This will be the basis for comparison with the NC files, so it has to be an exact match for the machine.

For a complete explanation see the appropriate section in this manual

5.4.3 Check Stations

After we have created the **Simulation Setup** we can check NC files for discrepancies.
To do this load an NC file, select this option and the **Turret Management** window will open:



5.4.3.1 Simulation tooling

This is the **Tools in Turret** section of the dialog and it shows the Simulation's tooling setup – which tools are specified for which station.

As noted, this is supposed to be an exact representation of the actual tooling in the machine.

5.4.3.2 NC tooling

This is the **New Tools** section of the dialog and it shows the tooling as specified in the NC file, and if there are any changes that have to be made in order for the NC to run correctly on the machine.

This can be based on a setup file in ***cncKad***, or simply defined by whoever processed the part.
If there are any changes to be made, they are shown in the **Operation** column:

- **Blank** – this station is not used in the NC.
- **OK** – this station matches the setup:

Operation	NC Station
OK	228

- **OK (NC Change)** – there is a discrepancy with the setup, but it can be corrected in the NC:

Operation	NC Station
OK (NC Changes)	315

- **Move from xxx** – the operator must make a change in the machine and switch between the tool in station xxx (e.g., 141) and this tool:

Operation	NC Station
Move from 141	256

- **CHANGE** – the operator must add this tool to the machine:

Operation	NC Station
CHANGE	132

5.4.3.3 Settings and definitions

These allow you to change tooling definitions, display modes and the NC file.

5.4.3.3.1 Change Station for New Tool

This button is used to move tools between stations in the Simulation's setup.
Click on one of the stations and click the button – a dialog will open showing which stations are available to accept this tool.

Only **move** them – **not** change.
To change the tool, use the **Change Turret Tool** button.

There are several restrictions for moving the tools:

- You can only move tools that are used in the current NC.
- You can move a tool only into an appropriate station – for a C sized tool, only C and larger stations will be available.

  There is also a matter of tool rotation – a tool in an **Index station** can only be moved to another **Index station**.
- You cannot change to a station that is being used in the NC.

If you do not follow these restrictions, you will either get a dialog without any stations, or none at all.

5.4.3.3.2 Change Turret Tool

This button is used to change the Simulation's setup – add or change tools in a specific station.

5.4.3.3.3 Display Mode

These settings determine which stations will be shown.

5.4.3.3.3.1 All Turret

Shows all the stations in the turret.

5.4.3.3.3.2 Used Tools Only

Shows only those stations that are used for the NC (this includes those that have to be changed in the setup).

5.4.3.3.3.3 Changes Only

Shows only those stations that require changing tools in the setup.

5.4.3.3.4 Confirm All Turret Changes, Correct NC

This will make all the changes needed to punch the part correctly:

- Add the necessary tools to the setup.
- Correct the stations in the NC to match those in the (newly changed) setup.

5.4.3.3.5 Close, Do Not Update

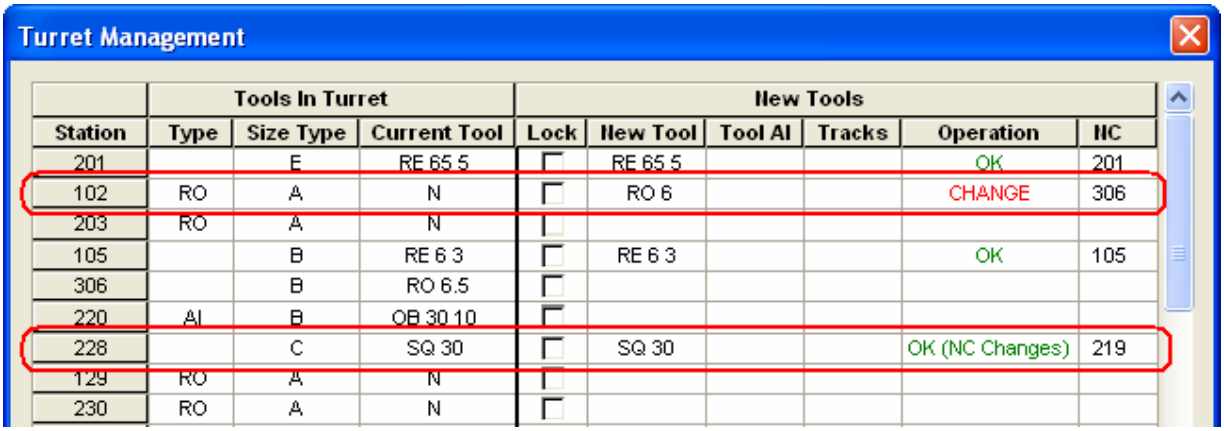
This will close the dialog without making any changes.

5.4.4 Tooling Changes Example

Below we have a section of an NC file showing station assignment as it was generated by ***cncKad***:

```
10  (*CLAMPS 100.0 401.0)
11  (MAT Steel 3.0 )
12  (*T372 RO 4.2 )
13  (*T271 RO 4.5 )
14  (*T170 RO 5 )
15  (*T105 RE 6 3 )
16  (*T256 RE 25 4 AI )
17  (*T306 SQ 30 )
18  (*T201 RE 65 5 )
19  (*T219 RO 6 )
20  G92X1830. Y1270.
21  G90
```

When we use the **Check Stations** option, we get the following:



We see results for four tools:

- RE 65 5 and RE 6 3 are in their correct stations.
- SQ 30 is in the wrong station, but this does not require any change in the turret.
- RO 6 is missing from the setup (and thus from the machine) and must be added by the operator.

After we confirm the changes, the code changes to reflect the corrected station assignments:

10	(*CLAMPS 100.0 401.0)
11	(MAT Steel 3.0)
12	(*T372 RO 4.2)
13	(*T271 RO 4.5)
14	(*T170 RO 5)
15	(*T105 RE 6 3)
16	(*T256 RE 25 4 AI)
17	(*T228 SQ 30)
18	(*T201 RE 65 5)
19	(*T102 RO 6)
20	G92X1830. Y1270.
21	G90

6 Appendices

These topics provide explanations regarding some more advanced features in ***cncKad***.

6.1 The Report File

Below you will find explanations concerning **Report Files** and directions of how to create them.

6.1.1 Small Introduction

A *Report File* is a document that provides information about a part created in ***cncKad***.

You can use the default Report Templates supplied with ***cncKad***'s installation, or you can create your own ones.

In order to create your own template you have to go through three stages:

- Creating a template
- Debugging it
- Testing it with ***cncKad***

In the following pages we will be using several concepts that are important for creating good templates:

- **Plaintext** – A file format that contains only text in a single font and nothing else; when viewing such a file with an editor, what you see is what you get. Windows supplies two plaintext editors – ***Notepad*** and DOS ***Edit***.
- **RTF** – A general format used by text editors to create documents with graphics and fonts. When viewed in ***Word*** or ***WordPad*** you will see the graphics and the fonts, but if you view such a file with a plaintext editor you will see that it is comprised of coded text that tells ***WordPad*** how to display the document.
- **DOC** – A format used by Microsoft ***Word*** to create documents that contain complex graphics and fonts. When viewing such a file with a plaintext editor you will see a lot of gibberish with some regular text strewn among it.
- **Attribute** – A characteristic of a part, such as its size, the material it is made of, which tools are used to process it and so on.
- **Token** – A sequence of letters or numbers beginning with "@"; tokens cannot contain spaces, and **must have** a space after them. Each token is unique and represents a specific attribute of a part.
  For available tokens go to: **Common Tokens** and **Additional Tokens** sections of this chapter.
- **Post Process** – Taking a draft of a part in DFT form and turning it into an NC file that can be sent to a machine. During the Post Process, ***cncKad*** keeps track of the various attributes of the part and each of these features is saved as a variable. These variables can be accessed and shown in the template you will create.
- **Template** – A file in plaintext, RTF or DOC formats containing a document that uses tokens to display information about a part. Each token represents a specific variable. During the Post Process, when the template is loaded into ***cncKad***, the post processor recognizes these tokens in the document, and replaces them with the values of the actual variables of part's attributes. After the Post Processing is over the template is saved under the part's name, with an extension denoting it as a **Report File**.

6.1.2 Types of templates

In general, a template consists of pairings of an Attribute **description** followed by the **token** for this attribute, for example:

Draft Number: @DRAWNUM

or

Customer Name: @CUST

Before beginning with template creation, you must decide what kind of format you want to make.

There are three main types of templates, with pros and cons for each:

- **Plaintext** – very easy to create and debug, but limitations in font and graphics make this kind of template plain looking.
  Go to A plaintext template example section and see an example.
- **RTF** – you can create very attractive templates, with different colors and fonts for highlighting key attributes, tables to delineate groups of attributes, etc. Creating this kind of template is a complex process, and it is hard to debug.
  Go to An RTF example section and see an example.
- **DOC** – this format allows you to utilize all the options available from ***Word*** – fonts, tables, colors, backgrounds, etc. These are the most attractive Reports, and are very easy to create, but owing to the complexity of this file format, it takes ***cncKad*** longer to generate the actual Reports during the Post Processing stage.
  Go to A DOC example section and see an example.

There is another type of template used in ***cncKad***, but it is not advisable to create templates of this type. These are DOS text templates, created with DOS *Edit*.

6.1.3 Creating the Template

Before starting a template, you should decide first which attributes of the part you want to display, and make a rough draft of it on paper.

6.1.3.1 Naming a Template

Whichever type of template you create, there is a certain naming convention you must follow in order to allow ***cncKad*** to use it fully.

The format for template names is:

RPT_xxx_name.*

Where:

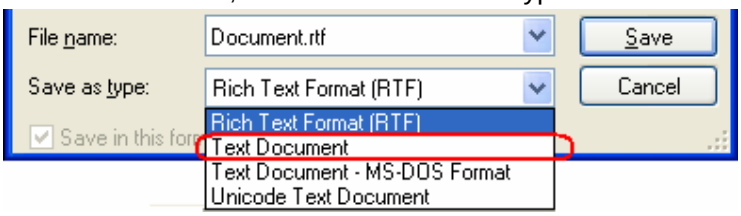
- "xxx" denotes language. Use the first three letter of the English name of the language – POR for Portuguese, SPA for Spanish, etc.
- "*" is one of the four formats used by ***cncKad*** – DOC, RTF, TXT and OEM.

It is of course possible to give different names for templates, but if you do this ***cncKad*** will not be able to recognize it automatically in the **NC Report Template Tab** of the Machine Settings dialog. See the appropriate section in Settings chapter of *Drafting and Nesting Manual*.

6.1.3.2 Creating a Plaintext Template

1. Open an editor, preferably a plaintext editor such as ***Notepad***.

2. Type in the attributes you have chosen, leaving space for a token following the attribute. You can use dashes and underscores to delineate attributes.
3. After each attribute write the corresponding token, making sure you leave at least one free space before the latter one. After each token you must leave a certain size of free space, to make a room for the attribute's value.
4. The size you must leave for each token varies – for a Time token (e.g. **@PUNCH_T**) you need 5 characters (mm:ss), whereas for a descriptive token (e.g. **@DESCR**) you will want to insert several words (30-70 characters). When designing your template it is a good practice to write down the size you allot for each token, so that you will know in the future not to attempt to insert too many characters in its place
5. Save the template as Text Document (.txt) in the **Machines** folder of ***cncKad*** (to see its location, go to **Settings => Machine Settings => Machine** tab **=> Machine Settings => Machine Path**). If you are working in a plaintext editor (***Notepad*** etc.) simply save the document to the above mentioned folder; if you are using ***WordPad***, you will have to specify the *Save as Type* as "Text Document", since there are no file type extensions displayed:



6. Go over the template again, checking that each token has at least one space preceding it and at least one following it.
7. That's it – you are done. Jump to the Testing your Template section.

6.1.3.3 Creating an RTF Template

1. Open ***Word***.
2. Make your template, incorporating the attributes you have chosen; you can use different colors, font sizes and types, and tables. When drawing tables, be aware you can only use the plain line type.
3. After each attribute write the corresponding token, making sure you leave at least one free space before the latter one. After each token you must leave a certain size of free space, to make a room for the attribute's value. If you have incorporated tables, move their limits to accommodate for it.
4. Because in this format you will probably wish to use tables, it is imperative to note the following:
   The size you must leave for each token varies – for a Time token (e.g. **@PUNCH_T**) you need five characters (mm:ss), whereas for a descriptive token (e.g. **@DESCR**) you will want to insert several words (30-70 characters). When designing your template it is a good practice to write down the size you allot for each token, so that you will know in future not to attempt to insert too many characters in its place.
5. Save your template into the **Machines** folder of ***cncKad*** (to see its location, go to **Settings => Machine Settings => Machine** tab **=> Machine Settings => Machine Path**), *making sure you save it as an RTF file*.
6. Open the template with ***WordPad***, and save it again. This removes all the extraneous coding ***Word*** inserted into it. Close the document.
7. Open the template in a plaintext editor.

8. Go over the document, and make sure that every token is followed by a space. You cannot do this in ***WordPad***, as the RTF format contains a lot of hidden code that only appears in plaintext.
9. Save the document. You don't have to worry about formats – it will be saved as an RTF.

6.1.3.4 Creating a DOC Template

Before beginning:

- Always start from one of the Templates supplied with ***cncKad***, and use the tables included in it (if you need tables).
- Do not attempt to create new tables for Parts or Tools – in the supplied tables there is an underlying code, which you would need for report to work.

1. Open ***Word***.
2. Open an existing Template and use **Save As…** option to create a new template (observing the Naming Conventions).
3. Make the changes and additions you want, incorporating the Attributes you have chosen; you can use different colors, font sizes and types, and tables.
4. Save your Template in the **Machines** folder of ***cncKad*** (to see its location, go to **Settings => Machine Settings => Machine** tab **=> Machine Settings => Machine Path**).

6.1.4 Testing your Template

Open ***cncKad*** and go to **Settings => Machine Settings => NC Report Template** tab.

In the **File Name field**, click the dropdown menu and choose your new template (assuming you have used the naming convention explained above).

Open a previously created part, and run the Post Processor by holding down the **Ctrl** key and clicking the  button in the Common Toolbar. After the post process is over, go to **File => Print Preview**, and choose the **Doc Report File** option in the Select Print Mode dialog.

A new window will open showing a Report File that was produced using your template. Go over it carefully, making sure all the values are in their proper places; this is especially important for RTF templates.

Common problems:

1. In a Text/RTF/DOC template, attribute's value isn't shown in the report.
   Resolution – reopen the template in a plaintext editor and place a space before the token.
2. In a Text/RTF/DOC template, an attribute's token is shown instead of its value.
   Resolution – reopen the template in a plaintext editor and place a space after the token.
3. In an RTF/DOC template, an attribute has "jumped" a cell in a table.
   Resolution – check that you have allotted the correct free space for the attribute preceding the one that jumped.
4. In a plaintext template, all the lines and text are out of place.
   Resolution – ***cncKad***'s plaintext templates are read in DOS, so you must correct your template with ***DOS Edit***; to do this, go to **Settings => Machine Settings => Machine tab => Machine Settings**. In the **Selected Machines** window choose your machine and then find your template in the **Edit**

Machine Files dropdown menu; click **Edit** button and your template will open in DOS ***Edit***. Correct all the offsets and save the template.

6.1.5 Tokens in cncKad

Here you will find lists of cncKad tokens that can be used for creating template:

6.1.5.1 Common Tokens

Following is a list of the most common tokens:

Token	Attribute
ANGLE	The tool's angle (Punch)
CLAMPS	The position of the clamps
CLEARANCE	Tool Clearance values: Min, Best, Max
CLMP_QTY	Number of clamps
CUST	The name of the part's customer
CUT_T	Time taken up by cutting (Laser)
CUT_TP	Cutting time per Part: to be used in a Nesting Line (Laser) Gives the cutting time only if the part has a Sub-Routine
CUT_WAY	The length of the cutting path (Laser)
DATE	Date
DATE_T	Date with Time
DESCR	The part's description
DRAWNUM	The draft's serial number
FILE	The NC file's name and partial path
LENGTH	The tool's length (Punch)
MATERIAL	The part's material
MODEL	The machine's model
MOVE_LEN	The rapid movement path (Laser)
MOVE_T	The rapid movement time
NESTING	This token denotes if this is a nesting
NOTE1	The order number of the part
NOTE2	The draft's version
NOTE3	General notes
NUM_OF_SHEETS	Number of Sheets
P_PREV	Part's preview
PART_DIM	The part's dimensions
PART_NUM	The total number of parts
PIERC_NUM	The number of pierces (Laser)
PIERC_T	The piercing time (Laser)
PROC_T	Total time for the processing
PROGLNG	The program's length
PROGNUM	The program's serial number

PROGR	The programmer's name
PUNCH_T	The time taken up by punching (Punch)
QTY	The number of parts in a Nest
REPOSNUM	The number of repositions
ROTTUR	Sheet's rotation/turn
SHEETD	The sheet's dimensions
STAT	The tool's station number (Punch)
STROKES	The number of hits per tool (Punch)
THK	The sheet's thickness
TC_T	The time taken up by tool changes (Punch)
TL_DIE	Dies for the Tool
TOOL_NUM	The total number of tools used (Punch)
TYPES	The tool's type (Punch)
WIDTH	The tool's width (Punch)
PB_NAME	Part's file name, without the extension
PB_DX	Size of Part in X
PB_DY	Size of Part in Y
PB_AREA	Part Area: when the Nest is generated by ***Auto Nest***, this is an exact area, otherwise – area of bounding rectangle
PRT_WGHT	Part Weight in Kg, e.g.: 2.120 kg When the Nest is generated by ***Auto Nest***, this is an exact Weight, otherwise – Weight of bounding rectangle
PB_WU	According to current Units, e.g.: For mm: 2.120 kg For Inch: 4.664 lbs
PB_WKG	Kg without units string: 2.120
PB_NUM	Part Running Number
SHT_WGHT	Sheet Weight in Kg: 52.50 kg
SHT_WU	According to current Units, e.g.: For mm: 52.50 kg For Inch: 115.50 lbs
SHT_WKG	Kg without units string, e.g.: 52.50
START_CORNER	Program Origin, with the following values: 0 - Top-Left Origin 1 - Bottom-Left Origin 2 - Bottom-Right Origin 3 - Top-Right Origin
X_START_CORNER	The Starting Corner X value
Y_START_CORNER	The Starting Corner Y value
Decimal Digits:	If you want to use in an RTF Report Template, **ALWAYS** use as follows:
@#S1 = @SET_FIXED_DECIMAL_DIGITS{ 3 }	At start of Report, after an Empty line!
@#S1 = @SET_FIXED_DECIMAL_DIGITS{ -1 }	At End of Report
SET_FIXED_DECIMAL_DIGITS{ 3 }	Generate 3 decimal digits, e.g.: 35.000

SET_FIXED_DECIMAL_DIGITS{ -1 }	Return to original decimal settings, e.g.: 35.

6.1.5.2 Additional Tokens

Below you can see the list of additional tokens:

Token	Attribute
ACC	Acceleration in X in mm / sec²
ACCX	Acceleration in X in mm / sec² (the same as ACC token)
ACCY	Acceleration in Y in mm / sec²
AR_SHT_THCK	Sheet directives for ARIESII directives
AR_TBLOCK	Tool blocks for old ARIES 245 directives
ARII_METAL	Material Type for ARIESII
ARIIPROGR	Programmer name (for ARIESII)
AR2STAT_NUM	Tool blocks for Nisshinbo directives
NI_TBLOCK	Tool blocks for ARIES 245-II and ARIES 255 directives
CALL_NCSEC	Call to NCSEC from model (MDL)
CL1	First clamp
CL2	Second clamp
CL3	Third clamp
CL4	Fourth clamp
CL1F	Clamp 1-st position on X with floating point
CL2F	Clamp 2-nd position on X with floating point
CL3F	Clamp 3-rd position on X with floating point
CL4F	Clamp 4-th position on X with floating point
CL1N	Hopper number of the 1-st clamp
CL2N	Hopper number of the 2-nd clamp
CL3N	Hopper number of the 3-rd clamp
CL4N	Hopper number of the 4-th clamp
CLAMPS_MOVED	The value will be equal 1 if ANY clamp has been moved
CLAMPS_CHECK	The value will be equal 1 if the clamps position should be checked
CL_ARII	Clamps position (for ARIESII)
SHT_EFF	Sheet efficiency (in percentage)
CUPS	Cups value for FinnPower when loading sheet
CUT_PRESS	Cutting pressure
DBLSLASH	Generate double slash '//'
DELAYEDPUNCH	Notifies when the punch is delayed
DFT_PATH	Path of DFT file
DFT_XOFS	DFT offset on X
DFT_YOFS	DFT offset on Y
F_HEADPOS	Focus the position of cutting head
F_PRE_PRG	Pre Program Functions

F_POST_PRG	Post Program Functions
FNAME	File name (without extensions)
FNAME_WITH_	Like @FNAME with replacing all Non-Alphanumeric signs into '_'
FEED_MAX	The maximum machine feed
FEED_X_MAX	The maximum machine feed on X
FEED_Y_MAX	The maximum machine feed on Y
FIRST_X	First punch position on X
FIRST_Y	First punch position on Y
FTL_BDC	Bottom Dead Center for Forming tools
FTL_TDC	Top Dead Center for Forming tools
GAP	Gap
GAS	The character of the gas
GLOB_COOLING_TIME	Cooling time from Global Cut
GLOB_CUT_SENSOR	Final cutting sensor
GLOB_PIERCE_SENSOR	Final piercing sensor
LAS	Gives information about laser (from Material Technology Table)
GEOM	Gives information about geometry (from Material Technology Table)
ENGRAVE_QTY	Quantity of engraved elements
PIERCE_TYPE_QNT	Number of types of piercing
LCURLY	Generate '{' left brackets
STATION_IN_SETUP	Station in setup; the value will be 1, if the specific tool is also in same station in the current setup
IS_STATION_LOCKED	The value will be 1 if the specific tool is locked in the station
LENS_NUMB	Number of cutting lenses
THIS_LINE_NUM	Number of current line
LINE_NUM_STEP	Step of the line numbering
LOADING_MODE	Loading mode: 0=Manual, 1=Auto
UNLOADING_MODE	Unloading mode: 0=Manual, 1=Auto
MATNUMBER	Material's number
MAT_TONNAGE	Tonnage of material
DENSITY	Density of material
CLR_MIN	Minimal clearance
CLR_BEST	Best clearance
CLR_MAX	Maximal clearance
MATMNAME	Machine's name (Material parameters)
MATCLASS	Material class: 0=Steel, 1=Stainless 2=Aluminium
MAT_MAX_WIDTH	Material Max Width for Dynamic Pierce
MAT_MACH_NAME2	Machine's Name 2

MAT_ID	Material ID
MACHOFSX	Machine offset on X
MACHOFSY	Machine offset on Y
MTG	Material Gage
MEASURE_SHEET	Measure sheet: 0=do NOT measure, 1=Measure
MEASURE_SHEET_TYPE	Type of Sheet Measuring
MEASURE_CORNER1_X	Measure 1-st Corner on X
MEASURE_CORNER1_Y	Measure 1-st Corner on Y
EDGE1_LENGTH_X	First Edge Length on X
EDGE1_LENGTH_Y	First Edge Length on Y
MEASURE_CORNER2_X	Measure 2-nd Corner on X
MEASURE_CORNER2_Y	Measure 2-nd Corner on Y
EDGE2_LENGTH_X	Second Edge Length on X
EDGE2_LENGTH_Y	Second Edge Length on Y
MEASURE_HOLE1_X	Measure 1-st Hole on X
MEASURE_HOLE1_Y	Measure 1-st Hole on Y
MEASURE_HOLE1_DIA	Measure diameter of the 1-st Hole
MEASURE_HOLE2_X	Measure 2-nd Hole on X
MEASURE_HOLE2_Y	Measure 2-nd Hole on Y
MEASURE_HOLE2_DIA	Measure diameter of the 2-nd Hole
MM_TO_UNITS	Convert millimeters to inches. For example: @MM_TO_UNITS{ 254 } will generate 254 for mm, and 10 for Inch
TURRET	Turret name
NCEXT	NC extension only
NCPATH	Partial NC path
NESTING	Nesting
NOTEARII	Note for ARIESII
NPART	The name of the part
NUM_ON_X	Number of parts on X
NUM_ON_Y	Number of parts on Y
PART_PATH	Partial path without extension
PB_N	Current Part Number (in Multiple Parts)
PB_PRC_NUM	Part: quantity of piercing
PB_AREA_NO_UNITS	Part: area without units
PB_SUM_PART_AREA	Total parts area
PB_WGHT_NU	Part: weight without units
PB_SUM_PART_WEIGHT	Total parts weight
ALL_VAC_FIELDS1	Notice all vacuum fields being used in the first arm of Part Unloader
ALL_VAC_FIELDS2	Notice all vacuum fields being used in the second arm of

	Part Unloader
AREA_UN	Area units
WEIGHT_UN	Weight units
MAT_THK_ID	Material thickness ID
CUT_WP	Cutting way of current part
PB_WORK_ORDER	Part's working order
PAU_STOP_PUN	The punch can be stopped from NCSEC or from Function
PAU_PUSHOUT_CENTER_PUN	The punch can be pushed out on center from NCSEC or from Function
PAU_PUSHOUT_RIGHT_PUN	The punch can be pushed out on right from NCSEC or from Function
PAU_PUSHOUT_LEFT_PUN	The punch can be pushed out on left from NCSEC or from Function
PAU_PUSHOUT_NOMOVE_PUN	The punch can be pushed out, without movement, from NCSEC or from Function
PAU_PUSHOUT_FIXED_PUN	The punch can be set as fixed from NCSEC or from Function
PAU_STOP_CUT	The cut can be stopped from NCSEC or from Function
PAU_PUSHOUT_CENTER_CUT	The cut can be pushed out on center from NCSEC or from Function
PAU_PUSHOUT_RIGHT_CUT	The cut can be pushed out on right from NCSEC or from Function
PAU_PUSHOUT_LEFT_CUT	The cut can be pushed out on left from NCSEC or from Function
PAU_PUSHOUT_NOMOVE_CUT	The cut can be pushed out, without movement, from NCSEC or from Function
PAU_PUSHOUT_FIXED_CUT	The cut can be set as fixed from NCSEC or from Function
PATH_NAME_EXT	Partial path, name and extension (DFT or NST)
PIN_NUM	Pin number
PRE_PROG_FUNC_1	Pre Program Functions 1
PRE_PROG_FUNC_2	Pre Program Functions 2
QTY_OF_SUBS	Quantity of Sub-Routines in current NC
QTY_OF_STATIONS	Total number of Stations in the Turret
QTY_TOOLS	Real number of used tools; equals the number of used stations. Example: RO 6.2 RE 50 5 RE 50.001 5 RE 50 5 Seq=99 QTY_TOOLS = 2 TOOL_NUM = 4
QTY_FORMING_TOOLS	Number of the forming tools
NAME_FORMING_TOOLS	Name of the forming tool

