

**DwgTools**

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## INSTALLING THE APPLICATION

### *Before installing **DwgTools**. System Requirements*

- \* Windows 2000, XP, Vista
- \* Pentium III or Pentium Pro (preferred), or more
- \* minimum 1GB of RAM (2GB recommended for blocks in 3D)
- \* 1600 x 1200 screen resolution
- \* 2,933 GB of free space on hard disk.
- \* Mouse
- \* Printer
- \* **AutoCAD** 2010 to 2012

### *Installing the program **DwgTools***

1. Run the setup.exe file by double clicking on it.
2. Answer the various questions that will make the installation program.

The application creates shortcuts to the main module in the Start Menu>> Programs of Windows

Similarly, a shortcut is created on the desktop to start the program.



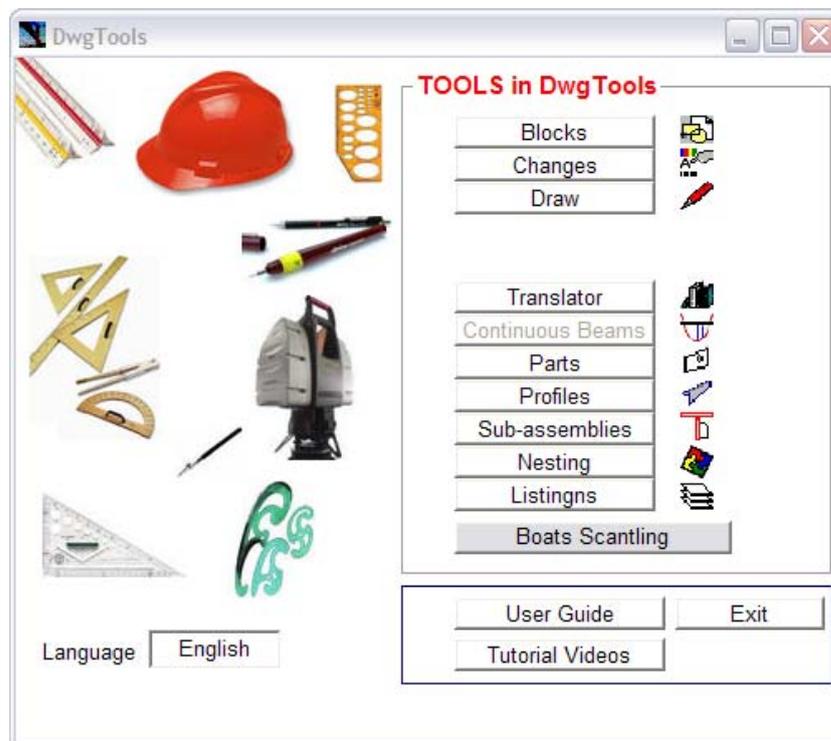
## 0 PRESENTATION

DwgTools get to automate most of the work a draftsman of marine structures has to perform to make constructive drawings for assemblies and sub-assemblies.

Also, for flat parts, facilitates nesting of them in plates and processes these sheets creating files with code for CNC machines.

It is divided into several modules, each one for the specific tasks required, and all of them, but the one called SCT, work in the AutoCAD screen.

This division makes the operation of the application much more agile, reduces the computer memory, the user works with the least possible number of open windows and the curve of time-learning is much smoother.



The initial screen gives access to each of the modules.

On the next page summarizes the action of each one of them.

### BUTTONS EXPLANATION

<b>User Guide</b>	Opens USER GUIDE (pdf format) in a separate window.
<b>Tutorial Videos</b>	Starts an independent application, the help system, with texts and videos
<b>Exit</b>	Closes the application

## MODULES IN DwgTools

icon	NAME	TASKS PERFORMED
	DwgTools	Starter module of the application. Allows activation to acquire the permanent license
	Blocks	Working with blocks. Creation of new blocks with attributes. Inserting existing blocks. Counts and related existing blocks in a drawing
	Changes	Handling, automatically, all existing graphic objects on a drawing
	Draw	Automatically draws of most parts and shapes that a designer of structures uses
	Translator	Translates texts on a drawing from English to Spanish. Translator has several languages Dictionaries
	Frames and Beams	Calculation of simple continuous beams and frames, by Cross and 3 moments method. Estimated also the module, inertia, weight, etc. of any section drawn in AutoCAD. Main Frame modulus.
	Parts	Definition of parts from plates .
	Profiles	Definition of profiles
	Sub-assemblies	Creation of sub-assemblies
	Nesting	Nesting of flat parts and generation of code for CNC cutting machines.
	Lists	Working with multiple listings generated by the application
		Small boats scantling according to ISO Standards 12215-5, 12215-6 and 12215-8
<p><i>In addition to these "core" modules, DwgTools has two "auxiliary" modules, to which we access through the toolbar created in AutoCAD.</i></p>		
	Utilities	Several options to automatically change data in drawings and sketches
	Printouts	Management of DwgTools printouts, full automated process.

# 1 FIRSTS STEPS WITH DWGTOOLS

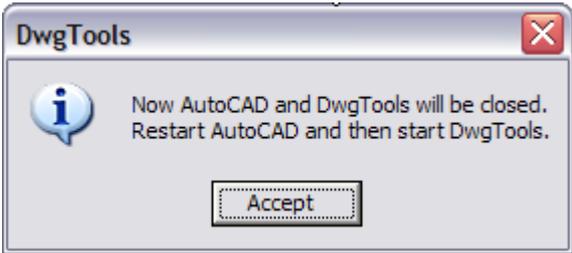
The first time, DwgTools has to be started as described herein, with AUTOCAD ON.

- 1.- Start AutoCAD
- 2.- In the Windows Desktop, you will find a shortcut to **DwgTools**. Use it to start the application.



Look for this icon

In a few seconds the following message :



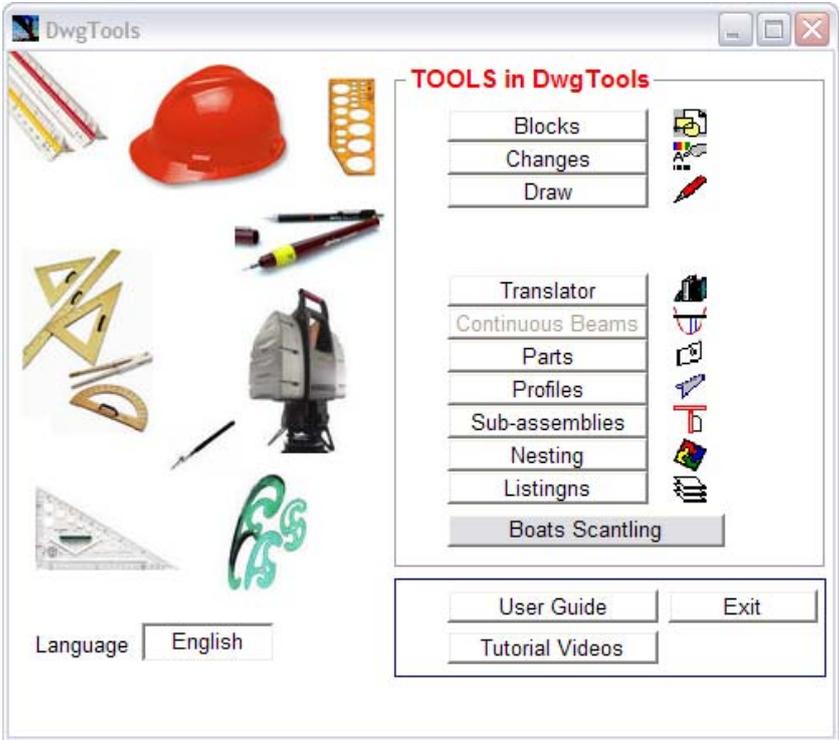
That means that DwgTools has successfully connected with AutoCAD and that communication is established between both applications.

Press the button **Accept**.

From this moment, when you start DwgTools, the window of the application is shown :

The buttons on it give access to the relevant modules, which will be described later in more detail and summary of which was made on page 1.

In the bottom label of the sale will indicate the remaining days of testing period.



**First thing to do is selecting LANGUAGE** to be used by the application.

Clicking on the text box you will get the desired language.



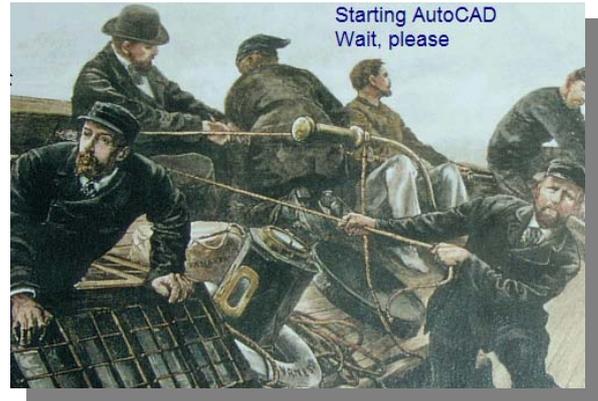
## NOW, AS A PRACTICE, PROCEED AS FOLLOWS :

Close AutoCAD and DwgTools.

Re-activate the **DwgTools** icon on the Desktop.

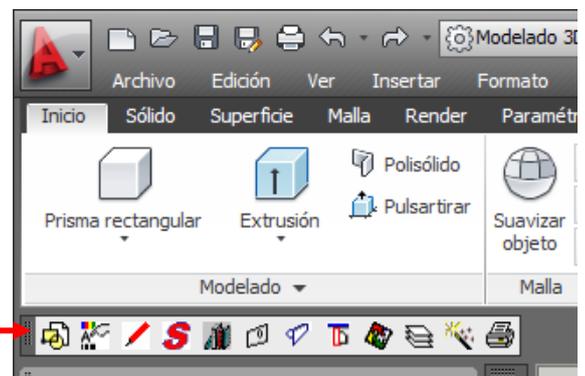
The application starts normally, displaying the window shown on the previous page, without starting AutoCAD, as the login module does not need it for himself

However, as the other modules do need, when you press any button, the application does start automatically AutoCAD and displays the picture on the right.:



When the process finishes we can see that between the AutoCAD toolbars appears the one of **DwgTools**

with buttons that, even from here, support booting our application modules.



We can therefore begin to work in two ways:

### 1. AutoCAD is running (normal procedure recommended)

- a. Start DwgTools.
- b. Select in the toolbar that has appeared in the **AutoCAD** window, the module you want to work with, pressing on the appropriate button



### 2. AutoCAD is NOT yet running (procedure illustrated at the top of page)

- a. Start DwgTools, from the Desktop.
- b. Choose the module you are going to work, from the toolbar of AutoCAD

In both cases, to change the module to work, simply activate the icon in the toolbar of AutoCAD.

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**NOTE:** You should end up with a module (close), before starting a new module.

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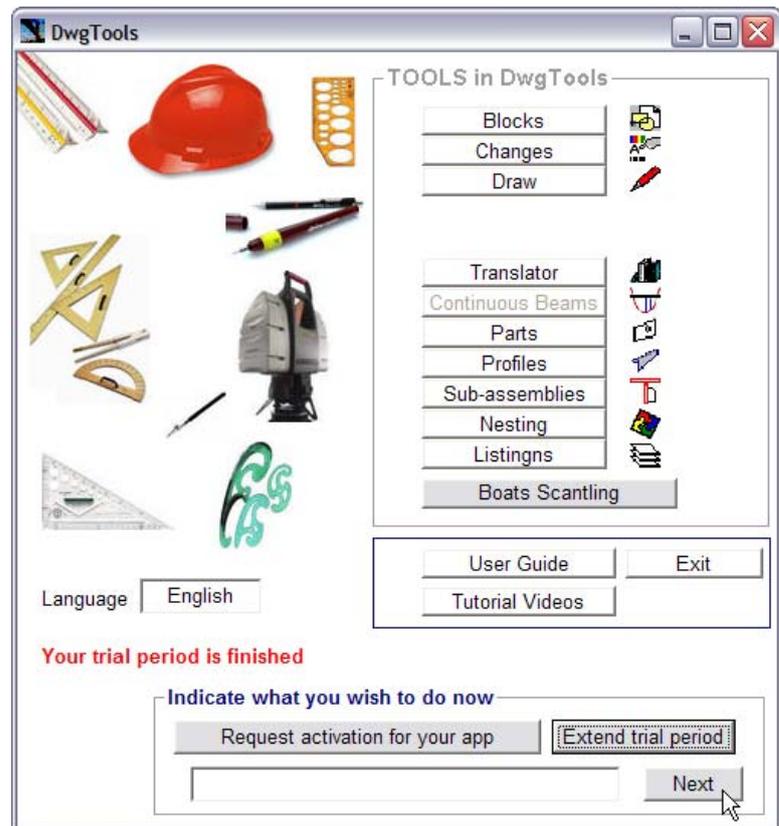
## 2 ACTIVATION OF THE APPLICATION

At the bottom of the module label home, the display shows the days remaining to complete the testing period. At the end of it, if you want to continue working with the application, must be requested :

1. An extension of the trial period
2. The final activation of the individual license

At any time during your trial period, you can request the permanent activation of DwgTools

When the trial period ends, the start window suffers a slight modification.



### 1. REQUEST EXTENSION OF TRIAL PERIOD

To prolong the test, you must explain your case to your provider, if he agrees, will send a code consisting of letters and numbers.

After receiving the code, you must act on the **Extend trial period** button

1. the lower text box will be shown.
2. enter therein the code you got and press **Next**

## 2.- FINAL ACTIVATION REQUEST

You must act on the button  (see previous page)

In this new window, check the option **Request activation code**.

And press the button **Next**.

The window changes its layout and text boxes will appear which you must fill in your details

**User data**

Name (*) .....	<input type="text"/>
Surname (*) .....	<input type="text"/>
Company .....	<input type="text"/>
Address	
st/sq/av/road .....	<input type="text"/>
N° .....	<input type="text"/>
Place .....	<input type="text"/>
State .....	<input type="text"/>
Zip code .....	<input type="text"/>
Country .....	<input type="text"/>
Telephone number (*) .....	<input type="text"/>
Fax .....	<input type="text"/>
e.mail (*) .....	<input type="text"/>



Once completed the data, press the **Generate Request** button.

The application tries to automatically create an email in Microsoft Outlook so you can send it, as it is, to your dealer.

In the event of an error, you must create this email, attach the file called **SolicitudDwgT.act**, that was saved in the directory of the application and send it to TAN S.L. : [657677483@orange.es](mailto:657677483@orange.es), [tansl@wanadoo.es](mailto:tansl@wanadoo.es)

Soon you will receive same via a file called **DwgTools.aut** that, once stored in **C: \**, will allow you unlimited use of the application.

### IMPORTANT NOTE

The file *SolicitudDwgT.act* contains your data, the identification number of your hard disk and Serial Number of your AutoCAD. All combine to generate a key specific to your computer and AutoCAD. That is, the program will not work on another hard disk or another AutoCAD.

### 3 CREATING AND HANDLING BLOCKS

Icon in *AutoCAD* Tool Bar

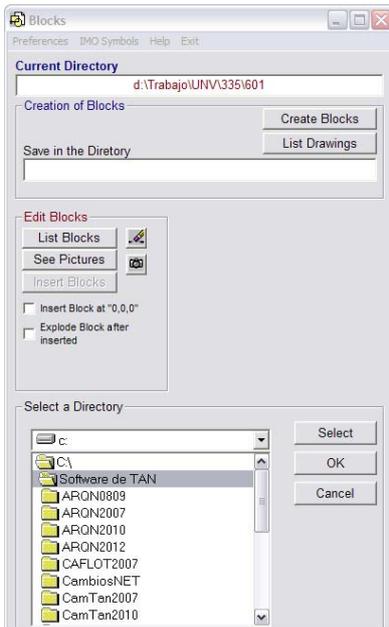


The initial window of the module is as follows:

This module is intended to assist in the creation of new blocks in the management of existing as well as offering several options that are convenient for the delineation of plans in general.

The building blocks can be made directly from AutoCAD. But the blocks created with this module have, among others, two important advantages over those:

- Once created, the block belongs to the current drawing, but is saved in the previously selected directory as a separate drawing. Thus it is available to be inserted in any drawing, without opening the plane that contains it.
- When created, it is created a "picture", visible with the help of this module, which greatly facilitates the task of choosing the right block to insert.



The **Current Directory** indicates where they are stored blocks with which to work.

The label **Save in the Directory** indicates where to save the blocks we have created.

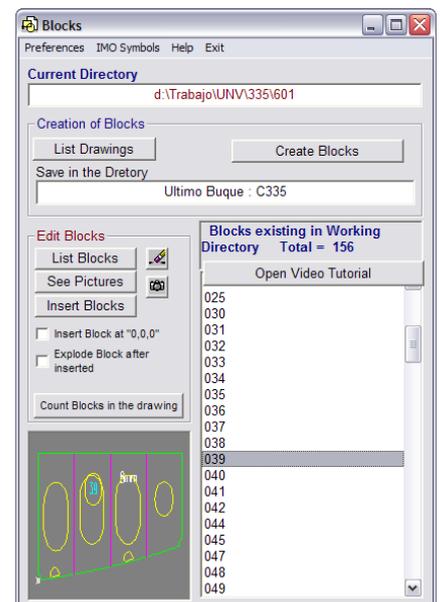
When you click on it, a directory tree appears, by which we can select the folder to save the new blocks.

