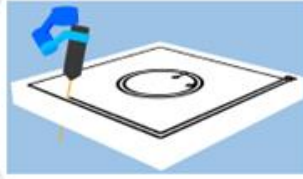# AJAN ®

Elektronik Servis San. Ve Tic. Ltd. Şti.

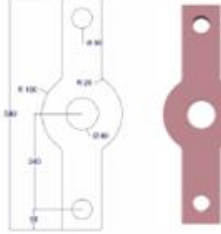**3D SİMULASYON**

AJANCAM YERLEŞİM VE OPTİMİZASYON
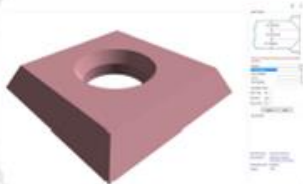
AJANCAD TASARIM

KESİM RAPORU

AÇILI KESİM

AJANCAM

Plasma/Laser

Automatic Nesting and Optimization Software

User Guide

INFORMATION: You will find the necessary information,training video this address

www.ajancam.com

Settings: It's the menu for language,units and display.

Material Management: It's the data base for the default amperage, material, kerfs and cutting speeds.

Help: It the online help menu which includes text and video tutorials and shortkeys .

Simulation: It's the cutting process demonstration to help finding un wanted errors during the real cutting proccess.

Simulate MPG Files: It's used to simulate a previous saved cutting job

Stock Management: Sheet stock view,edit and file export (.xls extension)

Consumable Cost: Input is made consumable costs to calculate the cost cutting

AJAN CAM software is actually consisting of 3 modules:

- **AJAN CAD:** It's the drafting interface use to draw parts for cutting.

- **AJAN CAM:** It's the nesting module used to nest the different parts and create the G codes to be send to the machine.

- **AJAN PIPE:** this module is used to prepare .dxf files ready to cut for pipes intersected with each other in many different ways.

Command Side Bar Mutual in AJAN CAD and AJAN CAM

AJANCAD and AJANCAM are used in the common

Below are descriptions of the command
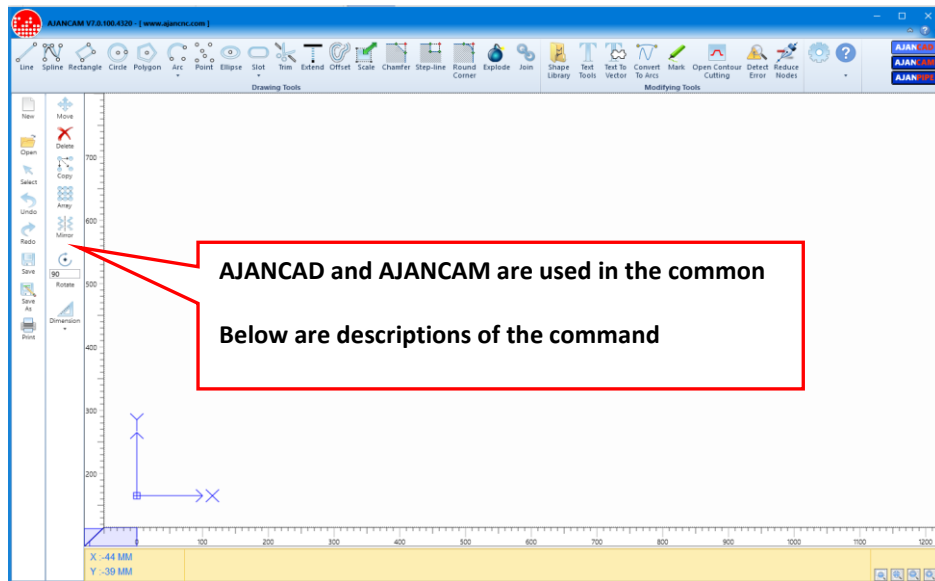
NEW: For new drafting page.

OPEN: To open a drawing file from a folder. (Extensions can be open .dxf/.mpg/.jpg/.png/.gif/.bmp/.cca)

SELECT: To select a vector or a part.

UNDO: To go back one step.

REDO: To go forward one step.

SAVE: This command saves the changes made on the drawings or new drawings.

SAVE AS: This command will save the drawing with different name.

PRINT: This command will print what is on the drawing page.

MOVE: This command will move the selected part to any place on the drafting page.