QTY_TAPPING_TOOLS	Number of tapping tools
QTY_MAX_TAPPING_TOOLS	Maximum Number of tapping tools allowed
QTY_LAS_TOOLS	Number of active laser stations
RAINER_TOOLS	Generates tools for Rainer machine (in the beginning of the NC)
REL_PATH	Partial (Relative) Path with NC extension
REPOSLAST	Last Reposition in X
REPOSTOTALPREVTRANS	Total value of reposition of previous Transformation
REPOSTOTALCUR	The Total value of reposition up to this point
REPOSFIRST	First reposition in X
REPOS2	Second reposition in X
REPOS3	Third reposition in X
REPOS4	Fourth reposition in X
REPOS5	Fifth reposition in X
TRANSTYPE1	First transformation type
TRANSTYPE2	Second transformation type
TRANSNAME1	First transformation name
TRANSNAME2	Second transformation name
END_OFFS_DX	Sheet End Offset in DX
END_OFFS_DY	Sheet End Offset in DY
SET_DECIMAL_MODE	Set Decimal Mode: 0- Always, 1- Never, 2- Only if needed
SET_FIELD_SIZE	Set Field Size
SET_FIXED_FIELDS	Set Fixed Fields
SET_G0123	Set the current interpolation. For example: @SET_G0123{0} to set G00, or to force interpolation on next command: @SET_G0123{ -1 }
SET_LINE_NUMBERING	Set line numbering: 0- OFF, 1- ON For example: @SET_LINE_NUMBERING{0} means: do NOT put line numbers
SET_PUNCH_MODE	Set Punch Mode: 0- OFF, 1- ON
SET_STATION	Set the current station. If you want to force Tool Change, use -1: @SET_STATION{ -1 }
SET_X_HEAD	Set the Head in X. For example: @SET_X_HEAD{130} or: @SET_X_HEAD{@#F1}
SET_Y_HEAD	Set the Head in Y
SET_X_MACH_OFS	Set machine offset on X
SET_Y_MACH_OFS	Set machine offset on Y
SFEVAL	Make some calculation; the result is a decimal number, according to the current MDL precision and number format. For example: @SIEVAL{ @SHTY / 2 }
SHEAR_IS_USED	Notices if the Shear function is being used or not
SHEETDIM	The dimensions of the sheet
SHEETGAP	Sheet Gap

SHT_TYPE	Sheet Type: 0=Normal, 1=Formed, 2=Irregular
UsedArea	The area of the bounding rectangle of the parts on the sheet
SHT_SIZE_ID	Sheet size ID
SHT_XOFS	Sheet offset on X
SHT_YOFS	Sheet offset on Y
SHTX	Sheet size in X
SHTY	Sheet size in Y
SIEVAL	Make some calculation, the result is Integer. For example: @SIEVAL{ @THK * 10 }
SHEXEVAL	Convert the decimal value to hexadecimal value
SIZE_AND_DX	Part size and distance between parts on X
SIZE_AND_DY	Part size and distance between parts on Y
ST_ABC	Station Size Letter
ST_ABC_NUM	Station Size letter, then Station Number. E.g.: B 314
STATION_NUM	Number of the station
STATION_AI	Auto Index Station: 1 for AI Station, 0 for Standard Station
STATION_MULTY	Notifies if there is a Multi Tool Station
ST_OFFSET_X	Station offset on X
ST_OFFSET_Y	Station offset on Y
SUBS_DEFS	Code for creating subroutine definitions in the NC program
SUBS_CALLS	Code for creating subroutine calls in the NC program
TC600ID	ID for Tools: a special TC600 feature
TAP_NUM	Number of Tapping tool
TAP_NUM_ROT	Number of rotations of Tapping tool
TAP_ROT_SPEED	Speed of rotations of Tapping tool
SEC_TO_TIME	Converting seconds to time: For example: 6005 => 100:05
TIME_TO_SEC	Converting time to seconds: For example: 100:05 => 6005
THK2	Thickness displayed with two decimal digits
US_THK	Thickness displayed with one decimal digit
TL_CHANGE_POS	Change tool position
TOOL_LINE	The line of the tool (in NC)
TOOL_N	The number of the tool
TOOL_FNAME	Brings name of Special (F) tool. E.g.: BANANA.T ->"BANANA"
TRAVEL_GAP	Travel gap
TRAVEL_GAP_HOLES	Travel gap in cutting of the holes
TRAVEL_GAP_PARTS	Travel gap in cutting of the parts
END_GAP	End of the gap
TSEQ	Tool Sequence number
TTLSTRK	Total number of strokes with tools
TL_TYPES	Tool types defined by two letters
TOOL_CODE	Tool Code. Codes are as follows:

	1 - RO (ROUND) 2 - RE (RECTANGULAR) -2 - SQ (SQUARE) 3 - OB (OBROUND) 5 - SD (SINGLE D) 6 - DD (DOUBLE D) 9 - F (TOOL FILE) 10 - CR (CORNER RADIUS) 12 - RR (ROUND RECTANGLE) 20 - B (BEAM - Laser, Plasma, Water, Flame, etc)
TOOL_TYPE	The shape type of the tool; the shapes parameters are as follows: TOOL_SHAPE_NONE=0, TOOL_SHAPE_CIRCLE=1, TOOL_SHAPE_RECT=2, OVAL=3, POLY=4, D=5, L=6, GRID=7, MULTIH=8, FILE=9, X=10, Q=11, ROUND_RECTANGLE=12, DRILL=13, MILL=14, MJ = 15, BEAM=20, SHEAR=30, SQUARE=-2, ROUND_SQUARE=-12, TRIANGLE = 16
TL_CMNT	The comment about the tool
TL_DIE	Tool Dies
TL_F	String of ALL the Tool Functions
TL_FIS	String of one Tool functions
TL_FIN	Number of one Tool functions
TL_WID	Wilson Wheel Tool ID: 1 – Shear F WW01 (Wilson Rolling Shear) 2 – Rib or RollBall (for Euromac) F WW02 (Wilson Rolling Rib) 3 – Offset F WW03 (Wilson Rolling Offset) 4 – Louver F WW04 (Wilson Rolling Louver) 5 – Pincher or Grooving MK (for Euromac) F WW05 (Wilson Rolling Pincher) 100 – Beading Trumpf F BEAD01.T 200 – Marking, RB for Euromac F MARK01.T
TL_S	Number of hits per this tool
TL_H	Tool height
TL_AREA	Tool area
TL_1	First tool size – length
TL_2	Second tool size – width; for CR – Cutting width
TL_3	Third tool size; for CR and RR – Radius size
X_HEAD	Current Head Position (in Sheet Coordinate X)
Y_HEAD	Current Head Position (in Sheet Coordinate Y)
Z_HEAD	Current Head Position (in Sheet Coordinate Z)
X_HEAD_OFS	Current Head Position (in Machine Offset Coordinate X)
Y_HEAD_OFS	Current Head Position (in Machine Offset Coordinate Y)
XLOADPOS	Loading position on X
XPIN	Pin position on X
XRANGE	Machine range on X
YRANGE	Machine range on Y

X_PROG_ORIGIN	Program origin in axis X
Y_PROG_ORIGIN	Program origin in axis Y
UNLOAD_X_DIM	Unload in dimension X
UNLOAD_Y_DIM	Unload in dimension Y
UNLOAD_X_COR	Unload in corner X
T_DIA	Diameter of the tube
T_WIDTH	Width of the tube
T_HEIGHT	Height of the tube
T_COR_RAD	Tube Corner Radius
T_TYPE	Tube profile type
T_PROFILE	Tube profile string
T_PROC_LEN	Tube processed length
T_SUP1	Tube Support Place 1
T_SUP2	Tube Support Place 2
T_SUP3	Tube Support Place 3
T_SUP4	Tube Support Place 4
T_SUP5	Tube Support Place 5
T_CLAMP_TECH	Tube clamping technique
T_PROC_TYPE	Tube processing type
T_WORK_POS	Working Position of the tube
T_SUP_LOAD_POS	Support loading position: 1= right side; -1= left side
T_SUP_DEAD_MINUS_X	Support Dead Zone Minus X
T_SUP_DEAD_PLUS_X	Support Dead Zone Plus Y
T_STAND_OFF1	Tube Nozzle Stand Off – Step1
T_STAND_OFF2	Tube Nozzle Stand Off – Step2
T_RED_CUT1	Reduce tube's cutting speed – Step 1
T_RED_CUT2	Reduce tube's cutting speed – Step 2
EMPTY	Create empty line in NC code
T_SUP_TYPE	Tube's support type
M_NUM_TEETH	Number of teeth in Milling
M_FEED_PER_TOOTH	Feed per tooth
M_CUT_SPEED	Cutting speed for Milling
M_T_TYPE	Milling tool type
UD_DATA1	Drawing Number (in User Data)
UD_DATA2	Project/Customer name (in User Data)
UD_DATA3	Programmer's name (in User Data)
UD_DATA4	Description (in User Data)
UD_DATA5	Order Number (in User Data)
UD_DATA6	Revision/Version number (in User Data)
UD_DATA7	Note (in User Data)

UD_TITLE1	User defined title for @UD_DATA1
UD_TITLE2	User defined title for @UD_DATA2
UD_TITLE3	User defined title for @UD_DATA3
UD_TITLE4	User defined title for @UD_DATA4
UD_TITLE5	User defined title for @UD_DATA5
UD_TITLE6	User defined title for @UD_DATA6
UD_TITLE7	User defined title for @UD_DATA7
CUR_GAS	Gas used for cutting current sheet
CUR_HEAD	Head used for processing current sheet
CUR_LENS	Lens used for cutting current sheet
GC_ENG_PRICE	Engineering rate
GC_PROG_PRICE	Programming rate
GC_MACH_HOUR_RATE	Hourly machining rate
GC_MACH_SETUP_RATE	Machine setup rate
GC_MACH_SETUP_TIME	Total machine setup time
GC_GRINDING_RATE	Hourly Grinding Rate
GC_PRICE_PER_BEND	Hourly rate per Bend
GC_TIME_PER_BEND	Bending time
GC_BEND_SETUP_RATE	Bend Setup Rate
GC_PRICE_PER_TAP	Hourly rate per Tap
GC_TIME_PER_TAP	Tapping time
GC_PRICE_PER_DRILL	Hourly rate per Drill
GC_TIME_PER_DRILL	Drilling time
C_PUNCH_SPEED	Punching speed
C_NIBBLE_SPEED	Nibbling speed
C_PUNCH_TIME	Total punching time
C_ENG_TIME	Total engineering time
C_PROG_TIME	Total programming time
C_MAT_TYPE	Material Type (in Estimation)
C_THICKNESS	Material Thickness (in Estimation)
C_MACH	Machine type name
C_GAS	Used gas type
C_HEAD	Used cutting head type
C_LENS	Used cutting lens size
C_CUT_SPEED	Machine's cutting speed
C_GRIND_TIME	Total Grinding Time
C_BEND_SETUP_TIME	Total Bend Setup Time
C_MAT_COST	Material price
C_NUM_PUNCHES	Quantity of Punches (in Estimation)
C_PIERCE_QTY	Total number of pierces

C_BOX_X	Sheet size in X (in Estimation)
C_BOX_Y	Part's bounding rectangle size in Y (in Estimation)
C_EFF	Sheet's efficiency in percentage
C_AREA_GR	Part's area gross (in Estimation)
C_AREA_NET	Part's area net (in Estimation)
C_WEIGHT_GR	Part's weight gross (in Estimation)
C_WEIGHT_NET	Part's weight net (in Estimation)
C_VAP_LEN	Vaporization path length
C_ENGRAVE_LEN	Engraving path length
C_CUT_LEN	Total cutting length
C_TRAVEL_LEN	Machine travel path length
C_CUT_TIME	Total cutting time
C_ENGRAVE_TIME	Total engraving time
C_VAP_TIME	Total vaporization time
C_PIERCE_TIME	Total piercing time
C_TRAVEL_TIME	Total machine travel time
C_GRIND_PRICE	Total Grinding Price
C_MAT_PRICE	Material Price (in Estimation)
C_DRILL_QNT	Quantity of Drills
C_TAP_QNT	Quantity of Taps
C_BEND_QNT	Quantity of Bends
C_DRILL_TIME	Total Drilling Time
C_TAP_TIME	Total Tapping Time
C_BEND_TIME	Total Bending Time
C_DRILL_PRICE	Total Drilling Price
C_TAP_PRICE	Total Tapping Price
C_BEND_SETUP_PRICE	Total Bend Setup Price
C_BEND_PRICE	Total Bending Price
C_CUT_MACH_TIME	Total processing time
C_TOTAL_MACH_TIME	Total machining time
C_ENG_COST	Total engineering cost
C_PROG_COST	Total programming cost
C_SETUP_COST	Total setup cost
C_MACH_COST	Total machine use cost
C_TOOL_QTY	Total number of used tools
C_TOOL_CHANGE_TIME	Tool change time (in Estimation)
C_TOT_COST_1	Total cost for one part with material
C_TOT_COST_NO_MAT_1	Total cost for one part without material
C_TOT_COST_10	Total cost for ten parts with material
C_TOT_COST_NO_MAT_10	Total cost for ten parts without material

C_TOT_COST_100	Total cost for hundred parts with material
C_TOT_COST_NO_MAT_100	Total cost for hundred parts without material
C_TOT_COST_1000	Total cost for thousand parts with material
C_TOT_COST_NO_MAT_1000	Total cost for thousand parts without material
C_TOT_COST_VAR	Total cost for desired amount of parts with material
C_TOT_COST_NO_MAT_VAR	Total cost for desired amount of parts without material
C_TOT_COST_VAR_AMOUNT	Any number of parts you wish to get the pricing for

6.1.6 ToolBlockOrder Model Key

The following section is for advanced users only.

One of the Keys in a machine's MDL file is **`ToolBlockOrder`**. This Key sets the ordering of the Tools:

Value	NC ordered by	Report file ordered by
0	NC order	NC order
1	NC order	TURRET order
2	TURRET order	NC order
3	TURRET order	TURRET order

- **NC order** means: the order in which the tools work.
- **Turret order** means: order by Stations – this is easier for doing the setup on the machine. Also, tools which are changed several times during the NC (when the tool is used with different Tool Sequence), will appear only once.

By default, this Key is set to "0".

6.1.7 Key Words for Tool List in NC and in Report

Supported in V6.3x.

You can put in the Report Template (and also in the HDR file) the following Key Words:

`@TL_F`	Tool Functions
`@TL_CMNT`	Tool Comment

This is useful for VIPROS machines for the special M-codes, and in other cases also where special M-codes, delays, feeds etc. are being used.
For example, in the HDR file:

Old HDR:

```
@TBLOCK_BEGIN
(*T@STATION_NUM @TOOL_LINE)
@TBLOCK_END
```

Change to:

```
@TBLOCK_BEGIN
(*T@STATION_NUM @TOOL_LINE @TL_F)
@TBLOCK_END
```

Or in the Report Template:

```
|ST No.| Type  |Length | Width|Angle  |   M   |Hits   |
|@STAT |@TYPES |@LENGTH|@WIDTH|@ANGLE | @TL_F | @TL_S |
```

For example:

```
...
(*SHEET 1000.0 1000.0 1.0)
(MAT St 37-2  SPC 1.0)
(*T202  RO 5)
(*T303  SQ 7 45)
(*T210  RE 28 3  AI)
(*T214  F LUVER.T  M536 , M13 PRESS MODE CANCEL)
(*T201  RE 65 5)
(*T218  RE 70 3 90  MEDIUM SPEED)
(*T307  F EMBOSS16.T  M549, M13 PRESS MODE CANCEL)
(*T218  RE 70 3 90  MEDIUM SPEED)
G92X1210.00 Y1270.00
G90
...
```

Another issue is the order of the tools (see the ToolBlockOrder Model Key section). Here you can see the same tool list as above, arranged by **Turret order**:

```
(*T201  RE 65 5)
(*T202  RO 5)
(*T303  SQ 7 45)
(*T307  F EMBOSS16.T  M549, M13 PRESS MODE CANCEL)
(*T210  RE 28 3  AI)
(*T214  F LUVER.T  M536 , M13 PRESS MODE CANCEL)
(*T218  RE 70 3 90  MEDIUM SPEED)
```

6.1.8 Template Examples

The following examples illustrate the differences between the types of templates, and present what can be done with each type.

6.1.8.1 A plaintext template example

The lines and words do not line up in this example; they will when ***cncKad,*** or you, will open the template in a text editor.

```
=== PART=================================================
|NC File Name: @FILE |Customer: @CUST                 |
|Draft Num: @DRAWNUM   |Date: @DATE |Version: @NOTE2      |
|Order Num: @NOTE1       |Description: @DESCR           |
=== SHEET ==============Number of Sheets: @NUM_OF_SHEETS =======
|Material: @MATERIAL    |Sheet Size: @SHEETD T= @THK    |
|Num of Parts: @PART_NUM   |Part Size: @PART_DIM       |
|Part Weight: @PRT_WGHT      |Sheet Weight: @SHT_WGHT    |
=== MACHINE ============================================
|Model: @MODEL         |Clearance: @CLEARANCE    |
|Rotate/Turn Sheet: @ROTTUR     |Clamps: @CLAMPS           |
|Num of Repositions: @REPOSNUM                          |
=== PROGRAM ============================================
|Number:@PROGNUM|Length:@PROGLNG|Programmer:@PROGR |Notes:@NOTE3|
=== JOB TIME ===========================================
|Travel: @MOVE_T|Tool Change: @TC_T|Punch:@PUNCH_T|Tot: @PROC_T |

=== USED TOOLS LIST (A total of @TOOL_NUM tools) ===============
| ST No. | Type     | Length    | Width    | Angle    | Hits      |
-----------------------------------------------------------------
| @STAT  | @TYPES   | @LENGTH   | @WIDTH   | @ANGLE   | @STROKES |
                                  | | | | | |TOTAL: @TTLSTRK |

      @NESTING
| PART NAME                                          | QUANTITY |
-------------------------------------------------------------------
| @NPART                                             |     @QTY |
```

6.1.8.2 An RTF example

PART

NC File Name: @FILE		**Number:** @PROGNUM
Order Num: @NOTE1	**Programmer:** @PROGR	**Date:** @DATE
Customer: @CUST	**Drawing:** @DRAWNUM	**Version:** @NOTE2
Description: @DESCR	**Notes:** @NOTE3	

SHEET **Number of Sheets:** @NUM_OF_SHEETS

Sheet Size: @SHEETD	**T=** @THK	**Sheet Weight:** @SHT_WGHT
Material: @MATERIAL	**Clearance:** @CLEARANCE	
Clamps: @CLAMPS	**Model:** @MODEL	
Num of Repositions: @REPOSNUM	**Rotate/Turn Sheet:** @ROTTUR	

JOB TIME

Travel: @MOVE_T	**Tool Change:** @TC_T	**Punch:** @PUNCH_T	**Tot:** @PROC_T

@NESTING

Part Name	Cutting Time	Quantity
@NPART	@CUT_TP	@QTY

6.1.8.3 A DOC Example

PART

NC File Name: @FILE		**Number:** @PROGNUM
Order Num: @NOTE1	**Programmer:** @PROGR	**Date:** @DATE
Customer: @CUST	**Drawing:** @DRAWNUM	**Version:** @NOTE2
Description: @DESCR		
Notes: @NOTE3		

SHEET **Number of Sheets:** @NUM_OF_SHEETS

Sheet Size: @SHEETD	**T=** @THK	**Sheet Weight:** @SHT_WGHT
Material: @MATERIAL	**Clearance:** @CLEARANCE	
Clamps: @CLAMPS	**Model:** @MODEL	
Num of Repositions: @REPOSNUM	**Rotate/Turn Sheet:** @ROTTUR	

JOB TIME

Travel: @MOVE_T	**Tool Change:** @TC_T	**Punch:** @PUNCH_T	**Total:** @PROC_T

	Part Name	DX	DY	QTY
@PB_N	@NPART	@PB_DX	@PB_DY	@QTY

USED TOOLS LIST (A total of @TOOL_NUM tools)

ST NO.	Type	Length	Width	Angle	Functions	Die	Strokes
@STAT	@TYPES	@LENGTH	@WIDTH	@ANGLE	@TL_F	@TL_DIE	@STROKES
TOTAL:	@TTLSTRK						

6.2 Creating New Functions

As explained previously in this manual, ***cncKad*** contains a list of pre-prepared functions that you can assign to a tool, punch or program.

If, however, you find that you are in need of a special function that does not appear in the list, you can create a new function to suit your requirements.

6.2.1 The Function's Structure

Each Function is comprised of three parts:

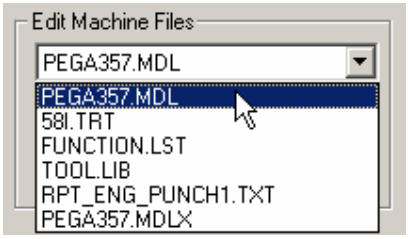
- **ID number** – this number is always preceded by `/N` .

  There cannot be two functions with the same ID number.
- **Name** – this is will be the Function's identification in ***cncKad***.
- **Code** – the actions to perform when this Function is called.

6.2.2 Making the Function available for cncKad

The Functions are defined in the machine's MDL file; the name of this file varies according to the machine, but it is always named according to the following pattern: **machine_name.MDL** (in this example the machine is a Vipros 367 Queen, and so the file is called **VIPR367Q.MDL**).

To access this file, go to the **Settings => Machine Settings =>** Machine tab **=>** Machine Settings dialog and mark the machine you are using, and from the dropdown menu shown below choose machine's .MDL file and click **Edit** button.



The Editor will open the file and you will be able to write there a new function, using the following format:

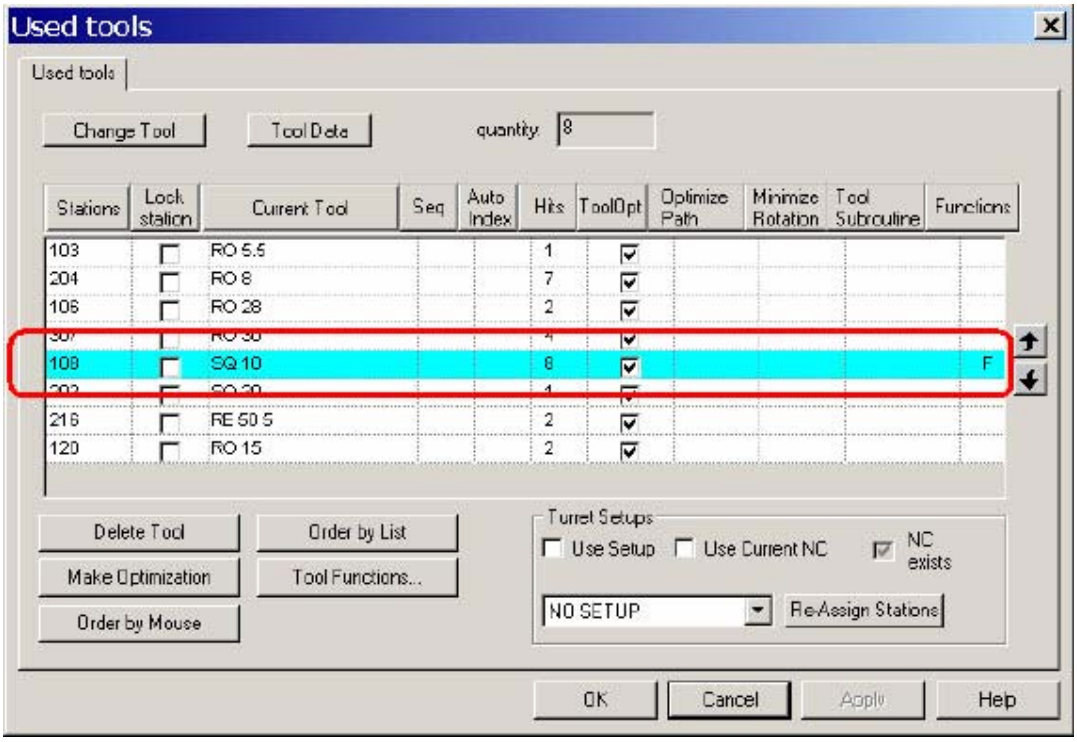
```
/N 221 NAME: "Lubrication ON"
M212
G04 X0.5
```

In the above example, 221 is the identification number, Lubrication ON is the name, and the two other lines are the actions to perform.

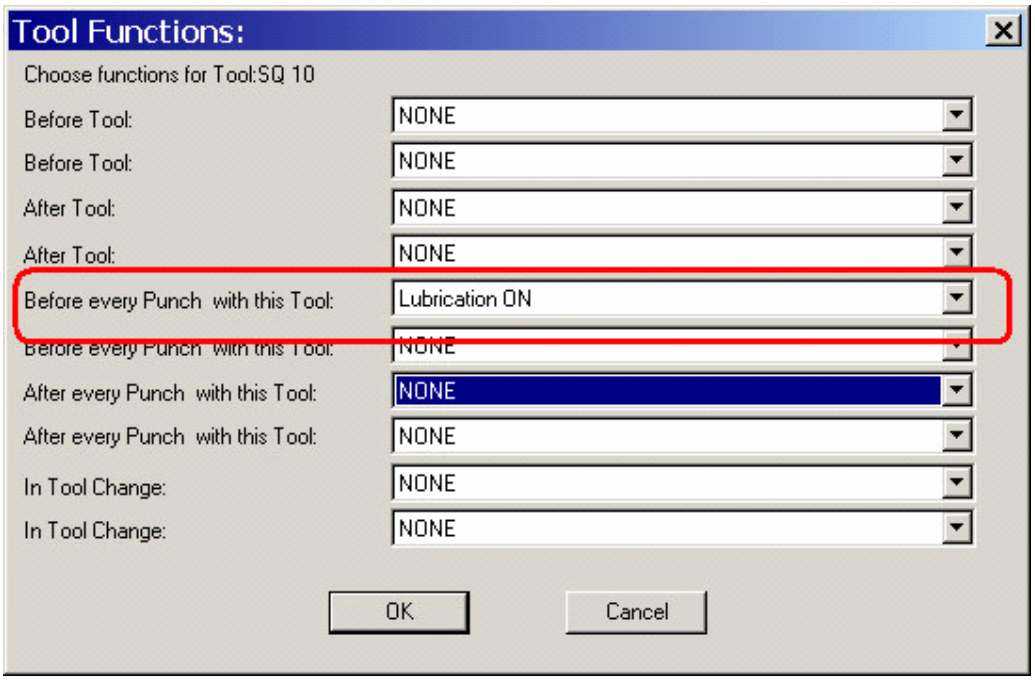
6.2.3 Adding the New Function in the Code

In this example we will insert the new function before every punch made with an **SQ 10** tool.