In the same way, clicking on the label **Current Directory**, using a similar dialog box, we can change the folder from which we want to find and insert existing blocks.

Two options are available but, although similar, are different:

- The **List Drawings** button provides a list of all existing drawings in the working directory, whether or not generated with this module.
- The **List Blocks** button lists only the drawings that have "photo." Some have been created as blocks for this module but others, such as parts of the structure that also have a "picture", have been created with other application modules.

Both produce a list of their names. Clicking on one of them, when there is



a photo, you can see it in the window below, left. In addition **Insert Block** button is now activated. Pressing the drawing mentioned here is inserted into the current sheet as a block.

When the drawing exists but has no picture (is not a block), the **Insert Block** button is not activated, and therefore, we can not insert it in the plane.

The imported drawing is inserted directly into the origin, if the box for that purpose is checked. Otherwise the application will ask you to indicate in the display the insertion point.

Another possibility is to explode the block, once inserted, to work with him. Simply select the appropriate box.

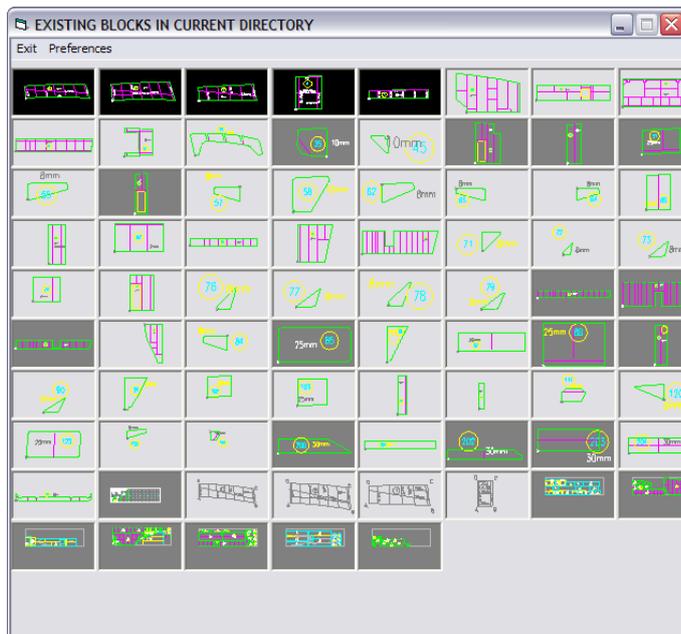


**Count blocks in the drawing** button creates a list of blocks in the drawing and how many instances of each one. Note that AutoCAD uses some objects, called "blocks", that are not visible but are in the drawing. Thus the ratio of the drawing blocks can have names that are not expected to find



Pointing to the list some of the drawings has no photo, and then pressing this button will create a photo of them. You can select several at once and the application will load them one by one. You only have to indicate the insertion point of each block..

Button **See Pictures** opens another window with photos of the drawings in the working directory.



Clicking on one of these images, the drawing is inserted in the current plane..

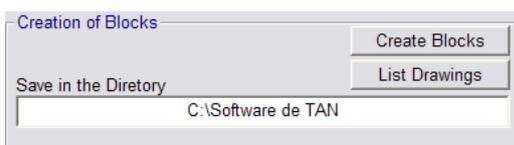
We will have to indicate the insertion point and angle of rotation.

If it's a block with attributes, we must also give the values for them.

The result is the same as that achieved with the **Insert block** button. In this case, however, we can see not only the picture of the marked object in the list, but of all objects in the directory.

### Creating new blocks

The first thing to do is specify the directory, folder, we want to store them.



The remaining process is exactly like with AutoCAD :

1. Draw objects.
2. Define the attributes and properties.
3. Click the button **Create Blocks**.
4. The application asked to point out the objects and attributes that make up a name for the block and its insertion point.

As mentioned previously, the block will be stored in the folder as a *dwg* file of AutoCAD, and a picture of it will be created, allowing easy identification. We do not need to remember his name because the image will be enough.

## 4 CHANGES IN GRAPHICAL OBJECTS

Icon in *AutoCAD* Tool Bar



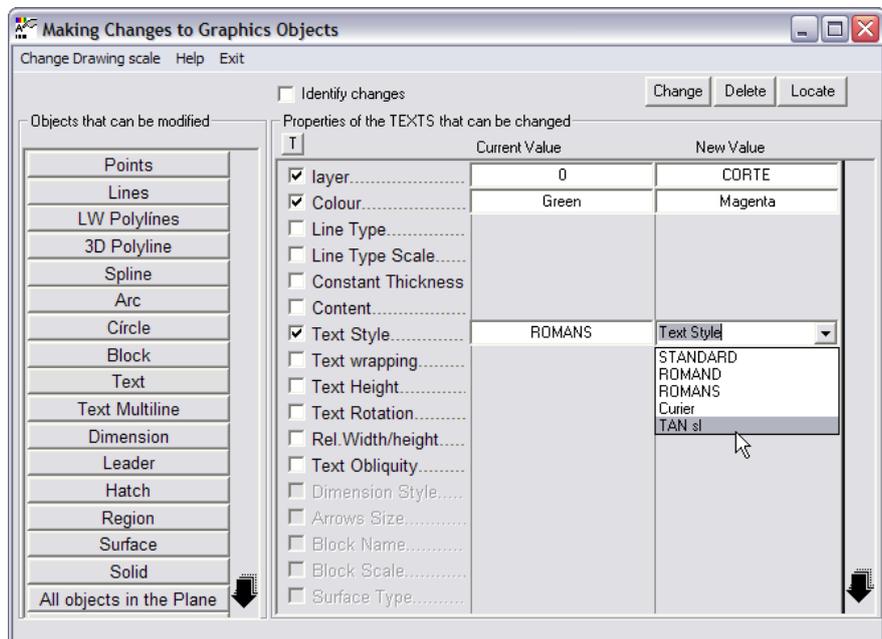
The general appearance of the module is as follows :

The left column is a list of all graphical objects that can exist on a drawing.

The right column represents the properties that can be changed on each object.

Not all objects have the same properties so that, when you press the button of one of the objects in the list, check boxes with one or other properties are activated on the right side of the window.

After choosing an object, you have to activate the cells whose properties you want to change. At that time opened a box to indicate the current value and the new value of the property.



Double-clicking on them, become visible popup windows that let you choose from among those existing in the drawing, the present value of the property and the new value to be taken.

Once selected you need, you can accomplish three things (the example corresponds to the selection of the figure. Previously has been pressed the Text button on the left column)

- **Change** : all text in layer ZERO, BLUE, ROMANS style, will be passed to the layer TRACE, RED will be drawn and his style will be changed to Arial. Other texts that do not meet these three conditions, and of course all other objects in your drawing will not be modified.
- **Delete** : all text in layer ZERO, BLUE and ROMANS style, will be deleted from the drawing.
- **Locate** : all text in layer ZERO, BLUE and style ROMANS, shall be marked by a line from the origin to its insertion point

### Header Menu Change Drawing scale

The general plan of the whole and, perhaps, someone else, are draw in AutoCAD at SCALE 1: 1. However they are printed at 1/50, for example. Therefore, to start them, you must choose a specific size for text, for dimensions and arrows, etc., so they are readable to the scale on which to print the drawing. (The template drawing "DwgToolModel.dwg" already has all those parameters ready to print to scale : 1/50)

If for any reason, you had to change the scale originally envisaged, would have to change all that size because if not done, would a plane slightly presentable. The work, done by hand, is endless. With this option the application will do the job automatically. In a window, which opens to the effect, we will indicate the current scale and the new one.

#### NOTE :

By loading this module, the application reads all text styles, line styles, of dimensions, layer name, etc.. existing in the current plane. This way you can prepare lists of things modifiable (*something that will slow the module load process when the drawing contains much information*)

This list of properties may change from one drawing to another. So once you have completed work on a plane, you must close the form, open another drawing to modify and restart the module.

This module provides support for drawing the parts of the current block in the normal work in 2D but also provides great help in making 3D models.

The initial window of the module is as follows :

- Multiple drawing aids, creation of profile passes, ties, consoles, etc. automatically.
- Create parametric parts: by giving different values to its parameters, totally changes the geometry of the part.

The drawing of new parts is supported by the building standards that are assigned to the current block.

Therefore, if these standards do not exist, should be created at this time. You may not create any piece without them.

The header menu Standards opens a window needed to create them.

The header toolbar only has, until now, 5 options:

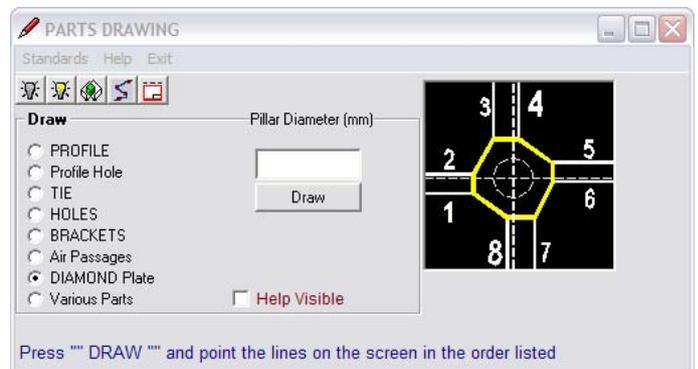
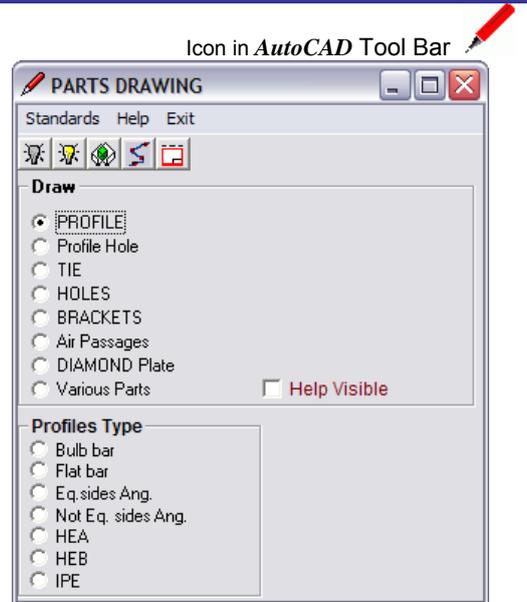
-  hides the objects that we have pointed on the screen.
-  (only active when there are hidden objects) returns to make visible the hidden objects
-  checks if two solid objects, identified in the screen, interfere with each other
-  Several operations with splines
-  Inserts the "template" file of DwgTools, which contains styles of text, dimensions, lines, blocks, etc..

Noting some of the options, new windows are activated to complete the necessary data for drawing.

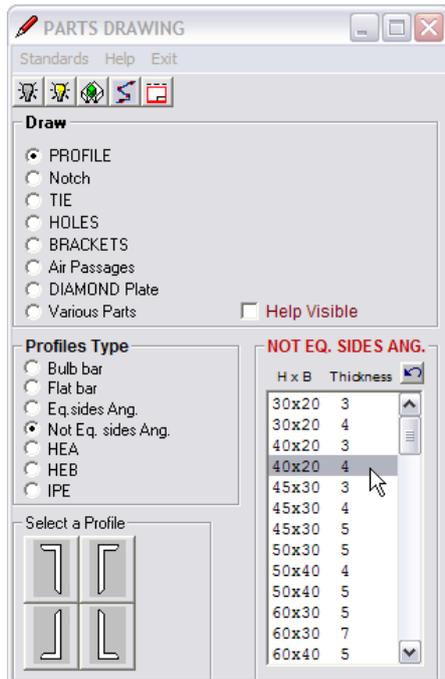
Most of these pieces are based on existing lines in the current drawing. In that case, we give indications of the lines that are necessary and in what order we should point them out.

The example of figure is to draw a diamond plate to join the wing plates of a beam and a deck girder. You must indicate the diameter of the pillar to be supporting it, Click Draw button and point out on the screen the lines representing the two wings, in the order shown in the figure.

The application will draw the corresponding diamond plate.



## PROFILE



The option PROFILE will draw the cross section of a profile or flat bar. When selected, the window expands with additional dialogue box where you can select the type and size of the required profile.

All types of normal profiles in shipbuilding are covered.

Depending on the chosen type, a list of standard dimensions of the type is displayed.

You must choose one of them, and then press the button that indicates which position should be drawn.

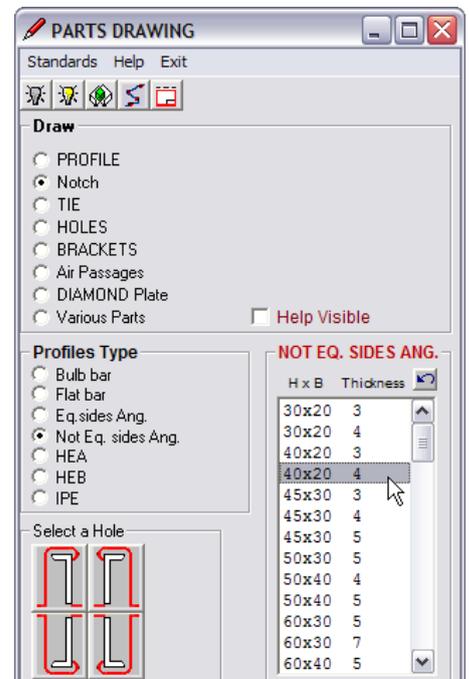
The application only will ask us to indicate the insertion point on the screen. It will draw the selected section in true scale.

Drawing a **NOTCH** is exactly the same :

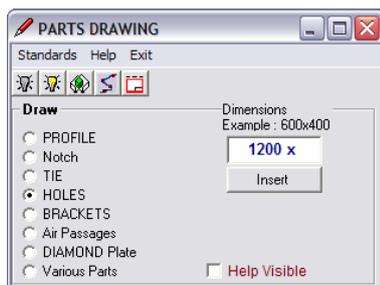
- Select the type, size and position of the profile and press the button.
- Mark the insertion point which has to be that of a previously defined profile.

## TIE

The same procedure is used to draw a tight tie.



## HOLES



To draw MAN holes, only to indicate, as shown in the image, its dimensions and press **Insert**

As always, it is noted on the screen the point where we want to insert.

The application will draw an oval in OVERALL dimensions: those marked on the box

The first dimension is the height

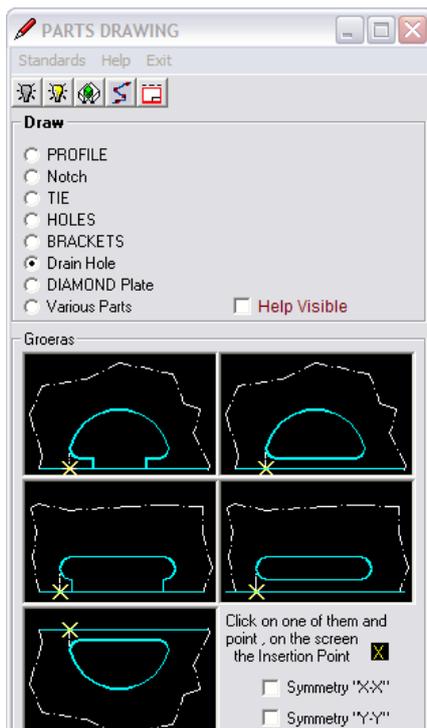
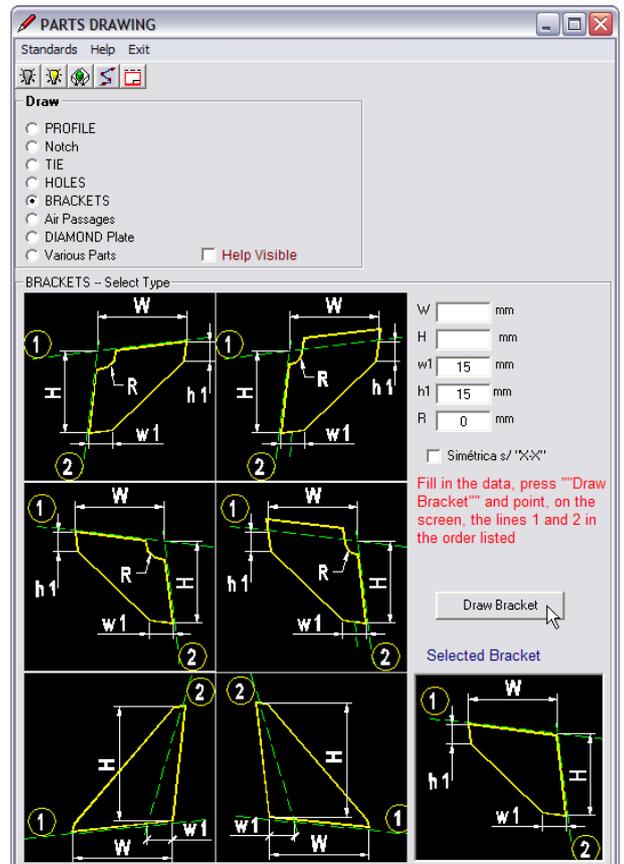
## BRACKETS

For the draft of consoles, once chosen the type of the same, among the 6 figures on the left, they will be shown the boxes will need to enter the data that define it.

