AJANCAM

Automatic Nesting and Optimization Software

User Guide

Options: It's the main menu to specify the dimensions of the sheet metal,the amparage,lead in –lead out and the cutting strategies.

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

Material Management: It's the data base for the default amparage, 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

<u>**AJAN CAM software is actually consisting of 3 modules:**</u>

- **AJAN CAD:** It's the drafting interface use to td 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 Mutaul in AJAN CAD and AJAN CAM

CAD and CAM are used in the common

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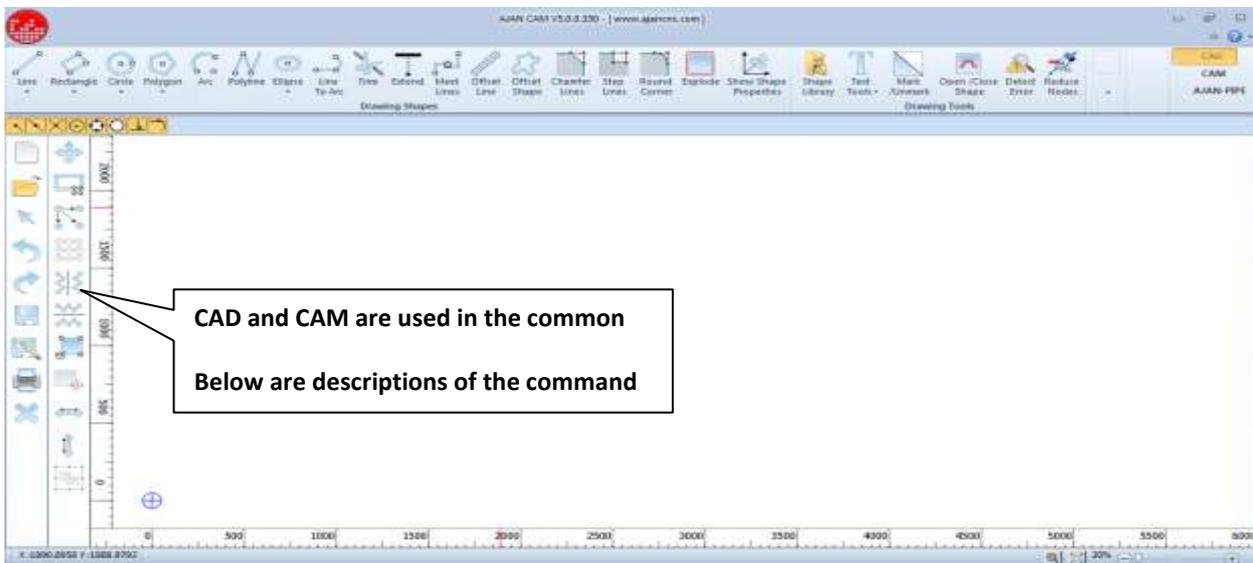
New: For new drafting page.

Open: To open a drawing file from a folder.

Select: To select a vector or apart.

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.

Exit: This command will terminate the program.

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

Delete: This command will delete any selected vector or part.

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

Copy as Array: This command will copy the selected shape as an array in the x and Y directions.

Vertical Mirror: This command will mirror a selected shape according to a vertical axis.

Horizantal Mirror: This command will mirror a selected shape according to a horizantal axis.

Rotate: The selected shape can be rotated by using the mouse.

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

**Vertical Dimension:** This command gives the dimension between two selected points on the Y axis of any shape.

Horizontal Dimension: This command gives the dimension between two selected points on the X axis of any shape.

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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.