In the **Used Tools** dialog, mark the Turret Station holding the **SQ10** tool:

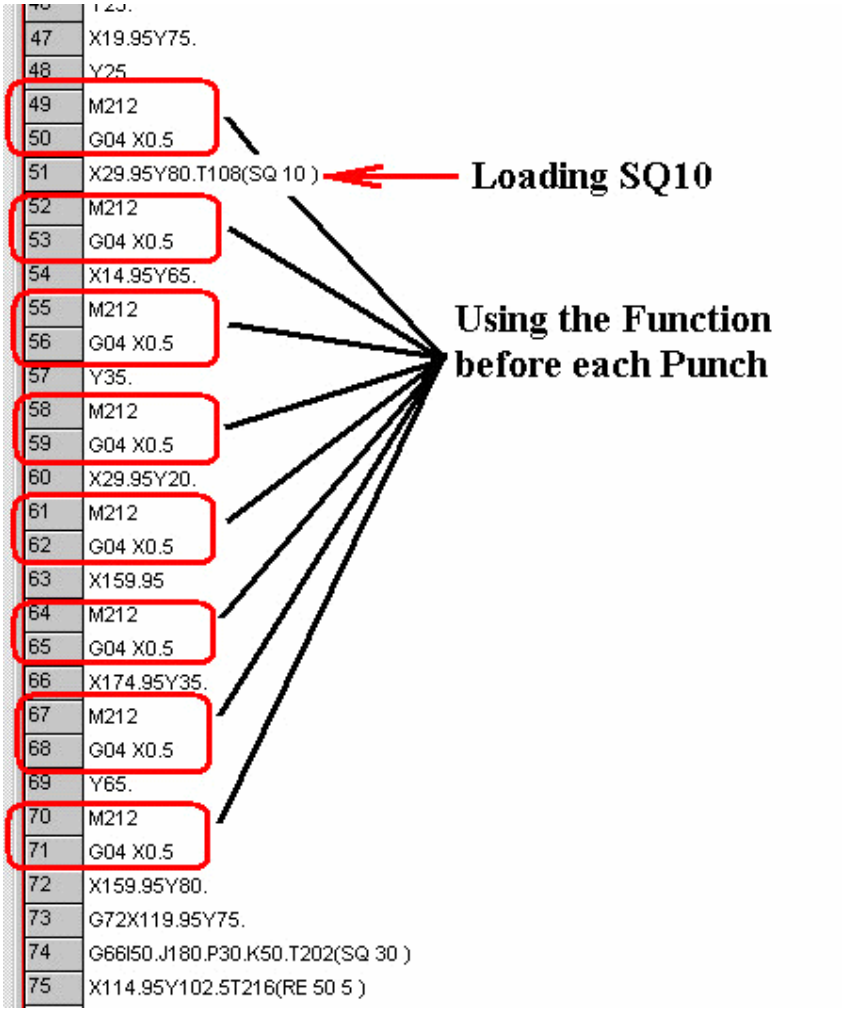


Click **Tool Functions...** button and the following dialog will open:



Define when you want this function to be used, and click **OK** button.

You will be able to see the results of this insertion after you create the NC code – the lines of code written in the MDL file now precede each punch made with the **SQ 10** tool:

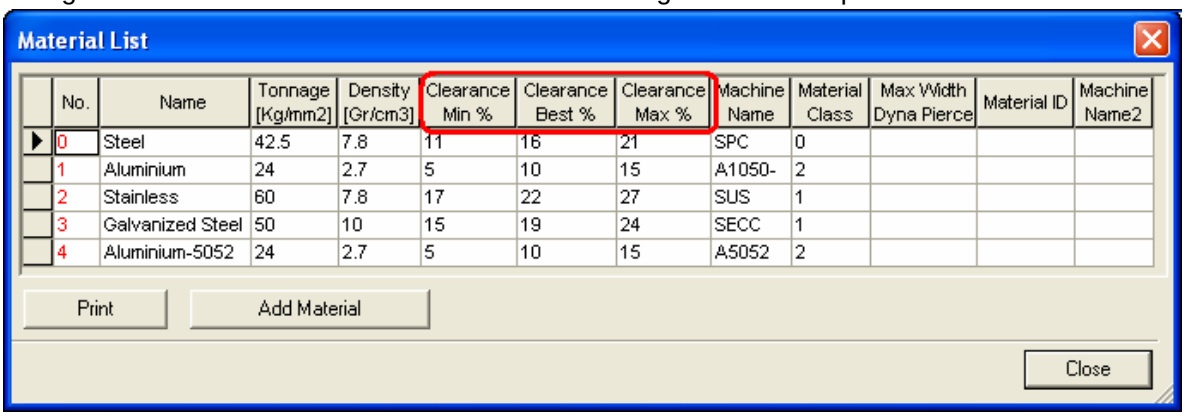


6.3 Tool Clearance

The following section is for Punch and Combination Machines only.

The **Tool Clearance** refers to the clearance between tool and die, with respect of the material.

To access the Clearance settings, go to the **Material** tab in the **Workspace Settings** dialog and click the **Edit Material** button. The following window will open:



As you can see, for each material you can define a **Minimum**, **Best** and **Maximum** values as percentages of sheet thickness.

The values for each material were taken from the Amada charts, but you can modify them to comply with the recommendations for your specific machine.

The values for the **Clearance** are indicated in the Report File, with the **@CLEARANCE** token.

6.4 Repositioning principles

Depending on your machine, the table size and the sheet size, your machine will have to reposition the clamps when working on a sheet that is longer than the table. Another reason to reposition is if you want to place punches in the clamps' **Dead Zones**.

Normally, the Post Processor generates Reposition and Head Position (during the Reposition) commands automatically, however you can add manual repositions from the **Reposition and Transformation** option of the CAM menu.

6.4.1 Clamp Avoidance

cncKad permits the user to place parts that are going to be processed in the area of, or even beneath, the clamps. This option allows manufacturers that produce grids of holes on a complete sheet, or nest multiple parts on a single sheet, to maximize sheet utilization by punching in the areas around and beneath the clamps, areas that are otherwise not accessible.

This option is activated automatically, as soon as the user positions a CAM in the area of the clamps. During the creation of the NC program, the post processor will calculate the reposition data, tool size and Dead Zone data and create a program that institutes repositioning as is needed, so as not to damage the clamps.

The number of repositions in a program is determined by several factors:

- The table size
- The sheet size
- The size of the Dead Zone

If the sheet selected is less than, or equal to, the table size – there will only be one or two repositions. If the sheet selected is larger than the table – there will be two or more repositions, as needed.

When a tool was selected (for punching), its Dead Zone is represented on the graphic creation screen as a blue rectangle surrounding the clamps.

While the program does most of the work and calculations for the repositioning, there are still some points that the user must take into consideration when using this option. Understanding and utilizing these points is crucial to the success of the automatic clamp avoidance algorithm, executed by the post processor.

For all machines the basic rules are much the same:

- The sheet is punched at the origin of the sheet and in the area where the clamps will be after the first reposition. Having executed a reposition, the program will punch the area where the clamps were before the reposition and will continue until the next reposition or to the end of the sheet.
- The position of a clamp is determined by the distance of its center from the left edge of the sheet.
- The *general* Dead Zone is determined according to the largest Dead Zone of the used tools.

- The minimum sheet size for a machine with two clamps is 3.5 times the Dead Zone, and for three clamps – 5.5 times the Dead Zone.
- Initially, the Post Processor will try to use the clamp positions set by the user, but if unsuccessful, the positions will be changed automatically.

6.4.2 Repositioning Example Tables

Setting the distances between the clamps is crucial, therefore we will give a few examples of machines with two clamps and with three clamps:

Amada 40" Tables (Pega 344, Octo 344)			Table Size = 1016	
Tool Size	Dead Zone	Clamp 1	Clamp 2	Clamp 3
A,B	160, 180	270	700-730	None
C	200	300	700	None
D	240	240	700	None

Amada 50" Tables (Aries 245,255, Pega 345, Vipros 255)			Table Size = 1270	
Tool Size	Dead Zone	Clamp 1	Clamp 2	Clamp 3
A, B	160,180	270	630-1000	None
C	200	300	700-970	None
D	240	360	840-910	None

Amada 70" Tables (Pega 357, Vipros 357)			Table Size = 1840	
Tool Size	Dead Zone	Clamp 1	Clamp 2	Clamp 3
A, B	180	270	630	990-1570
C	200	300	700	1100-1540
D	240	360	840	1320-1718

All models with table length 2000			Table Size = 2000	
Tool Size	Dead Zone	Clamp 1	Clamp 2	Clamp 3
A	180	270	630	990-1730
B	200	300	700	1100-1700
C	240	360	840	1320-1640
D	280	420	980	1540-1580

6.4.3 Automatic Repositioning for Small Sheets

As mentioned above, the minimum sheet size for utilizing the automatic Clamp Avoidance (for a machine with two clamps) is 3.5 times the Dead Zone.

For sheets smaller than this, a different strategy is needed. The initial position for both clamps is on the *right* side of the sheet and the left side is processed, and then both clamps are moved to the *left* and then the right side is processed.

This strategy is possible under these restrictions:

- The combined Dead Zones will cover less than half the sheet (otherwise there will be an overlap of Dead Zones after the repositioning).
- The maximum repositioning movement will be less than the distance from the sheet's right edge to the edge of the Table.

The following example illustrates these principles:

Table size – 1250

Dead Zone – 200

Sheet size – 600

(Automatic Clamp Avoidance not possible: 200x3.5 = 700)

Proposed solution:

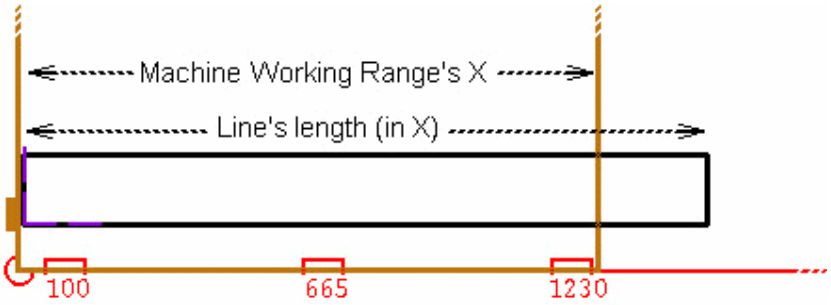
Positioning	Clamp 1	Clamp 2	Area processed
Initial	400	540	0 - 300
After reposition	60	200	300 - 600

The reposition in this case will be negative: -340mm.

It is important to note that the initial position of the clamps is at the user's discretion – they can be placed close together, with a resulting small reposition but loss of sheet stability, or they can be far apart (under the above mentioned restrictions), with a large reposition but a more stable sheet.

6.5 Laser Processing Long Parts

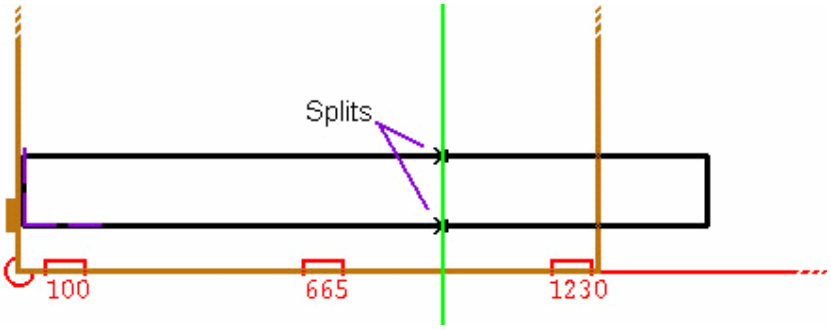
The problem of processing long parts comes up when a contour in the part includes a Line entity with an *X length* larger than the machine's *Working Range*:



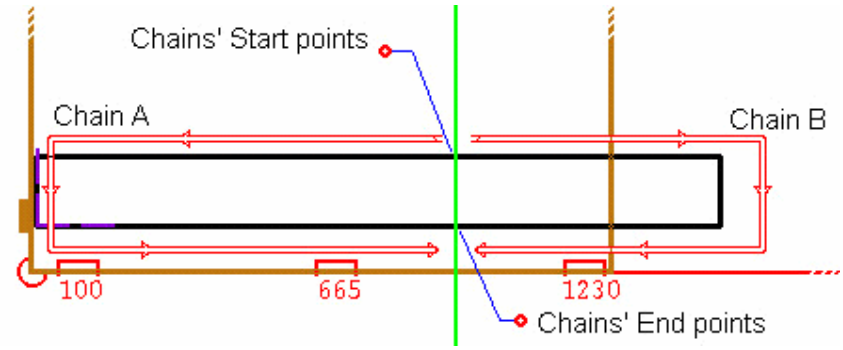
The solution is in two stages:

- Use the **Split** command, from Edit toolbar, to break up the long Line entities into segments smaller than the Working Range.

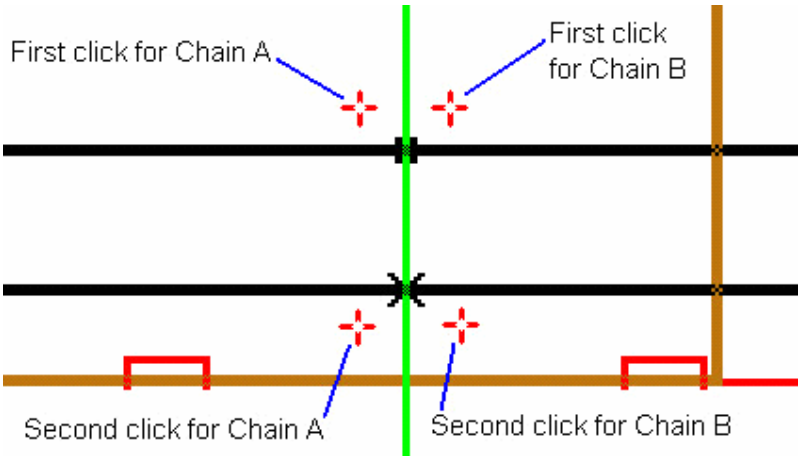
In order to make the procedure easier, first place a **Help Line** where you want to create the Splits; the simplest way is to set **Snap=Center** for the Help Line, and then return to **Snap=Auto** for the Splits:



- Process the part's contour as two Chains, where the Chains begin and end at the Splits:



It is imperative to be precise about the clicking positions when creating the chains:



After you go through this procedure, ***cncKad*** will be able to insert repositions as necessary during the Post Processing.

This procedure is necessary for **all** Lines longer than the Working Range.

6.6 Wilson Wheel Programming in cncKad

The following topics explain how to use Wilson Wheel tools with ***cncKad***.

6.6.1 Tools

The standard tool library supplied with ***cncKad*** contains the following Wilson Wheel tools:

- **`WW01.T`** - Wilson Rolling Shear
- **`WW02.T`** - Wilson Rolling Rib
- **`WW03.T`** - Wilson Rolling Offset
- **`WW04.T`** - Wilson Rolling Louver
- **`WW05.T`** - Wilson Rolling Pincher

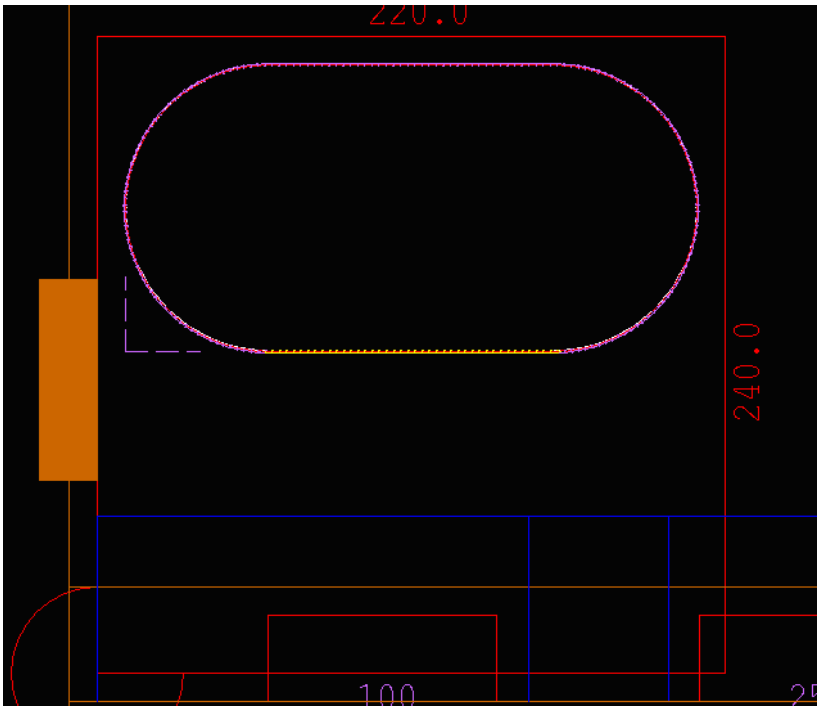
For use with Fanuc controls they are named this way:

- **`WW01`**
- **`WW02`**
- **`WW03`** and so on.

For use with Siemens and Nisshnibo, they look this way:

- **WW11**
- **WW12**
- **WW13** and so on.

Below you can see an example of Rounded Rectangle processing (200mm x 100mm, R=50mm) with WW12:



6.6.2 Programming

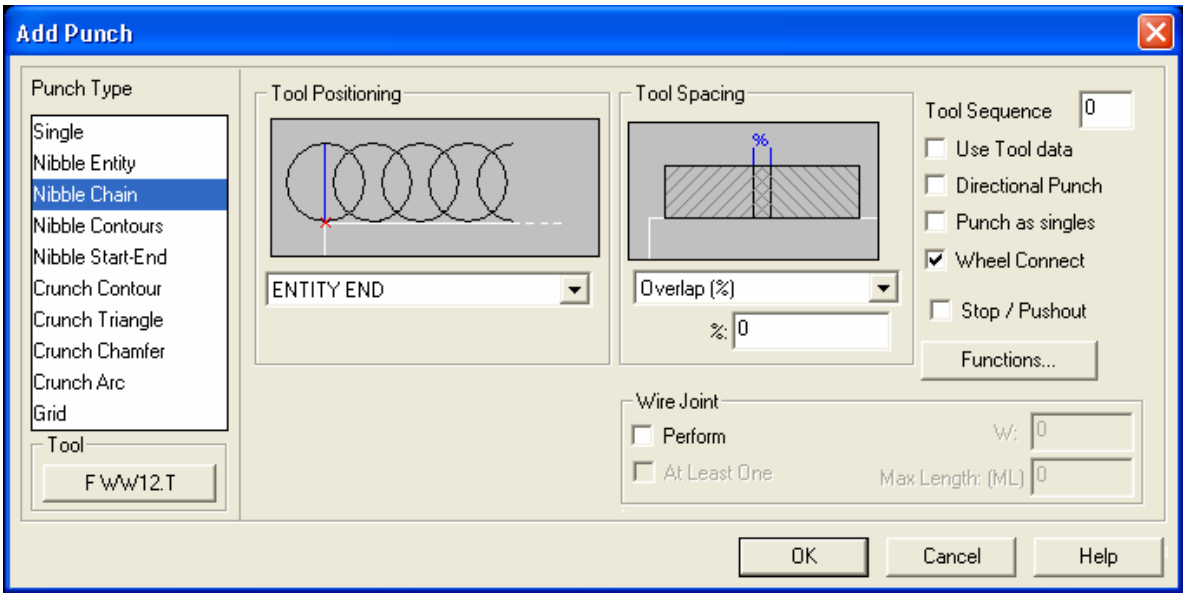
=+Nibble Entity, Nibble Chain or Nibble Contour.

Make sure **Index**=**ON** (to enable Rotation of the Wheel).

If you want to work on a "smooth" contour (no sharp corners), for example, **Round Rectangle**, you can use the

`[ x ] Connect`

so the Wheel stays down from cut to cut.



Wheel cutting Speed and Depth can be set for the tool and for individual cuts (if, for example, you want to reduce the speed at arcs compared with lines). When adding the Wheel in Add Punch dialog, mark the Use Tool Data checkbox and there, on the **Type Properties** tab you will be able to enter Speed and Depth values.

> Checking the Use Tool Data box is a shortcut to Tool Data option of Tool Library dialog.

Plunge-In:

For machines that support Plunge-In, select a line in the contour to be processed and split, for example into two halves. Start a cut on one half and finish it on the other. Using **Edit CAM** option, click the half you started cutting from, change to Offset and set a Plunge-In distance of, for example 10mm.

6.6.3 Common Cuts

When you wish to have Common Cuts for Wheel Tools, simply set the distance between Parts (**dx** or **dy** or both) to 0.01.

Then, while running the Post-Processor (NC), make sure that in the last screen (Post-Processor Options) the automatic testing of common cuts is checked.

```
[ x ] Automatic testing of common-cuts
```

6.6.4 Customization

Customization for the default values is done in the MDL file.

However, anytime you use a Wheel on your part, you can set the parameters to suit the specific cut. For more information on this issue go to **Type Properties** section of Tools Menu chapter of this manual.

6.6.4.1 Customization using Functions

- **`Function 101`** – used to start WW01
- **`Function 102`** – used to start WW02
- **`Function 103`** – used to start WW03
- **`Function 104`** – used to start WW04
- **`Function 105`** – used to start WW05

- **`Function 99`** – used to End all the Wheel Tools
- **`Function 100`** – Wheel Connect (keep cutting/forming, do NOT lift-up Head)

6.6.4.2 Customization using NCSEC

- **`[NCSEC_WW_Start]`** to start any WW tool (Functions 101 -105 will NOT be used)
- **`[NCSEC_WW_End]`** to end any WW tool (Function 99 will NOT be used)
- **`[NCSEC_WW_Down]`** lower the Head to start cutting/forming
- **`[NCSEC_WW_Up]`** lift-up the Head to Stop cutting/forming

6.6.5 Cutting/Forming Speed

In the NCSEC or Function for the Wheel Start, normally some Feed command is placed, e.g.

```
F4000
```

This will be the **default speed** of the Tool.

Normally, the machine operator will adjust it on the machine (see also the Use Tool Data option).

6.6.6 The Wilson Wheel Section in the MDL file.

```
[Wilson]
BaseFuncWilsonStart = 100          // function 100 is: Continue Wilson (NO RAM up)
;                                  // function 101 is Wilson Rolling Shear Start Commands.
;                                  // They are inserted as Tool-Change function automatically.
;                                  // 102 is Rolling Rib Start Commands,
;                                  // 103 is Offset,
;                                  // 104 Louver
;                                  // 105 Pincher
ControlSupportG02G03 = Yes         // can we use G02 and G03 or we have to use
;                                  // Nibble commands
ShedMaterialTravel = 10            // When working with WW01 (Rolling Shear = Cutting),
;                                  // whenever finishing a cut and Lifting Up the Head, a
;                                  // 10mm travel is done to "Shake away" the Sheet
;                                  // from the Wheel.
```

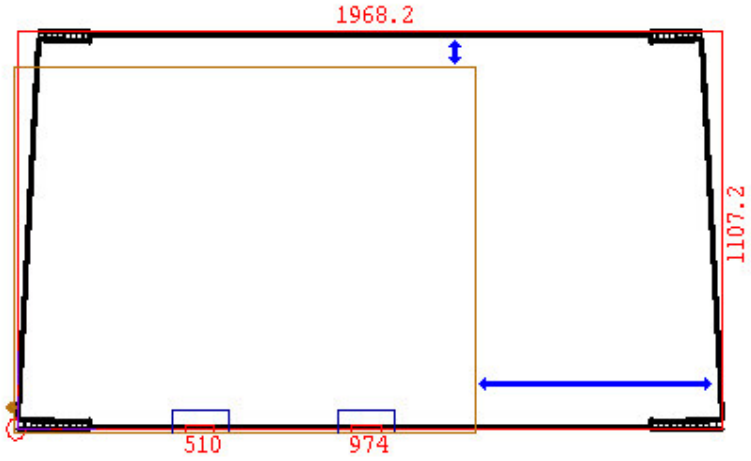
6.7 Using Sheet Transformation

6.7.1 Processing with a Sheet Transformation

Often the sheet in use extends beyond the machine's **Working Range**. When this happens in the X range, the solution is to use a **Reposition**, however when the sheet extends over the Y range, the solution is using a **Transformation**.

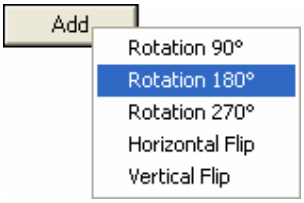
- **The problem:**

Processing a 2000mm x 1100mm part on an Amada Aries245 (Working Range 1000x1270):



Note above the arrows marking the sheet's extension beyond machine's Working Range.

- **The solution:**
  - Click the **Reposition and Transformation** button - 
  - Use the **Add** button to add a 180° rotation:



  - In the **Transformations** section of the dialog select the **Rotation 180°** entry, and then click the **Set CAMs to Current** button, to select the parts you want to process when the sheet is in this position. For that sake you can use any of the standard Selection Methods (Single, Window, etc.).
  - Now you have two Transformations; note that when you click on either of them, the CAMs that are defined for it will be highlighted.

After you generate the NC, you will see that there is a **Stop Machine** (M00) inserted in order to allow rotating the sheet before the machining continues.

You can also use **Sheet Rotation** when there is a problem with the clamps' Dead Zones and the sheet is too small for using a Reposition – first process *away* from the clamps, rotate the sheet and process the rest.

6.7.2 Sheet Transformation with Reposition

Sheet Transformation can be used in conjunction with **Reposition** in the following way:

- **The problem:**

Processing a 2000mm x 1200mm sheet on an Amada Pega344 (Working Range 1000x1000).

- **The solution:**
  - Process the bottom 600mm, using **Repositions** for **Clamp Avoidance** and in order to reach the X=2000 position.
  - Insert a **Rotation 180°** type **Transformation**, and process the remainder of the sheet.

6.8 The NC to DFT feature

This section explains the steps needed to turn a given NC file into a DFT file.

6.8.1 Preparing the GCD file

To use the ***NC to DFT*** option, some changes must be made to the header of the NC – several comment lines with information must be added.

For example, the original GCD file contained the following lines:

```
(AD1-OS)
G92X1830.00Y1270.00
G90
U1
X24.5Y120.T230
V1
G73X763.Y563.Q3W1
G73Q4W1
G73Q2W1
U2
X92.Y62.T150
V2
G73Q3W2
G73Q4W2
G73Q2W2
U3
G72X46.Y46.
G66I50.J270.P-30.Q-80.T237
V3
G73Q3W3
G73Q4W3
G73Q2W3
G50
```

To use this option, the following lines (marked with the asterisk) have to be added:

```
(AD1-OS)
(*MODEL PEGA357 58I)
(*SHEET 750.0 550.0 1.0)
```

```
(*CLAMPS 290.0 440.0)
(*T230 RO 10)
(*T150 RO 20)
(*T237 RE 80 30)
G92X1830.00Y1270.00
G90
U1
X24.5Y120.T230
V1
G73X763.Y563.Q3W1
G73Q4W1
G73Q2W1
U2
X92.Y62.T150
V2
G73Q3W2
G73Q4W2
G73Q2W2
U3
G72X46.Y46.
G66I50.J270.P-30.Q-80.T237
V3
G73Q3W3
G73Q4W3
G73Q2W3
G50
```

In this case, the comment lines are added after the first line, the one with the *Part Name*. They contain information about the *Sheet size*, *Clamps* position, and *Tools*.

If the customer uses a Standard Setup, then a Setup file can be created as explained later on.