The default values adopted for heels and radii are those given in Constructive Standards, but can be changed by acting on the boxes.

Once you press the **Draw Bracket** button, you should note on the screen the lines that limit the console (baseline), in the order that each figure shows.

Depending, of course, the position of baselines, the console will result in different shapes but always adapted to such lines.



## DRAIN - AIR HOLES

Just point out the drain/air hole figure and indicate on the screen the insertion point.

The dimensions required to create each of them has already been defined in one of the panels of **Constructive Standards**.

In the image of each one is shown the relative position of the insertion point in each case.

Checking the **Symmetry** boxes, the application to draw the symmetrical drain hole of the one marked on the figure.

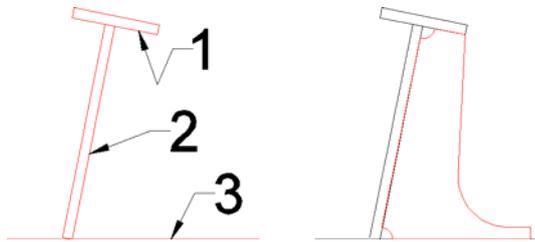
## VARIOUS PARTS

Attached window opens in which you can select the type of piece you need to draw. There are 4 different types.

By selecting one of them, opens another figure with a series of boxes that allow us to enter the parameters that define the part.

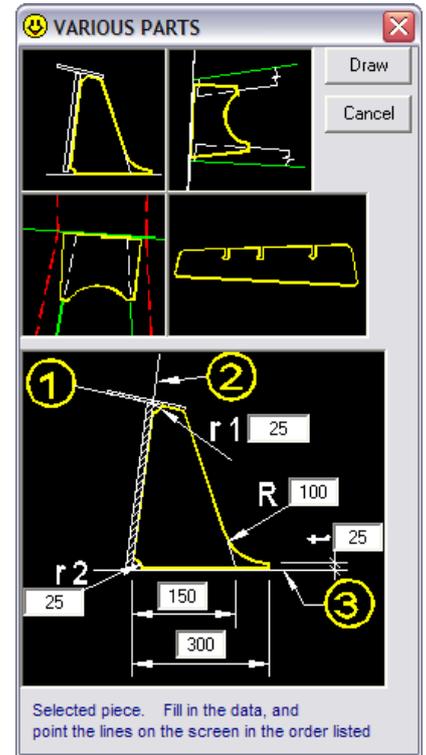
Press **Draw** button, and point on the screen base lines that will support the new part. Should be stated in the order indicated in each figure by the numbers circled.

The example shows the lines as given in the plane that we are creating and the final result.



elements in the drawing

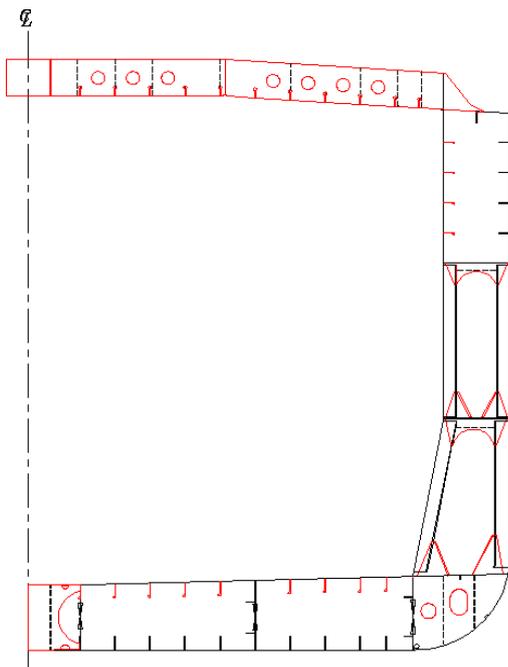
part created automatically by the application




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IN ALL CASES, LABELS WILL APPEAR IN CORRESPONDING WINDOWS WITH SEVERAL MESSAGES INDICATING WHAT TO DO AND WHAT INFORMATION MUST BE PROVIDED FOR EACH ITEM.

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EXAMPLE :

All items in red can be drawn automatically by the application.

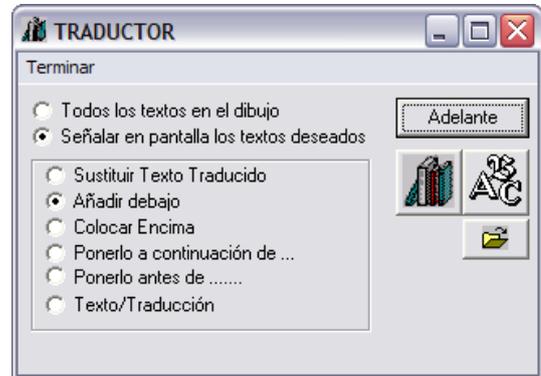


Translates, from Spanish to English, texts contained in the active drawing in the AutoCAD screen.

Can be done :

- Automatically all existing texts in the drawing, or
- Pointing out only several texts, or
- Pointing to a single text..

In any case, the new text replaces the old one or will be placed in either position on the same, according to the option you have chosen.



The program has an extensive collection of texts already translated.

This may be :

- Individual word
- Complete phrases.
- Abbreviations. Example: "M.D "(instead of main deck)

When we point a text on the screen, the application searches in its database and write the translation in AutoCAD screen.



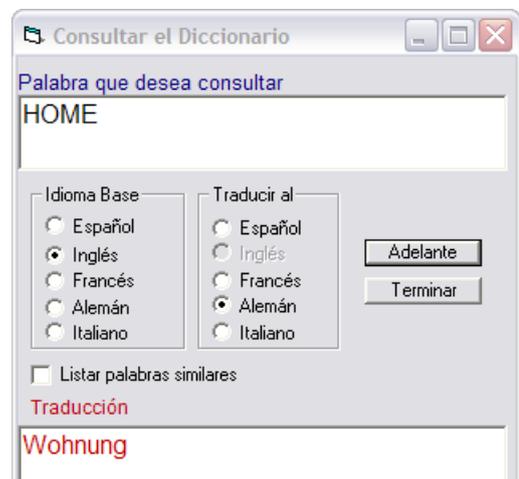
If it is a new word, not previously existing in the database, the program will ask us to made the translation.

The **Adelante** button will write it on the drawing.

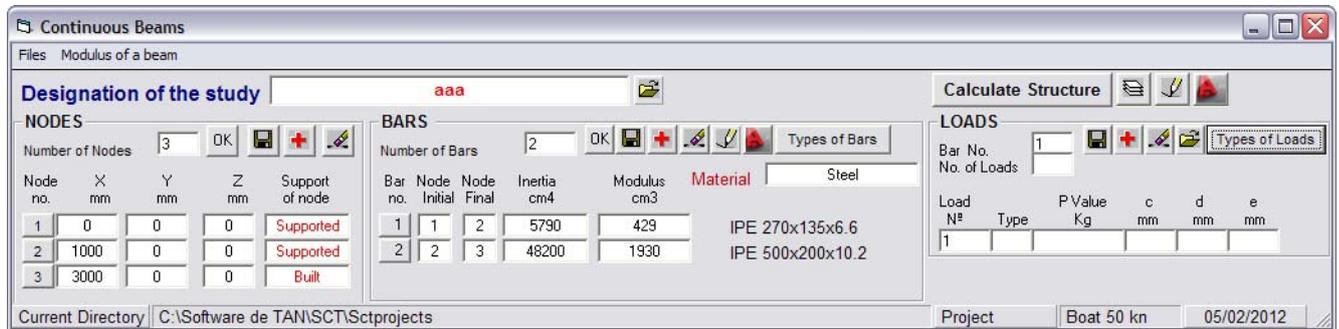
The **Agregar** button will cause the word, or phrase, is incorporated into the database.

It is important, therefore, to call the same things always by the same name, so that the application can process them. If once we wrote M.D. and save the translation into the database, and the next time the call it Main D., the application will not find the translation of the second one.

Furthermore, with the button  we access to several, no maritime, dictionaries, that are included in the software, and that allow us multiple translations between 5 languages.



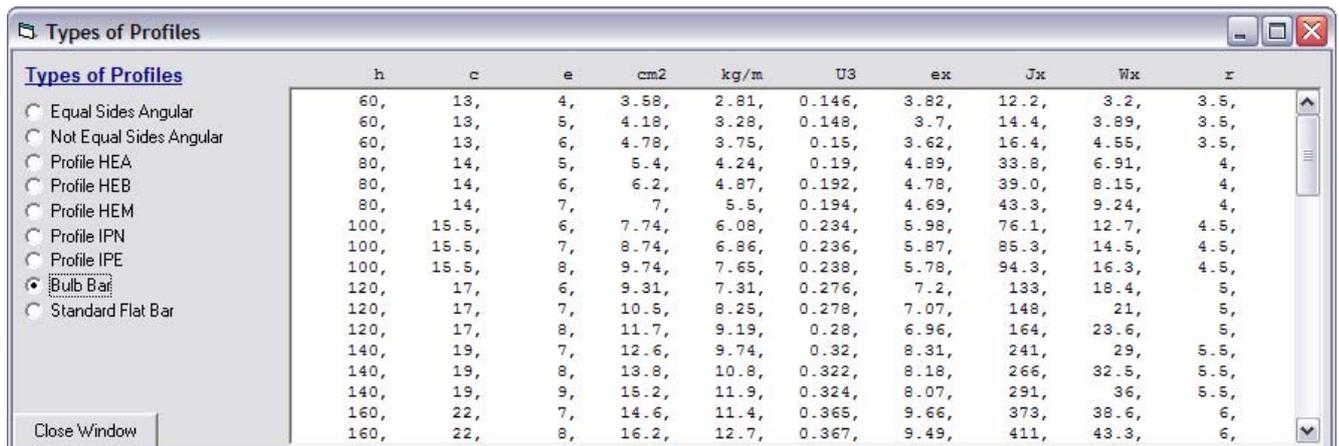
.Aspect of the window



Can be raised structures formed by various beams consisting of any type of profile. Reactions are calculated in the various nodes, bending moments and shear forces in each beam.

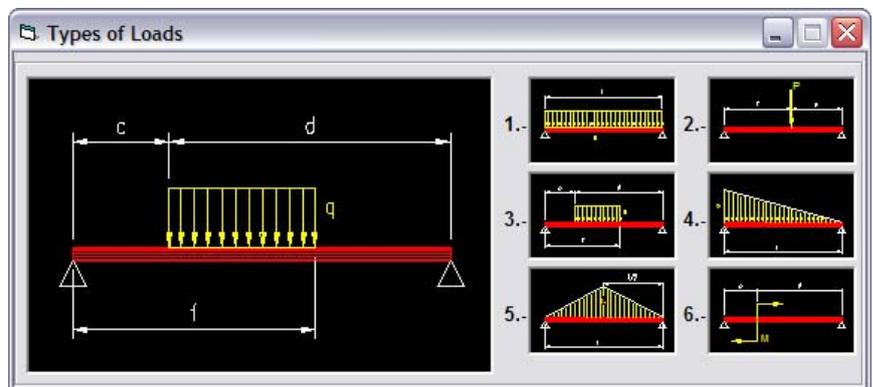
You can study continuous beams or simple frames on a plane or in 3D.

The program provides in its database most types of profiles used in marine structures so that the incorporation of their properties, inertia, module, etc., is very comfortable.



For other non-standard type profiles, the application will help u s by calculating the physical properties of any cross section we have previously drawn in AutoCAD.

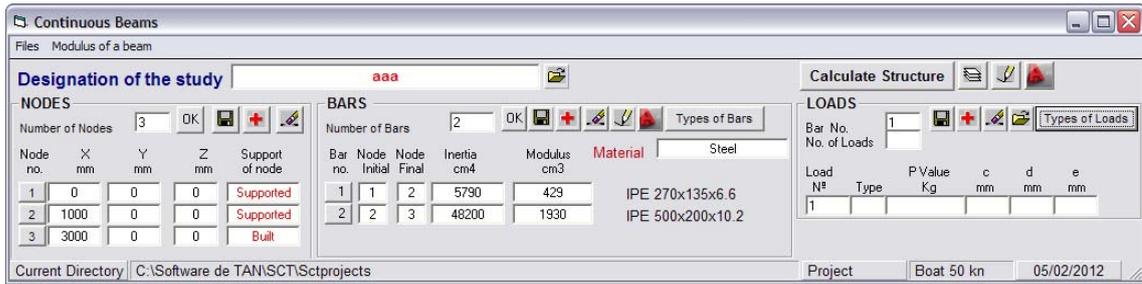
Loads types, applicable to any of the beams, that are n ormal for this type of studies.



The results appear in a special window of the application, in a table with the values and graphically. Also available in AutoCAD, where the application automatically draws them.



## REVIEW OF THE WINDOW OF WORK

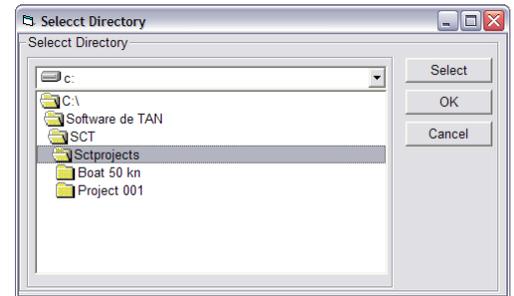


The bottom status bar indicates the directory and the project you're working



By clicking on the button, a window with a directory tree, similar to Windows Explorer, change directory if needed.

Now we can click on this button that opens a dialog box with existing projects in the above directory. Now we can choose the project on which we need to work.



Within the same project can be several studies.

The current study is the one that appears in the text box labeled **Designation of the study**

The header of the window contains several menus

### Files

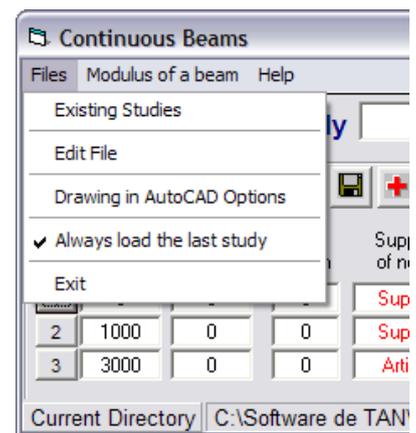
**Existing Study** : allows you to load on the screen data from a study conducted previously and stored in the current project directory.

**Edit File** : of any existing study, in order to viewing or manipulating it.

**Options for drawing in AutoCAD** : results can be drawn automatically, if desired, in AutoCAD. With this option, choose the colors and line types for drawing beams, Bending Moment, etc..

**Always load the last study** : when starting, the module will load the last study in which we worked.

**Exit** : closes the modulus Beams.



**Module of a Beam** : calculates the modulus, inertia and other physical properties of the cross section of any beam we have previously drawn in AutoCAD.

The module of the Main Transversal Frame can be calculated with this option.

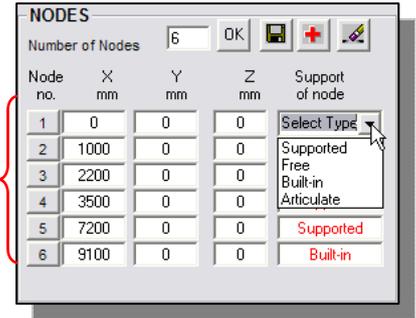
## WORKING WITH NODES

In the box labeled **Number of Nodes** indicated the total number of nodes that form the structure and click **OK**

4 cells per node appear, 3 of them to indicate the coordinates (X, Y, Z) of each of the nodes. The 4th box is used to indicate the type of support of each. Clicking on this a drop-down list appears with four options. By clicking on it, the box corresponding to the node will be completed.

Once completed the data from all nodes, should be saved

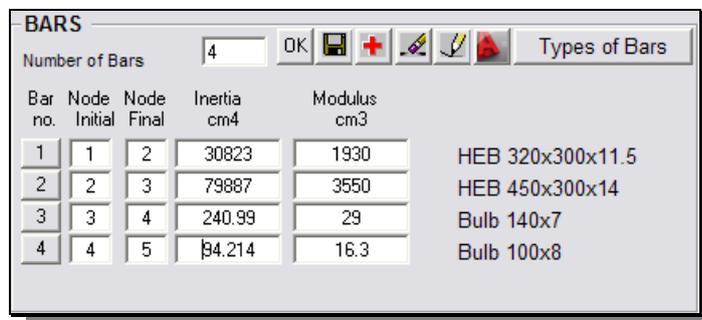
Until you save this data, it will not be activated the check boxes to define the bars. (see below)



Adds nodes in the existing list. Click on the button before of which we need to insert it, then we press **+**.

Deletes a node. Press this button and then the button with the number of the node we want to delete..