1. Select Object
2. Select command
3. Select referance point
4. Input length and angle,then press ENTER

DELETE: This command will delete any selected vector or shape.

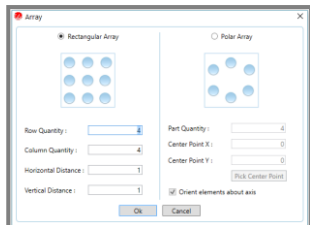**COPY:** This command will copy a selected shape to a specified location.

1. Select Object
2. Select command

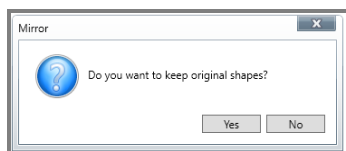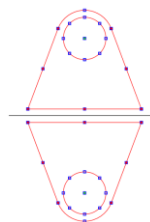3. Select referance point
4. Input length and angle,then press ENTER

COPY AS ARRAY: This command will copy the selected shape as an array in the x and Y directions.

1. Select Object
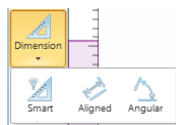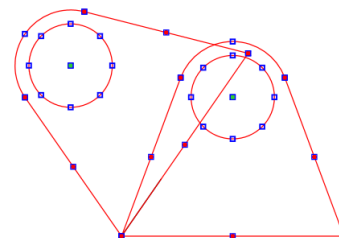2. Select command
3. Input variables

MIRROR: This command will mirror a selected shape according to a vertical/horizontal axis.

1. Select Object
2. Select command
3. Select (mirror line) first referance point
4. Select (mirror line) second referance point
5. Select the option you want to apply in the following pop-up window

ROTATE: If you want to rotate any shape with exact angle you can use this command and add the angle value in the window.

1. Select Object
2. Select command
3. Select referance point
4. Input rotate angle, then press ENTER (Keyboard shortcut **HOME** and **END**)

DIMENSIONS: This command gives the dimension between two selected points on the X/Y axis of any shape.

SMART: Select command,select object. Dimension between automatic start and end point

**ALIGNED:** Select command,select start and second point

ANGULAR: Select command,select start and second object

MAKE GROUP: This command will help you group any shapes together and make them like one entity, you can see them in the lower side of the screen after grouping and you can use them when you do nesting so this command will speed up your work.

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THE AJANCAD MODULE

The CAD interface consisting of the mobility command tools which is all organized in the side bar and the drafting commands on the main menu also the modify commands in the main menu to.

By pressing on the Module buttons

You can move from module to another easily...
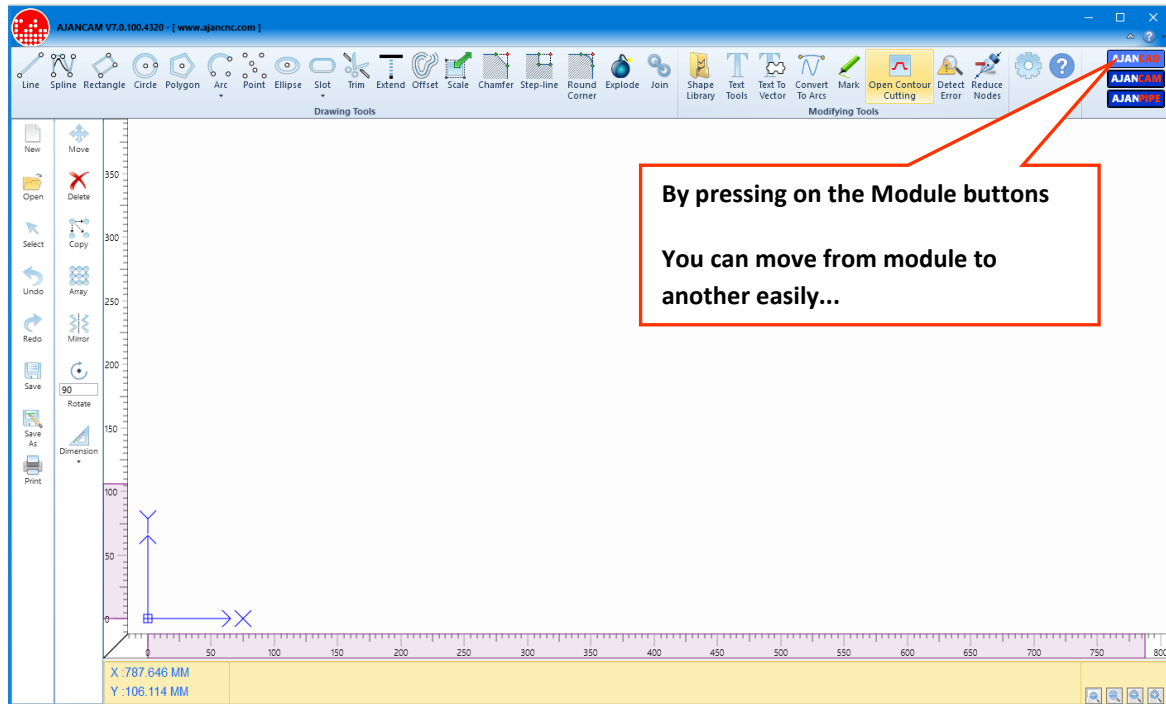
LINE: Manual line is to draw sketch style lines with left click of the mouse to specify the start and end points of a line.