These new comment lines can be stored in a file, for example in **HDR.PNC** .

In such a case, the steps to prepare the GCD file are as follow:

1. Open Metalix Editor.
2. Open HDR.PNC
3. Mark all the lines and select **Edit => Copy** (or **Ctrl+C**)
4. Open the GCD file.
5. Place the cursor at the start of the second line (following the *Part Name*)
6. Select **Edit => Paste** (or **Ctrl+V**)
7. Correct *Sheet Size*, *Clamps*, and *Tools* if needed.
8. Use **File => Save As** with the same name, but with extension **.PNC**

6.8.2 Loading the prepared NC file

Open any part in ***cncKad***, click **NC** icon in order to generate an NC File, and then go to the ***Simulation*** program. There select **File => Open**, and load the **.PNC** file you prepared just before.

You will be able to see the Part on the Graphic Simulation screen, **Run** it slowly or **Line by Line** (**Line**=**ON**). It is also possible to make manual corrections to the NC file, and see and verify them immediately in the Graphic Simulation window.

After making changes to any line, remember to press **Enter** key in order to confirm the change.

6.8.3 Creating a drawing (DFT)

After finishing the steps described above, you can select **File => NC to DFT** option in the ***Simulation*** program. This will generate a Drawing with the CAMs already in place!

Now you can close the ***Simulation***, return to the main ***cncKad*** screen, and use **File => Open Part** to load the newly created DFT file.

Note that the generated DFT will normally NOT have a valid geometry, therefore it is recommended that you correct it manually.

6.8.4 Creating a Setup file

In ***cncKad*** go to **Tools => Turret Setups** and create your standard **Setup File**, for example:

PEGA357.PET

For more information on **Setup file** including the procedure of creating a new Setup and editing an existing one, see the appropriate sections of Tools Menu chapter of this manual.

In order to use a Setup file, you must add the following lines (marked with asterisk) in your **HDR.PNC** file:

```
(*MODEL PEGA357 58I)
(*SHEET 750.0 550.0 1.0)
(*CLAMPS 290.0 440.0)
(*SETUP PEGA357.PET)
```

You must also remove the **T lines** (the lines that specify Tools).

If for Part processing the Tools are being used ONLY from the Setup, then adding these 4 lines is all you need to do. However if tools for Part processing are being used both from the Setup, AND a few tools NOT from the Setup ("Non-Standard Tools"), you should add the **T lines** only for the "Non-Standard Tools".

For example, the following tools are being used to process the Part:

RO 6.5
RO 4
SQ 12.7
OB 32 8 90
RE 50 5
RE 50 5 90

And from among them, only the OB is "Non-Standard Tool". So in this case your NC header will look this way:

```
(*MODEL PEGA357 58I)
(*SHEET 750.0 550.0 1.0)
(*CLAMPS 290.0 440.0)
(*SETUP PEGA357.PET)
(*T303 OB 32 8 90)
```

6.9 Laser Common Cuts

The Laser Common Cut feature is being used to process the edges of two adjoining parts with the same cut; instead of cutting each part individually, the following procedure is implemented:

- When a part is being processed, ***cncKad*** checks if any juncture on the part's contour is also the starting point for a **future** cut – one for cutting the contour of an **adjoining** part.
- When ***cncKad*** prepares to cut this adjoining part, it begins from the Preparatory Cut (using an *Overlap*, to create a smooth transition), made in the direction of the future cut, thus avoiding the need to pierce the sheet again.

The **advantages** of using this method are:

- Reduced machining time.
- Savings in cutting \gas and wearing of parts.
- Reduced material wastage.

Using Laser Common Cuts is recommended in the following cases:

- Parts with long, straight contours.
- Parts cut from thick sheets, where the cutting speed is low.
- Parts with contours that will be vaporized, as these require two passes, resulting in a double reduction of machining time and gas.

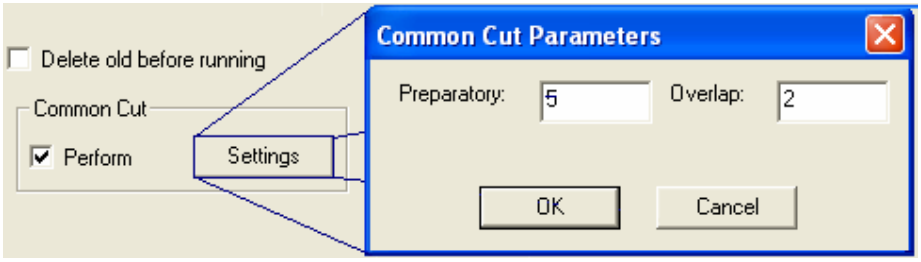
6.9.1 Preparing a Geometry for Common Cut

In order to use Laser Common Cut on geometry, it is compulsory to prepare geometry particularly for it, first; there are two options of doing this:

- **Array** a part (using the **Part Array** option from Transform menu) with the **dX** and **dY** distances set at Beam diameter (see the Example below).
- Load parts into a Nest in ***Auto Nest***, set the Buffer size at half the Beam diameter (e.g. for a Beam with a 0.2 diameter use a 0.1 Buffer), and then send the entire Nest to ***cncKad*** as a DFT file.

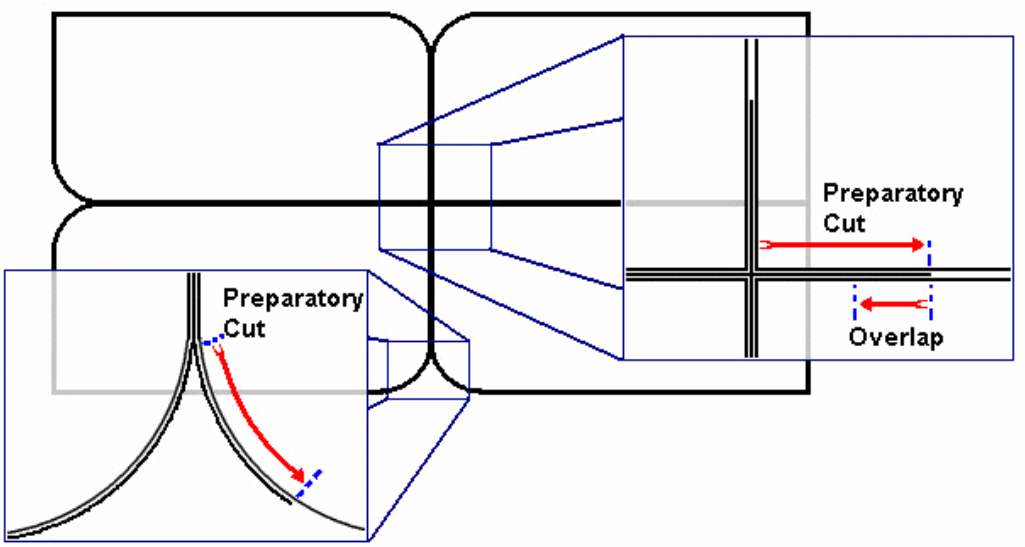
6.9.2 Preparatory Cut

In the **Auto Cut** dialog, on **Auto Cut** tab mark the **Perform** checkbox and click the **Settings** button, like shown below. The Common Cut Parameters dialog will open:



From this dialog you can set the parameters governing this option:

- **Preparatory** (Cut Length) – in order to minimize the need of piercing, the Post Processor will create a **Preparatory Cut** in the positions where a new cut will start.
  The example below shows how the bottom-left part will be cut:



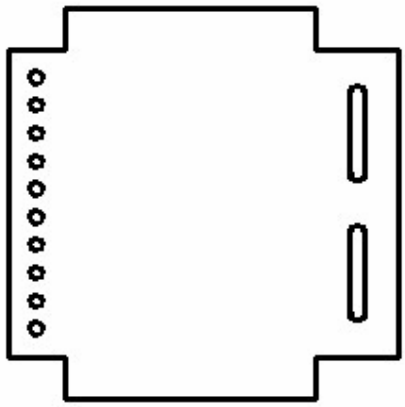
Notice the enlarged sections – when arriving at the middle juncture, the cutting is extended *upwards* and *to the right*, to prepare for cutting the bottom-right and top-right parts. The same action is being taken at the bottom juncture.

- **Overlap** – this is the distance from the end of the **Preparatory Cut**, to where the actual cutting will start when processing will resume on the adjoining parts.

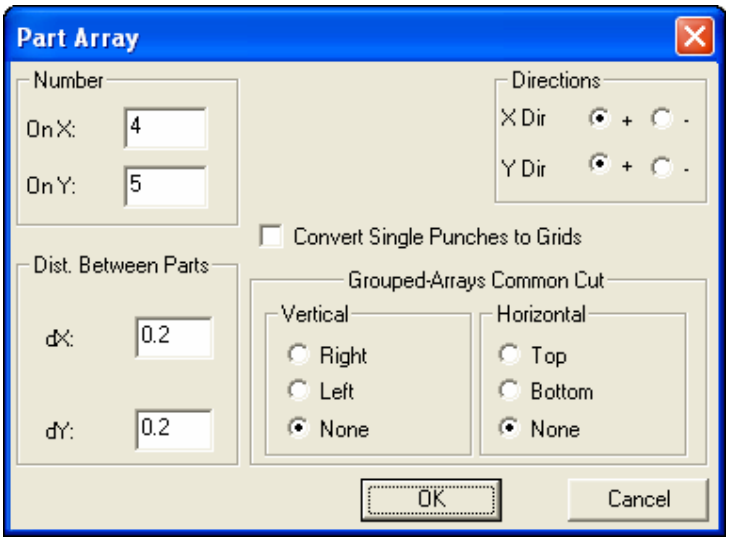
6.9.3 Example

This feature can only be used on complete geometries, where the distance between parts has been set to the Beam Diameter.

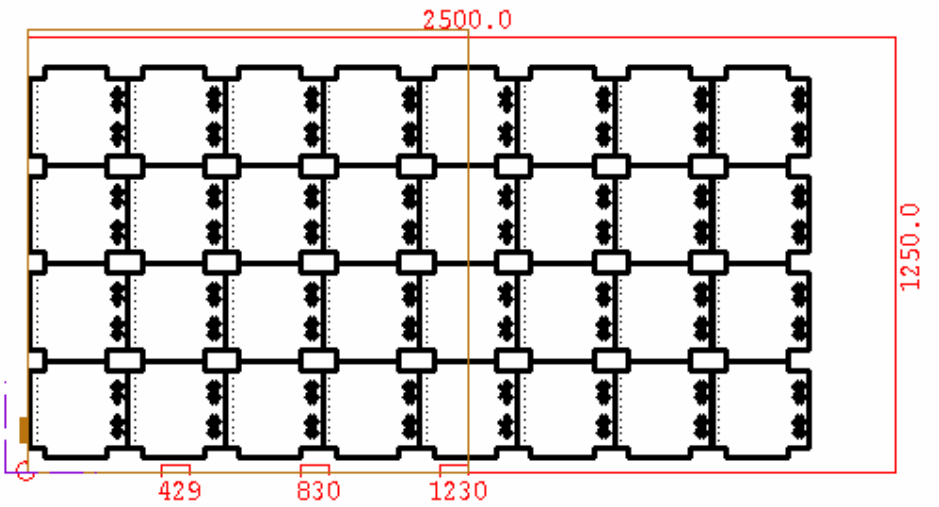
In this example we will make an Array of the following part:



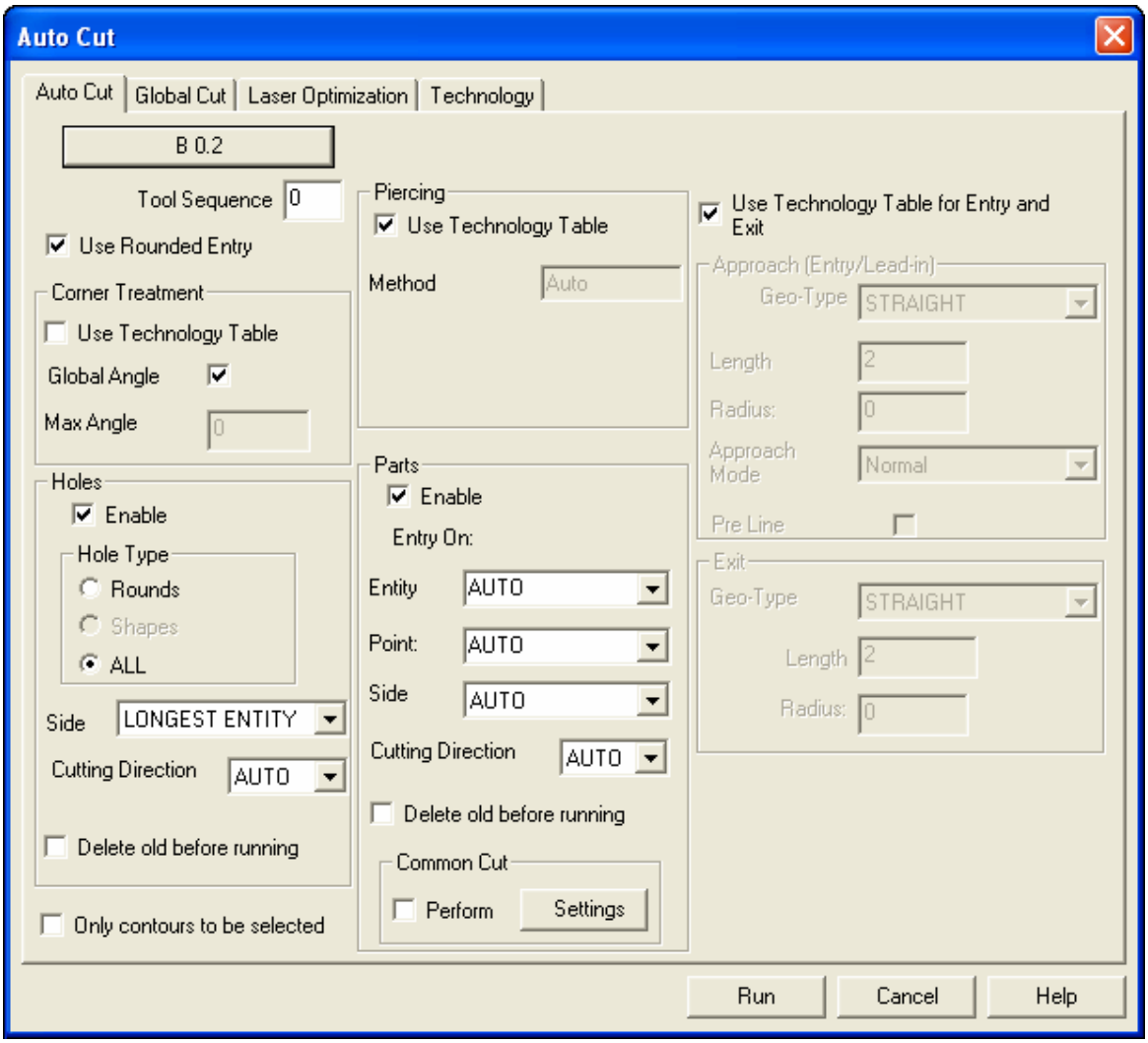
To array the part we will use the **Part Array** option from **Transform** menu. After choosing it, click the contour of the part; the entire part will be highlighted in blue color. Press **Enter** key in order to confirm the selection. As soon as you do so, the following dialog will open:



Fill out the parameters definitions as shown above; note that in the Distance Between Parts section, **dX** and **dY** are both set to **0.2**, which is the Beam diameter value.
Click the **OK** button, and the part will be arrayed on the sheet according to the above definitions:

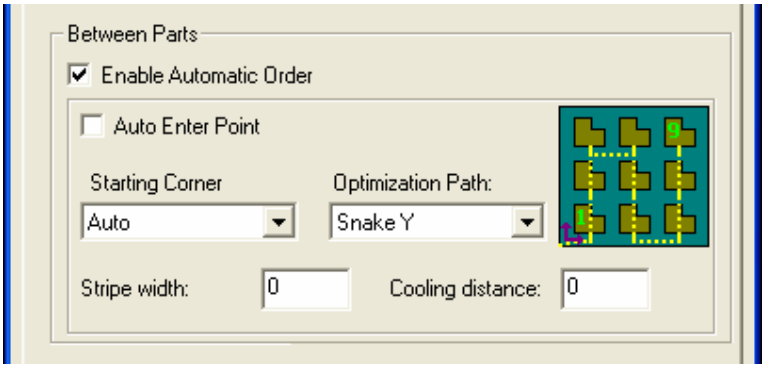


Now it is time to process the sheet; click the Auto Cut button. When the following dialog will open, select the **Auto Cut tab**:



In this tab, select the **Perform** checkbox in the Common Cut section, and click the **Settings** button. After doing so, the Common Cut Parameters dialog will open. Enter the settings you desire, and click **OK** button.

Now go to the **Laser Optimization** tab and set the path **Between Parts** as either **Snake X** or **Snake Y**:



When you have finished, click the **Run** button. The entire sheet will be processed, including the inner contours of the arrayed part. If you don't want to process the inner contours, you should check the **Only contours to be selected** option on the Auto Cut tab.

Use the **Ctrl+B** key shortcut (or **View => Full Cut Tool Width**) for switching the Beam's width *on* and *off*, to make sure the cuts are processing the arrayed part correctly.

In order to view the effect of using the Common Cut function on the actual processing, we will use the ***cncKad***'s **Simulation** program. To do that – generate NC for the part, and then go to the Simulation window.

Run the Simulation, and **Pause** after the contours of the first part have been processed. **Zoom** onto the intersections with the adjoining parts, and note the Preparatory Cuts.

6.10 Additional Options in cncKad's Machine files

6.10.1 Additions in the MDL file

These changes have been introduced from ***cncKad*** version 6.41:

6.10.1.1 New MDL Keys

- `NCUnitsMode = ?` with values as follow:
  - 0 – Use DFT units
  - 1 – Always generate mm
  - 2 – Always generate Inch
- `MPCornerMode = ?` with values as follow:
  - 0 – **Q1** is at bottom-left, and **Dy** is positive
  - 1 – **Q1** is at top-left, and **Dy** is negative
- `AssignStationsMode = ?` with values as follow:
  - 0 – Allow smaller tools into bigger stations
  - 1 – Each tool size **MUST** fit the Station Size
- **`[SendToDiskette]`**
  - `CreateFolderIfNeeded = No` // if Yes, the Target Folder will be created, if does not exist.
  - `AskForConfirmation = No` // if Yes, user is asked to confirm copying
  - `CopyNCReportToo = No` // if Yes, the NC Report file (if exists) is copied to the same destination as NC.

6.10.1.2 Functions have moved to MDL files

Functions Names are now specified in the MDL, and each MDL file contains only those functions that belong to specific machine:

```
...
N 0 BEGIN
/N 1 NAME: "0.5 SEC DELAY"
G04 X0.5
/N 2 NAME: "1.0 SEC DELAY"
G04 X1.
/N 3 NAME: "2.0 SEC DELAY"
G04 X2.
/N 4 NAME: "5.0 SEC DELAY"
G04 X5.
/N 5 NAME: "LOW SPEED"
F4
```

```
/N 6 NAME: "MEDIUM SPEED"
F3
/N 7 NAME: "HIGH SPEED"
F2
/N 8 NAME: "MAXIMUM SPEED"
F1
/N 9 NAME: "OPTIONAL STOP"
M01
...
```

6.10.1.3 Functions can be used as Pre-Macro or Post-Macro

```
/N 1082 #S! NAME: "M82 Macro Only"
M82
/N 1083 #S! NAME: "M83 Macro Only"
M83
```

If "M82 Macro Only" is used as Pre-Tool, and "M83 Macro Only" is used as Post-Tool and for example, U3-V3 is the Macro for that Tool, you will get:

```
M82
U2
….
V2
G76 W2 Q1
M83
```

6.10.1.4 Delete comments from NC

Some machines do not accept the comments that appear between brackets in the NC code. In the MDL file, you can set the deletion of these comments as they are sent to a diskette.

All the following fields can be copied from the **Example Model File** (Examp.mdl) situated in the **Machines** folder, if they are not already existent in the MDL file:

In your machine's MDL file, go to the section `[SendToDiskette]`
and add `New Key =`

Enter: `Remove Comments = Yes`
`CopySrc2Trg = Yes`

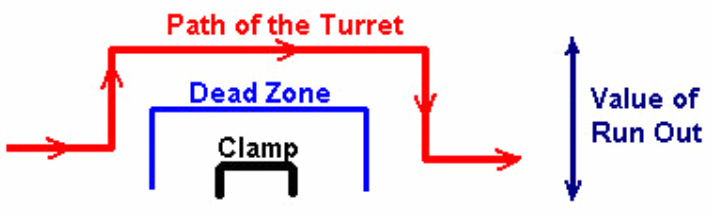
You can define the Target Extension as you wish, for example: `trgext = nc`

> If the NC code is modified in the Simulation screen, that change will be automatically updated to the **.nc** file.

6.10.2 Run Out in cncKad

A **Run Out** means that the Turret never passes over a Clamp. This result is achieved in the following way:

When the Turret approaches a Clamp, the Sheet moves away in **Y**, bypasses the Clamp in **X** and then comes back in **Y**:



To define a Run Out you must edit the machine's MDL file, in the section called `General Info`.

There are 3 key values to consider:

```
…
[General Info]
RunOutRequired = No
YExtraForRunOut = 0
MinYForRunOut = 0
…
```

- **RunOutRequired** – choose **Yes** or **No** to use a **Run Out** option.
- **YExtraForRunOut** – when using the Run Out option, this value (in mm) will be added to the Dead Zone as defined in the Turret file.
- **MinYForRunOu** – this defines the minimum value (in mm) for the Run Out, even when the Dead Zone **+** `YExtraForRunOut` are less than this value.

If you are using Inches as your System Units, you must still define the values in **mm**; these will be automatically converted by ***cncKad***.

For Example for the following definitions:

```
…
[General Info]
RunOutRequired = Yes
YExtraForRunOut = 40
MinYForRunOut = 200
…
```

The result will be that a Run Out will **always** be used, with a value of 200mm (~ 7.9") or Dead Zone **+** 40 (~ 1.6"), whichever is greater.

6.10.3 Support for very big arcs

When the NC codes for generating Arcs contain Angles, we can have an accuracy problem when the Arc Radius is very big.

This happens in all FANUC controlled machines (Amada, Murata, FinnPower, LVD, and more) and in some other controls as well.

The accuracy problem is now resolved automatically by adding the following Keys into the MDL file:

```
[NC_GENERATION]
ArcNCByAngles=Yes       // supported from V8.0: correct NC
                           for very big Arcs
AnglePrecision= 0.01
```

6.10.4 Additions in the TRT files

For **Round Only** stations, it is now possible to set:

- 0 – Any Shape

- 1 – Only Rounds
- 90 – Used for an **Auto Index** station; allows rotating the index station only at 90 degrees

For more information on these features see the Edit Turret section on Tool Menu chapter of this manual.

6.10.5 Support for Program Origin

To enable/disable support for Program Origin, do the following:

- Set the following two Keys in the machine MDL files `[Machine Info]`section:
- `SetOriginEnable = ?` with values as follow:
  - 0 – disable,
  - 1 – enable, but default is false,
  - 2 – enable
- `InitialOffset = ?` with values as follow for the initial point:
  - 0 – Top-Left Origin, i.e. (0, Sheet Y size)
  - 1 – Bottom-Left Origin, i.e. (0, 0)
  - 2 – Bottom-Right Origin, i.e. (Sheet X size, 0)
  - 3 – Top-Right Origin, i.e. (Sheet X size, Sheet Y size)

The program uses for the latter mentioned Key:

- Incremental programming – used to set the Initial Head Position.
- Absolute programming – used to set the Machine Offset.

- In the **DIR_xxx.HDR** file change the SHEET directive as follows:

```
(*SHEET @SHTX @SHTY @THK @MATNUMBER @START_CORNER
@X_START_CORNER @Y_START_CORNER)
```

cncKad will change its coordinates according to new program origin.

6.10.6 MDLX per machine file

Some of the functions defined in the MDL file have been moved to a file with **MDLX** extension –this file is also per machine, and it includes all the machine parameters that are editable via ***cncKad***.

6.11 Inches users upgrading from previous versions to 7.xx

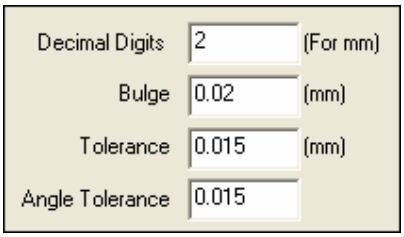
The following information is intended for ***cncKad*** users who are upgrading to version 7.xx from a previous one.

For customers who are using more than one **System Units** set (mm / Inch): you must make sure that the following values are correct:

6.11.1 Values that are per Working Unit

These parameters are defined for each **Working Unit**, and in order for ***cncKad*** to function properly, they must be set separately for each one of them in the **Settings => Workspace Settings => Working Defaults** tab:

- Decimal Digits
- Bulge
- Tolerance
- Angle Tolerance



You only **must** set these values **once** for Working Unit; thereafter the values are saved by ***cncKad***.

6.11.2 Values translated when switching between Working Units

Each time you switch from one Working Unit to another, some values are being "translated" – multiplied or divided by the **mm**-to-**inch** ratio. Therefore every time you make the switch, you should go over these values to make sure they have been translated correctly:

- In the **Settings => Workspace Settings** dialog:
  - Material tab – Default Material => Sheet Thickness.
  - Display tab – Text Size and Arrow Size.
- In the **Settings => Machine Settings** dialog:
  - **Cutting Parameters** tab (for Laser machine users):
    - ~ Tool Offsets defaults – **Start** and **End**.
    - ~ Entry Length.
    - ~ Exit Length.
  - Auto tab – Auto Processing => Min Part Size.
  - **Machine** tab – Sheet and Part parameters.
- In the **Auto Punch** dialog (for Punch machine users):
  - In the Auto Punch Parameters tab –
    - ~ Tool spacing defaults – Minimum tool overlap and Scallop for round tools.
    - ~ Cutting tools restrictions – Minimum and Maximum **Widths**.
    - ~ Notches – Minimum and Maximum **Offsets** .
  - In the **Holes** tab:
    - ~ Crunch Holes up to – **Width** and **Length**.
    - ~ Wire Joint – **Width** and **Place Every**.
- Common Cut Parameters (for Laser machine users) – **Auto Cut => Auto Cut tab => Common Cut** section **=> Settings** button **=> Preparatory** and **Overlap** parameters.

6.12 cncKad V.8's new Material List

This section is meant for laser cutting machines. It describes how to start working with the New Material List, being used in version 8.0 and higher ones:

This section is intended only for those users, who have upgraded to version 8 (and higher) from previous versions.

6.12.1 Introduction

Beginning with version 8.0, ***cncKad*** will be using a new format for its Material List – where in previous versions there were 15, 20, or even more materials, now there will usually be 5-10.