## DEFINITION OF BARS

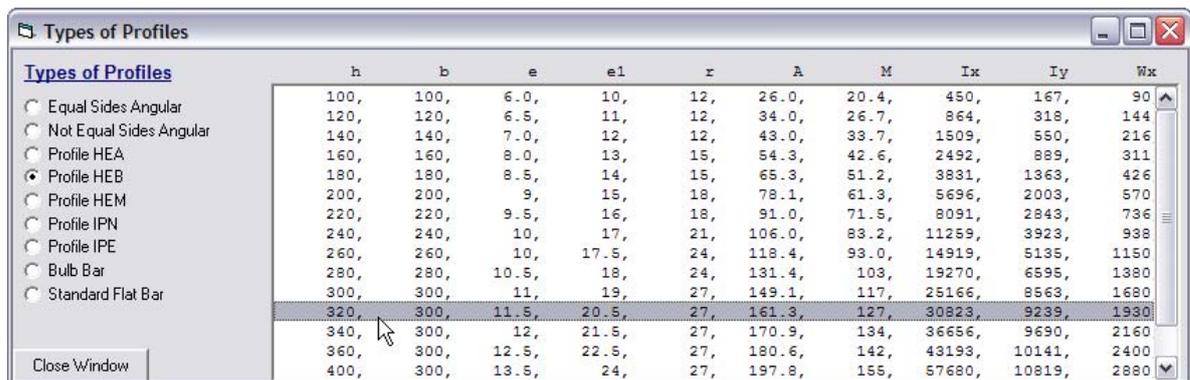


Indicate the number of beams that make up the structure and click **OK**.

4 boxes appear for each beam, so that to put the ends nodes of each one, its inertia and its module.

The inertia and modulus, when it is very special beams, must introduce by hand, once they have been calculated. For this purpose you can use the header menu seen above.

In the case of beams formed by standard profiles, the method is greatly simplified: Acting on the button **Type of Bar**, it will be displayed a window with a list of the standard profiles included in the database of the application.



The procedure is as follows:

1. We mark the on the corresponding button of the beam **1** o **2**, etc.
2. In the types list on the left note the desired type.
3. In the profile list of such type, note the one with appropriate dimensions.

Boxes of inertia and modulus of the beam are filled with the values of selected profile.

Mark another button or .. **2**.... and repeat the process.

**DO NOT FORGET TO SAVE THE BEAMS.**  Until you do, check boxes to define the loads will not be activated.

## BUTTONS OF THE WINDOW **BARS**



After entering the number of bars, this button prepares and numbers the boxes for every bars data.



Stores the data of the bars. Only after that will activate the check boxes to define the loads of bars.



Add a new bar before the button **1** o **2**, etc that we must mark before pressing this button



Deletes the bar whose number you have marked before.



Draw the structure, without loads, in the auxiliary drawing window of the modulus.



Draw the structure, without loads, in AutoCAD

Types of Bars

Open the window with the standard profile types, as seen on the previous page.

## DEFINITIO OF LOADS ON EACH BAR

After saving the configuration of the bars, the window is activated to define the loads on each one of them.

For each bar we have to indicate the number of loads acting on it

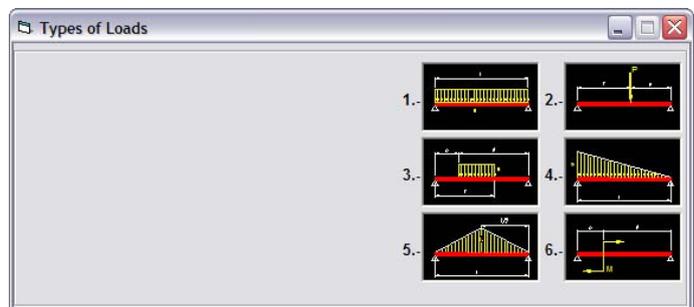
It will be shown on which bar we are working with and the number of the load we have to define.

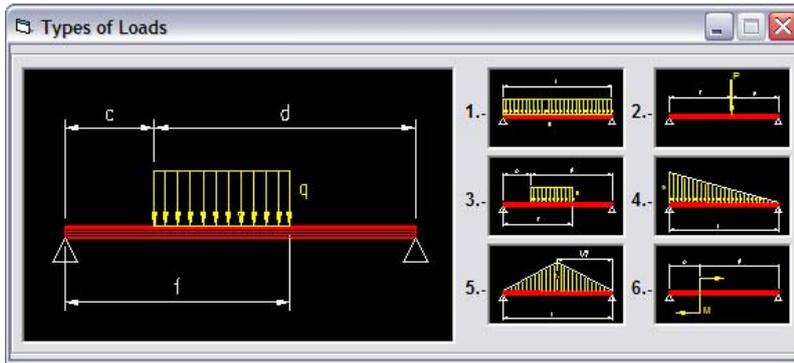
Load N°	Type	P Value Kg	c mm	d mm	e mm
4					

The type of load is indicated by pressing the button **Types of Loads**

Following window is shown.

Here we click on the desi red type of load. The window will now show us the dimensions necessary to define it, while the type is written in the second section of the window LOADS.





These dimensions, as well as the value of the load, we introduce them by hand in the window boxes LOADS.

Each load, once defined, must be **SAVED**

This done, the application moves to the next load on the current bar or to the next bar.

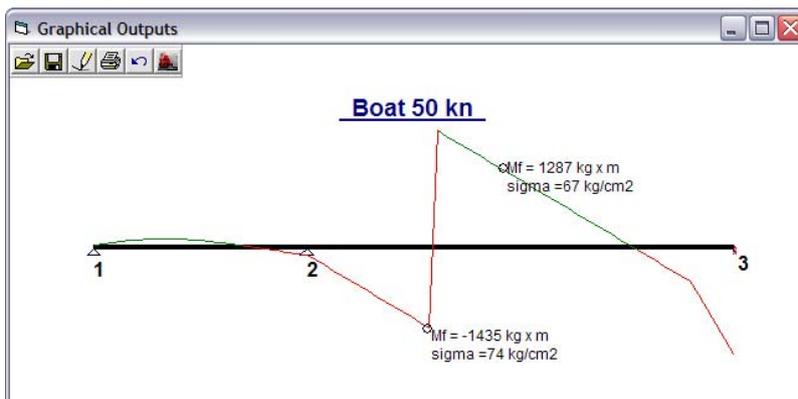
### BUTTONS ON THE WINDOW **LOADS**

- Saves data of the load just defined.
- Adds a new load on the current bar.
- Deletes current load.
- Opens an auxiliary window with loads defined and saved, so far.
- Tipos de Cargas** Open the window with types of loads.

Having done all this, you press the button

The results appear in two windows,

A graphical window



- Opens another file of results
- Saves changes.
- Redraws curves
- Print
- Exit
- Draws bars and curves in AutoCAD

Pointing the cursor over a point on the curve and pressing the right mouse button, the settings of the Bending moment and tension at that point in the beam will appear.

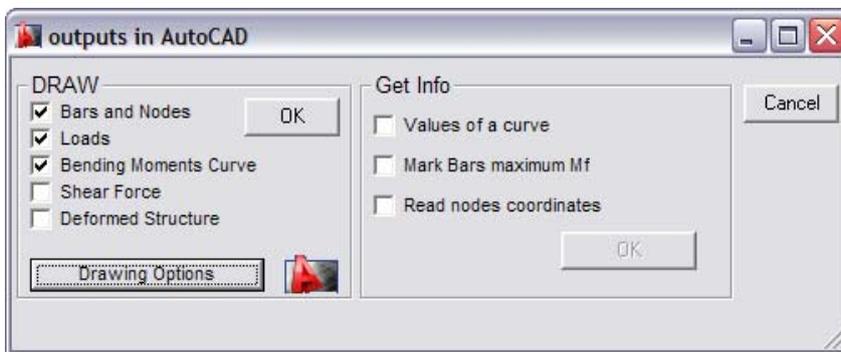
And another text window

Data and Outputs										
Save as Types of Bars										
Boat 50 kn : aaa										
Node No	X (cm)	Y (cm)	Z (cm)	Support						
1	0	0	0	Supported						
2	1000	0	0	Supported						
3	3000	0	0	Built						
Steel										
Bar No	InNode	FinNode	Isupport	Fsupport	Bar type	Length	Inertia	Rigidity(I/l)	Modulus	
1	1	2	Supported	Supported	IFE 270x135x6.6	1000	5790	5.79	429	
2	2	3	Supported	Built	IFE 500x200x10.2	2000	48200	24.1	1930	
Mf nodes (kg x m)	Reactions on nodes (kg)			Max.Bars Bend.Mmnt (kg x m)						
Mf 1= 0	R 1= 281			Barra 1= -169 kg x m a 1000 mm del nudo 1 Sigma max. =-39.4 kg/cm2						
Mf 2= -169	R 2= -2495			Barra 2= 1953 kg x m a 612 mm del nudo 2 Sigma max. = 101.2 kg/cm2						
Mf 3= -1858	R 3= 6169									

-  Opens another file of results
-  Save changes
-  Looks for a text on the screen
-  Closes the window
-  Passes results into MS Word
-  Increases fonts size.
-  Print

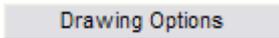
 Draw bars and curves in AutoCAD

Pressing this button, the following window will be shown, where you select what you want to draw.



Pressing  curves, whose cells have been selected, a re drawn.

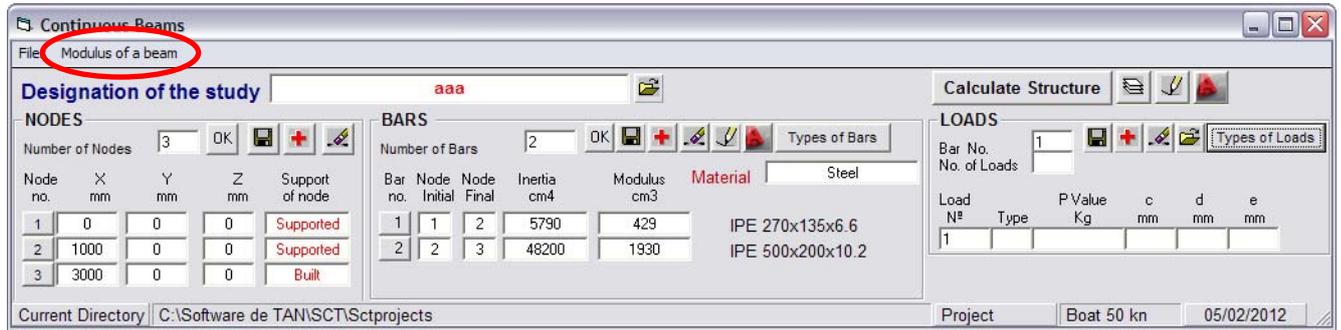
Once the drawing is done, the right options allow us to obtain information from its objects.

The button  opens the window on the right in which indicate the colors and types of lines for each of the elements in the drawing.

Drawing in AutoCAD Options		
Item	Color	Type of Line
Vigas	Red	continua
Cargas Puntuales	Cyan	continua
Carga Uniforme	Yellow	rayitas
M. Flectores	Yellow	rayitas
Esf. Cortantes	Green	continua
Close Window		Save

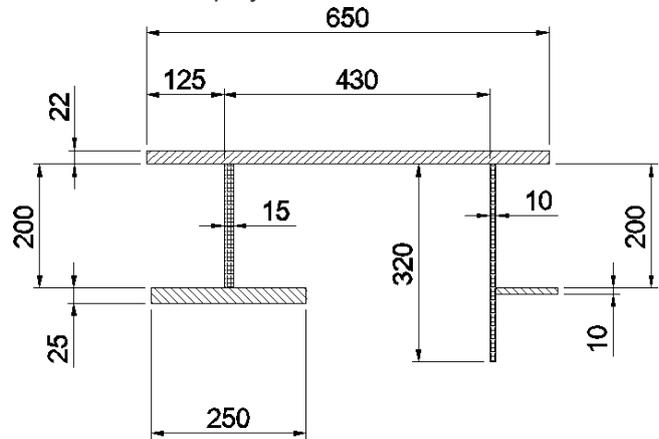
## COMPUTATION OF THE MODULE OF A BEAM

It is an option that appears in the header menu of the application window.

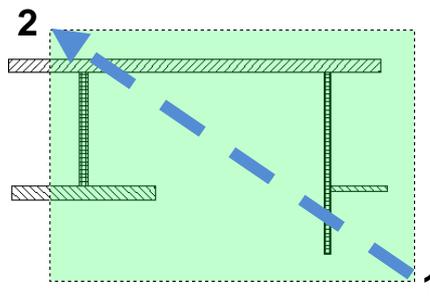


The beam in question must have been previously drawn in AutoCAD. Must be drawn in true scale, dimensions in millimeters, and each of its elements must be a closed polyline.

1. Draw the beam of the figure

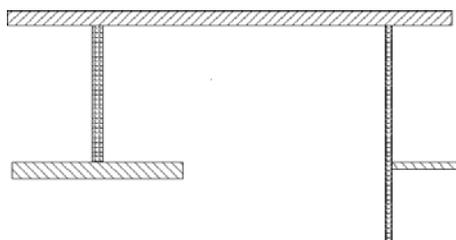


2. Press on the menu **Module of a beam**.
3. In the AutoCAD screen select all elements of the beam. For example, click on point 1 and, without releasing the mouse button, move the cursor to point 2



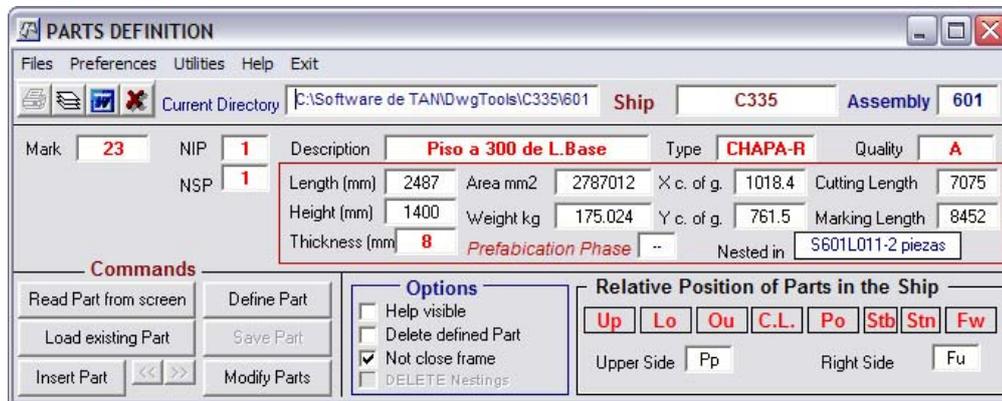
4. Now you are prompted to enter another point on the screen where you want to insert the results

**Final result :**

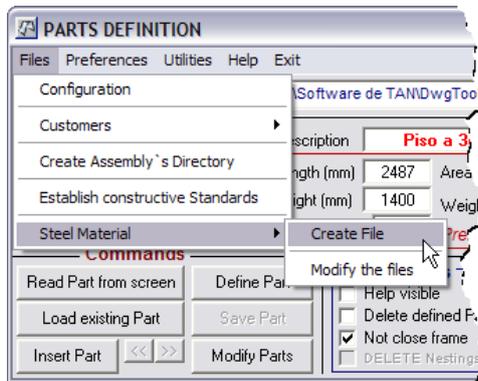


**Area = 277.5 cm<sup>2</sup>**  
**Neutral Axis = 10.08 cm a parte sup.**  
**Inertia = 30357.0927 cm<sup>4</sup>**  
**Module to the upper edge of the plate = 3010.657 cm<sup>3</sup>**  
**Module to the lowest edge = 1258.753 cm<sup>3</sup>**  
**Weigth section = 217.837 kg/ml**

Aspect of the module's window :



**Header Menus :**



**Configuration**, opens a window to introduce User Name and his Authorization Code.

**Customers** to add or modify customer data or customer to select the current job.

**Create Assembly's Directory**, in the case of a new block within the current boat.

**Establish constructive Standards** to create or modify this file.

**Steel Material**, opens a window that will detail the number of plates, by thicknesses and grades intended for the block. Later, as we nesting parts, the application will count down the listing and we can always know how many we have at any moment.

The last part of the menu plays the most recent directory. Clicking on one of them, will become the current customer and ship.

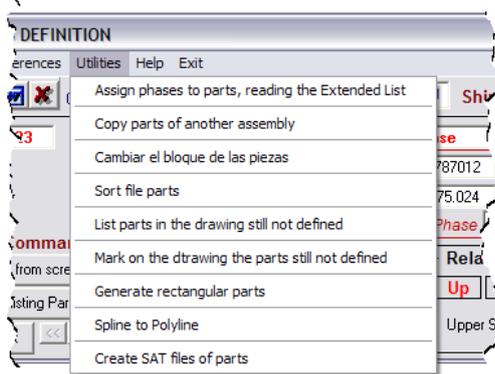


The **Preferences** menu opens a dialog box where you can specify colors to use when creating new parts: color for the outline of the part and color for lines indicating the marks, ie traces of other parts on the current one.

Parts are typically defined by means of polylines, which must be closed. In some cases you may want to define it by a spline.

However, we must bear in mind that cutting machines only understand arcs and straight sections. Therefore the final contours of the pieces must be always formed by polylines.

One option that is in the module is to automatically convert a spline into a polyline, which is a collection of straight sections forming a single closed object. Unless stated otherwise, the application will create polylines with very short runs which is very accurate but consumes lots of memory; file processing for numerical control cutting can be very slow. Even the CNC file size may become larger than the buffer of the cutting machine. Therefore, it is desirable to indicate a higher value for the length of these segments, depending on the experience of each designer. This value is shown in Fig box and pressing the OK button will save the preferences.