Click for video applications: http://www.ajancam.com/en/line.php

1. Select command
2. Select start point (Press F8 orthogonal line)
3. Input length and angle,then press ENTER

SPLINE:

1. Select command
2. Select start point
3. Select next point
4. Press ENTER to end the process

RECTANGLE: Spesify the start point of the rectangle with left mouse click and the same thing for the end point press ESC to escape the command.

Click for video applications: http://www.ajancam.com/en/rectangle.php

1. Select command
2. Select start point
3. Input length and width

CIRCLE: To draw a sketch mode circle c lick the command and then click any where on the screen to specify the center of the circle then click again to specify the radius.

Click for video applications: http://www.ajancam.com/en/circle.php

1. Select command
2. Select center point
3. Input diameter or radius value,then press ENTER

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POLYGON: To draw a poligon in the sketch mode sslect the Poligon command a window will appear asking you about the number of poligon sides .enter the ,number press Ok and click on the screen to specify the center of the poligon and rotate the mouse to rotate the poligon , press escape to quit the command.

Click for video applications: **http://www.ajancam.com/en/polygon.php**
1. Input sides quantity
2. Select center point
3. Input radius value

ARC: Select the command click to specify the center of the arc, click to specify the radius, and click to specify the starting angle and click again to specify the end angle.
1. Select command
2. Select center point
3. Input radius
4. Input end angle

POINT: Shapes used to put a point on the desired area.Point mark by selected the marking/open contour cutting applied.

ELLIPSE: Select the command,click to specify the ellipce center click to specify the width and the height of the ellipse.

Click for video applications: **http://www.ajancam.com/en/ellipse.php**
1. Select command
2. Select center point
3. Input minor diameter
4. Input major diameter

SLOT: It is a way to create slots.
1. Select command
2. Select start point
3. Input length,width

RADIUS SLOT:
1. Select command
2. Select center point
3. Enter values in the window below
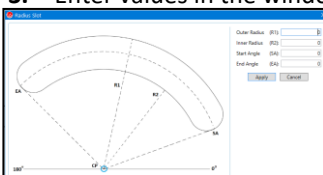
TRIM: This command is used to trim and delete the extra unwanted lines.

Click for video applications: **http://www.ajancam.com/en/trim.php**
1. Select command
2. Trimed to select the desired object

EXTEND: This command will help to connect any non-parallel two lines so we select the line that we want to extend and press enter then select the line you want to extend to.

Click for video applications: **http://www.ajancam.com/en/extend.php**
1. Select command
2. Select the object to be (extended to) and press ENTER
3. Click on the object you want to extend

OFFSET: Select the shape you want to offset click offset icon a window will open put the value of the offset and specify the direction of the offset.

Click for video applications: http://www.ajancam.com/en/offset.php

1. Select shape
2. Select command
3. Input the offset value
4. Select offset side

SCALE: The selected object is proportional scaling

1. Select shape
2. Select command
3. Select reference point
4. Input scale value

CHAMFER: Select the corner you want to chamfer put the chamfer values in the opened window and press ok.

Click for video applications: http://www.ajancam.com/en/chamfer.php

1. Select command
2. Select first edge
3. Select second edge
4. Input the dimension

STEP-LINE: Select the corner you want to make a step put the step values in the opened window and press ok.