This is because in the past, the list contained the same materials, but with different Gas, Head or Lens definitions. e.g.:

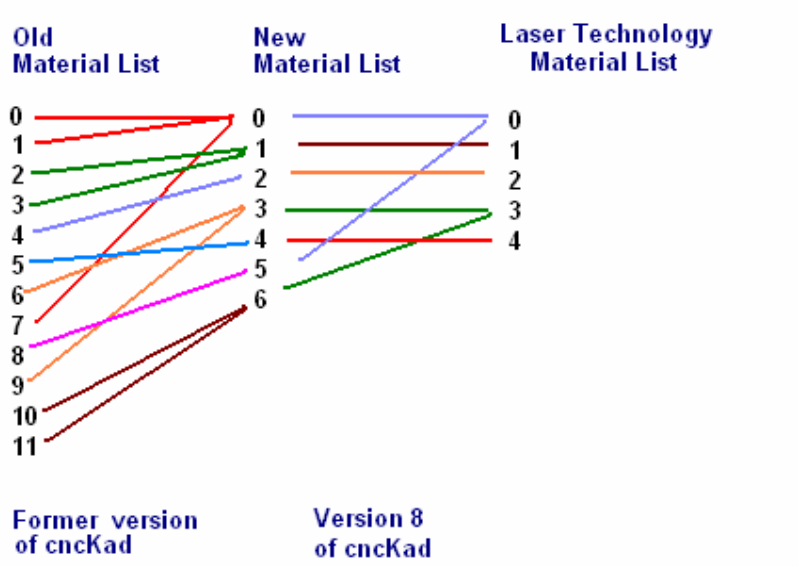
6	X 5 CrNi 18 9 SUS O2
7	X 5 CrNi 18 9 C-SUS N2

In this example the same material is defined twice – once for cutting with Oxygen, and once with Nitrogen. In the new list, for each Material the **Gas**, **Head** and **Lens** definitions can be selected separately; this means a shorter list, but much more flexibility.

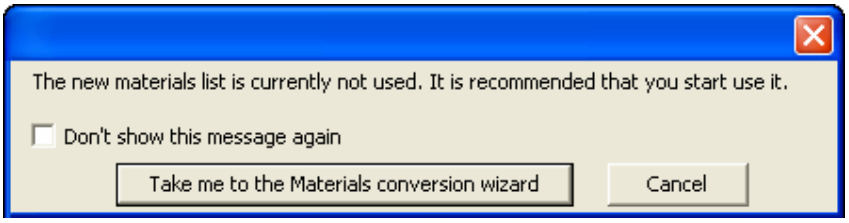
In order to use the new list you will need to create a **Conversion Table**, that will allow ***cncKad*** to select the correct material from the **new** list, when you will open a part created in a **previous** version (and therefore having an old material defined for it).

The conversion process, described on the successive pages, is done when you open ***cncKad*** for the first time, after installing the new version. But can still change the conversion settings at any time. For more information on how to do this go to **Changing the Conversion Definitions** section of this chapter.

The scheme of converting materials looks this way:



When you open ***cncKad***, after the installation completes, a window will open giving you the option to convert your old list to the new format:



We highly recommend going through with the conversion process. If you are dissatisfied with the results, you can always return to using the old list at any time.

If you are not interested in converting your old list, check the **Don't show this message again** option and click the **Cancel** button.

To start the conversion, click the **Take me to the materials conversion wizard** button. The following message will appear:

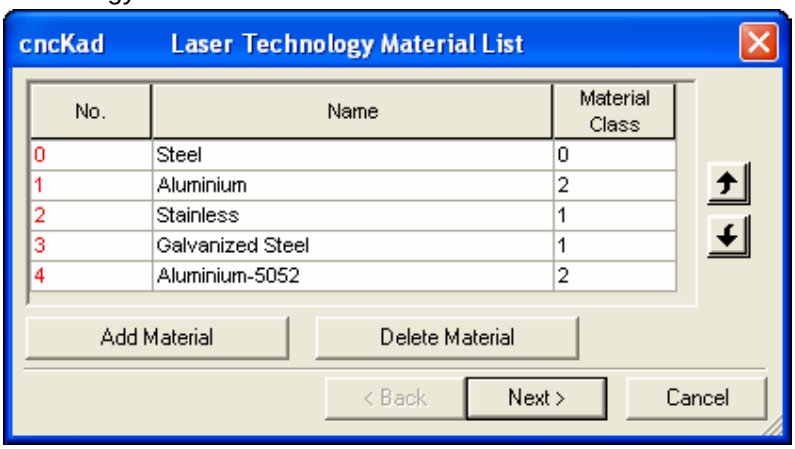
Click **OK** and the Conversion Wizard will open.

The conversion consists of two stages:

- Choosing the **Materials List** for the new list.
- Converting materials using the **Materials Conversion Table**.

6.12.2 Laser Technology Material List

When the Conversion Wizard opens for the first time, the first dialog you will see is the Laser Technology Material List. Each material shown in this list has its own laser technology table.

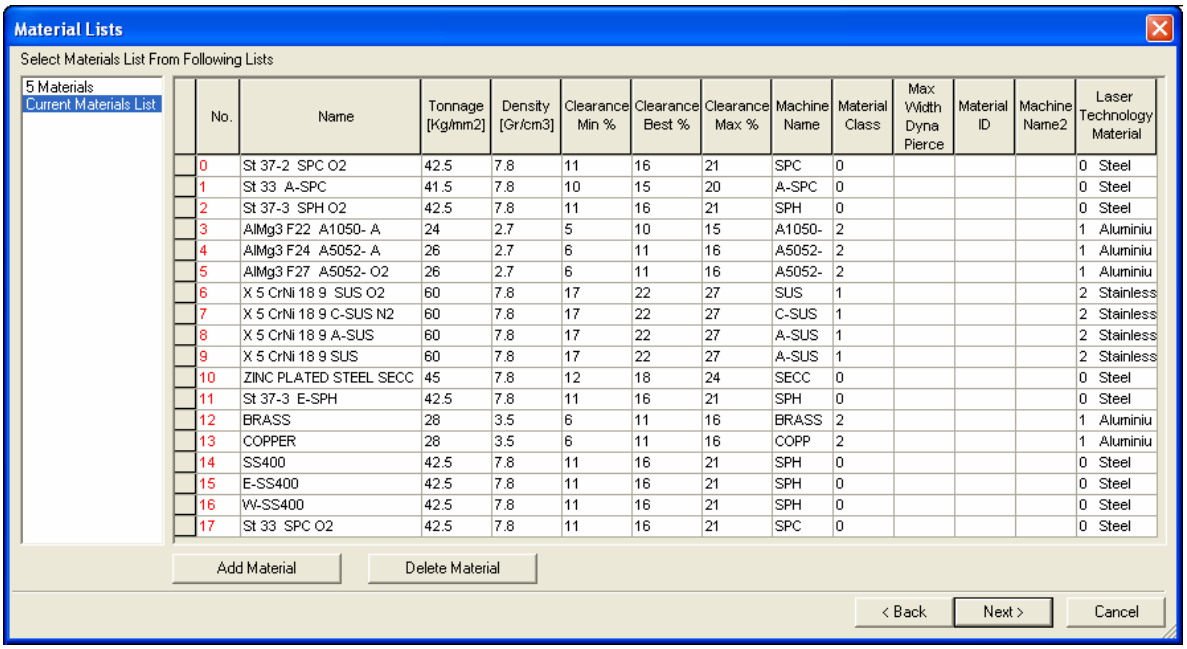


If this list is not extensive enough for your use, simply **add** other materials.

In order to move to the next step, click **Next** button.

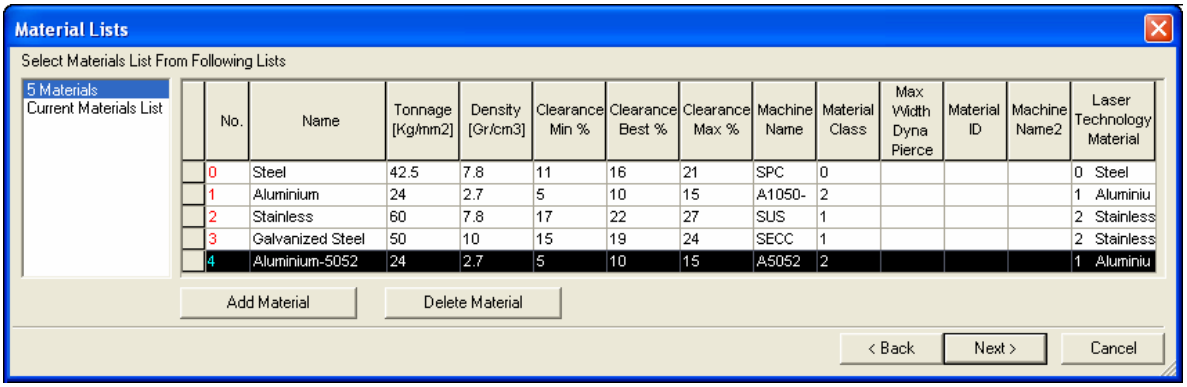
6.12.3 Material Lists

Now you will see two Material Lists: a long one, containing the materials used in your previous version of ***cncKad***:



	No.	Name	Tonnage [Kg/mm2]	Density [Gr/cm3]	Clearance Min %	Clearance Best %	Clearance Max %	Machine Name	Material Class	Max Width Dyna Pierce	Material ID	Machine Name2	Laser Technology Material
	0	St 37-2 SPC O2	42.5	7.8	11	16	21	SPC	0				0 Steel
	1	St 33 A-SPC	41.5	7.8	10	15	20	A-SPC	0				0 Steel
	2	St 37-3 SPH O2	42.5	7.8	11	16	21	SPH	0				0 Steel
	3	AlMg3 F22 A1050- A	24	2.7	5	10	15	A1050-	2				1 Aluminiu
	4	AlMg3 F24 A5052- A	26	2.7	6	11	16	A5052-	2				1 Aluminiu
	5	AlMg3 F27 A5052- O2	26	2.7	6	11	16	A5052-	2				1 Aluminiu
	6	X 5 CrNi 18 9 SUS O2	60	7.8	17	22	27	SUS	1				2 Stainless
	7	X 5 CrNi 18 9 C-SUS N2	60	7.8	17	22	27	C-SUS	1				2 Stainless
	8	X 5 CrNi 18 9 A-SUS	60	7.8	17	22	27	A-SUS	1				2 Stainless
	9	X 5 CrNi 18 9 SUS	60	7.8	17	22	27	A-SUS	1				2 Stainless
	10	ZINC PLATED STEEL SECC	45	7.8	12	18	24	SECC	0				0 Steel
	11	St 37-3 E-SPH	42.5	7.8	11	16	21	SPH	0				0 Steel
	12	BRASS	28	3.5	6	11	16	BRASS	2				1 Aluminiu
	13	COPPER	28	3.5	6	11	16	COPP	2				1 Aluminiu
	14	SS400	42.5	7.8	11	16	21	SPH	0				0 Steel
	15	E-SS400	42.5	7.8	11	16	21	SPH	0				0 Steel
	16	W-SS400	42.5	7.8	11	16	21	SPH	0				0 Steel
	17	St 33 SPC O2	42.5	7.8	11	16	21	SPC	0				0 Steel

and a new, short one, consisting of just five default materials:



	No.	Name	Tonnage [Kg/mm2]	Density [Gr/cm3]	Clearance Min %	Clearance Best %	Clearance Max %	Machine Name	Material Class	Max Width Dyna Pierce	Material ID	Machine Name2	Laser Technology Material
	0	Steel	42.5	7.8	11	16	21	SPC	0				0 Steel
	1	Aluminium	24	2.7	5	10	15	A1050-	2				1 Aluminiu
	2	Stainless	60	7.8	17	22	27	SUS	1				2 Stainless
	3	Galvanized Steel	50	10	15	19	24	SECC	1				2 Stainless
	4	Aluminium-5052	24	2.7	5	10	15	A5052	2				1 Aluminiu

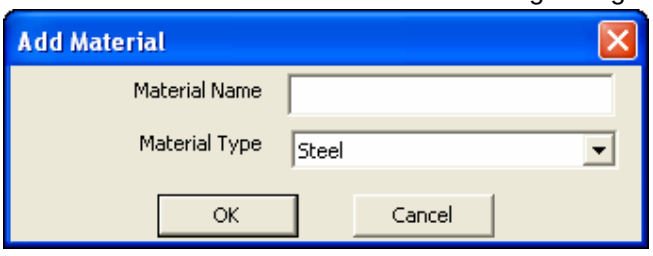
The user chooses which of the lists he wants to be used as a New Material List. In order to confirm your choice and move on to the **Materials Conversion** stage, click **Next** button.

If this list is not extensive enough for your use, you can simply **add** other materials.

6.12.3.1 Adding Materials to Lists

The instructions given in this section apply for adding materials both to Laser Technology Material List as well as to Material Lists.

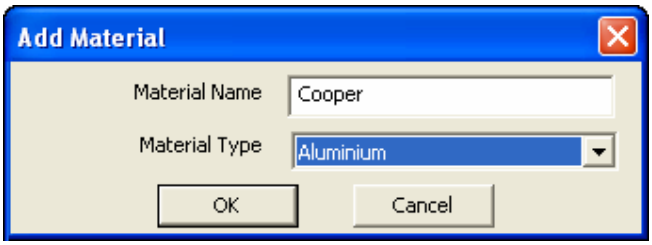
Click the **Add Material** button. The following dialog will appear:



Enter the new material's name and select the base Material Type for it:

This Material Type will be used to set the initial attributes for the new material – Tonnage, Density, Clearance, etc., so you should select the Type, that resembles the new material the most.

For example, when adding Copper to the list, you will probably select Aluminum as the base Type:



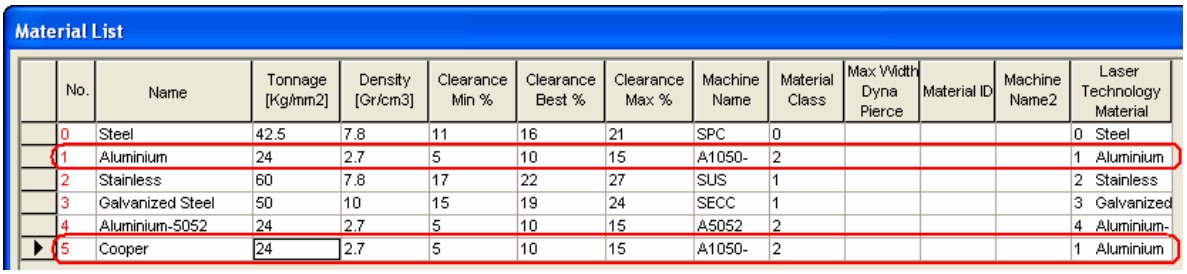
As a result, *Copper* will be added to the list with *Aluminum*'s attributes:

- In Laser Technology Material List:

cncKad2006 Laser Technology Material List

No.	Name	Material Class
0	Steel	0
1	Aluminium	2
2	Stainless	1
3	Galvanized Steel	1
4	Aluminium-5052	2
5	Cooper	2

- In Material List:



Material List

	No.	Name	Tonnage [Kg/mm2]	Density [Gr/cm3]	Clearance Min %	Clearance Best %	Clearance Max %	Machine Name	Material Class	Max Width Dyna Pierce	Material ID	Machine Name2	Laser Technology Material
	0	Steel	42.5	7.8	11	16	21	SPC	0				0 Steel
	1	Aluminium	24	2.7	5	10	15	A1050-	2				1 Aluminium
	2	Stainless	60	7.8	17	22	27	SUS	1				2 Stainless
	3	Galvanized Steel	50	10	15	19	24	SECC	1				3 Galvanized
	4	Aluminium-5052	24	2.7	5	10	15	A5052	2				4 Aluminium-
▶	5	Cooper	24	2.7	5	10	15	A1050-	2				1 Aluminium

You can now change the attributes, so that they reflect Copper's true nature – click inside a table cell and enter the correct value.

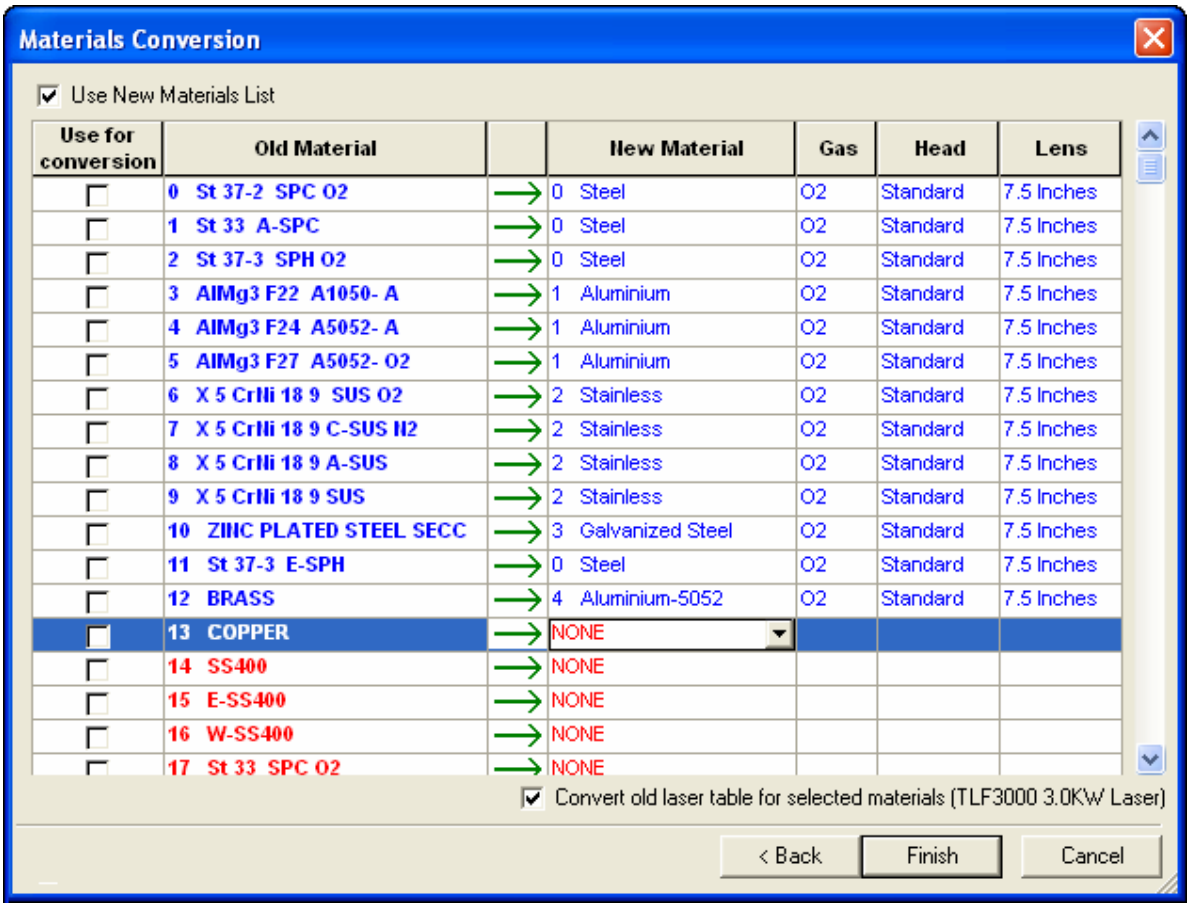
Click **Next** to continue.

6.12.4 Materials Conversion Table

This table tells ***cncKad*** how to treat materials from the old list. This means that when you open an **old** file, it stands to reason it will have an **old material** defined for it; this table tells ***cncKad*** which **new** Material, **Gas**, **Head** and **Lens** to select for it instead.

6.12.4.1 Selecting a New Material for the Old one

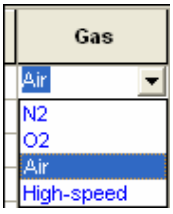
To start material conversion, mark the **Use New Materials List** checkbox at the top of the dialog. Now, for each Old Material, select a New Material from the dropdown list:



If you anyways want your new material list to look like the old one, don't check the **Use New Materials List** option. The materials will be automatically copied from one column to another.

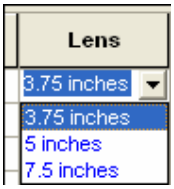
After selecting a New Material, you can also change the definitions for Gas, Head and Lens, like shown below:

Selecting a Gas:



Selecting a Head:

Selecting a Lens:



After defining a new materials for all the old ones (you cannot continue unless **all** materials are defined), click the **Finish** button to create the **Conversion Table**.

6.12.4.2 Use for conversion

When you enable this option, ***cncKad*** reads the Laser Technology Material List and looks for which material there are no assignments. When it finds such, then *automatically* it marks it with "**V**". The selected material will replace the standard cutting definitions with the ones from the Old Material List.

For example, the first time you select Stainless as the New Material this option will be checked automatically, if the new material does not have cutting data in the laser technology cutting table, but it will not be checked for subsequent selections of Stainless (or if there is already an entry for it within the table).
This is an useful feature for experienced users who have calibrated their old laser cutting tables and know the specific parameters needed for them.

This option will be used only if the **Convert old laser table for selected materials** checkbox is selected.

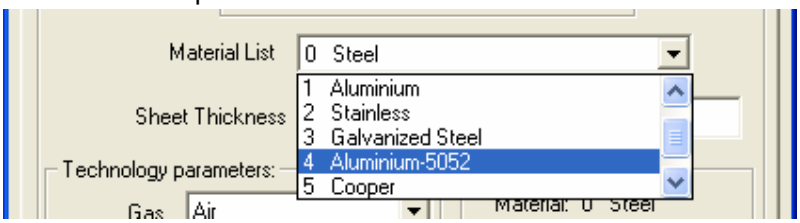
6.12.4.3 Convert old laser table for selected materials

When this checkbox is selected next to some materials, ***cncKad*** knows that it should take all the data from the old (pre-existing) version with all the cutting parameters and copy it to the new table. When this checkbox is unselected the ***cncKad*** will use the defaults supplied with the software system.

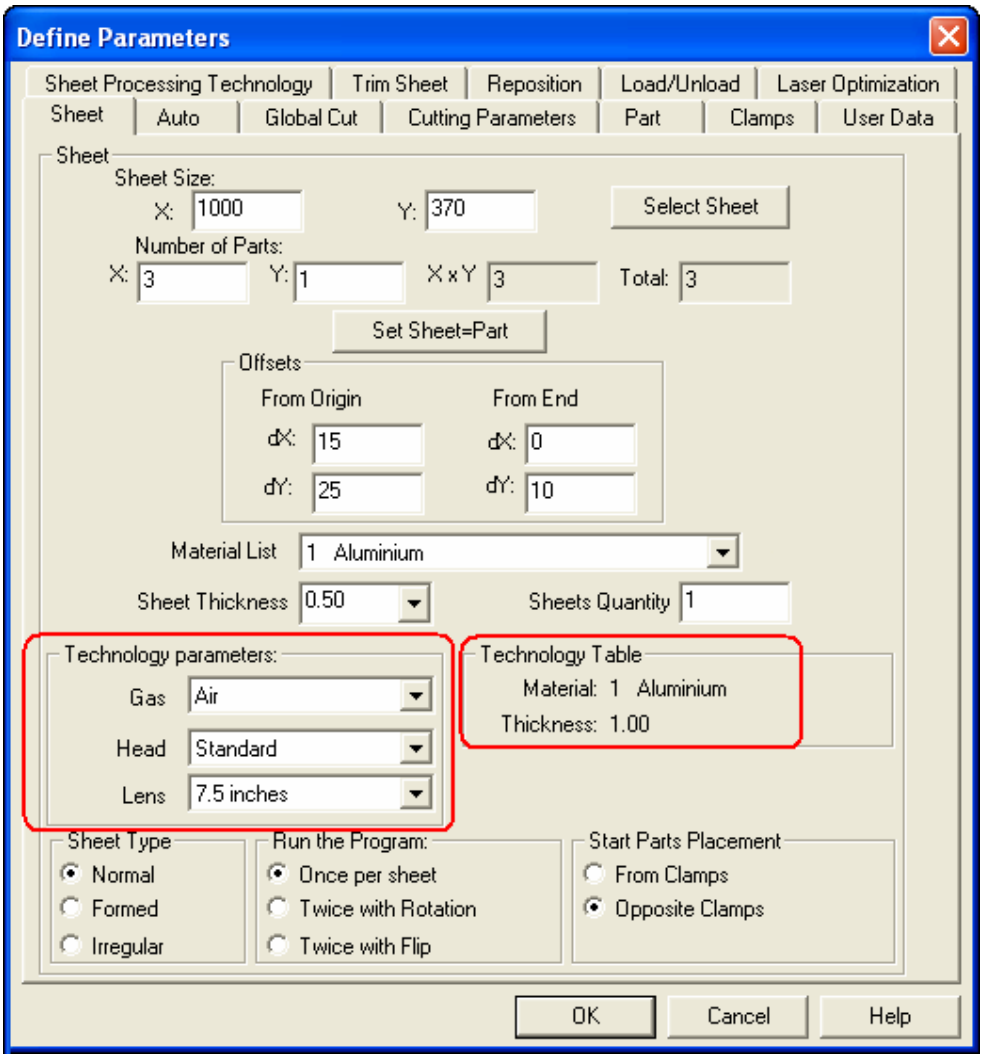
This will be done only for those materials, where the **Use for conversion** option is selected.

6.12.4.4 Using the new list

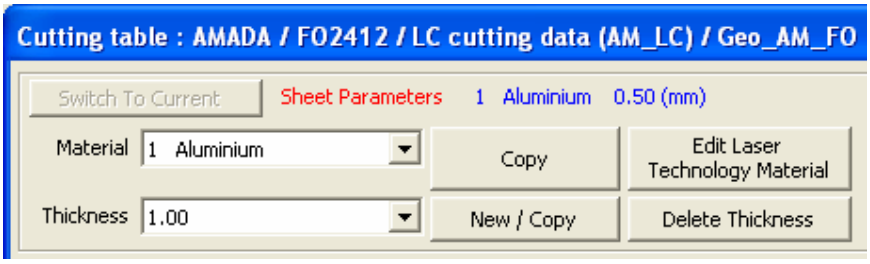
After the conversion is complete, ***cncKad*** will open and you from now on you will be able to use the new material list. Go to Sheet tab of Set Sheet and Clamps dialog and see the dropdown Material List:



See the picture shown below.
In the **Technology Parameters** section you will see other material definitions like Gas, Head and Lens and from dropdown lists will be able to choose various options for them, if it will be possible.
In the **Technology Table** section you will see the laser technology parameters for your settings or material type and sheet thickness. Notice that in our example data displayed here differs from the settings entered in above Sheet Thickness field. This is because in the Laser Technology Table there is no such thickness for this material type. So it offers you the closest value existing in the table for this material.



Now, when you'll go to the **Laser Technology Table** you will see that thickness presented here is the same as in Technology Table section of Sheet tab, presented above. However in the bottom part of the dialog you will see the actual **Sheet Parameters** displayed in blue color.



6.12.5 Changing the Conversion Definitions

You can change the conversion settings at any time, by going to the **Settings** => **Workspace Settings** => **Material** tab and clicking the **Convert Materials** button. This will re-open the Conversion Wizard, and you will be able to re-define the conversions.