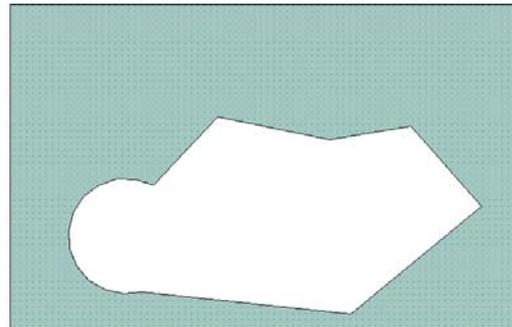
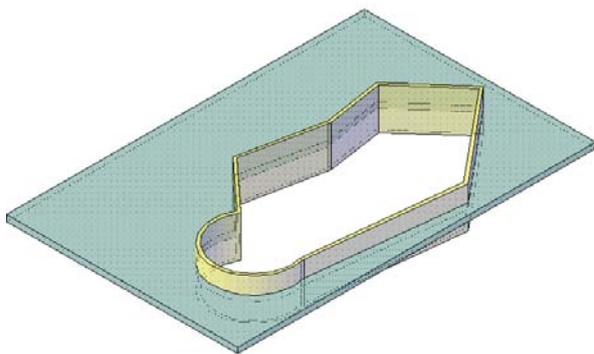
**Assign phases to parts ...** usually, once all the information of the assembly is finished, a technician from the workshop assigns to each piece its prefabrication phase. This assignment, if done in the same .xls listing file created by DwgTools, can be transferred to each of the pieces, completing its information.

**Copy parts of another assembly,** copies parts from an assembly that we have previously defined. Changes in the base part the assembly's code and saves it in the directory of the current assembly. There should not be copied manually as there is no convenient way to change the codes of each part. If needed, the **Change Part's assembly** option, will assign to the parts that we point out on the screen, the current assembly code.

The pieces can be defined in any order and the application will apply to them to the parts file of the assembly by adding parts in the same order in which we define. Although at times DwgTools order this file, if you notice that the order is not correct, the menu **Sort file Parts** will carry out this action.

**List parts in the drawing still not defined** compares the file with parts we are creating with the drawing of the assembly and shows us the missing pieces to be defined in a list that can be printed. The menu at the

**Mark on the drawing the parts still not defined** draws a line from the origin to each of the missing parts.



Let's suppose that we need to develop the bead to the hole of Fig. It is a bead of 350 mm height. If we use the **Generate rectangular parts** menu, the program will develop a rectangular piece of 350 mm in height and total length of the contour of lightening.

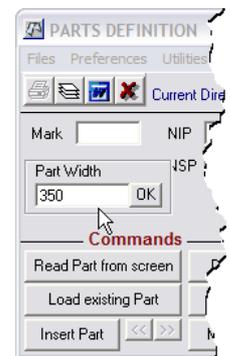


To do this, after pressing the menu, a button appears with the height, by default, will have the bangs (150 in this case)

Right-clicking on the button in the left figure, opens a box where you type out the necessary width, 350.

Then press the **OK** button and note the shape of the hole in the screen.

The application creates a rectangular piece of length equal to the perimeter of the hole.



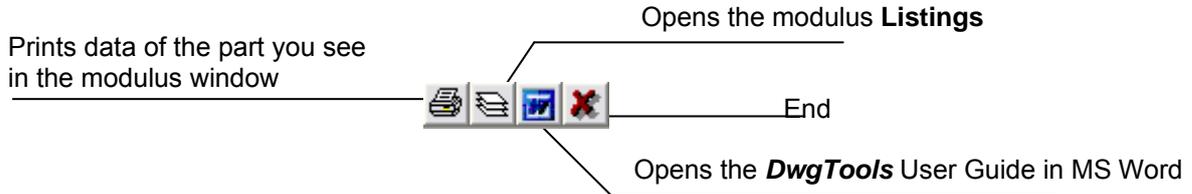
**Spline to Polyline,** transforms any spline into a polyline, polygon composed of straight lines, each section of the length that you specified in **Preferences**.

**Create SAT files of parts,** creates a file of this type for each one of the pieces, with the thickness thereof. These files can be exported to other CAD-CAM application that needs solid models of parts.

**Help** opens, in MS Word, the User Guide of the modulus

**Exit** ends with this modulus

**Toolbar :**

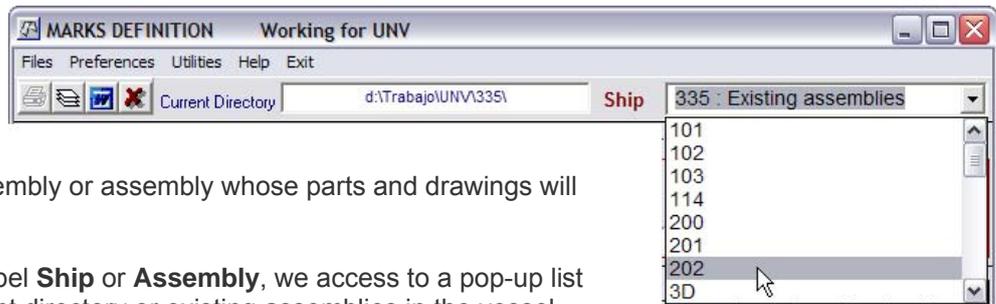


**General Data**



**Current Directory** indicates the full pathname of the directory in which we work. Double clicking on this label, opens a window that will allow you to change the working.directory

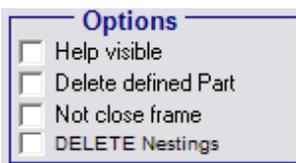
**Ship** is the name of the subdirecto ry, particularly for each boat or work, intended to store their data and assemblies information.



**Assembly** : subassembly or assembly whose parts and drawings will have access..

By clicking on the label **Ship** or **Assembly**, we access to a pop-up list with ships in the client directory or existing assemblies in the vessel, which allows us to choose at all times the correct directory..

**Options**



**Help Visible** : when active, the buttons do not perform the command assigned to them but, pressing each one of them, we get some explanations about what they do and how to get it.

**Delete defined Part** : once defined and saved, the piece is removed from the display. So we know at all times what parts are still not defined.

Every time a piece is defined by default dialog box disappears **Definition pieces**. If we want to remain it visible on screen, we select the **Not close frame** box.



When you build a piece by copying an existing one, also the data (s), nesting (s), in which the base part is nested are copied. **DELETE Nestings** delete this data for the new part.

## Command

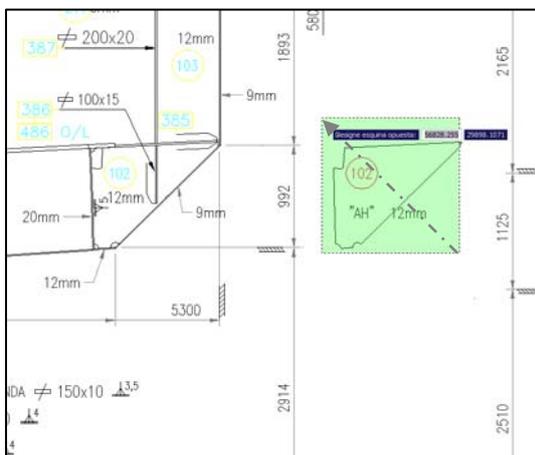


**Read Part from screen** : pointing a part on the screen, from the previously defined, and all its properties are shown. It is useful to define a new part from a similar existing one, because the boxes are filled in automatically. Just change some of them by hand and vary the pattern you need, and save it with another tag number.

**Load existing Part** : the list of defined parts will be shown. Indicates one of them and opens the drawing thereof. The picture we had on the screen disappears. The selected part data are shown in corresponding boxes.

**Insert Part** : same list as above will be shown; pointing to a part, it will be inserted into the current drawing. The current drawing does not disappear, unlike the previous case.

**Define Part** : It should be pointed on the screen all the lines, marks, text, etc. forming part of the piece. The closed contours, internal to the piece, will be considered as lightening. Lines or open contours are considered as "marks".



**DwgTools** calculates the physical properties of the same and fill the boxes. Must be filled in by hand the missing boxes and then click **Save Part** to save the part as a separate drawing.

Although there are many things in the plane, only the objects in the selected set will be considered as elements of the part.

The figure indicated the outline of the piece, the assembly with its mark and a text that indicates the quality of the part and its thickness. With AutoCAD's options "capture" or "window" you can comfortably selected the desired objects.

The shape of the piece **MUST** be a **CLOSED** polyline.

**Delete Part** : deletes an existing part. Deletes all pictures and references of part existing in any other file.

## Relative position of the Parts in the Ship

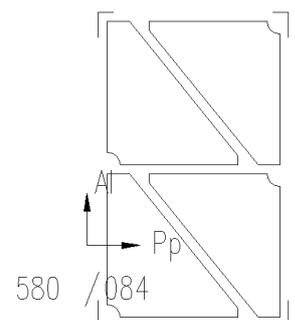
We'll draw each piece in the position that is most comfortable for us. We must, therefore, indicate its actual position on board. We'll make by checking the relative position of the top and right edges of the piece, as we have drawn it.



By clicking on the labels on **Upper Side** or **Right Side** we'll activate them (the box to enter the data will be gray). Then clicking on the labels Up Lo Ou C.L. ... the desired position will be marked.

Code	Meaning
Up	Upper side
Lo	Lower side
Ou	Outer side
C.L.	Center Line
Po	Port side
Stb	Starboard
Stn	Stern side
Fw	Forward side

When you insert each piece in its nesting, a mark appears, according with what we stated at the time to define it, and according to the preceding code symbols, which help you situate it, once cut, in the right position on its assembly..



## Defining Parts.

You must draw the outline of each piece with its holes, drain holes, with the greatest accuracy. Each contour, both exterior and interior (holes), you **MUST** be a CLOSED POLYLINE.

The contours fall within the main contour are interpreted by **DwgTools** as lightening of the piece.



Molded lines will be marked, including any information you want to incorporate into the design of the part. In particular **MUST** be provided in one piece of text the thickness and quality of the piece (see figure)

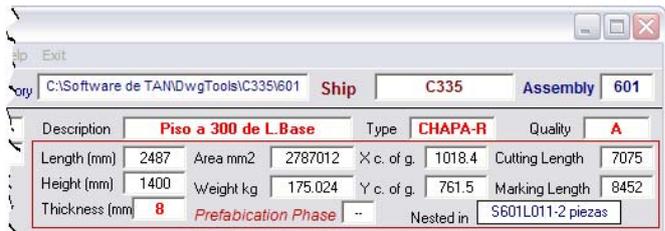
**DwgTools** has specific blocks to give the brand of each piece. The part numbering is done according to the following criteria:

Part Mark No.	Part Type
1 to 299	parts obtained from plates
300 to 499	flat bar
500 to 699	profiles (bulb, angle, etc.)
700 to 999	pipes, pillars

Thus **DwgTools** recognizes the type of part through its tag number.

Pressing on **Define Part** button, we are required to point on display all lines, text, etc. comprising one piece. All useful information is displayed will be read by **DwgTools** saving us having to fill the boxes with the data. It was also automatically calculate the area, weight and c. of g. of the piece and the cutting length as well as the path length of marking. This will appear in the dialog boxes.

The boxes inside the red box are filled in automatically, as well as the brand and quality of the piece, if specified in the drawing.



The relative position of the piece should be noted by hand, indicating whether the top of the piece, as shown in the drawing, corresponds to the position up, down, etc.. on board and if its right side is forward, aft, etc..

Once we agree with everything we need to **Save Part**, to create its drawing and give it its own. If any important data is missing, the program will ask you to enter it.

**NOTE :** **DwgTools** generates a toolbar on the AutoCAD screen, allowing to directly insert some blocks, useful in part definition.

Icono	Action
	inserts the drawing template with entire set of blocks
	inserts the block <b>thickness</b>
	inserts the symbol <b>flat bar</b>
	inserts the symbol <b>bulb bar</b>
	gives <b>number to marks</b>
	Indicate <b>relative position</b> of the part
	gives <b>number to trofiles and flat bars</b>
	<b>automatic welding</b> of plates
	<b>non automatic</b> welding of plates, with edges preparation
	T welding <b>throat</b>
	auxiliary <b>arrows</b>

Some blocks have attributes that must be filled in by hand at the time of insertion.

To do this, previously, insert the drawing model in the current plane, like any other block.

In any picture that we are we can insert the drawing model, pressing . It has not only blocks as mentioned but all line types, dimensions, fonts, etc.. necessary for the delineation of any plane. It acts as a "template" drawing.

On several occasions we have said that, when making a choice, a list of parts will be shown. This list is presented in a window that looks like this

**LIST of PARTS in the ASSEMBLY 611**

Options

Parts Filtered 54  
Weight = 9302.028 kg

Mark	Designation	Qty	Length x Width x h	No P	Weight (kg)
10	Forro Pantoque	A	1674 x 1689 9	1+1	167.054
14	Quilla Plana	A	2859 x 500 9	1+0	100.98
15	Forro Fondo	A	2858 x 2131 9	1+1	414.876
16	Forro Fondo	A	1704 x 2047 9	1+1	241.211
18	Forro Costado	A	2350 x 2149 9	1+1	292.69
21	Piso a 3500 de L.Base, en Er.	A	7238 x 2392 8	1+0	929.249
22	Piso a 3500 de L.Base, en Er.	A	2450 x 2400 8	1+0	354.408
23	Piso inclinado bajo eje	A	1850 x 1188 8	1+1	137.992
30	Plancha de cubierta	A	2350 x 1325 8	1+1	195.543
31	Plancha de Cta, Babor	A	2350 x 2650 8	1+0	265.012
34	Plancha de cubierta en Er.	A	2350 x 2650 8	1+0	391.087
35	Piso a 3500 de L.B, Er	A	912 x 610 8	1+0	34.937
41	Vagra en L.C. e/c5 y 9	A	2392 x 1945 8	1+0	240.596
48	Choque a 950 de L.C. e/c6-7	A	592 x 1355 8	1+1	47.619
49	Choque a 950 de L.C. e/c 8-9	A	338 x 1783 8	1+1	32.8
62	Choque a 2400 de L.C. e/c 5-6	A	592 x 1226 8	1+1	42.935
63	Choque a 2400 de L.C. e/c 8-9	A	300 x 385 8	1+1	3.168
72	Choque a 4200 de L.C. e/c 5-6	A	592 x 1065 8	1+1	37.159
76	Choque a 3600 de L.C. e/c 5-6	A	592 x 1119 8	1+1	39.091
115	Varenga Cd. 6	A	1788 x 1431 8	1+1	127.567
116	Varenga Cd. 6	A	1192 x 1288 8	1+1	66.13
117	Varenga Cd. 6	A	2300 x 1181 8	1+1	96.903
118	Varenga Cd. 7	A	1788 x 1596 8	1+1	145.694
119	Varenga Cd. 7	A	1192 x 1251 8	1+1	63.132
120	Varenga Cd. 7	A	2300 x 1330 8	1+1	117.673
121	Varenga Cd. 8	A	1788 x 1769 8	1+1	164.091
122	Varenga Cd. 8	A	350 x 339 8	1+1	4.026
123	Varenga Cd. 8	A	2300 x 1486 8	1+1	139.539
125	Plancha Mro. 9	A	721 x 4153 8	1+1	186.285

Pieza N° 115

**FILTERS for the Listing**

- Length =< of
- Length => of
- Width =< of
- Width => of
- Thcknss =< of
- Thcknss => of
- Qty
- Sub-Assemblies
- Already nested
- Not nested

Accept Clear All Close

**NOTE** : in the figure all options are open, but NOT always all of them are visible

It usually appears defined parts list so far, with their main features.

Pointing to one in the list we will get also the sketch of the part on the small screen to the right.

Clicking the right mouse button on the part in the list, properties boxes, in the main window of the module, will be filled. They see ALL the properties of the part and the cliche (s) in which it was nested.

Depending on the item that has appeared, the button on the top right will take one aspect or another, to continue the initial command.