Click for video applications: http://www.ajancam.com/en/stepline.php

1. Select command
2. Select first edge
3. Select second edge
4. Input the dimension

ROUND CORNER: Select the command and select the corner you want to round up put the Radus in the opened window in press **OK**

Click for video applications: http://www.ajancam.com/en/round.php

1. Select command
2. Select first edge
3. Select second edge
4. Input radius value

EXPLODE: It's used to separate all the pedited lines in a drawing.

JOIN: Select command,select objects. Lines and arcs can be joined to polylines if a polyline is selected as the source object.

SHAPE LIBRARY: A library of ready-made shapes. You can edit the dimensions.
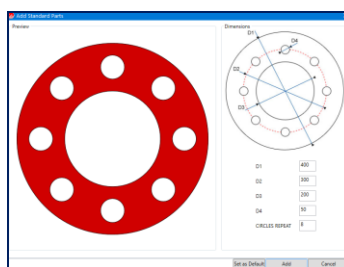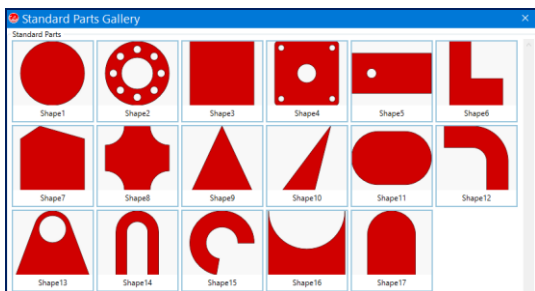
TEXT TOOLS: With this command you can write letters, numbers and symbols on the sheet metal.

- Select the TEXT command in the opened window enter the text in any language
- You will see the text on the screen. click on the text and a window will open you can change the size, font and type of the text
- To change the font from TRUE TYPE FONT to a vector select convert to vector icon.

TEXT MARK: If you want to mark text, numbers and symbols use the TEXT MARK icon to change to line style vector

1. Select TEXT TOOLS command
2. Input the variables
3. Select **OK**
4. Select start point
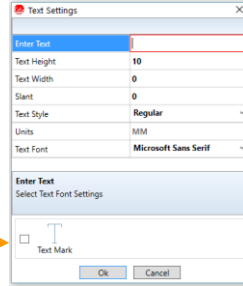
Marking the check box for writing

CONVERT TO ARC: Spline objects to convert minimum arc vector

1. Select shape
2. Select command
3. Enter tolerance (Recommended tolerance is 0.5mm)
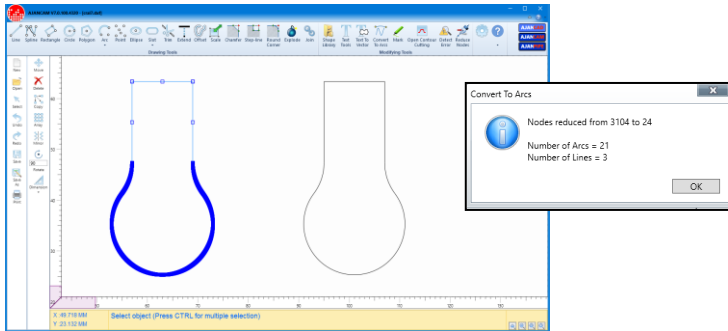
DETECT ERROR: This command will detect any error in the shape like gabs between continuous vectors, overlapping lines and intersected vectors and mark them with symbols as it shown below and can be fixed according to the tolerance value you put.
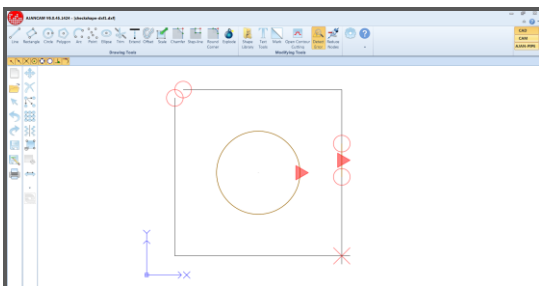