You can also use this to return to using the old List, but there really is no point of doing this – if you take the time to do the Conversion correctly, you will get your old data integrated into the new list.

This action is **not** recommended.

To return to using the old materials List:

1. Open the Wizard as mentioned above
2. Click the **Next** button
3. Unselect the **Use New Materials List** checkbox and click the **Finish** button.

In this **Material** tab you can also **Edit Materials** as well as **Edit Laser Technology Materials**, by clicking on the appropriate buttons. When you do, you will be directed correspondingly to and to Material List or to Laser Technology Material List. You can add new materials here, as well as edit their parameters, if applicable.

6.13 The cncKad Service Pack

The **cncKad Service Pack** is an application that updates ***cncKad*** installations (seats) with the latest versions of program files. These updated versions may include new features, fixes for bugs, etc.

6.13.1 Installing the cncKad Service Pack

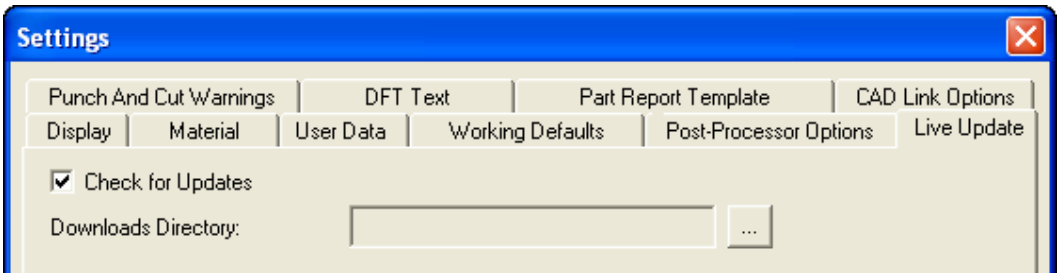
To use the Service Pack for updating ***cncKad*** you need to do two things:

- Place the **Service Pack** application on a local/network drive.
- Set ***cncKad*** to use this application for updating.

For the first of these, we recommend saving the application to a dedicated directory on the PC or on the server, as shown in the examples presented below.

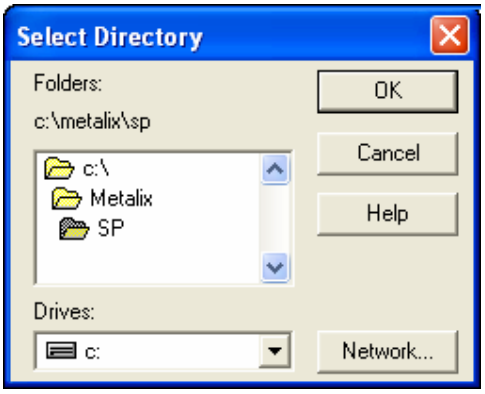
After saving the Service Pack to this directory, do the following:

Open ***cncKad***, go to **Settings => Workspace Settings** and select the **Live Update** tab:

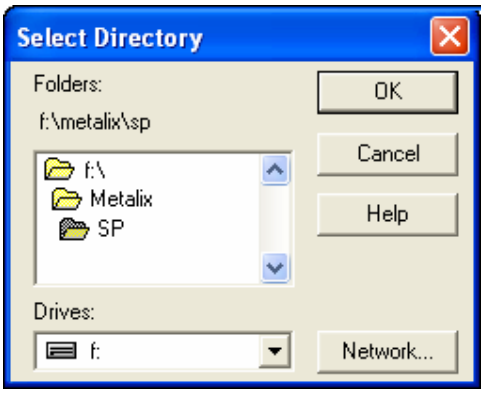


Mark the **Check for Updates** checkbox, and then click the ... button for browsing to the **Service Pack**'s directory:

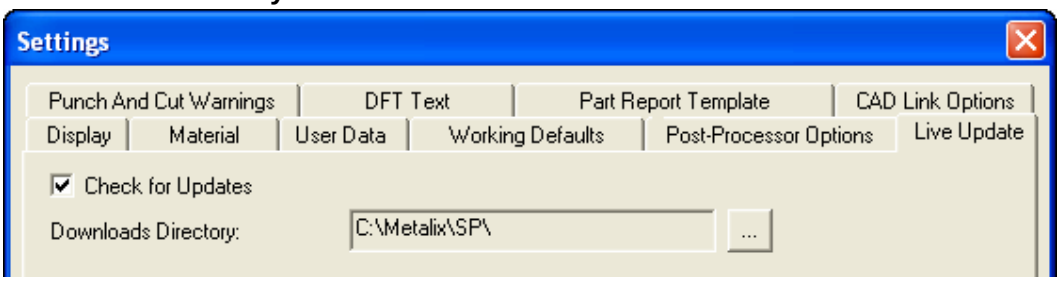
If this directory is on local PC, searching it might look this way:



If the directory is on a network drive, then the browsing may give the following result (depending on the drive letter):



Regardless the **Service Pack'**s directory location, its path will be shown in the **Downloads' Directory** field:



After making sure that the path is correct, do not forget to click the **OK** button, to confirm activating the Update feature.

Thereafter, each time ***cncKad*** is opened, it will check for updates and install them when appropriate.

6.13.2 Working with networked seats

When there are several ***cncKad*** seats working together on a network, we recommend saving a **single** copy of the Service Pack on a network drive, and then following the procedure described above for each of the seats.

There is no need for each seat to save an individual copy of the Service Pack – they all can use the same one! This will ensure that all the seats are updated to the same version, and will also protect against cross-version compatibility issues.

> If there is a Firewall protecting the network, it must be set to allow the ***cncKad*** seats to access the Service Pack's directory.

6.14 The cncKad Update Manager

This section gives explanations about ***cncKad***'s Update Manager.

6.14.1 What is the Update Manager

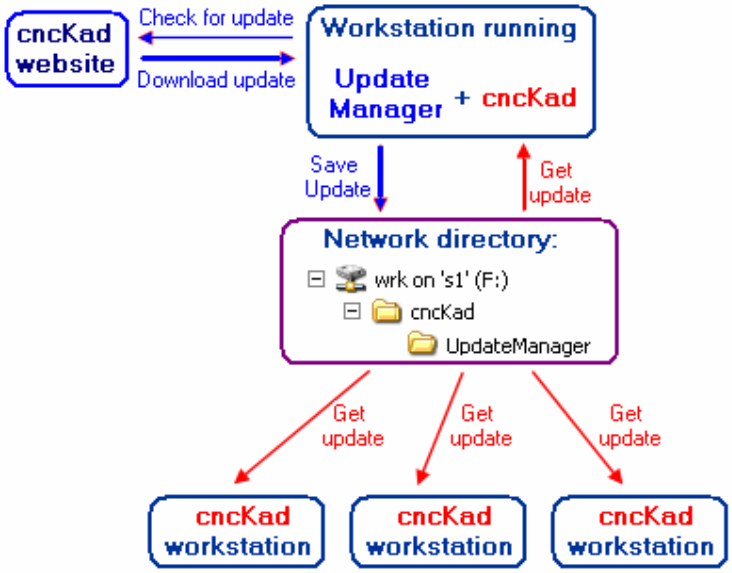
The **cncKad Update Manager** is an application that allows your workstation to keep up with the most recent updates released for ***cncKad***. This is done by downloading updates from the ***cncKad*** website and installing them to the local workstation(s).

Likewise it was for **Service Pack** installation, here there are also two basic configurations for working with the Update Manager:

- **Local** – this configuration is straightforward – a single workstation runs the Update Manager and applies the updates.

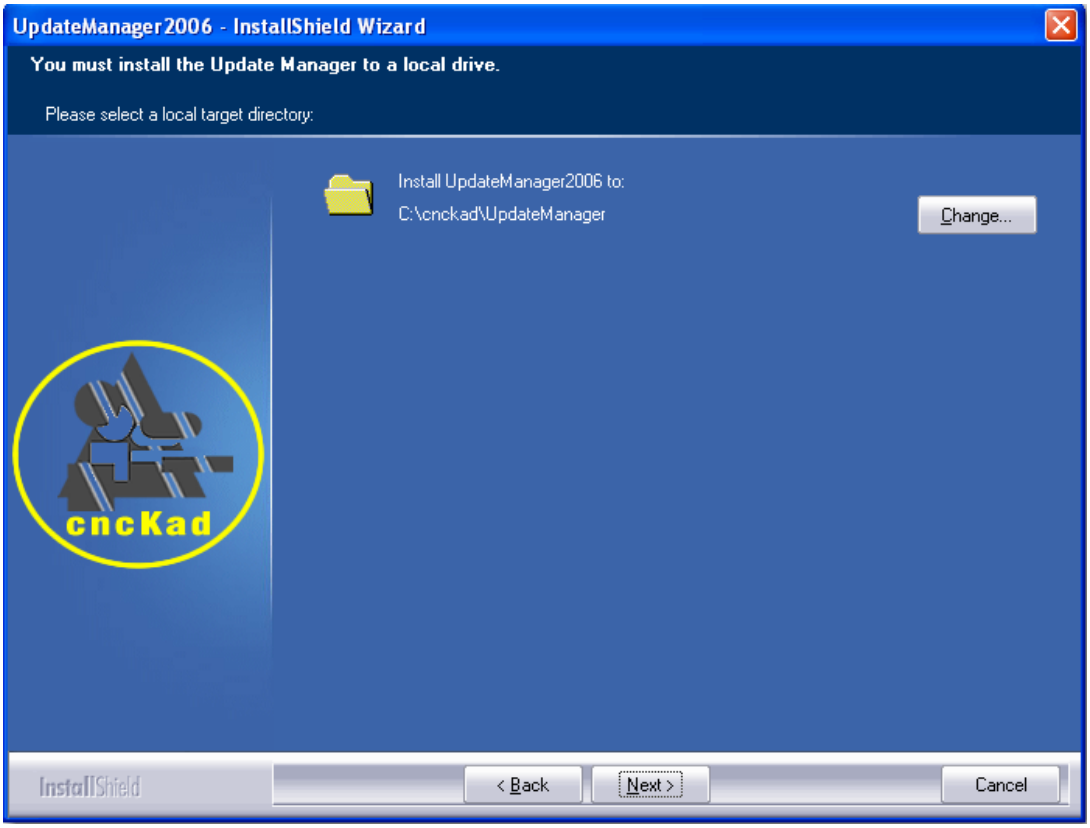
- **Network** – this configuration is more complex – one workstation runs the Update Manager, but the updates are downloaded from the website to a network directory, and ***cncKad*** gets the update from this directory for each workstation connected on a network.

The diagram below explains this process:



6.14.2 Installing the Update Manager

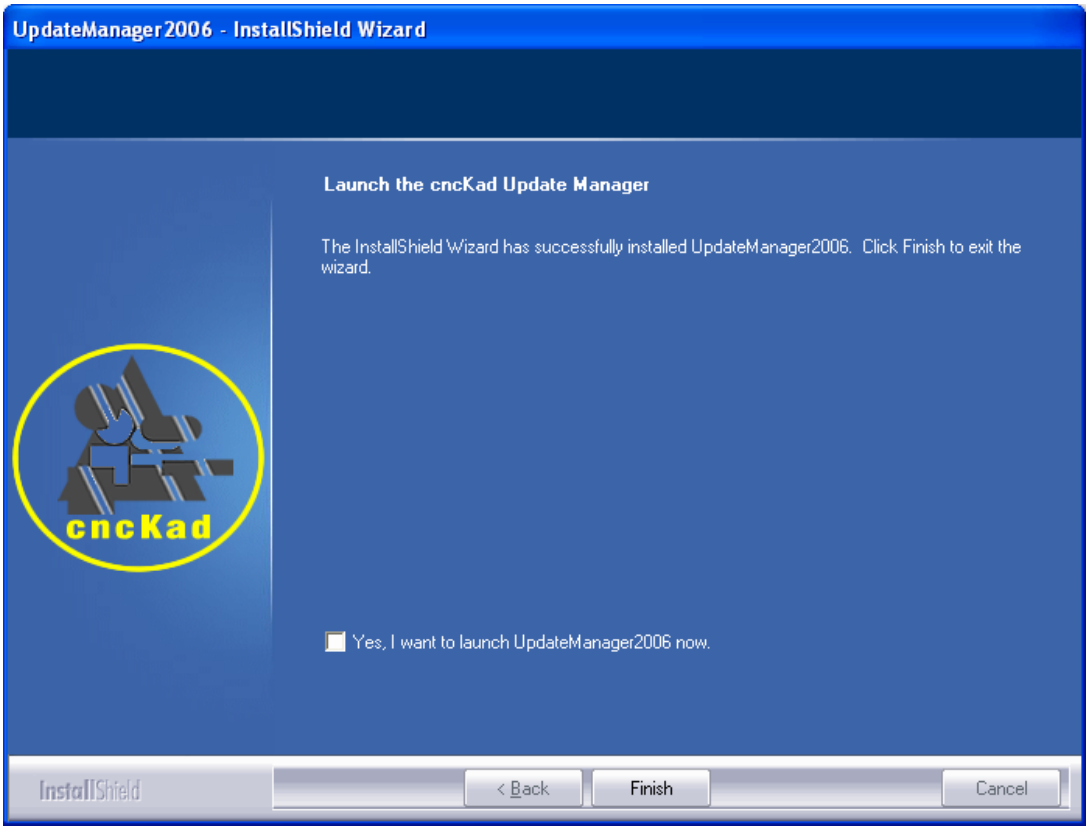
After starting the **cncKad Update Manager** installation you will get the following window:



Here you will see the default installation directory. If you want to install Update Manager elsewhere, click the **Change** button and choose a different location.

> You must install **Update Manager** on a computer that has an internet connection.

Click the **Next** button, and you will be presented with the following window:



Select the **Yes, I want to launch Update Manager2006 now** checkbox, click the **Finish** button and the Update Manager will open.

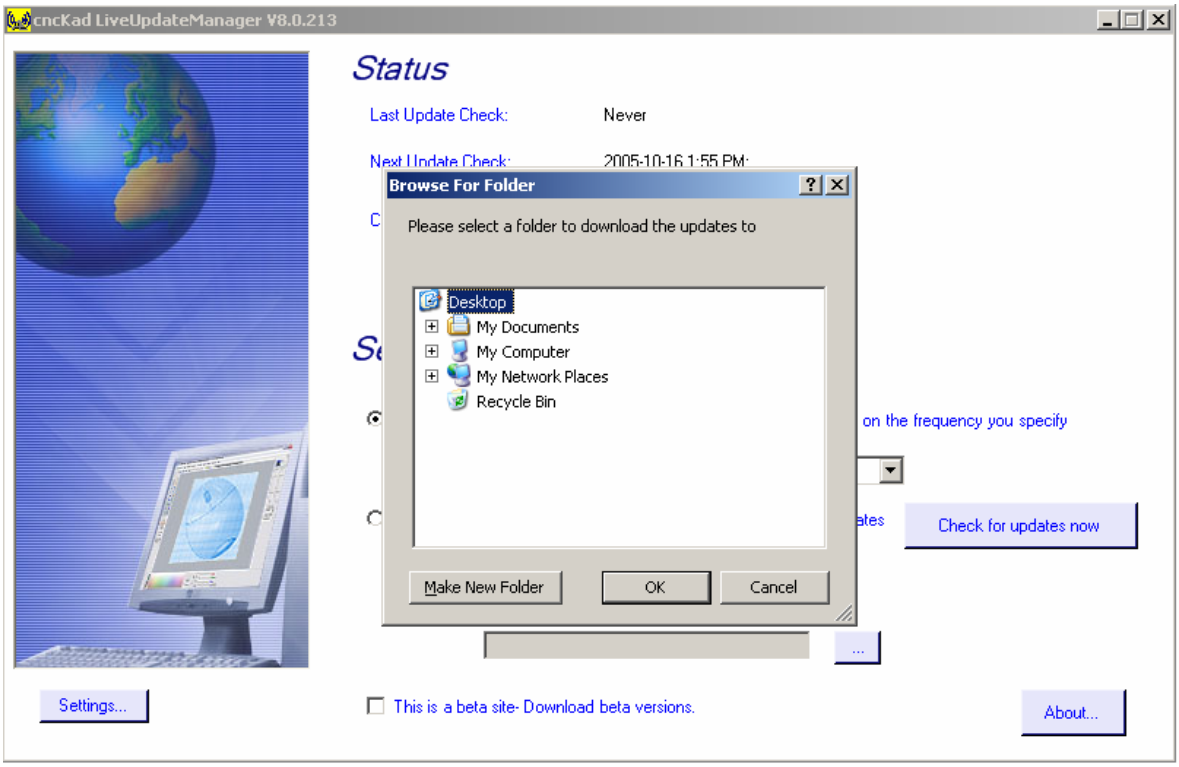
6.14.3 Configuring the Update Manager

These are the three stages to configuring your Update Manager:

- Selecting a download path
- Entering a Customer ID
- Setting the update check interval

6.14.3.1 Selecting a download path

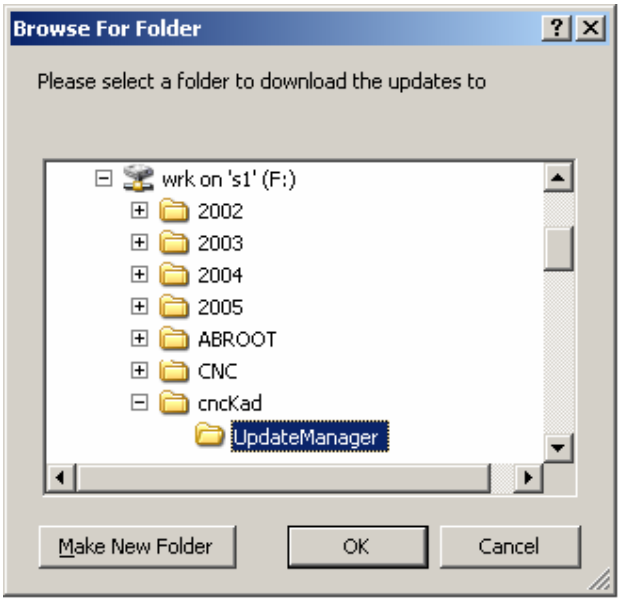
When the Update Manager opens for the first time, you will be asked to select a location for storing the updates after downloading them:



At this point you must decide if this is a **local** or **network** configuration (as explained **above**):

- **Local** – select a local directory, e.g. C:\cncKad\SP
- **Network** – select a network directory.

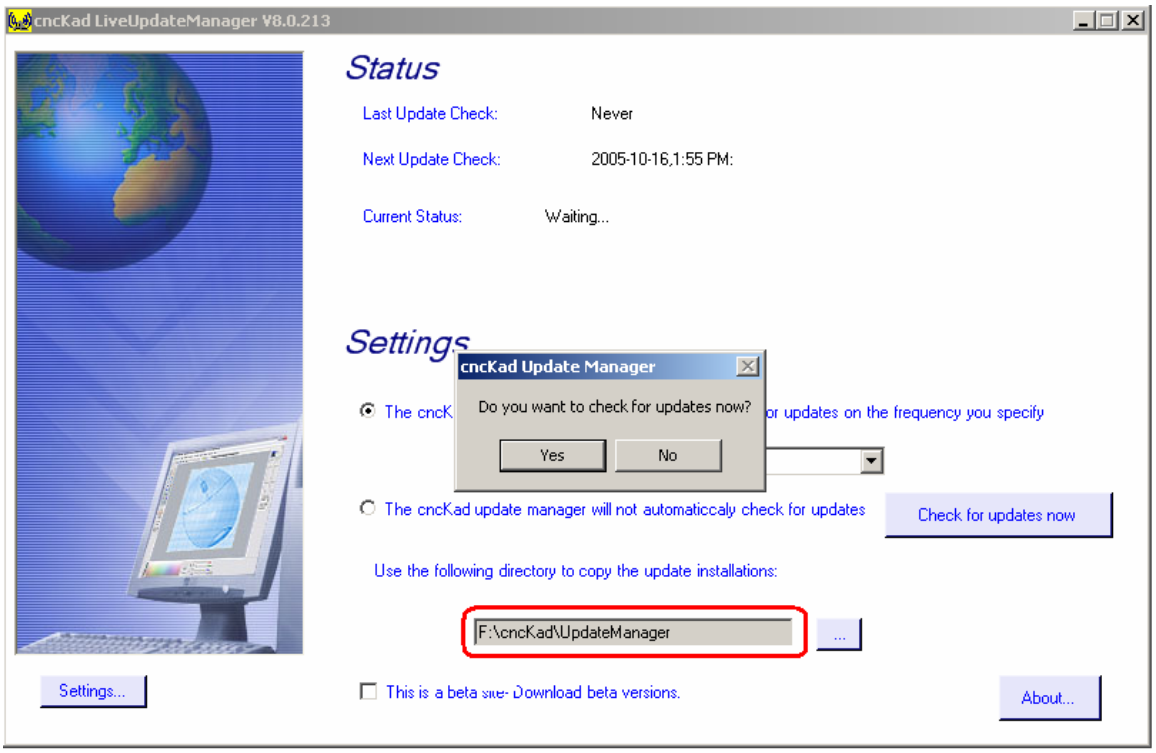
For this example we will choose a **network** configuration, and select the folder *F:\cncKad\Update Manager*:



It is compulsory that this directory can be accessed from all the computers connected to the network; otherwise they will **NOT** receive the updates!

Click the **OK** button and the directory path will be shown on the display, as marked on the picture below. You can change this path at any time from this dialog.

After confirming the path, a new window will open, asking if we want to check for updates now:

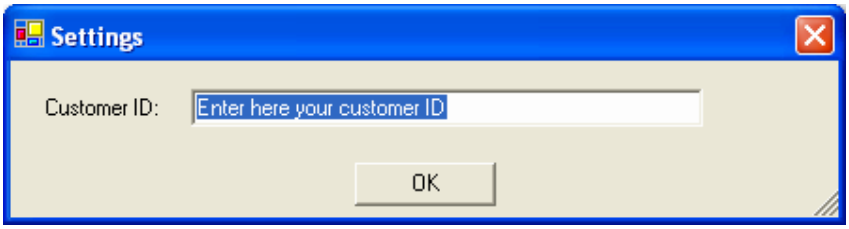


At this point choose **No**, because in order to download updates you must have valid **Customer ID** for your Update Manager.

6.14.3.2 Entering a Customer ID

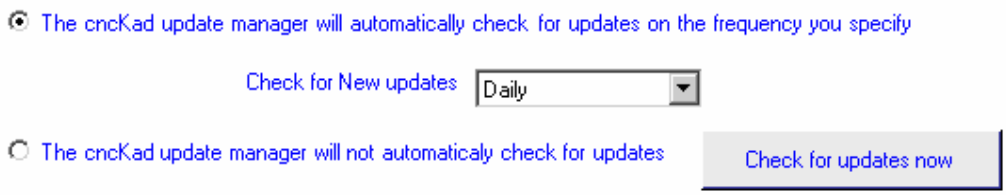
If you try getting updates without first entering a Customer ID, you will receive the following message:

In order to enter your Customer ID, click the **Settings…** button, and when the dialog shown below will appear, type-in the ID number and click **OK**.



6.14.3.3 Setting the update check interval

In the main window you can set the interval for checking if there are updates:



There are several options for this interval:

- **Automatic checks** – you can specify the frequency of checking for updates from one of two options – **weekly** or **daily:**

When choosing this option, in the top of the dialog you will see when the **Last Update Check** was done and when the next one is scheduled for.

- **Manual checks** – in this case a check for updates will be performed only when you click the **Check for updates now** button.

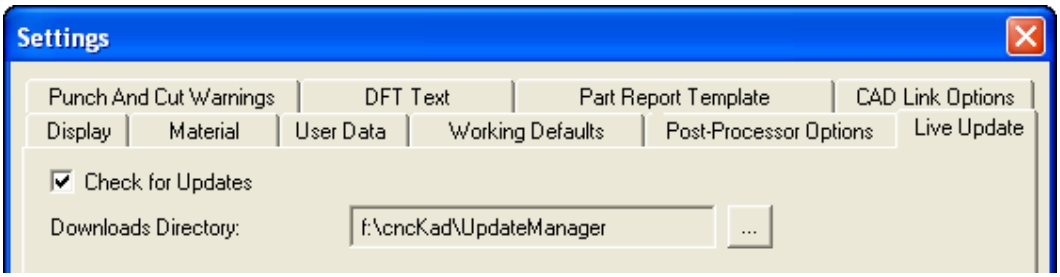
6.14.4 Configuring the cncKad workstation

The following configuration must be done for **every** workstation.

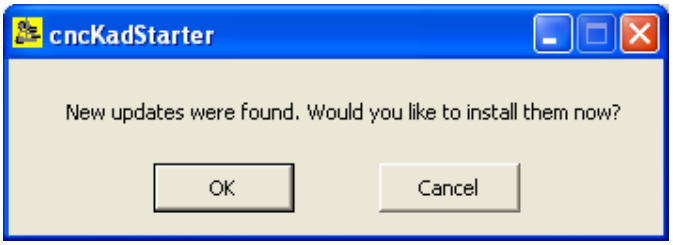
In ***cncKad***, go to the **Settings => Workspace Settings => Live Update** tab.

To receive updates, do the following:

- Select the **Check for Updates** option.
- Browse to the directory in which the program will be looking for updates; this will be the directory selected as the destination for downloading the updates (in our example this is: *F:\cncKad\Update Manager*):



The next time when you open ***cncKad***, the Update Manager will check for the updates and if it finds any, the following message will appear:



When you click **OK** button, the updates will be automatically installed on your computer, and ***cncKad*** will open.

6.15 Configuring your system to send Problem Report

In order to send **Problem Reports**, your Windows system must have email capability and be configured to send emails.

The email capability issue depends on the specific conditions at your workstation (e.g. whether the PC is networked, if it is behind a firewall, etc.). If you are not sure, consult your System Administrator.

To configure your Windows system in order to send Problem Reports files, do the following:

6.15.1 Open the Control Panel

The procedure to open the **Control Panel** varies, depending on the Windows version you are using.