Header menu **Options** will display a box where we can select a series of filters for parts. This means that the list will only show those parts that meet the chosen filter. In the figure we have indicated the parts with thickness not less than 7 mm and only these are listed

## Tool Bar



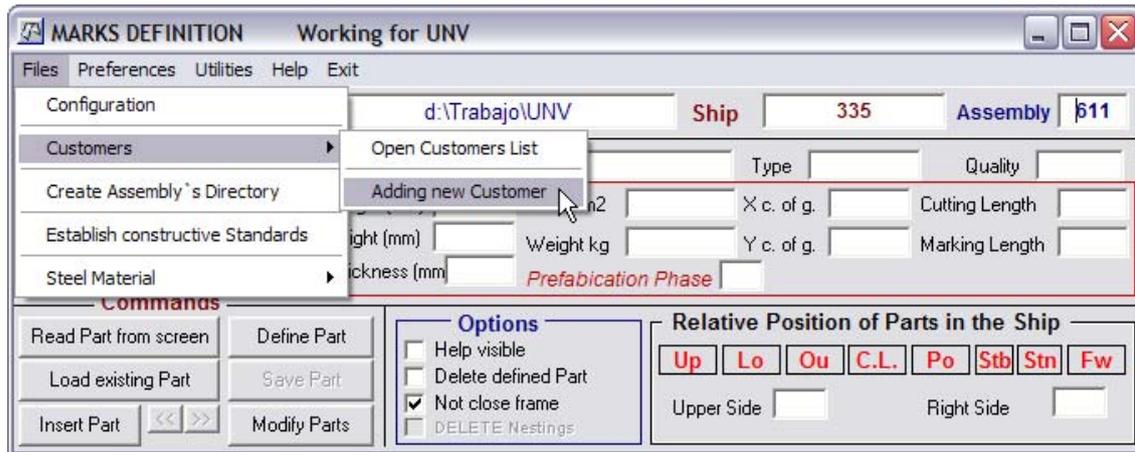
- Does the same effect as pressing the right mouse button on a part of the list. Their properties appear in the boxes of the main module.
- Prints the list you see on screen. (For a full parts listing : go to the module LISTINGS)
- Deleted from the database, the piece who have previously been pointed.
- Closes the window with the parts list

## 9.1 WORKING WITH PARTS

DwgTools organization considers, in the module PARTS, that every potential customer has a specific directory for it, in which will be stored, in different folders, their different ships, and within each of these will create a specific folder for each one of the building assemblies.

Our own shipyard, if this were the case, will be a "CUSTOMER" in this organization.

Therefore, when starting the module for the first time, the application tells us that we need to create the customers file.

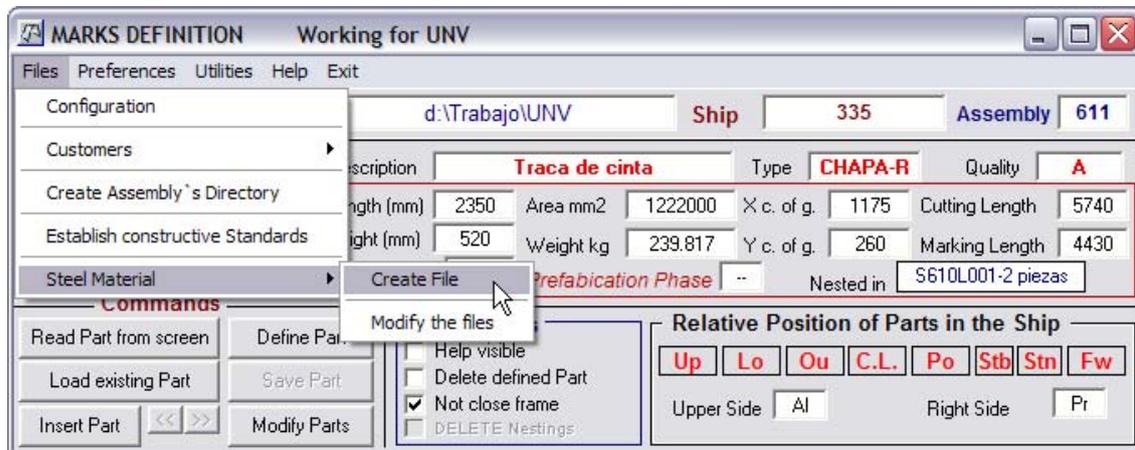


In the video Tutorial of module DwgTools, one can see how to create the directory of a new customer.

Parts module also has an option among their header menus showing this example.

Once created the client and its directory, we will indicate in the boxes prepared for this purpose, identification of the vessel and the assembly you want to work with. The header menu **Create Assembly's Directory** will be created corresponding folders.

Although not essential, it is convenient to establish, at this point, the file with steel material we have planned for the assembly in which we have begun working.



For each block we'll indicate how many plates, dimensions and thickness for each one, we have provided for each destination.

**Example :**

Number	Length	Width	Thickness	Qty	Used	Remaining	Destination
10	8000	2400	9	A	3	7	Shell
7	8000	2400	12	A	7	0	Botton shell
2	8000	2400	25	A	1	1	Bulwark
4	9000	2700	8	A	4	0	Main Deck
1	8000	2500	8	A			Deck at 3500 from B.Line
1	9000	2700	10	A	1	0	Central Girder
2	8000	2400	10	A	2	0	Girders between Fr -5 to 5
1	8000	2500	7	A			Girders Fr. 5/9
6	8000	2500	12	A	5	1	Floors Fr -4 to 4
1	10000	2700	7	A	1	0	Bulkhead Fr. 5
1	10000	2700	8	A	1	0	Bulkhead Fr. 5
5	8000	2400	8	A	4	1	Floors Fr. 6, 7 y 8
1	8000	2400	7	A			Bulkhead Fr. 9

Thus, as we consume plates for every cliche, the system will take note of consumption, and let us know of surplus of material at all times. Also notify us if we try to nest a piece on a plate of different thickness to yours.

In the video Tutorial, in module DwgTools, you can see how to create the file for steel plates.

Done all this, we can begin to define the parts of the assembly.

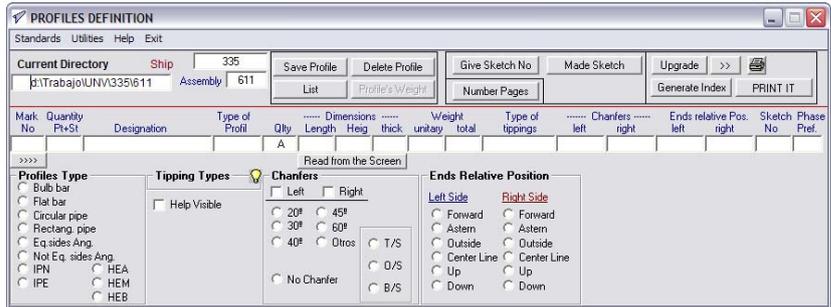
Look at the Video Tutorial to see how to draw a piece with the module DRAWING, and how to process it, with the module PARTS for inclusion in the database of the assembly.

In the AutoCAD toolbar click the icon  that leads to the module for profile definition.

The module window opens with the directory where you've been working previously.

Must be completed for each profile, the data requested in the empty boxes, before pressing the button **Save Profile**

We will see that there are options that facilitate data entry



In the following pages are shown the profiles that we use as an example, with their characteristics, pinching end and chamfers on them.

Profile mark 500 (see picture in next page), is a deck beam, in frame 35, centered in the frame, profile type bulb bar 80 x 5, 1582 mm length, without pinching nor chamfers at its ends. There exists just one as described, none symmetrical to it. Looking at it from the ship's bow its left end goes to starboard and right end towards Port side. So, in **Ends Relative Position** will indicate OUT, OUT

Profile mark 501 is a deck beam located al port side, profile type bulb bar of 80 x 5, and 509 mm length, without pinching nor chamfers at its ends. There exists just one as described, none symmetrical to it. Looking at it from the ship's bow its left end goes to center line and right end towards Port side. So, in **Ends Relative Position** we'll indicate C.Line, OUT

Profile mark 526 is a stringer in longitudinal bulkhead al 2600 mm from C.L., profile type bulb bar 80 x 5, 1470 mm length, with pitched left end. There are two pieces as described ant two more in the symmetrical bulkhead. That's why we write 2+2 (2 equal and 2 symmetric). Looking at them from starboard, their left end is towards the ship's stern and the right end towards the bow. In **Ends Relative Position** we'll indicate Stern, Bow.

In this way, one by one, we'll fulfill boxes for each profile. But before we continue we will learn how to use the facilities that a system DwgTools presents.

We have already discussed that should have finished the drawing of the assembly, before proceeding to define parts, profiles, etc. In our case this level is the 111012.dwg contained in the directory **Tutorial**.

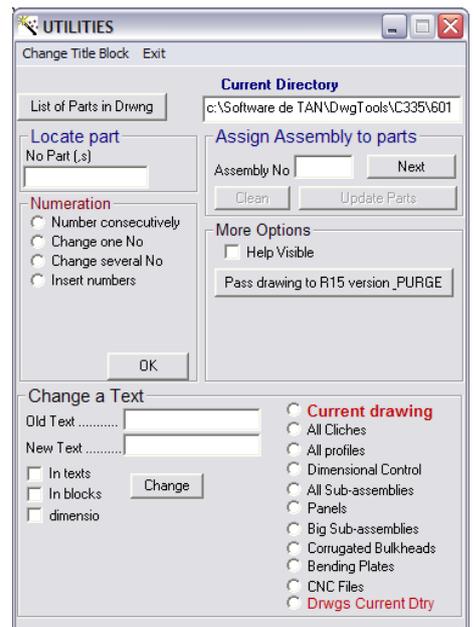
Save, if not previously done, the drawing 111012.dwg in the directory of the current assembly.

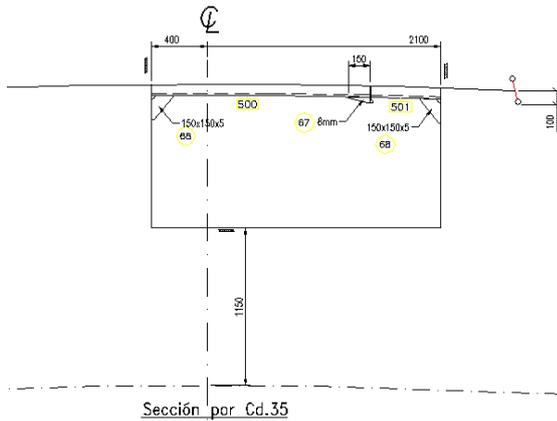
The overall plan, as it will in many cases, has been developed by someone else and therefore we are not familiar with it. The first thing you should know is how many parts and profiles are defined, their location and number of each mark. Click **Exit** on module **Profiles** and see how it works :

1. Open drawing 111012.dwg with AutoCAD.
2. In AutoCAD's toolbar  press the button  to start the module. (Look at the end of this Guide, where that module is described).
3. Press, in Utilities, the button **List Drawing Parts**. You will get following list by the System printer

PLATES MATERIAL						PROFILES																		
1	2	3	4	5	10	500	501	502	503	504	505	506												
11	12	13	14	15	16	507	508	509	510	511	512	513												
17	18	19	20	21	22	514	515	516	517	518	519	520												
23	24	25	50	51	52	521	522	523	524	525	526	527												
53	54	55	56	57	58	528	529	530	531	532	533	534												
59	60	61	62	63	64	535	536	537	538	539	540	541												
65	66	67	68	69	70	542	543	544	545	546	547	548												
71						549	550	551	552	553	554	555												
						556	557	558	559	700	701	702												
						703	704	705	706	707	708	709												
						710																		

4. In the window **Locate Part** write the number 500. The button **OK** will activate to start the search. Click on it.





Every time DwgTools finds a block mark 500, it draws a line from the origin to its insertion point. With this we can ascertain the location and amount of existing pieces of the same mark.

One of the located pieces will be the one in the accompanying figure.

If the plane is drawn accurately, simply ask for the properties of the beam line to get its entire length. This case is of 1582 mm.

#### NOTES :

The program indicates the number of blocks found in the drawing with the mark No 500, which does not have to match the number of pieces of the same mark.

You can only find and count the blocks created by DwgTools, which are contained in the toolbar **Blocks**.

You finish with module Utilities and start again Profiles. Let us define profiles.

Profile mark 500 :

Hand fill the boxes with the tag number, quantity (equal + symmetric) and Designation.

#### Profiles Type

In the window with Profiles Type, check the box Bulb Bar. Press now the mouse's right button. A drop-down list will appear, with all possible dimensions for bulb bars.

Point to the bulb 80 x 5. Boxes of this size will be filled.

Press  to close the box

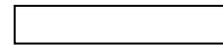
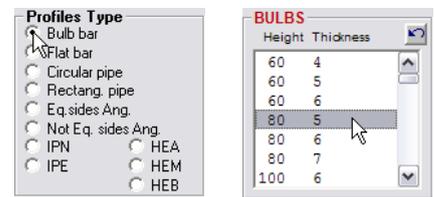
#### Length

Enter the profile length : 1582

#### Tipping Types

The ends are not blunt.

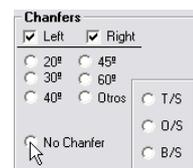
Press the bulb in the **Tipping Types** window. Among all the options that appear, point to the figure for **Normal** tipping. (Row 7, column 2). The figure is highlighted. Now press the Select button on the lower left corner and then the **Close** button.



#### Chamfers

Has no chamfer at its ends so that, in the window chamfers:

- Check the boxes LEFT. and Right.
- Click No Chamfers



#### Ends Relative Position

In the corresponding window, check :

- Left End. : Out
- Right End : Out

Activating the left side, also the box on the right side will be activated automatically, with the opposite option. In most cases this is correct, but not in the current case. To avoid this use, to point out, the right button of the "mouse" instead of left one. Only the chosen end box of the profile will be completed, right or left.



**NOT to fill** the box that says **Sketch No**. Later we will see that is automatically fulfilled.

As you press the various buttons, check boxes will be completed with the data.

If everything is OK, click the **Save Profile** button to be recorded in the database of the assembly. Before doing so, DwgTools calculate the weight, discounting the material corresponding to the trimmings it may have.

Click the button  to define next profile

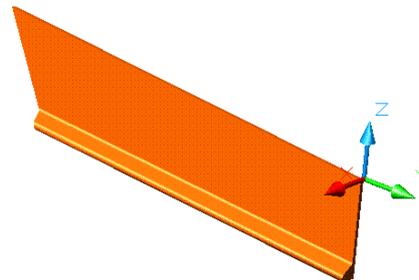
Now You define, similar way, profiles 501 to 517.

Profile No 518 :

Again, call the Utilities module to locate it.  
Is at all similar except the type of crop ends. There is, however, a circumstance that should be explained.

To work comfortably on a table processing, the profiles must be placed with the bulb upwards. Therefore, to prepare the sketch drawing of the profile, in general, be represented by the bulb upwards.

The rule has been followed in the drawing of the assembly is to look at pieces from the bow and from starboard. If you were to represent the profile seen from the bow, the bulb would be hidden. System must be changed to suit processing needs. As a result, the left side of the profile will be OUT and the right is C.Line.

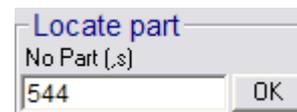


The profile goes up the side of the Bridge that, in that area, it is tilted. Needs an emergence, on the left side, to save this inclination. So far only indicate emergence on the left side, corresponding to the pattern in row 1, column 1 of the trimmings dialogue box. We could also have pointed out the drawing of row 1, column 5, blunt Left Symmetric, but then the picture in the drawing should be done with the head of the bulb at the top..

Profile No 544 :

We'll locate it on the drawing by the **Utilities** module. (see at the end of this Guide).

Write the number and press **OK** to start search.



The application will tell us he has found 4 blocks to the 544 mark, pointing them with a yellow line. When analyzed we'll see that it is a profile and its symmetrical, appearing several times in the drawing.

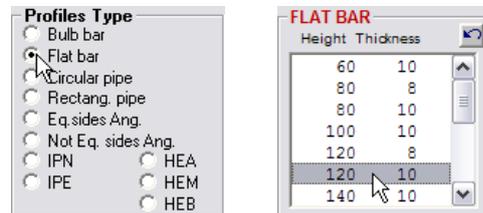
Hand fill the boxes with the tag number, quantity (equal + symmetric) and Designation.

**Profile Type**

In the window Profile Types, we'll check the box Flat Bar.

Now press the Right button of the "mouse", a drop-down list appears with all possible dimensions of bars.

Note the 120 x 8, whereby the boxes are filled in these dimensions. With the button  the list will disappear.



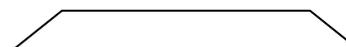
**Length**

Enter the profile length : 1252

**Tipping**

The ends are as shown in attached figure.

Press the bulb in the **Pinching Types** window. Of all the options that appear, point to the figure for **Symmetric Double Tipping**. (Row 2, column 5). The figure is highlighted. Now press the **Select** button on the lower left corner and then the **Close** button.

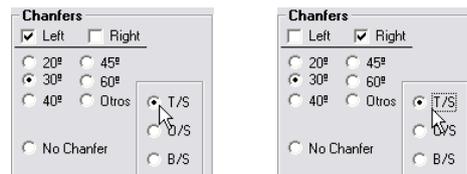


**Chamfers**

snipped ends with 30° this side (visible side)

1. Check Left Side  
Check 30°  
Press on the label T/S (this side)

2. Same with Right side.



### Ends Relative Position

In the corresponding window, check :

- Left Side : Out

Activating the left side, also the box on the right side will be activated automatically, with the opposite option. In this cases this is correct.



Practice all you want with the rest of the profiles. DwgTools will be creating profiles file.

To continue, you must copy the file "perfiles.new" which is in the TUTORIAL directory in the subdirectory of the assembly 012. Doing so will replace the file created during practice for this new one.