Circle symbol is describing an open vector

Triangle symbol is describing the overlapping

**The X symbol is describing the intersection**
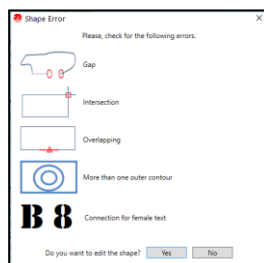
REDUCE NODES: In some DXF files imported from other drawing software's like Corel Draw the number of nodes that creates the drawing is too high because is consists of many small lines. And this will make it hard for the program to read it and the machine to cut it. So what we want to do is to reduce the nodes and we can do that by entering a tolerance value in the box and in the bottom of the screen we can see the contour numbers before reduction and after reduction.
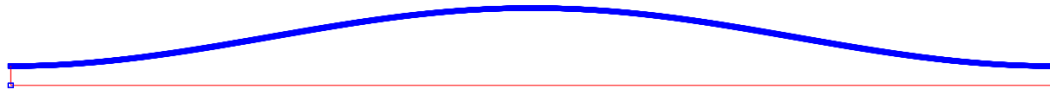
This corel draw DXF file has 2436 nodes
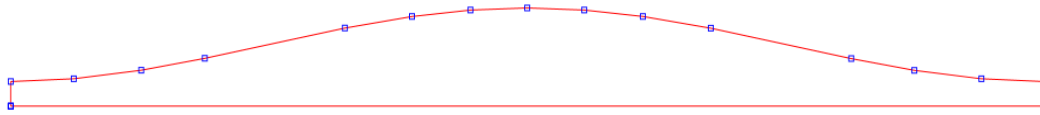
After applying the command, the nodes number drops to 19

MARKING: Is the process of scratching the sheet metal with low amperage instead of cutting it, and it's used to put some information or signs on the cut parts.

The process:

Open any DXF file in the ajan CAD module and select the shape you want to mark and press marking command

You will notice that the shape color has been changed to green.

Save the drawing and nest it in the CAM module. When you cut the part, you will notice all the shapes with green color is marked only.
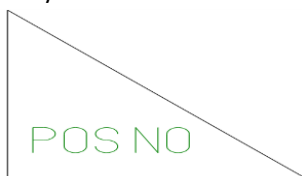
GREEN COLOR MARKING
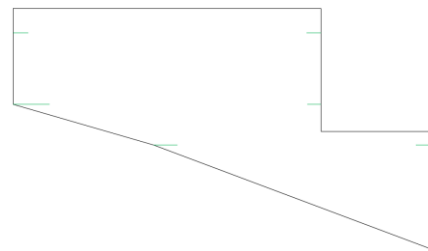
OPEN CONTOUR CUTTING: The cam module will accept all the closed contours if there Any open contour in the shape will be refused and an error massage Will appear to let the CAM module understand the open contours we need to select them when we draw them and Press OPEN CONTOUR CUTTING icon so the color of the contour will change to yellow which means any red Contour is an open contour.

1. Open the drawing in AJANCAD
2. Select the shape
3. Press OPEN CONTOUR CUTTING and you will notice that the color changed to yellow
   Save it and open it in the CAM module.