- For Win98\ME:

  Go to **Start => Settings =>Control Panel**

- For Win2000:

  Go to **Start => Settings =>Control Panel**

- For WinXP:

  Go to **Start => Control Panel** (or **Start => Settings =>Control Panel** depending on which type of *View* you have chosen for the *Start menu*)

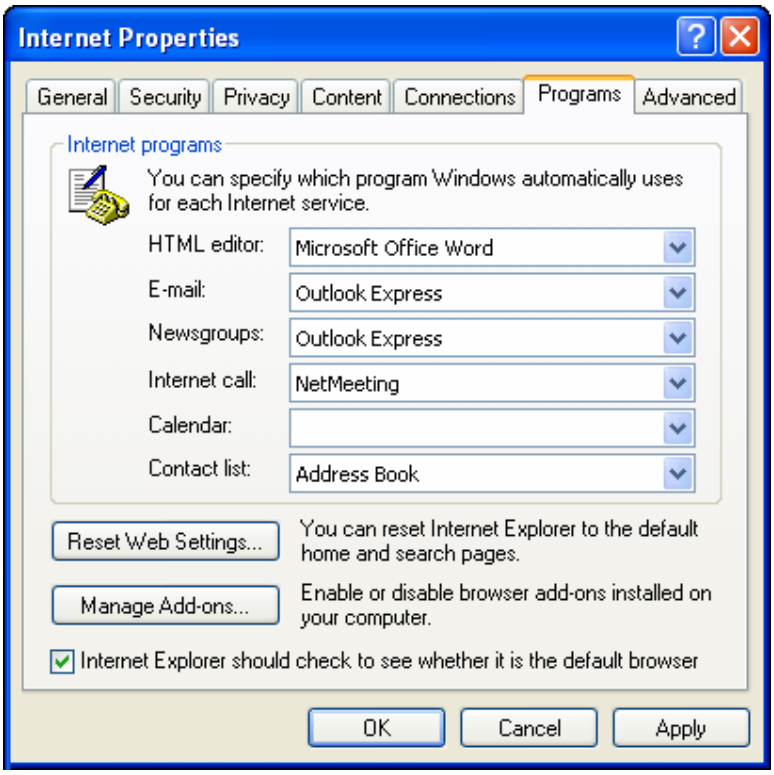
6.15.2 Open the Internet Options dialog

From the **Control Panel**, select the **Internet Options** icon:

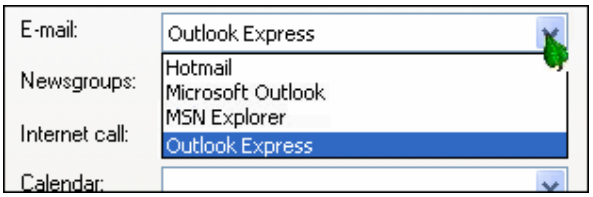
You can also open the **Internet Explorer** and select this option from the **Tools** menu:

6.15.3 Select a default application

In the **Internet Options** dialog, go to the **Programs** tab:



Use the **E-mail** dropdown menu to select the program you would like to handle your outgoing email massages:



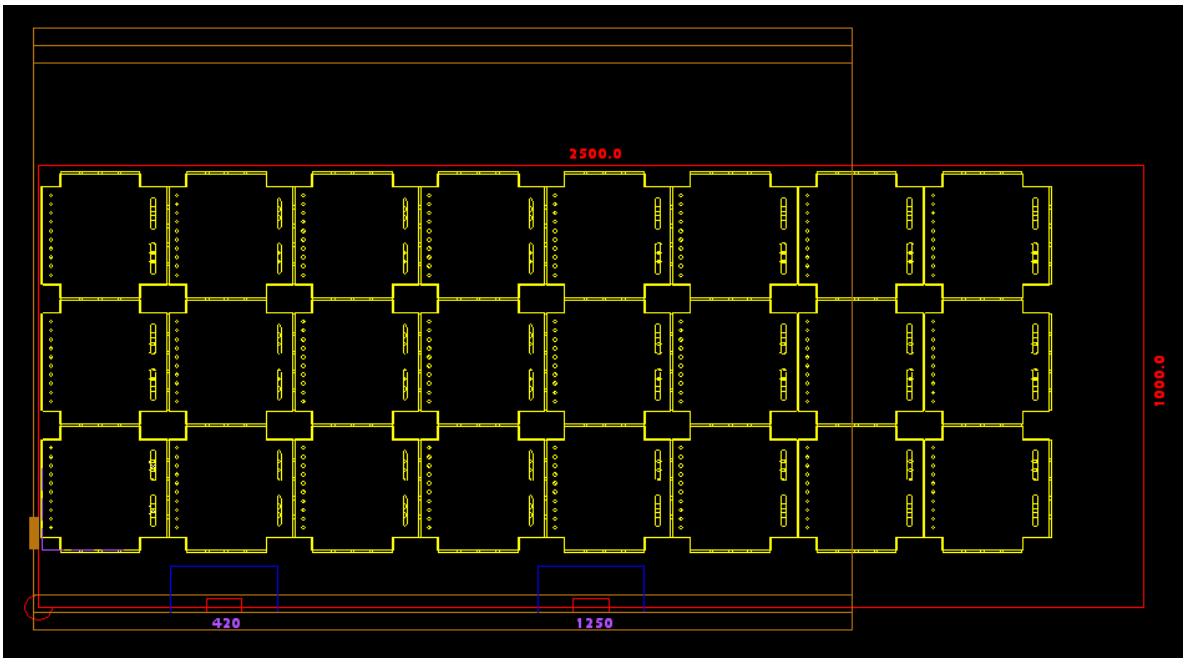
Don't forget to click the **OK** button to confirm your selection.

More information on **Sending Problem Report** you will find in the **Help Menu** section of **Introduction** chapter of ***Drafting and Nesting Manual***.

6.16 Multiple Parts with Reposition

- **The Problem:**

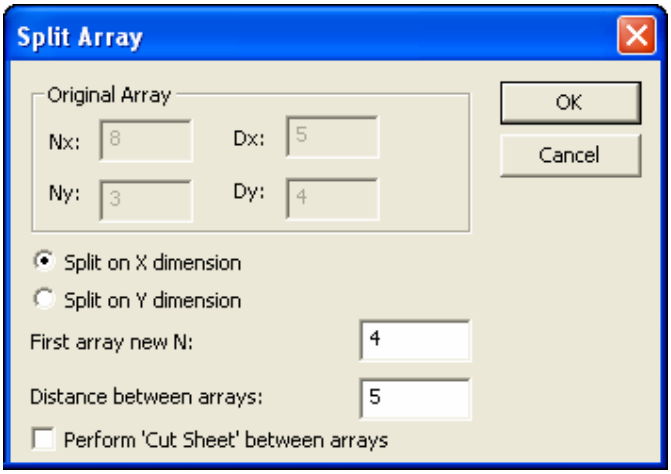
When working with Multiple Parts on a sheet bigger then the working range, as illustrated on the example below, you encounter the following problem:



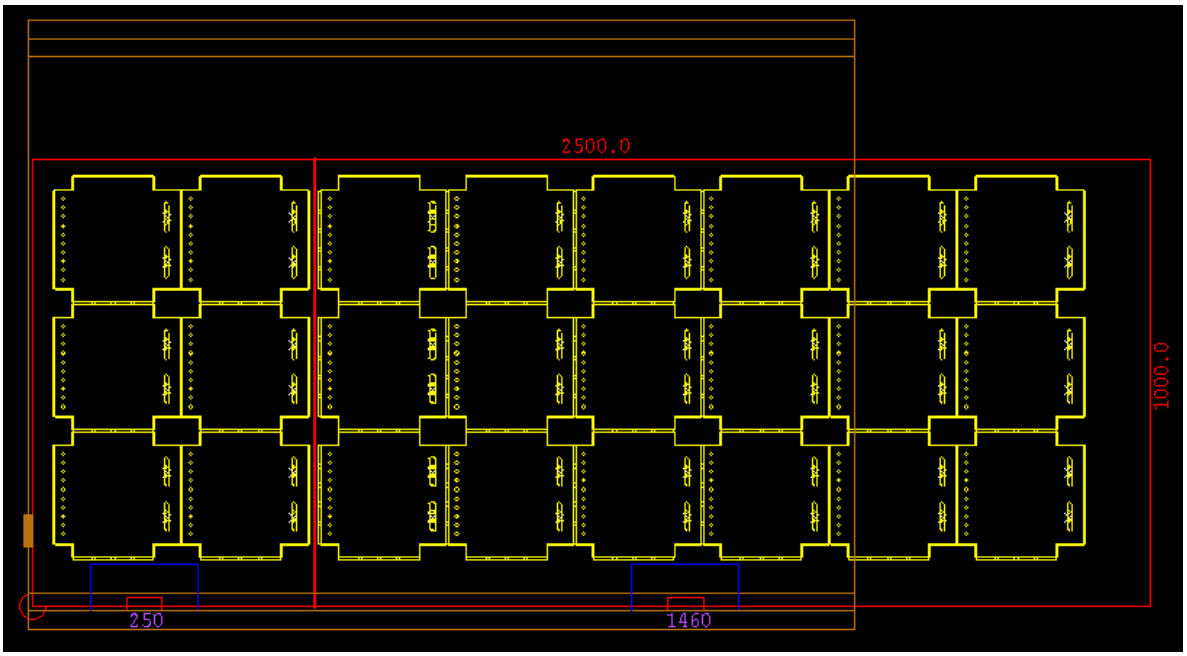
There are 8 columns of parts on the sheet. Suppose you will process the first 3 columns, make a reposition, and then process the remaining 5 columns.
While processing the last 5 columns, the first 3 columns are on **MicroJoints** – so they can easily get disconnected.

- **The Solution:**

Lay out the sheet the normal way, including Common Cuts. Alter the DFT file into nest, using the command **Convert to Nest** from File menu. As a result you will get a NST file laid out identically to your Multiparts on DFT.
Now from **Nest menu** choose the command **Split Array**. Click the cursor anywhere on the array. The parts in the nest will be highlighted with red color, and the following window will appear:



For our example change the number of columns to **2**, set distance between arrays to **30mm** and check the option **Perform 'Cut Sheet' between arrays**. The number of new columns in new array should be the number of parts overhanging the table range. Confirm your choice, clicking **OK** button. You will get the following result:



Go to **Used Tools** dialog and move the separating tool (the one whose sequence is **99**) to the first position in the table. We want it to be the first tool to work **AFTER** the Reposition. Confirm your settings with **OK** button. Next generate NC.

When the simulation module window will open, watch the punching sequence in the Graphical Simulator:

1. The first two columns are being punched.
2. Reposition is being made.

   Pay attention to how the clamps are holding the sheet after repositioning. The left one should be on the right side of the Cut Sheet.

3. The Sheet is cut off and the machine stopped for the operator to remove the first two columns.
4. The last six columns are being punched.

6.17 Punching a Perforated Sheet in cncKad

The problem with perforated sheets is that when we try punching them, we often find that the material distorts, gaining a convex shape. This is dangerous and can cause a collision with the turret.

The solution is to avoid punching contiguous lines of the perforation, as described **further** in this section.

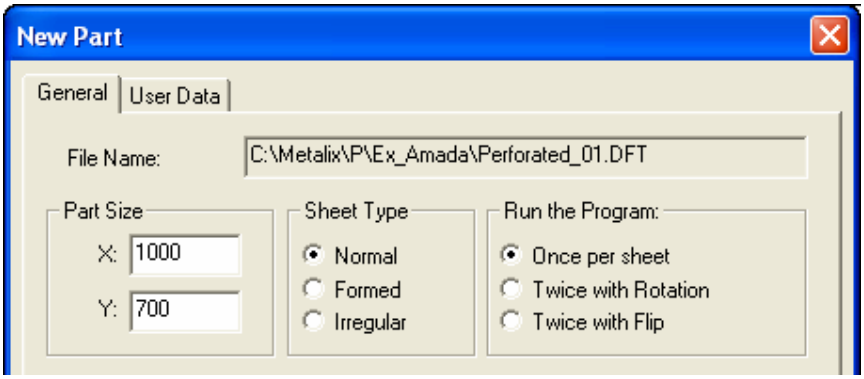
6.17.1 Creating a Perforated Sheet

Generally, you can get a perforated sheet in two ways:

- Importing a part into ***cncKad***, e.g. as a DXF file.
  If you have imported the part, go directly to the Punching a Perforated Sheet section.
- Creating one in ***cncKad***.
  If you want to create a part, continue in this section.

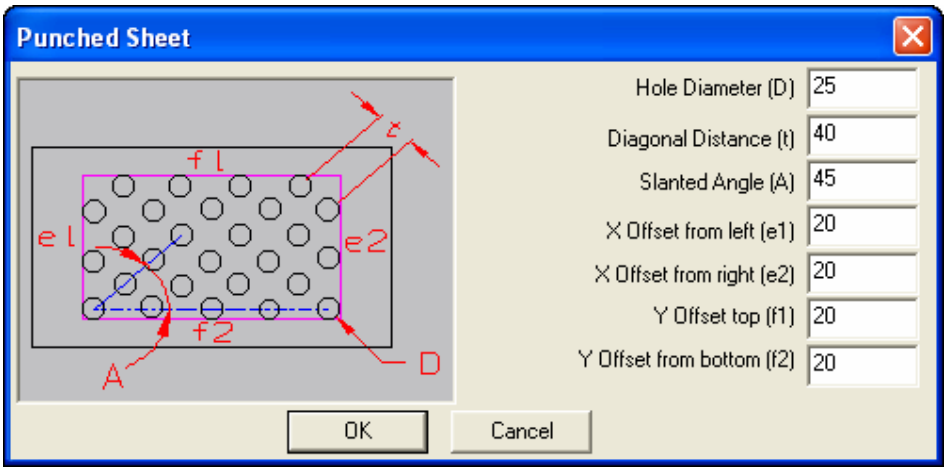
6.17.1.1 Sheet Dimensions

Start a regular part and give it the outer dimensions of your sheet. In our example the sheet sizes are: 1000 x 700:



6.17.1.2 Perforating the Sheet

Go to **Draw =>** Shapes **=>** **Punched Sheet** and define the perforation like shown below:

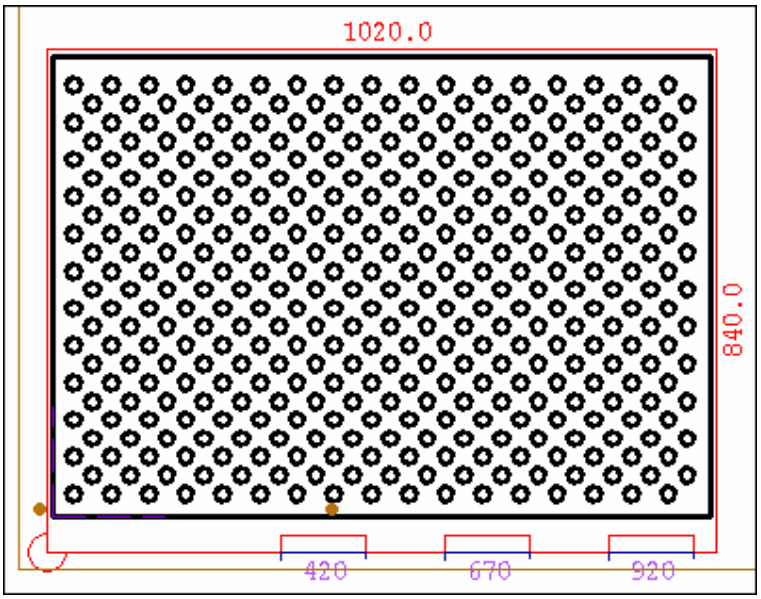


After clicking **OK** button, the sheet (or part's inner contour), will be filled with round holes. See the **example** presented below.

> If the **Hole Diameter** definition does not match any of the tools in your Library, you will be asked to add this tool.

6.17.1.3 Perforated Sheet example

The result is:



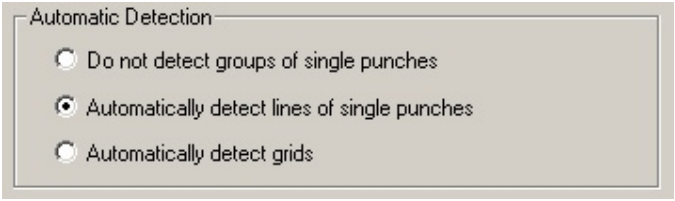
6.17.2 Punching a Perforated Sheet

If the perforated sheet was created with the **Punched Sheet** feature, go directly to the Optimizing the Path section, since the sheet is already punched.

If our sheet was imported, we will punch it now using Auto Punch feature, and then we will set a specific **Optimization Path** for this punching.

6.17.2.1 Using Auto Punch

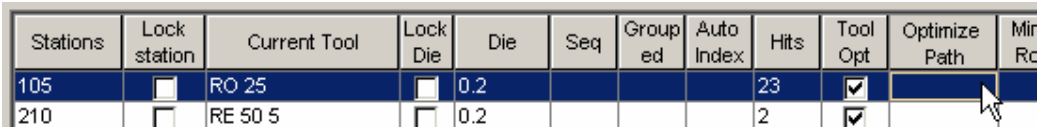
Open the Auto Punch dialog and go to the Holes tab. There in the **Automatic Detection** section, select the **Automatically detect lines of single punches** option:



Then click the **Auto** button to punch the part.

6.17.2.2 Optimizing the Path

Open the Used Tools dialog, find the tool you used for the perforation (in our example: **RO 25**) and click the **Optimize Path** cell for this line:



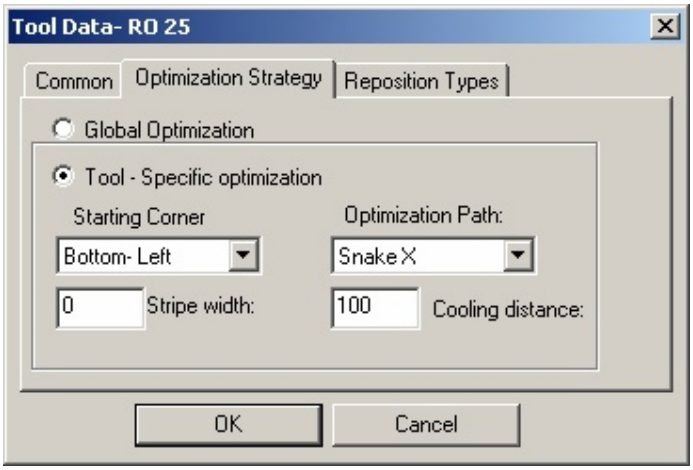
The Tool Data dialog will open on the **Optimization Strategy** tab.

Select the **Tool-specific optimization** option, and then proceed to define the optimization:

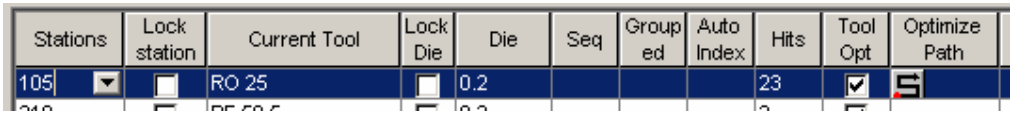
- **Starting Corner** – use the setting you usually use.
- **Optimization Path** – select **Snake X**.

- **Cooling distance** – this will define the distance between the snake's "curves", or in our case – how many rows to jump between passes.

For this example we will use a **Cooling Distance** of 100:



After clicking **OK** to confirm these settings, note that the **Optimize Path** column cell shows now the **Snake X** pictogram:



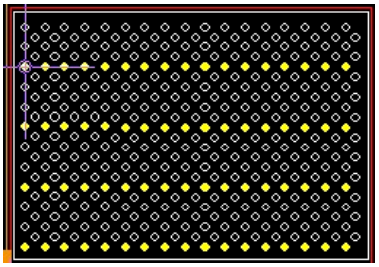
6.17.3 Testing the Optimization

Generate NC and run the Simulation.

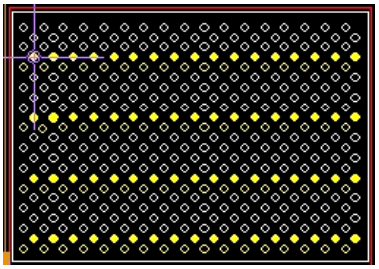
You may want to use the **Line = On** option to follow the optimization path.

We highlighted the specific lines being punched in each "curve" of the snake:

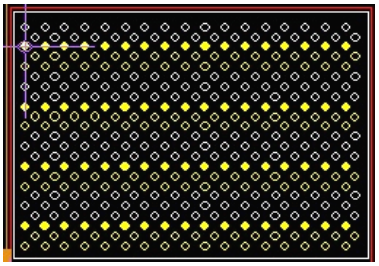
First pass – Lines 1, 7, 13 and 19:



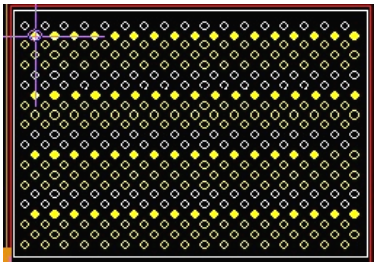
Second pass – Lines 2, 8, 14, and 20:



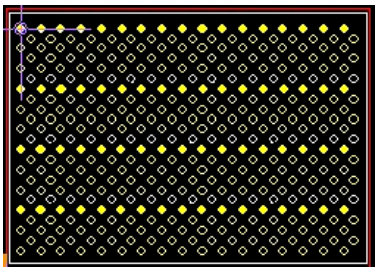
Third pass – Lines 3, 9, 15, and 21:



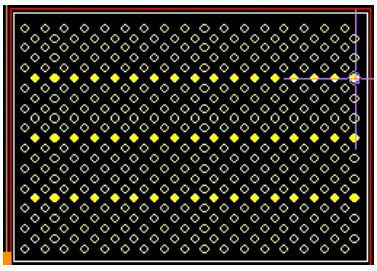
Fourth pass – Lines 4, 10, 16, and 22:



Fifth pass – Lines 5, 11, 17, and 23:



The last pass takes care of the rest of the lines:



6.18 Installing and Using the NetHASP

The NetHASP enables networked workstations to use a single **HASP** hard lock key in a floating license configuration.

We should distinguish between two types of PCs:

- The **License PC** – this is the PC where the **NetHASP** is installed.
- **Workstations** – these are the PCs running ***cncKad***.

6.18.1 Installing the NetHASP on the License PC

Plug-in the **NetHASP Key** to a PC, which will be your License PC.
It is recommended to connect the NetHASP Key on the system server but you can also connect it on a different PC that does not have ***cncKad*** on it; it is **compulsory** to make sure that this PC is always on and connected to the network!

> The License PC cannot run ***cncKad***.

6.18.1.1 Installing the NetHASP utilities

To install these utilities, insert the ***cncKad*** installation CD and click the **Net HASP** button.
The utilities that will be installed are:

- **License Manager** (lmsetup.exe):
  This application enables other PCs on the network to work with the NetHASP on the server.
  If this application is not running, you won't be able to run ***cncKad*** on any computer.
- **Aladdin's Monitor** (aksmon32.exe):
  You may run this application if you want to know, which PCs are currently using the NetHASP.

6.18.2 Using workstations with the NetHASP

If ***cncKad*** is already installed, simply disconnect the **Metalix** Keys (Time Keys or any other Key), and run ***cncKad***. Otherwise, install ***cncKad*** and start working with it.

6.18.2.1 Setting the NetHASP.ini file

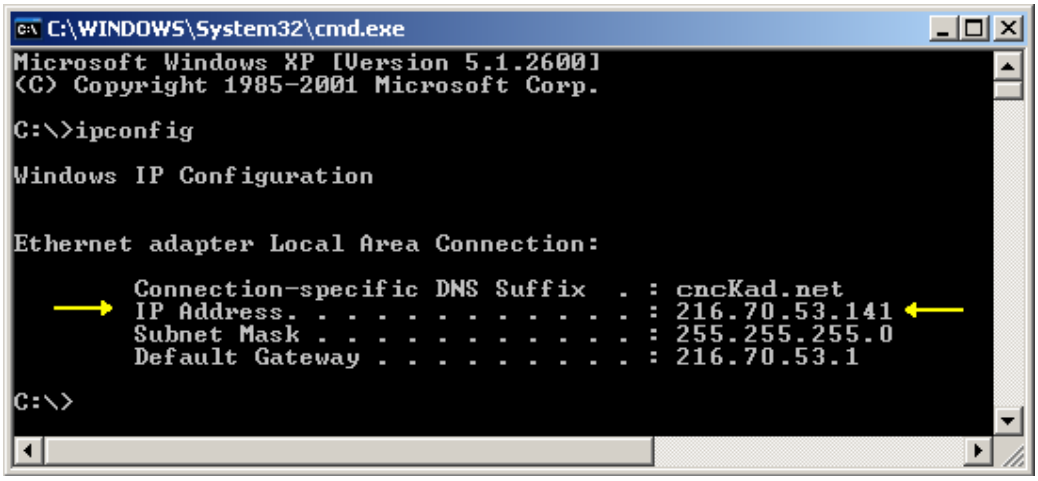
For faster operation over the network, it is recommended to set parameters in **nethasp.ini** on each workstation running ***cncKad***.

First, you need to know the IP address of the License PC:

To obtain the IP address:

1. Go to the **License PC**.
2. Click **Start => Run** and type `cmd`.
3. In the command line type `ipconfig` and press **Enter**.

You should get something like this:



The second line is the IP address (in the above case 216.70.53.141).

The **NetHASP.ini** file should be located in the ***cncKad*** folder (where the **gkadw.exe** file is located), for example:

C:\Metalix\cnckad2006.80

Double-click this file to open it, and modify it as shown below.
The important key is:

```
[NH_TCPIP]
;;NH_SERVER_ADDR = xx.xx.xx.xx;
```

- First, you must delete the ";;" signs from the beginning of the line.
- Next, instead of the `xx.xx.xx.xx` type the IP address of the **License PC**, e.g.:

  ```
  [NH_TCPIP]
  NH_SERVER_ADDR = 216.70.53.141;
  ```

- Save the file and you are done – close the file and open ***cncKad***.
- Remember to do this on **all** the PCs running ***cncKad***!

6.18.3 HASP FAQ

These are the common questions regarding using the NetHASP.

6.18.3.1 What is Error 129?

When you get this message, this means that:

- No plug is connected to the PC and License Manager was found.
- The PC, where the License Manager is running, is connected to a non NetHASP plug (like time or memo).

How to solve it:

- Replace the non NetHASP key with a NetHASP key on the computer where the License Manager is running.
- Use NetHASP.ini file.

6.18.3.2 "Cannot run cncKad with NetHASP connected directly."

If this message appears when running ***cncKad***, then there are two options:

- NetHASP is connected to the computer.
- HASP driver 5.12 is not installed.

How to solve it:

- Use local key (memo/ time) or use NetHASP on a different computer with License Manager.
- Install HASP driver 5.12.

6.18.4 How do I know on which computer the License Manager is running?

Run **Aladdin's Monitor** – it will show you exactly which stations are running a License Manager.

6.19 Creating a Trimmed Sheet

This section explains how to create a **Trimmed Sheet** in ***cncKad***.

6.19.1 Why Trim a Sheet

Trimming a sheet is very useful for those cases where the skeletonized sheet may cause a crash with the turret.

The basic principle is to process the sheet in a way that allows the removal of each individual part after it has been severed from the sheet. This removal can be done through a **Chute**, or manually, but the important issue is that the parts are not left on the table to rattle around.

For this example we will use the following parameters:

- Part Size: 300 x 200
- Sheet Size: 2000 x 1000
- Cutting Tool: **RE 65 5** (in an AutoIndex Station)
- Machine Working Range: At least 2000 in X

  All sizes are in mm.

6.19.2 Creating the initial part

Create the geometry and punch the holes, using the above mentioned **Cutting Tool**.

6.19.2.1 First Nibble

Add **Nibble Entity** with Tool Sequence of 1 to the right vertical side of the part.

Make sure you make this cut with an 3 mm offset on each side.