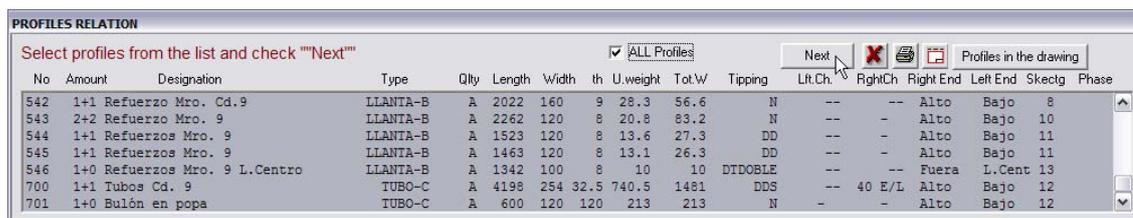
Press the **List** button and get the following window :

PROFILES RELATION																
Number of marks : 34			Number of parts : 44			Total Weight of profiles : 3004.3 kg			Profiles in the drawing							
No	Amount	Designation	Type	Qty	Length	Width	th	U.weight	Tot.W	Tipping	Lit.Ch.	RightCh	Right End	Left End	Skectg	Phase
308	3+0	Platabanda Esloras	LLANTA-P	A	2343	150	10	27.3	82.2	MDOBLE	--	--	Popa	Proa	1	
312	1+1	Platabanda consola marca 219	LLANTA-P	A	576	200	20	18.1	36.2	ESPEC	--	--	Popa	Proa	2	
313	1+1	Platab. esloras a 950 de L.C.	LLANTA-P	A	2089	150	10	24.3	48.7	MDOBLE	--	--	Popa	Proa	1	
333	1+1	Platabanda consola marca 226	LLANTA-P	A	583	200	20	18.3	36.6	ESPEC	--	--	Popa	Proa	2	
342	1+1	Platabanda consola marca 228	LLANTA-P	A	597	200	20	18.6	37.2	ESPEC	--	--	Popa	Proa	2	
343	1+1	Platabanda consola 223	LLANTA-P	A	600	200	20	18.8	37.6	ESPEC	--	--	Popa	Proa	2	
347	1+1	Platabanda consola marca 235	LLANTA-P	A	618	200	20	19.3	38.7	ESPEC	--	--	Popa	Proa	2	
348	1+1	Platabanda consola marca 235	LLANTA-P	A	637	200	20	20	40	ESPEC	--	--	Popa	Proa	2	
353	1+1	Platabanda consola marca 241	LLANTA-P	A	614	200	20	19.3	38.6	ESPEC	--	--	Popa	Proa	2	
400	1+1	Refuerzo varenga cd. 7	LLANTA-P	A	1480	80	8	7.2	14.5	DTDOBLE	--	--	Alto	Bajo	3	
401	1+1	Refuerzo varenga cd. 7	LLANTA-P	A	1149	80	8	5.5	11	DTDOBLE	--	--	Alto	Bajo	3	
402	1+1	Refuerzo varenga cd. 7	LLANTA-P	A	1227	80	8	6	12	DTDOBLE	--	--	Alto	Bajo	3	
403	1+1	Refuerzo varenga cd. 7	LLANTA-P	A	1170	80	8	5.7	11.4	DTDOBLE	--	--	Alto	Bajo	3	
404	1+1	Refuerzo varenga cd. 7	LLANTA-P	A	651	80	8	3.1	6.2	DTDOBLE	--	--	Alto	Bajo	3	
405	1+1	Refuerzo varenga cd. 8	LLANTA-P	A	1373	80	8	6.7	13.5	DTDOBLE	--	--	Alto	Bajo	3	
406	1+1	Refuerzo varenga cd. 8	LLANTA-P	A	1250	80	8	6	12	DTDOBLE	--	--	Alto	Bajo	3	
407	1+1	Refuerzo varenga cd. 8	LLANTA-P	A	775	80	8	3.7	7.5	DTDOBLE	--	--	Alto	Bajo	3	
512	1+0	Refuerzo Long. escotilla	LLANTA-B	A	1963	160	9	27.5	27.5	N	--	--	Popa	Proa	4	
513	3+0	Baos Cd.s 6, 7 y 8 Br.	LLANTA-B	A	2862	120	8	26.1	78.2	RBDR	--	--	Fuera	L.Cent	5	
514	0+1	Refuerzo Long. escotilla	LLANTA-B	A	2333	160	9	32.7	32.7	N	--	--	Fuera	Proa	4	
515	3+0	Baos Cd.s 6, 7 y 8 Br.	LLANTA-B	A	1158	120	8	10.4	31.1	RBDR	--	--	Fuera	L.Cent	5	
516	3+0	Baos Cd.s 6, 7 y 8 Er.	LLANTA-B	A	5240	120	8	48.2	144.5	N	--	--	L.Cent	Fuera	6	
517	1+0	Refuerzo proa escotilla	LLANTA-B	A	1420	120	8	12.9	12.9	NIRD	--	--	L.Cent	Fuera	7	
530	3+3	Cuadernas	LLANTA-B	A	2112	160	9	29.6	177.5	N	--	--	Alto	Bajo	8	
539	1+1	Refuerzo Mro. Cd.9	LLANTA-B	A	2262	160	9	31.6	63.2	N	--	--	Alto	Bajo	8	
540	1+1	Refuerzo Mro. 9	LLANTA-B	A	1191	120	8	10.9	21.8	RBD	--	--	L.Cent	Fuera	9	
541	1+1	Refuerzo Mro. Cd.9	LLANTA-B	A	1827	120	8	16.8	33.6	N	--	--	Alto	Bajo	10	
542	1+1	Refuerzo Mro. Cd.9	LLANTA-B	A	2022	160	9	28.3	56.6	N	--	--	Alto	Bajo	8	
543	2+2	Refuerzo Mro. 9	LLANTA-B	A	2262	120	8	20.8	83.2	N	--	--	Alto	Bajo	10	
544	1+1	Refuerzos Mro. 9	LLANTA-B	A	1523	120	8	13.6	27.3	DD	--	--	Alto	Bajo	11	
545	1+1	Refuerzos Mro. 9	LLANTA-B	A	1463	120	8	13.1	26.3	DD	--	--	Alto	Bajo	11	
546	1+0	Refuerzos Mro. 9 L.Centro	LLANTA-B	A	1342	100	8	10	10	DTDOBLE	--	--	Fuera	L.Cent	13	
700	1+1	Tubos Cd. 9	TUBO-C	A	4198	254	32.5	740.5	1481	DDS	--	40	E/L	Alto	Bajo	12
701	1+0	Bulón en popa	TUBO-C	A	600	120	120	213	213	N	--	--	Alto	Bajo	12	

After defining all profiles they should be drawn up sketches for the preparation of them in the Workshop. This requires

- Group bars that will be in the same sketch.
- To draw sketches one by one.

The first step we will, automatically, by clicking the button **Give Sketch No**



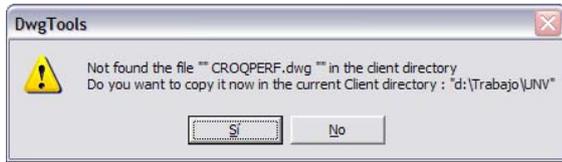
Activate the check box **ALL Profiles** and click **Next**. DwgTools analyzes the selected list of profiles, all in this case, groups that are similar and assigns different numbers to each of these groups

Once numbered, click the button **Make Sketch**.

A text box appears, blank, to indicate the first sketch we want to make. Key in  and press **Next**

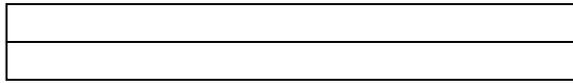


A warning appears, because the ten-plate drawing used to make the sketches has not been charged yet in the current Customer. Directory.

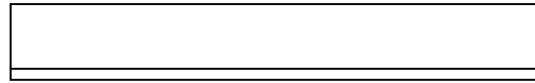


Press **YES** and the program will continue to make the profiles sketch # 1.

This case is very simple and the sketch is almost finished, but you have to give a light touch: locate the line that defines the head of the bulb



That is what we get



this is the correct sketch

We continue to create sketch # 2.

The button that before said **Next** now has got the text **Next Sketch**. We press it..



It generates the sketch and the same happens as in the former. Both sketches are virtually identical except that the **Relative Position** of the profiles is different

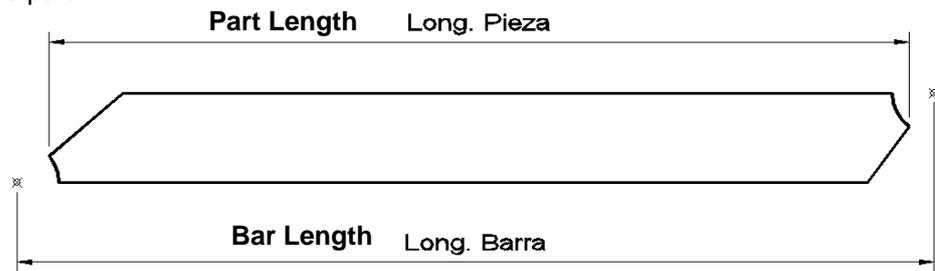
<p><b>Sketch # 3 :</b> The automatic drawing does not work. We must make a sketch as in Fig. The remaining information in the drawing is correct <b>A</b> is the deflection, in this case 80</p>	
<p><b>Sketch # 4 :</b> We need to change position of horizontal line</p>	
<p><b>Sketch # 5 :</b> Redo and complete sketch so that it is as in Fi</p>	
<p><b>Sketch # 6 :</b> modify slightly</p>	
<p><b>Sketch # 7 :</b> modify slightly</p>	

So you can continue till create all the sketches. Having completed all sketches, with the **Numbering Pages** button DwgTools will number consecutively all pages.

You can find sketches # 8 to 24 in compressed file **PERFILES.ZIP** located in folder *Tutorial*..

**NOTE :**

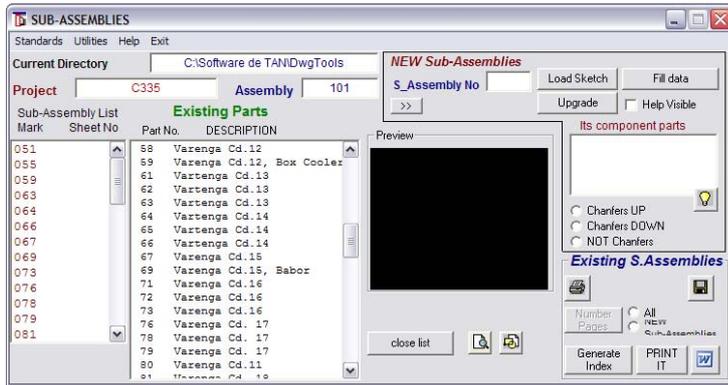
In the previous figures is shown what is the length to be given to define the profile: **TOTAL length of the bar** from which we'll get the PART. Next figure shows how the total length of the bar may NOT COINCIDE with the total length of the part..



Whenever we talk about length of the profiles will be referred to the length of the BAR, and this is also contained in all listings. The calculated weight by DwgTools, however, is correct because the radii, heels and trimming are discounted.

Prof. No	Qnt.	Designation	Type	Qty	Length	Width	t	Weight			Chanferss		Relative Position	
								unit.	total.	Trimming	Left	Right.	R. Side	L. Side
500	1+0	Bao Cd. 35	LLANTA-B	A	1582	80	5	6.7	6.7	N	-	-	Fuera	Fuera
501	1+0	Bao exterior cd.35	LLANTA-B	A	509	80	5	2.2	2.2	N	-	-	L.Cent	Fuera
502	1+0	Bao exterior cd.36 Br	LLANTA-B	A	1012	80	5	4.2	4.2	N	-	-	L.Cent	Fuera
503	1+0	Bao central Cd.36	LLANTA-B	A	2293	80	5	9.5	9.5	N	-	-	Fuera	Fuera
504	1+0	Bao exterior Cd.37 Br	LLANTA-B	A	1516	80	5	6.4	6.4	N	-	-	L.Cent	Fuera
505	1+0	Bao central Cd.37	LLANTA-B	A	2183	80	5	9.3	9.3	N	-	-	Fuera	Fuera
506	1+0	Bao exterior Cd.38 Br.	LLANTA-B	A	1571	80	5	6.7	6.7	N	-	-	L.Cent	Fuera
507	1+0	Bao central Cd.38	LLANTA-B	A	2072	80	5	8.8	8.8	N	-	-	Fuera	Fuera
508	1+0	Bao exterior Cd.39 Br.	LLANTA-B	A	1621	80	5	6.9	6.9	N	-	-	L.Cent	Fuera
509	1+0	Bao central Cd.39	LLANTA-B	A	1962	80	5	8.3	8.3	N	-	-	Fuera	Fuera
510	1+0	Bao exterior Cd.40 Br.	LLANTA-B	A	1674	80	5	7	7	N	-	-	L.Cent	Fuera
511	1+0	Bao central Cd.40	LLANTA-B	A	1851	80	5	7.7	7.7	N	-	-	Fuera	Fuera
512	1+0	Bao exterior Cd.41 Br.	LLANTA-B	A	1728	80	5	7.2	7.2	N	-	-	L.Cent	Fuera
513	1+0	Bao central Cd.41	LLANTA-B	A	1741	80	5	7.4	7.4	N	-	-	Fuera	Fuera
514	1+0	Bao exterior Cd.42 Br.	LLANTA-B	A	1781	80	5	7.5	7.5	N	-	-	L.Cent	Fuera
515	1+0	Bao central Cd.42	LLANTA-B	A	1631	80	5	6.9	6.9	N	-	-	Fuera	Fuera
516	1+0	Bao exterior Cd.43 Br.	LLANTA-B	A	1834	80	5	7.7	7.7	N	-	-	L.Cent	Fuera
517	1+0	Bao central Cd.43	LLANTA-B	A	1520	80	5	6.4	6.4	N	-	-	Fuera	Fuera
518	1+0	Bao exterior Cd.44 Br.	LLANTA-B	A	1483	80	5	6.2	6.2	DI	-	-	Fuera	L.Cent
519	1+0	Bao central Cd.44	LLANTA-B	A	1410	80	5	6	6	N	-	-	Fuera	Fuera
520	1+1	Rfzo. mro a 2600	LLANTA-B	A	2114	80	5	9	18	N	-	-	Alto	Bajo
521	1+1	Rfzo. mro a 2600	LLANTA-B	A	1170	80	5	5	10	RBI	-	-	Alto	Bajo
522	1+1	Rfzo. mro a 2600	LLANTA-B	A	1158	80	5	4.9	9.8	RBI	-	-	Alto	Bajo
523	1+1	Rfzo. mro a 2600	LLANTA-B	A	2120	80	5	9	18	N	-	-	Alto	Bajo
524	1+1	Rfzo. mro a 2600	LLANTA-B	A	1161	80	5	4.9	9.8	RBI	-	-	Alto	Bajo
525	1+1	Rfzo. mro a 2600	LLANTA-B	A	1144	80	5	4.7	9.5	DIS	-	-	Alto	Bajo
526	2+2	Rfzo. mro a 2600	LLANTA-B	A	1470	80	5	6.2	24.8	DDS	-	-	Popa	Proa
527	1+1	Rfzo. mro a 2600	LLANTA-B	A	1023	80	5	4.2	8.5	DDS	-	-	Popa	Proa
528	1+1	Rfzo. mro a 2600	LLANTA-B	A	1431	80	5	6	12	DDS	-	-	Popa	Proa
529	1+1	Rfzo. mro incl. Popa	LLANTA-B	A	1145	80	5	4.9	9.8	RBD	-	-	Bajo	Alto
530	2+2	Rfzo. mro incl. Popa	LLANTA-B	A	1154	80	5	4.7	18.8	DDOBLES	-	-	L.Cent	Fuera
531	1+0	Rfzo. Mro Popa	LLANTA-B	A	2075	80	5	8.8	8.8	N	-	-	Fuera	L.Cent
532	1+0	Rfzo. Mro Popa	LLANTA-B	A	963	80	5	4	4	RBD	-	-	Fuera	L.Cent
533	1+1	Rfzo. Mro Popa	LLANTA-B	A	1140	80	5	4.7	9.5	DDS	-	-	Bajo	Alto
534	1+0	Rfzo. Mro Popa	LLANTA-B	A	1146	80	5	4.7	4.7	DDS	-	-	Bajo	Alto
535	1+0	Rfzo. Mro Popa	LLANTA-B	A	1158	80	5	4.7	4.7	DDS	-	-	Bajo	Alto
536	1+0	Rfzo. Mro Popa	LLANTA-B	A	2100	80	5	8.9	8.9	N	-	-	L.Cent	Fuera
537	1+0	Eslora de Babor	LLANTA-B	A	5287	120	8	48.2	48.2	DD	-	-	Popa	Proa