THE CAD 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 easly...
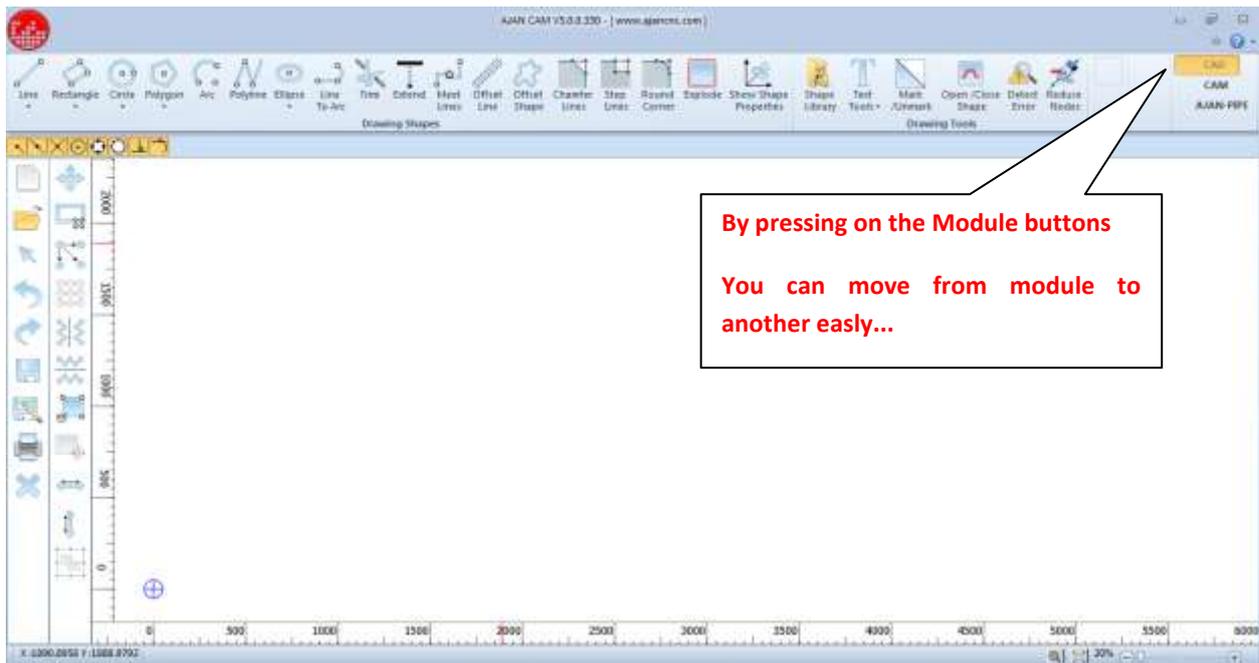
Line : Manual line is to draw sketch style lines with left click of the mouse to specify the start and end points of a line.

Line with Dialouge Input: If you want to give exact dimensions for the line.Clickon this command a dialog box will appear and you can put the angle and the length of the line.

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.

Rectangle with Dialog Input : To draw a rectangle with ecaxt dimension click the command and with left click of the maouse specify the start point a dialog window will appeare put the dimensions and click ok. Esc to escape the command.

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.

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Circle with Dialog Input: To draw a circle with exact dimension click this command and then click the center point a dialog window will open put the dimensions and click OK.

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.

Poligon with Dialog Input: Select the command click the center point a dialog window will open ,input all the dimensions and press OK.

Radius: 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.

Poly Line : Select the command and sketch the lines after each other.

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

Line – Arc : This command is used to convert the lines to arcs.

Trim: This command is used to trim and delete the extra unwanted lines.

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.

Meet Line: This command will connect the gabs in shapes with a line .

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.

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

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

Round Corner: Select the command and select the corner you want to round up put the Radus in the opened window in press Ok.

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

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Shape Properties: It is used to show all the them shape coordinates and dimensions values and to edit.

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.
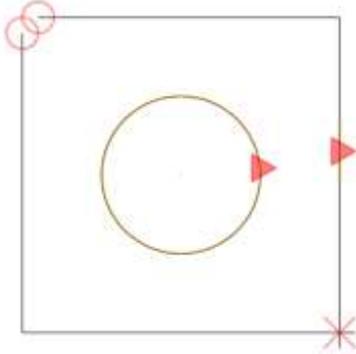
Circle symbol is describes an open vector

Triangle symbol is describing the overlapping

The X s ymbol 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
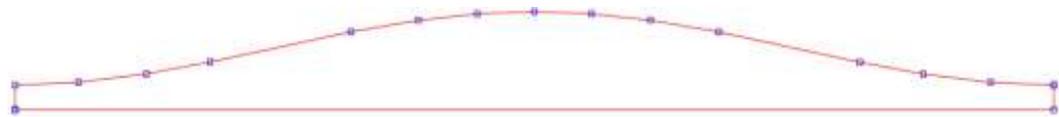
After applying the command the nodes number drops to 19

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 have 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.

POS NO

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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.
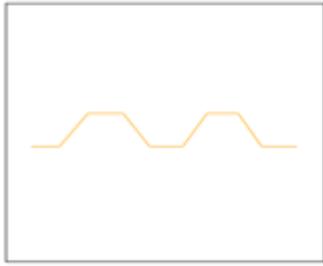
1. Open the drawing in ajancad

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.