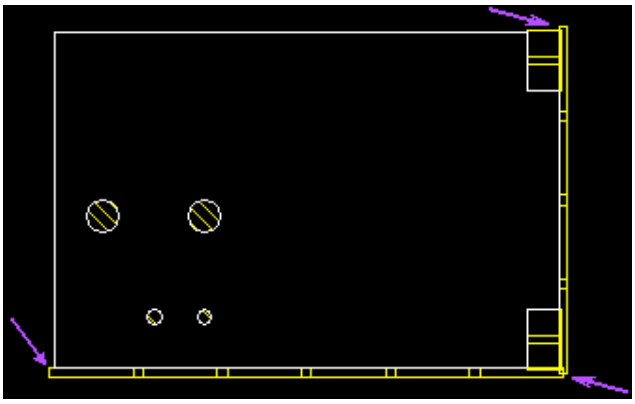
6.19.2.2 Second Nibble

Add **Nibble Entity** with a Tool Sequence of 2 to the bottom horizontal side of the part.

Make sure you make this cut with an 3 mm offset on each side.

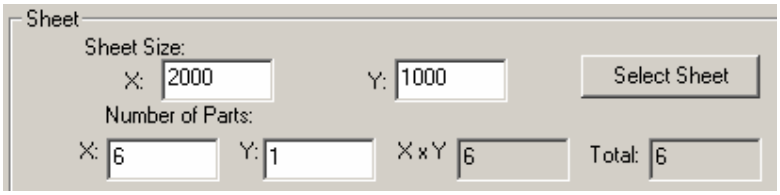
6.19.2.3 Current Part View

The results should be as seen on the below picture. Note the offsets of the Cutting Tool:



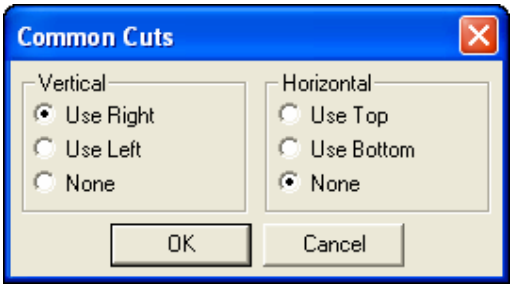
6.19.3 Defining the Sheet

Define the Sheet Size as 2000 x 1000, and set Number of Parts in Y as 1. As you can see, we now have a single line with 6 parts in it:

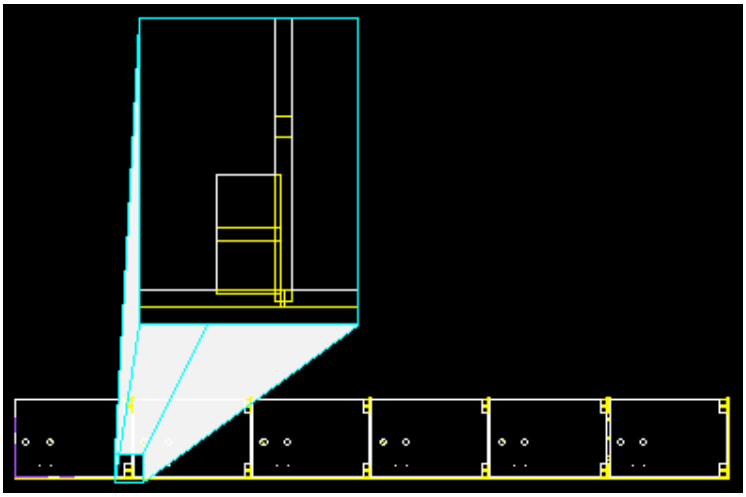


6.19.4 Defining the Common Cuts

Now, we will define **Common Cuts** for these parts – click the  button and set the parameters as shown below:

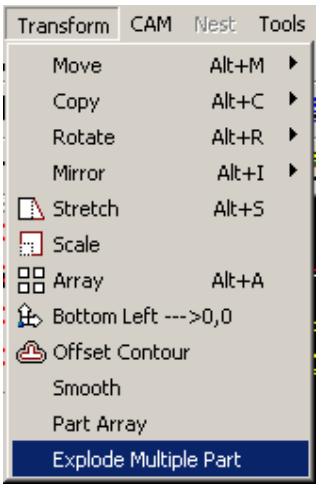


The result will be:

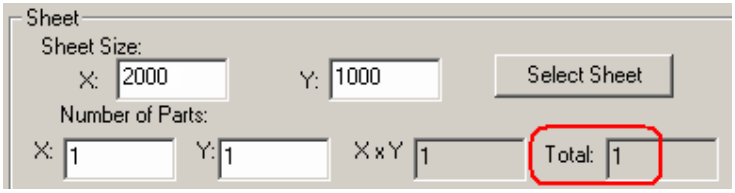


6.19.5 Using Explode Multiple Part

Now, select the **Explode Multiple Part** option from the **Transform** menu:



When you now view the **Sheet** tab in the **Set Sheet and Clamps** dialog (click the  button to open this dialog), you will see there is only one part currently on the sheet:

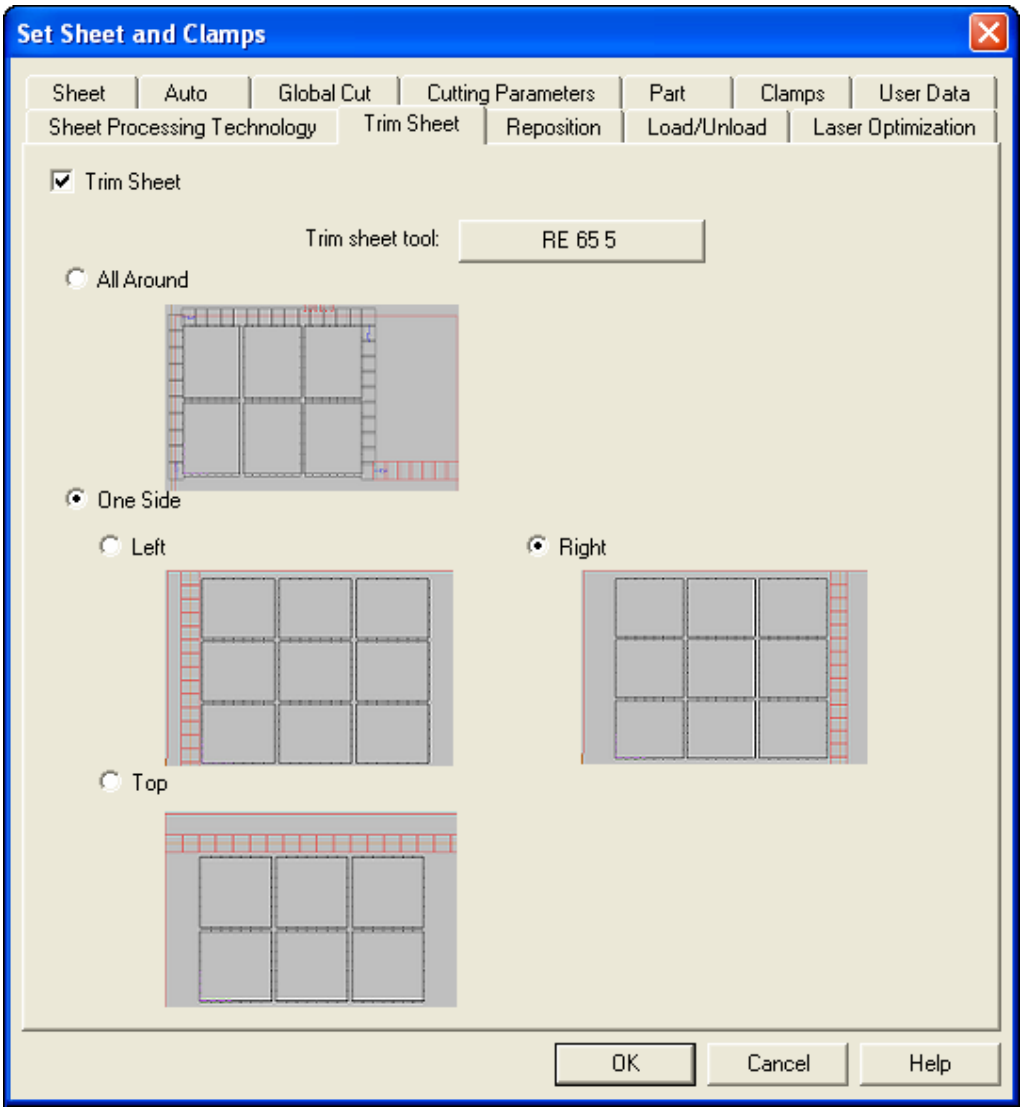


6.19.6 Deleting extraneous punches

Delete the punching made with **RE 65 5** tool, on the vertical right side of the part number 6 (the most right one) and **Redraw** the screen, using the  button.

6.19.7 Setting the Trimming

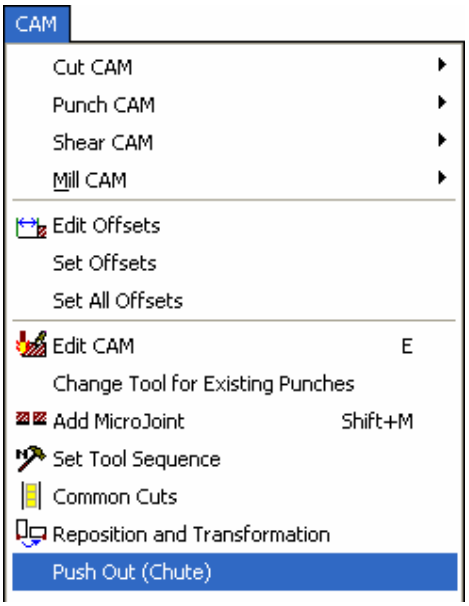
In the Set Sheet and Clamps go to the **Trim Sheet tab**, check **Trim Sheet** option, set the trimming tool to **RE 65 5**, choose the way you want the sheet to be trimmed and click **OK** button:



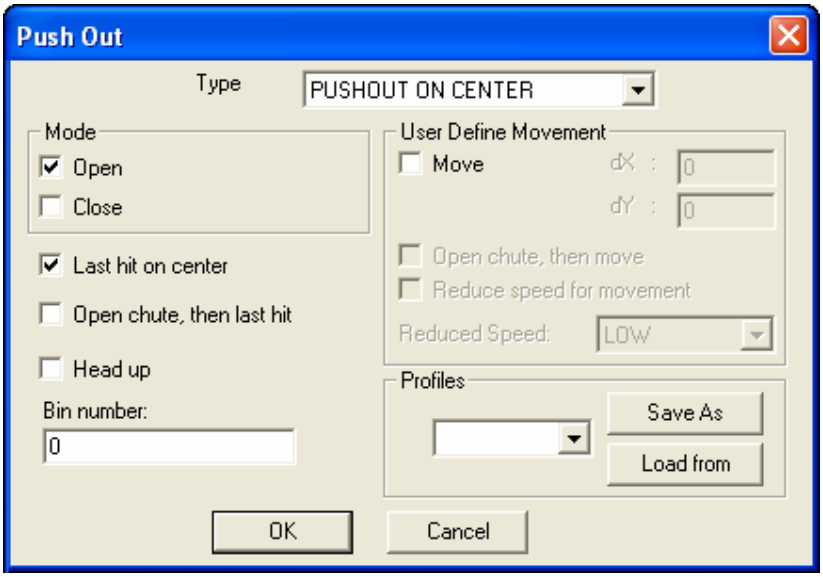
6.19.8 Adding Chute commands

We will now set **Chute (Push Out)** commands; in this example we will open the chute at part 1, and close it at part 6:

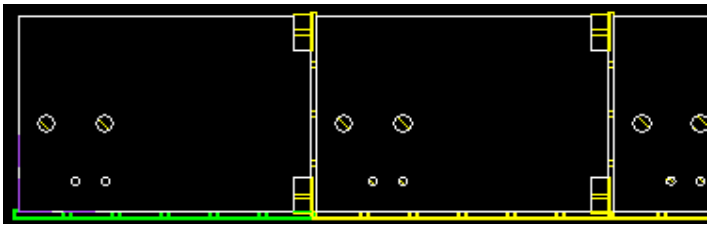
Select **CAM => Push Out (Chute)**:



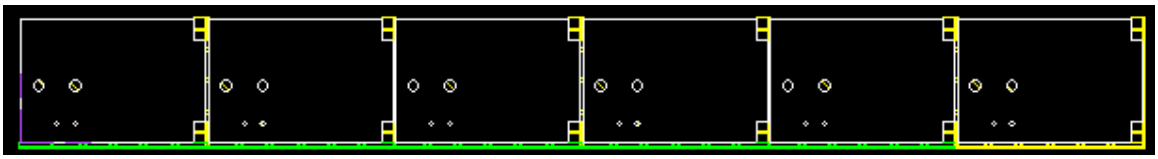
Set the parameters as shown below and click **OK** button:



Click on the bottom cut of part 1; it will change to green to signify the **Push Out** command:



Press on the **Space** bar to bring back the **Push Out** dialog, mark all the **open** options, click **OK** button and click on parts 2,3,4,5 (but **not** on part 6); the bottom cuts on all these parts will be colored green:



Press on the **Space** bar again, check the **Close** option, click **OK** button and then click on the last part (the 6-th one).

Normally, we will want to use the **Chute** command, but this can work with a **Stop machine** command as well.

6.19.9 Adding more parts to the Sheet

Open the **Sheet** tab in the Set Sheet and Clamps dialog and set the **Number of Parts** in **Y** to 100; this will ensure that the largest number of parts possible will be placed on the sheet.

Now we will define **Common Cuts** for the sheet – click the  button and set the options as shown below:

6.19.10 Setting the tools' order

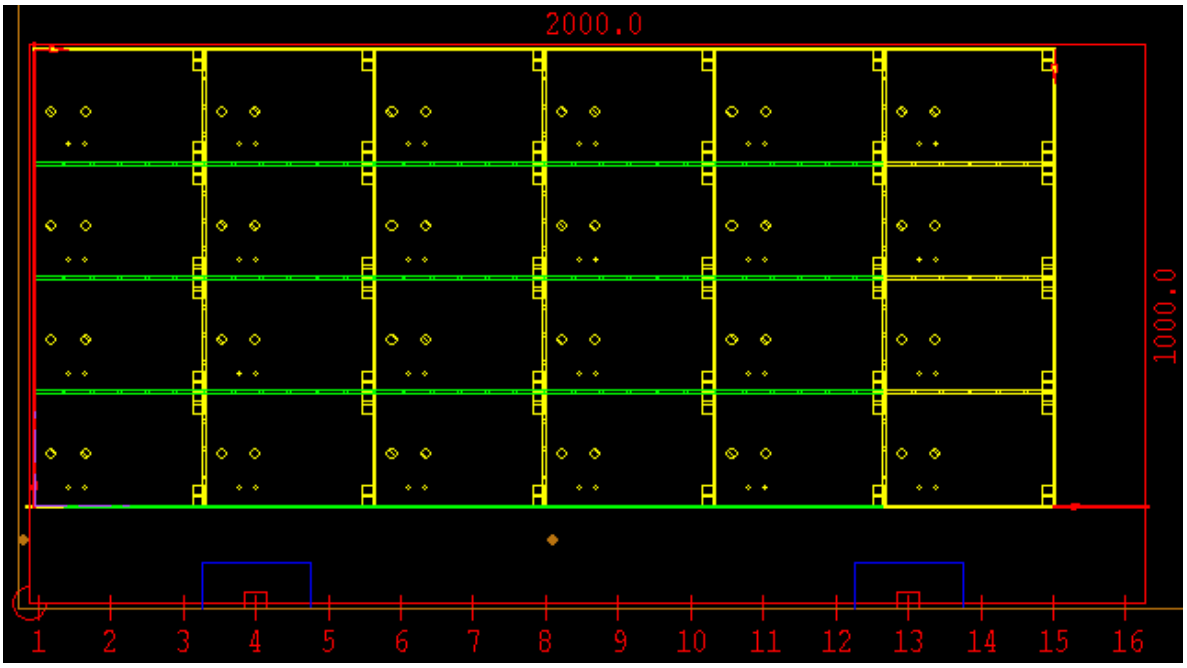
Now, we will set the **Cutting Tool** (**RE 65 5**) as the first tool to be used for processing the sheet; from the Common Toolbar click the **Used Tools** button  , and the **Used Tools** dialog will open, showing a list of the tools that will be used for processing the sheet.

Click on the **Cutting Tool** and using the arrows –  move it to the first position in the list:

Stations	Lock station	Current Tool	Seq	Grouped	Auto Index	Hits	Tool Opt	Optimize Path	Minimize Rotation	To Subr
1	☐	RE 65 5	-99		AI	5	☐		☐	
2	☐	RO 10				12	☑			
3	☐	RO 20				12	☑			
4	☐	SQ 20				12	☑			

The tool used to trim the sheet is designated as having a Tool Sequence of -99.

Congratulation, now you have a **Trimmed Sheet**:



You may want to go back to Set Sheet and Clamps **=>** **Sheet Processing Technology** tab, and select to work in **Stripes** (Line by Line) for all the tools, or just for the Cutting Tool.

6.20 Changing Turret Numbering

Each Turret has a default Numbering System that can be changed. In some cases it comes with a deferent system which is written in the Turret (*.trt) File.

To change the numbering for your turret, in Machine Settings dialog, go to Machine tab **=>** **Machine Settings** button. There **edit** your Master *.trt File and go to:

[TurretInfo]

```
[TurretInfo]
MultiToolNumbering = 1
```

If this line is not present in your turret file, and you wish to change station numbering, you should add it.

There are three ways of changing **Turret Numbering**:

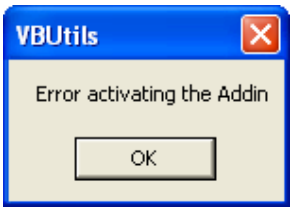
- **0** – This is normal numbering of multi tool station in turret file. That means that each Multi-tool station is represented by its base number with two or three digit prelude as indication. With this option it is possible to change Multi-tool numbering from the Tools Menu **=>** Turret Setups **=>** **Edit Turret** button.
- **1** – This is a sequential numbering of multi-tool station in turret file. This means that each Multi-tool station will be given the next number after the single stations, for example in 20-station turret the first Multi-tool stations will be numbered as follows: 21,22, 23 and so on. It won't be possible to change these station numbers manually.
- **2** – Here the number equals base station number (/K) * 100 + station index. That means that the first number of the Multi-tool station is the number of base station multiplied by a hundred and added on to the station index. With this option it is possible to change Multi-tool numbering from the Tools Menu **=>** Turret Setups **=>** **Edit Turret**.

6.21 Re-Enabling the DOC Report Addin

One of the features of ***cncKad*** is creating **Job Reports** from a Microsoft Word© template.

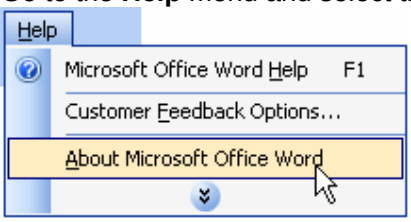
This feature may get disabled because of various inter-application data exchanges.

If this happens, the next time you try to generate a Job Report you will get this error message:

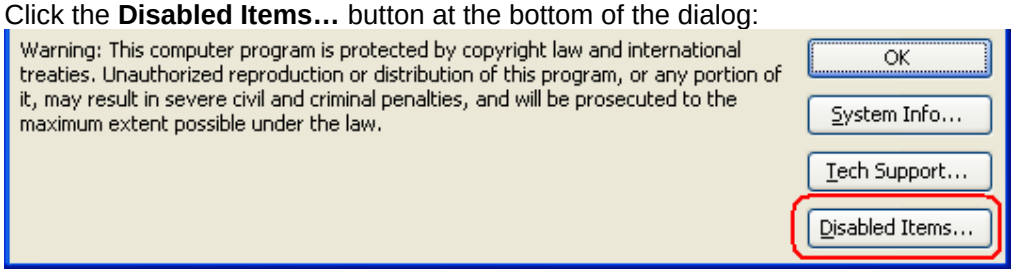


In such a case, use the following procedure:

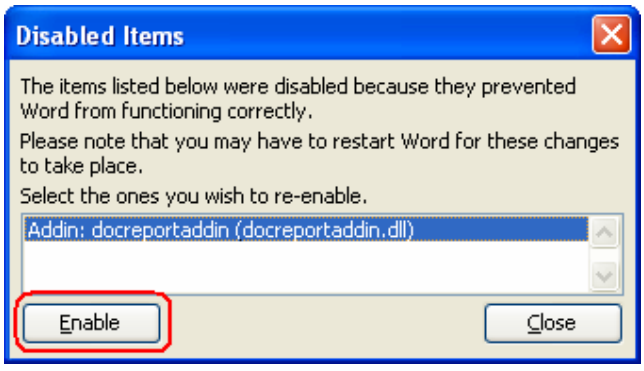
1. Restart your computer.
2. Open Microsoft Word©.
3. Go to the **Help** menu and select the **About…** item:



4. Click the **Disabled Items…** button at the bottom of the dialog:



5. In the Disabled Item dialog, select the **docreportaddin** item and click the **Enable** button:



   Do not select other items – they are there for a reason!

6. Close all dialogs and exit Word©.

Now you can generate **Job Reports** again.

6.22 Release Notes for BOSCHERT users

In this section you will find explanations and instructions on how to **Convert Old Boschert Tools** and adjusting them for work with ***cncKad***.

6.22.1 Boschert Tools

cncKad uses two files for tools:

- TOOLLIB.MDB – this one contains a list of **ALL** your tools
- LABOD.SET – this one contains definitions of **BOSCHERT** tool numbers

These files can be created manually (using any text editor or the editor that opens while clicking **Edit NC** in Files Menu in ***cncKad***), or automatically from BOSCHERT WZG file in the following way:

1. In your BOSCHERT machine, create a tool file and for **NEW** control. Now you will have a **TOOLS.BIN** file (in old controls it was: ECCO.WZG, TRI.WZG or PWR.WZG).
2. Place the diskette with the tool file in your PC.
3. From **Start => Programs => Metalix**, or from your Desktop, run **Labod Tool Convertor**.

The program will look for the Machine Tool File on the Diskette. If it won't find it, you will be asked to locate it manually. When the program will find it, it will select and automatically ascribe the name of the setup file: **LABOD.SET**. The ***Labod Tool Convertor*** will create both the LABOD.SET and the TOOL.LIB files.

6.22.1.1 Working with new setup file

Now, in ***cncKad*** go to: Tools **=>Turret Setups => Add/Create New**. You should see **LABOD.SET** file in the dialog. Select it by double-clicking on it, or by clicking it once and then clicking **Open** button. The **LABOD.SET** file will appear in the Select Setup file dropdown list.

When running the Post Processor (by clicking **NC** button), you will come to the Used Tools dialog, make sure that in **Turret Setups section**:

- **Use Current NC** box is checked,
- The **LABOD.SET** file is selected in the dropdown list

This procedure needs to be done just once. Later on, the correct numbers will be automatically ascribed to the tools.

6.22.1.2 Adding tool to machine

When you add a new tool to the machine station, you have two options:

- You can create an updated **TOOLS.BIN** or **WZG** file on diskette, and then run ***Labod Tool Convertor***
- In ***cncKad*** you can go to Tools **=>Turret Setups => Edit Setup** button and add the new tool.

6.22.2 ToolN setting in Tool Library

In the **Tool Library** ***Labod Tool Convertor*** automatically assigns, the ToolN for every tool, according to the turret station numbers they are in.

This option is designated in order not to use LABOD.SET at all.

6.22.3 Defining Tools' Angle

Tools for Boschert machines should be defined this: **RE 25 5** or this way: **RE 25 5 90**. You should always start naming the tool with the longer side.

> In old controls you could use tool angle **ONLY** for slanted tools. If you have a Rectangular tool of 25x5 at 90 degree, define it as **RE 5 25** (and **not** as RE 25 5 90).

6.22.4 Setting the Machine Version Parameter

Depending on the version of the machine controller being used – the program created within ***cncKad*** will have to be suitable. There are several options of setting the program version and they can be defined within the MDL file.

To set it correctly, first check the size of the PGM file. Then, go to: Settings **=>** Machine Settings **=>** Machine tab **=>** **Machine Settings** button. In the dialog which will open highlight your machine, and in Edit Machine Files select the **MDL** file you want to edit and click the **Edit** button.

In the Editor go to:

```
[Machine Info]
MachineVersion = 11
```

The **'MachineVersion'** parameter could be one of the following:

- 1: BOSCHERT PGM file, 26128 bytes
- 2: BOSCHERT PGM file, 34278 bytes
- 4: BOSCHERT PGM file, 34428 bytes
- 6: BOSCHERT PGM file, 34428 bytes (File Version: 7.72)

Versions 10 and upper ones are reserved for **New LABOD** format (which has no specific size for the file).

- 10: New BOSCHERT format (Ver.1), no specific size.
- 11: New BOSCHERT format (since 1999). Only ONE file is generated. Subroutine numbers are from 1 to 99.
- 12: New BOSCHERT format. Each subroutine is in a different file. Main file has 4 digits (like 1234) and subroutines filenames has 8 digits: first four are the same as the main file name (1234). The following two are the subroutine number (from 01 to 99) and the last two digits are 00. i.e. the first subroutine for 1234 is 12340100.

6.22.5 Offsets for Repeated Functions

If you want the offsets to be the same for **ALL** the tools, you should make sure that all tools start processing from the same corner (e.g. Top-Left) and have the same processing Path (e.g. Snake X).

This can be done in two ways:

- By setting the **corner** and **path** for Parts, Vertical Cuts and Horizontal Cuts in Set Sheet and Clamps **=>** **Sheet Processing Technology tab**.

  You can also set these parameters in Settings **=>** Machine Settings **=>** **Laser Optimization tab**. Then they will be the default for every part to be created.

- By setting **corner** and **path** for every tool in Tools **=>** **Used Tools**.

6.22.6 Plasma / Roller / Engrave

cncKad supports Plasma and Roller tools. When using plasma, it can cut the sheet or just engrave it. The **B** tool should be used for all these procedures.

- For Plasma:
  - Plasma tool declaration:
    B <beam width>
    For example (1 mm plasma):
    B 1
    If you want to engrave using the Plasma – use the command **Engrave Entities**, then the Set Number for Engrave will be used

- For Punch: we still use the **B** tool, and the **Add Cut** command:
  - **"Rollieren"** Roller tool declaration:
    B <roller width> N = 50
    For example (1 mm roller):
    B 1 N = 50
  - **"Gravieren"** Engrave tool declaration:
    B <roller width> N = 60
    For example (1 mm engraving):
    B 1 N = 60

Engrave text is also supported - simply use the **Engrave Text** feature in ***cncKad***.

6.22.7 Converting Plasma.bin to cncKad Laser Tables

More on this feature you will find in **Convert Plasma Table to cncKad** section of Settings Menu chapter of ***Drafting and Nesting Manual*** (go to: Settings **=>** Machine Settings **=> Cutting Parameters tab**).

7 Index