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AJANCAM MODULE

AJANCAM MODULE: In this module, you will be able to identify the specifications of the sheet metal, machine and material and all other variables.
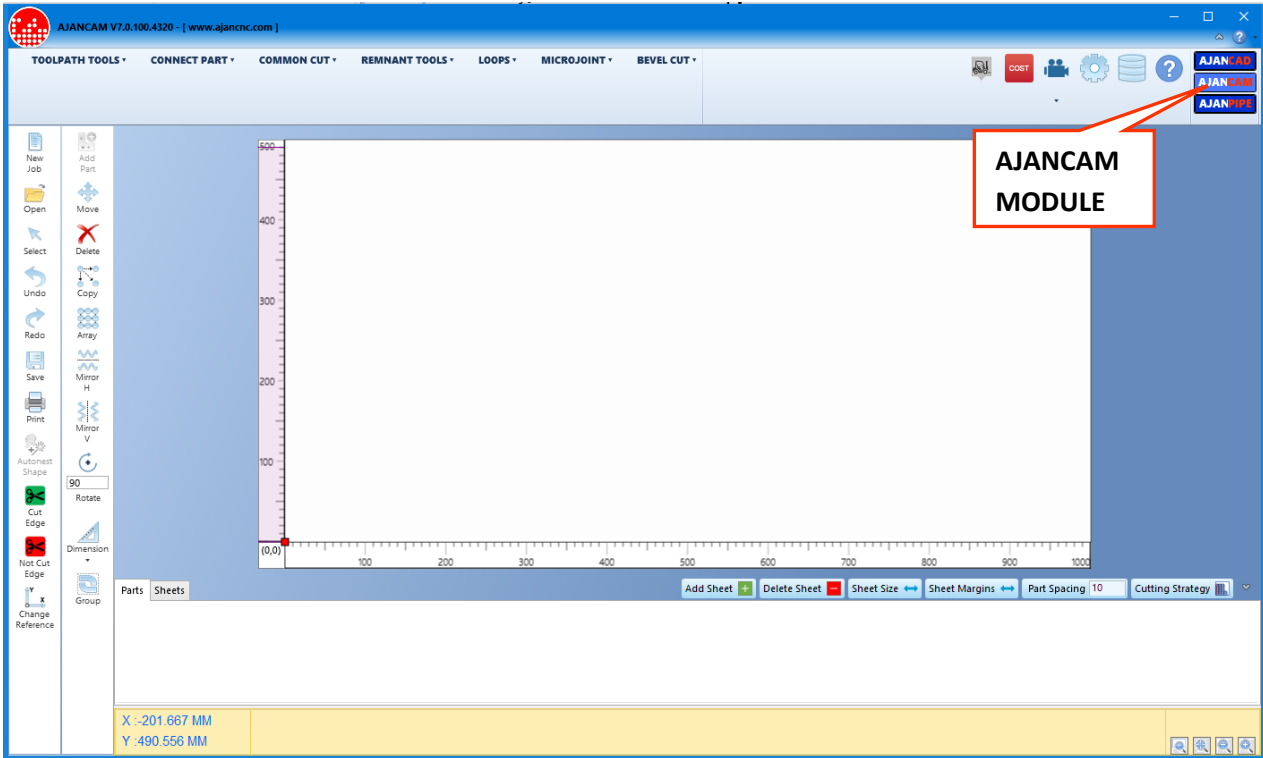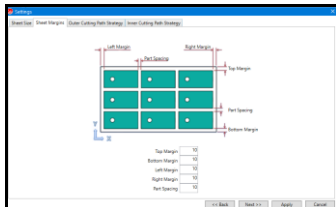

NEW JOB: Means new nesting job opens a sheet metal with specified dimension by you ready to be filled with parts to cut.
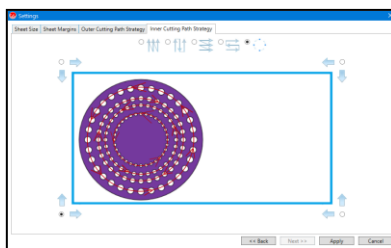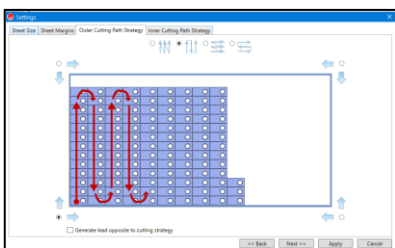
SHEET SIZE TAB: Select machine type, material, amperage, thickness, sheet length-width
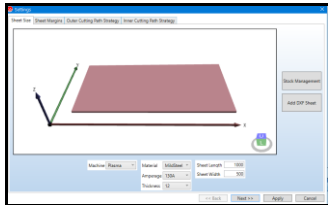
SHEET MARGINS TAB: Input top-bottom-left-right margins and part spacing

CUTTINGPATH STRATEGY TAB: Select cutting path strategy for outer/inner contour

ADD PART: If you nest some parts on the sheet metal and finish the process you can add an extra part that you may forgotten to the nest without the need to redo the whole nesting process from the begining again.

AUTONEST SHAPE: If you nest some parts manually by pressing this command you can nest the rest different parts automatically.

CUT EDGE: Only cut the selected edge

NOT CUT EDGE: Only not cut the selected edge

CHANGE REFERENCE: Select commad then with the following window selection detect new reference point
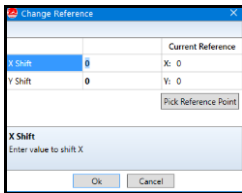

TOOLPATH MENU

AUTOMATIC TOOL PATH: This command will add cutting path to the nested parts according to the specified leadin – lead out values.

CLEAN TOOL PATH: This command will delete the whole tool path assigned.

MODIFY LEAD: Manually you can modifiy the length of the lead by extending or reducing its length.