TEXT

Text: 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 To Vector:** Used to change the text to dxf vector

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

Shape Library: Some of the standard shapes are saved in the memory of the software, you can change the shapes dimensions easily.

CAM MODULE

CAM MODULE: In this module you will be able to identify the specifications of the sheet metal, machine and material and all other variables.
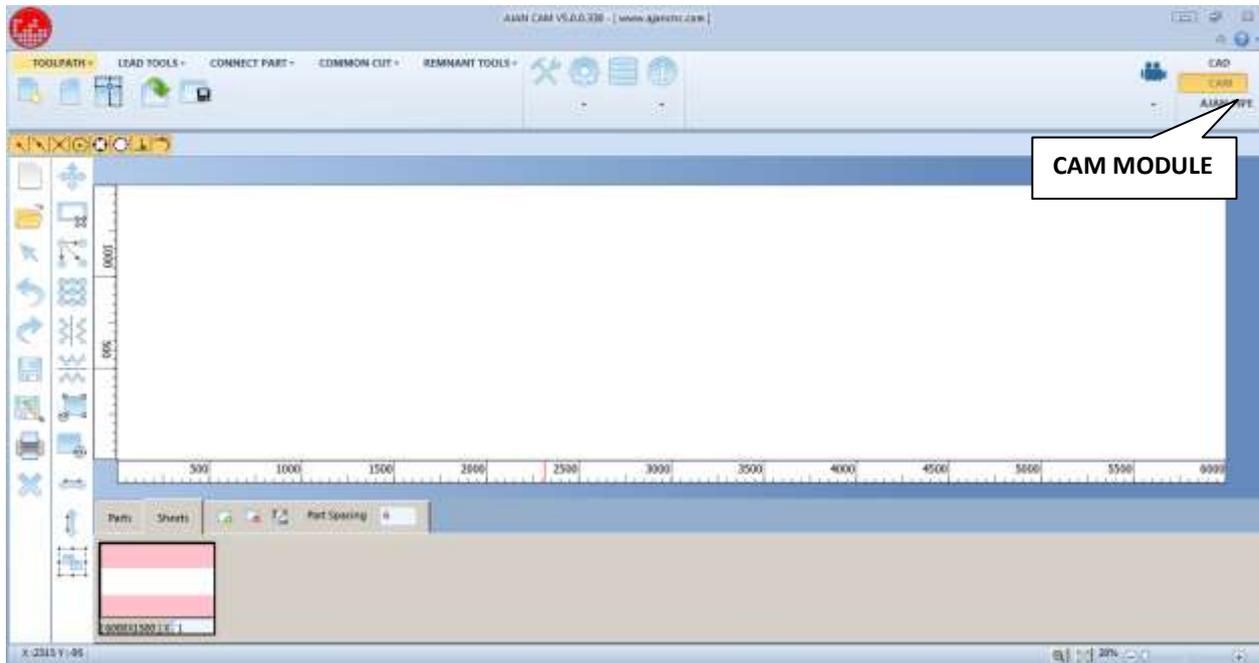
New Job: Means new nesting job opens a sheet metal with specified dimension by you ready to be filled with parts to cut.

Add Shape: 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 beginning again.

Autonest Remaining Shapes: If you nest some parts manually by pressing this command you can nest the rest different parts automatically.

Open Nesting: With this command you can open saved MPG files (nesting files) and you can do any changes on the nesting and save it again.

Save Nesting : This command will save the nesting in the memory.

TOOLPATH MENU

Autamtic Tool Path : This command will add cutting path to the nested parts according to the specified leadin – lead out values.

Delete Tool Path: This command will delete the whole tool path assigned.

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.

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.

Lead in /Lead Out Delete: Deletes the selected lead in / lead out by the mouse.

Modifay Lead: Manually you can modifiy the length of the lead by extending or reducing its length.

Lead Setting: Using this command you can adjust the lead in lead out values and specifications.

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.

Manuall Connect Part : Enter the connection length and by using the mouce specify the connection locations on the shapes.

COMMON CUT MENU:

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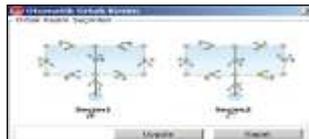
Manuall Common cut: This command will help you reduce the remnants and increase the life of your consumables If you are cutting parts has the same shape and size just select the command   choose the sides of the parts you want to connect manually.

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

Save Remnant: This command will save the part of the sheet metal left from the nesting to be used in afuture job.

 Delete Remnant: To delete the added reminant from the nestted sheet.

 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.

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