1. Select command (For shortcut:click right mouse on the leadin/leadout)
2. Leadin/leadout modify or change position
3. Select the application in the window
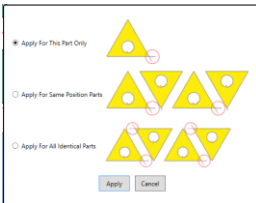
CONVERT TO LINE/ARC: You can change leadin/leadout style

1. Select command
2. Click on the leadin/leadout
3. Select the application in the window

DELETE LEADIN/LEADOUT: Deletes the selected lead in / lead out by the mouse.

1. Select command
2. Click on the leadin/leadout
3. Select the application in the window

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MANUAL CUT: By using the mouce This command gives you the freedom to add the lead in –lead out for each part seperatly and manually.You can use the inner contour and the outer contour seperatly.
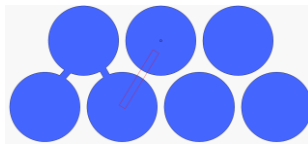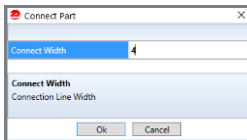
1. Select command
2. Select shape order

MANUAL PART CUT: By using the mouce you can use the sequence of the parts to be cut. In this command any shape you choose the lead in – lead out will be given automatically to the inner and outer contour.

CONNECT PART MENU:

CONNECT PART: To increase the life of the consumables you can use the connect parts command which will help you connect meny parts together and cut them in one time with one piercing process. Enter the length of the connection and select the parts that you want to connect and they will be connected automatically.

1. Select shape (CTRL+A all shape select)
2. Input the connection width,then select **OK**

MANUAL CONNECT PART: Enter the connection length and by using the mouce specify the connection locations on the shapes.

1. Select command
2. Select connection start point
3. Input the connection width,then select **OK**
4. Select connection end point

DELETE PART CONNECTION: Choose connection and delete.
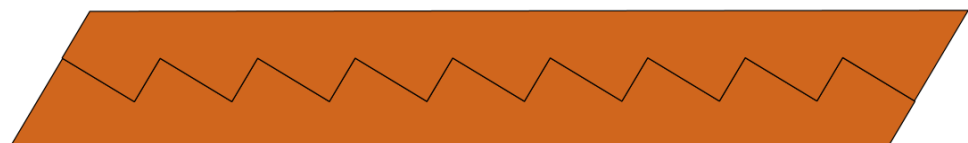
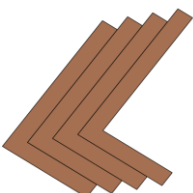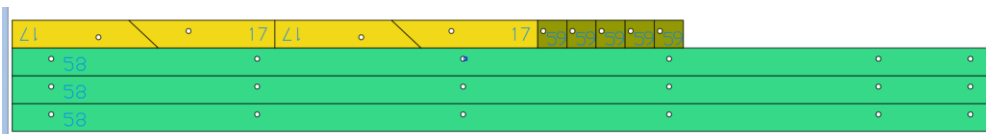**DELETE ALL:** Delete all the selected connection.

COMMON CUT MENU:

COMMON CUT: This command will help you reduce the remnants and increase the life of your consumables If you are cutting parts has the same shape and size just select the command and in the window choose the right strategy for you and press apply.

1. Select common cut command

2. Click the arrows to join any two sides
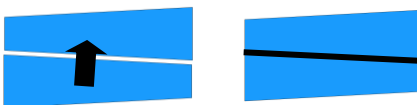
**NOTE: The same shape and can be applied in different shapes common cut**

RECTANGULAR COMMON CUT: This command will help you reduce the remnants and increase the life of your consumables If you are cutting parts has the same shape and size just select the command   choose the sides of the parts you want to connect manually.

1. Select rectangular common cut command
2. Select edge to start
3. Drag the mouse xy direction
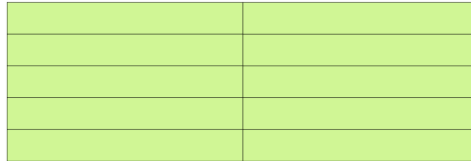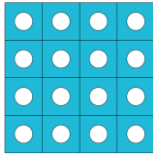
EXPLODE COMMON CUT: Common components applied individually separated segments

CUTTING PATH STRATEGY: Common cutting sequence is used to.
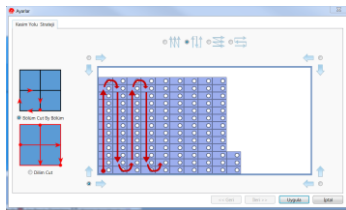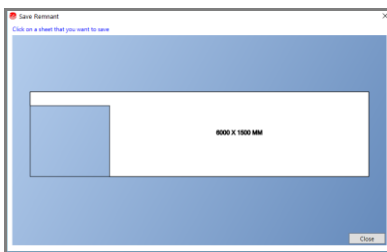
REMNANT CUTTING MENU:

AUTOMATIC REMNANT: This command will let you separate the un cut part of the sheet metal from the nested part automatically.

MANUAL REMNANT: This command will let you separate the un cut part of the sheet metal from the nested part manually
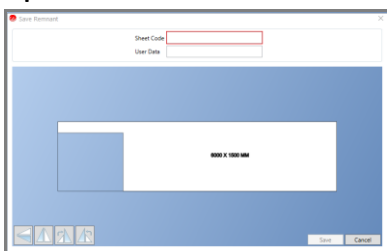1. Draw remnants path and press ENTER to finish

SAVE REMNANT: This command will save the part of the sheet metal left from the nesting to be used in afuture job.
1. Select command
2. Click on the sheet with the mouse

3. **Input sheet code**

DELETE REMNANT: To delete the added reminant from the nestted sheet.

DELETE ALL: All plates will delete remnant cutting applications

LOOPS MENU:

AUTO LOOP: Give all the sharp corners of the selected shape is transformed into.
1. Select shape (**CTRL** multiple select)
2. Select auto loop command
3. Select loop cutting strategy and input length value

MANUAL LOOP: Selected corners of the shape data is transformed into.
1. Select manual loop command
2. Select loop cutting strategy and input length value,select **OK**

3. Click the wanted corners

MODIFY LEAD: Used for manual editing with the mouse
1. Select command
2. Drag the mouse on the modify loop

DELETE LOOP: Deletes the selected loop by the mouse.

DELETE ALL: All plates will delete loop applications

MICROJOINT MENU:

MICROJOINT: Entered by the distance doesn't make the cutting process.
1. Enter microjoint length or used from database value
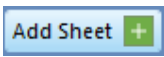2. Select any point on the cutting path

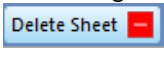**DELETE MICROJOINT:** Deletes the selected microjoint by the mouse.
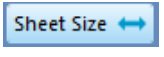1. Select command
2. Click on microjoint

DELETE ALL: All plates will delete microjoint applications

ADD SHEET: Is located In the lower side of the CAM module window ,use this command to add a new sheet to the nesting

DELETE SHEET: Is located In the lower side of the CAM module window,use this command to delete sheet from the nesting.
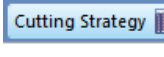
REDIFINE SHEET SIZE: This command located in the lower side of the cam module and used to show the exact area used in the nesting you are doing In case if you have a piece of sheet metal that can be used instead of a full sheet.

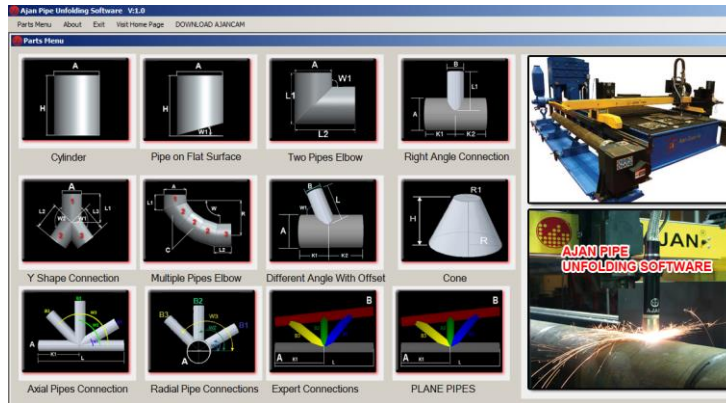**SHEET MARGINS:** Used to edit the sheet margins

PART SPACING: Used to edit the part spacing dimension

CUTTINGPATH STRATEGY: Used to edit cutting path strategy

AJAN PIPE UNFOLDING SOFTWARE

The software have ready to use pipe connection kits

To save the intersection drawing Press "DXF" and save the drawing in the DXF format.

Put the values you want in the boxes
Below, each one of these boxes represents a dimension on the drawing above.

To draw the intersection of the pipes according to the values you entered press "Calculate" and you will see the drawing in the right window