Prof. No	Qnt.	Designation	Type	Qty	Length	Width	t	Weight			Chanfersss		Relative Position	
								unit.	total.	Trimming	Left	Right.	R. Side	L. Side
538	1+0	Eslora de Estribor	LLANTA-B	A	4929	120	8	45	45	DD	-	-	Popa	Proa
539	1+1	Refzo. del Frontón	LLANTA-B	A	1122	80	5	4.7	9.5	ROMBOD	-	-	Alto	Bajo
540	1+1	Refzo. del Frontón	LLANTA-B	A	1125	80	5	4.7	9.5	ROMBOD	-	-	Alto	Bajo
541	1+1	Refzo. del Frontón	LLANTA-B	A	1149	80	5	4.9	9.8	ROMBOD	-	-	Alto	Bajo
542	1+1	Refzo. del Frontón	LLANTA-B	A	1140	80	5	4.7	9.5	ROMBOD	-	-	Alto	Bajo
543	1+0	Refzo. del Frontón	LLANTA-B	A	1136	80	5	4.7	4.7	ROMBOD	-	-	Alto	Bajo
544	1+1	Refzo. del Frontón	LLANTA-P	A	1252	120	8	8.9	17.8	DDOBLES	30E/L	30E/L	Fuera	L.Cent
545	1+0	Refzo. del Frontón	LLANTA-P	A	3350	120	8	24.6	24.6	DDOBLES	30E/L	30E/L	Fuera	L.Cent
546	1+1	Refzo. del Frontón	LLANTA-B	A	1160	80	5	4.9	9.8	ROMBOD	-	-	Alto	Bajo
547	1+1	Refzo. del Frontón	LLANTA-B	A	1245	80	5	5	10	DDOBLES	-	-	Fuera	L.Cent
548	1+1	Refzo. del Frontón	LLANTA-B	A	1088	80	5	4.5	9	ESPEC	-	-	Fuera	L.Cent
549	1+0	Refzo. del Frontón	LLANTA-B	A	1299	80	5	5.5	5.5	N	-	-	Fuera	Fuera
550	1+0	Rfzo. Mro Popa	LLANTA-B	A	805	80	5	3.2	3.2	N	-	-	Fuera	L.Cent
551	1+0	Bao exterior cd.36 Er	LLANTA-B	A	976	80	5	4	4	N	-	-	Fuera	L.Cent
552	1+0	Bao exterior Cd.37 Er	LLANTA-B	A	1484	80	5	6.2	6.2	N	-	-	Fuera	L.Cent
553	1+0	Bao exterior Cd.38 ER.	LLANTA-B	A	1541	80	5	6.5	6.5	N	-	-	Fuera	L.Cent
554	1+0	Bao exterior Cd.39 Er.	LLANTA-B	A	1598	80	5	6.7	6.7	N	-	-	Fuera	L.Cent
555	1+0	Bao exterior Cd.40 Er.	LLANTA-B	A	1655	80	5	7	7	N	-	-	Fuera	L.Cent
556	1+0	Bao exterior Cd.41 Er.	LLANTA-B	A	1712	80	5	7.2	7.2	N	-	-	Fuera	L.Cent
557	1+0	Bao exterior Cd.42 Er.	LLANTA-B	A	1770	80	5	7.5	7.5	N	-	-	Fuera	L.Cent
558	1+0	Bao exterior Cd.43 Er.	LLANTA-B	A	1827	80	5	7.7	7.7	N	-	-	Fuera	L.Cent
559	1+0	Bao exterior Cd.44 Er.	LLANTA-B	A	1483	80	5	6.2	6.2	DD	-	-	Fuera	L.Cent
700	1+1	Puntales	TUBO-C	A	2076	70	10	66.5	133.1	N	-	-	Alto	Bajo
701	1+1	Rfzo. SOMBRERETE PROA	TUBO-C	A	3405	20	10	25.1	50.2	N	-	-	Fuera	Fuera
702	1+1	Rfzo. Sombrerete	TUBO-C	A	1297	20	10	9.5	19.1	N	-	-	Fuera	L.Cent
703	1+1	Rfzo. Sombrerete Lateral	TUBO-C	A	3010	20	10	22.3	44.6	N	-	-	Proa	Popa
704	1+1	Rfzo. Sombrerete Lateral	TUBO-C	A	1507	20	10	11	22.1	N	-	-	Popa	Proa
705	1+0	Rfzo. Sombrerete Lateral	TUBO-C	A	3405	20	10	25.1	25.1	N	-	-	Fuera	Fuera
706	1+0	Rfzo. Sombrerete Lateral	TUBO-C	A	3404	20	10	25.1	25.1	N	-	-	Fuera	Fuera
707	1+1	Rfzo. Sombrerete Lateral	TUBO-C	A	1527	20	10	11.3	22.6	N	-	-	Popa	Proa
708	1+1	Rfzo. Sombrerete Lateral	TUBO-C	A	3680	20	10	27.1	54.2	N	-	-	Proa	Popa
709	1+1	Rfzo. Sombrerete	TUBO-C	A	1562	20	10	11.5	23.1	N	-	-	Fuera	L.Cent
710	3+0	Refzo. Barraganete	TUBO-C	A	647	20	10	4.7	14.4	N	-	-	Alto	Bajo



The module window appears with information on the last block we has been working with.

One list shows the names of the already defined sub-assemblies and the other one the realtion of existing parts in current assembly.

Each sub-assembly will have the name (mark) of the most significant piece, larger and heavier, among which it is composed.

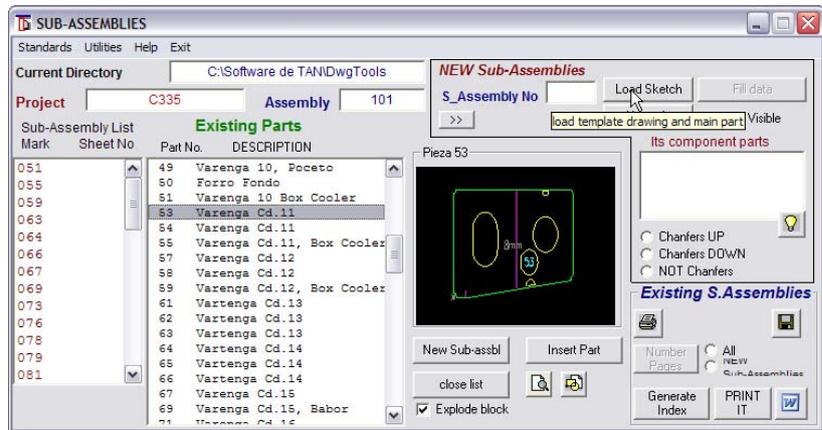
As in other modules, when you point a piece in the list, is displayed its preview.

In this module we could need a part with two purposes :

- To create a new sub-assembly
- To insert the part in the sketch of a previous sub-assembly.

That's why you can see two different buttons, one for each case.

If we click on **New Sub-Assembly** in the box **Sub-Assembly No** will appear the mark of the part that, being the main part, will give its name (mark) to the sub-assembly.



**The process of defining a sub-assembly is as follows :**

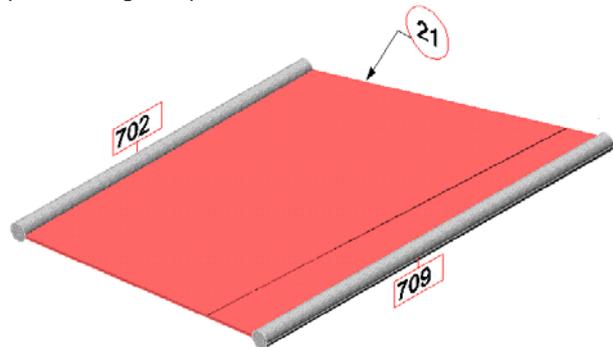
1. Indicate the dominant part, pointing to the list
2. Order New Sub-Assembly, pressing the button with that name.
3. Indicate the total parts and shapes that compose
4. Indicate the chamfers, if any, and where are they
5. Loading the Drawing
6. Fill in the data in the drawing

Let's define sub-assembly # 21. It is a piece of the bow cap consisting of 3 parts:

- Plate # 21, which gives the name to the collection
- Upstairs a round bar mark # 702
- Downstairs, round bar # 709

That's all there is to indicate, but you must do it in such a way that DwgTools can understand it.

It can be done in two ways :



**1st Procedure**

1. Indicate main part.
2. Ask for New Sub-assembly
3. In the window of the module, with the label "**Component Parts**" type the mark of main part and number of pieces of the same mark, separated by a hyphen. A comma (,) separation and number of another part, script, number of equal parts, and so on.

In this case : **21-1,702-1,709-1**

(If sub-assembly had 3 pieces of the mark 702 and 5 of 709 mark, we would have written: 21-1,702-3,709-5)

4. Press Load Drawing. DwgTools automatically will carry out the following
  - Open the ten-plate drawing for sub-assemblies and save it with the sub-assembly name in the current assembly directory.
  - Insert in this drawing the parts drawings. As many insertions, for each parts, as the number of parts we have indicated.
5. Insert blocks with the mark of any profile. For each brand, as many blocks as parts of it we have indicated.
6. Press **Complete Data**.

## 2nd Procedure :

1. Indicate the main part, 21, in the parts list.
2. Press New Sub-Assembly
3. Press Load Drawing. DwgTools, in this case, just :
  - Will open the template drawing and will save it, with the name of the sub-assembly, in current assembly Directory.
  - Will insert the drawing of the main part in the current drawing.
4. Insert, in the drawing, blocks showing marks and number of pieces in each mark on the sub-assembly :
  - Click the icon  in AutoCAD's toolbar. Specify the insertion point. X and Y scales are equal to 1 and, when asked for the number, enter 21 in the commands line of AutoCAD.
  - Click now the icon , proceed in similar way and enter number 702.
  - Click icon , etc. and enter number 709.
5. Click **Fill Data**.

### **NOTE :**

In this module, and on many other occasions, DwgTools examines the objects in a drawing and extracts data from them. There are two ways to allocate or assign information to an object :

- **Creating a block with its attributes.**  
When inserted into the drawing are prompted us to complete the information related to its attributes. Later, any application can read the information inserted blocks in that drawing just by knowing the name of the block and the attributes it has.
- **Assigning to an object what in AutoCAD is called XDATA.** Each object can have associated as many XDATA as you want. For example the polyline that defines the outline of a part have associated several more details: mark of the part, description, thickness, weight, etc. etc.

In reviewing a drawing, we check if an object has been created by DwgTools and if it has Xdata attached. If so, we read them, obtaining complete information of it, without having to search complex databases. Everything has "piggyback" their properties.

When an object has always the same shape but can change its properties, for example the title box of the plane, it is preferable to define it as a block and assign attributes, always the same, but their texts will have in each case the appropriate values.

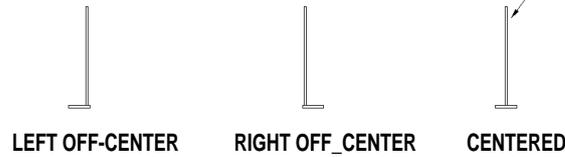
If the object, let's say a part, can have different shapes and different properties in each case, we need to assign to each form their particulars. The solution are XDATA.

In both cases, it is checked that the object has been created by DwgTools, discarding it if not.

This explains why it is essential to use the blocks and drawings created by the application. Moreover, and coming in the case of creation of sub-assemblies, if on the drawing there are more inserted blocks than parts in the sub-assembly, data obtained by DwgTools will be erroneous.

With both procedures we get generate much of what will be the drawing of the sub-assembly, but we'll often have to draw with AutoCAD objects, texts, dimensions, etc.. for complete information. The diversity of forms is so large that it can not be standardized over the creation process. Something else can be added, such as :

There are several block, stored in the template drawing, tan we can use :



Just inserting them, when needed, with AutoCAD's commands

If you point a sub-assembly on the list, its drawing will be loaded. Indicate, for example, sub-assembly # 65

Note the text in the upper left corner. There are the data about its component parts.

One of the options offered by the module is to update the drawing, which can be useful to reflect the changes we have done in the background, without having to redo the entire drawing. But we must use it carefully. Click the **Update** button and watch what happens with the earlier texts. They have doubled the quantity of each part.

This occurs because DwgTools, as explained before, reads the blocks which are displayed. In the example of the figure, to better explain the previous one, has been added a cross section which includes the numbers of each part. DwgTools, then, finds a mark No 65 in the elevation and another with the same number in the section, and concludes that the sub-assembly has two pieces of that mark. The same applies to the profiles. The total weight will be doubled and the total weld length also doubled. The problem is avoided if we only use DwgTools block number as many times as there are parts of a mark. The rest can indicate with texts in AutoCAD.

Mark	Qty.	Length	Width	Thickn.	Unit Weight
95	2	692	580	25	78.767
96	2	483	550	25	52.188
207	1	3876	558	30	489.528
701	3	763	508	32.5	271.8

No of EQUAL Sub-ASSBL : 1  
No of SIMMETRIC Sub-ASSBL : 0

ALL WELDING SHALL BE  $\nabla$ <sup>90°</sup> UNLESS OTHERWISE INDICATED

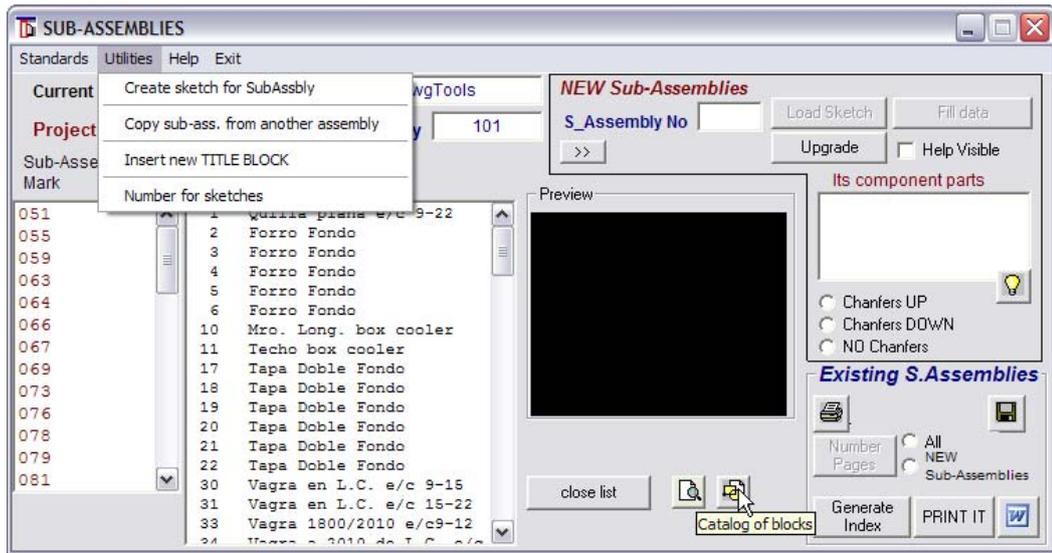
WEIGHT (Kgs.)		Mts. WELDING	C. of G.(mm.)		REF.	DATE	MODIFICATION	DATE	NAME	CHECKED
1566.833		9.673	X			May, 2012				
			Y							
			Z							

SCALE	DENOMINATION	DRAWING No.
v <sup>o</sup> .B <sup>o</sup>	Sub-ASSEMBLY No 207	111/603
	Flange Beam Fr.40	Sub-Assembly : 603
		MODIFICATION :

CONST.	C335	TAN s.l.
FILE		SHEET No. 20 OF 20

EXAMPLE information for a sub-assembly

## Header Menu UTILITIES



This module has several blocks that allow us to clarify some details about how to place the flange, the position of a part relative to another, etc.

They are accessed by pressing  (Catálogo of blocks).

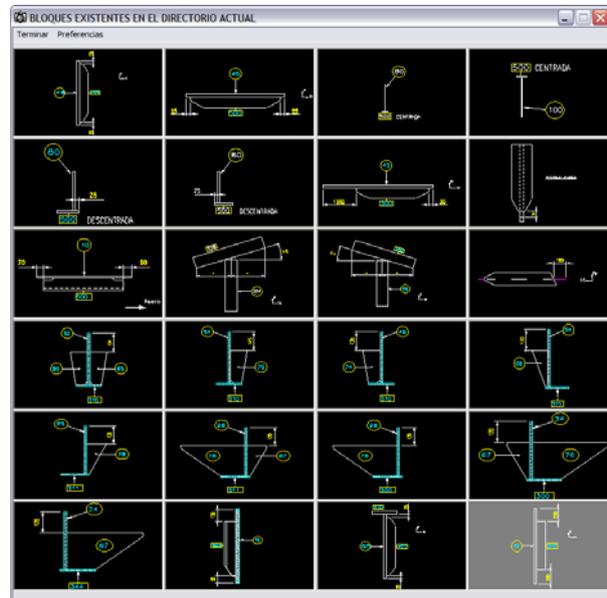
Generally one of these sketches will suffice to give further details of our sub-assembly.

If we were in the need to create a new sketch, use the header menu:

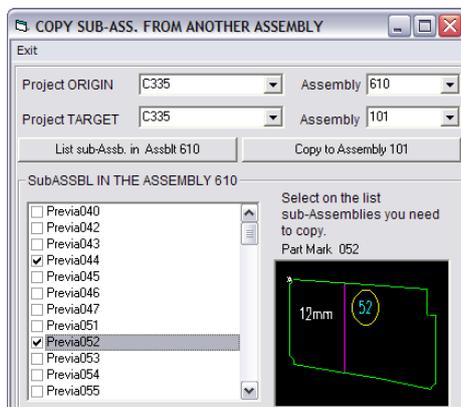
### Create Sketch for Sub-Assembly.

Clicking on it will open the module definition blocks (see Chapter 3).

Proceed exactly as explained in this chapter to create our new block.



The only difference is that we should not worry to indicate the current directory or the directory to save the block. The application will keep it in the directory for this type of blocks.



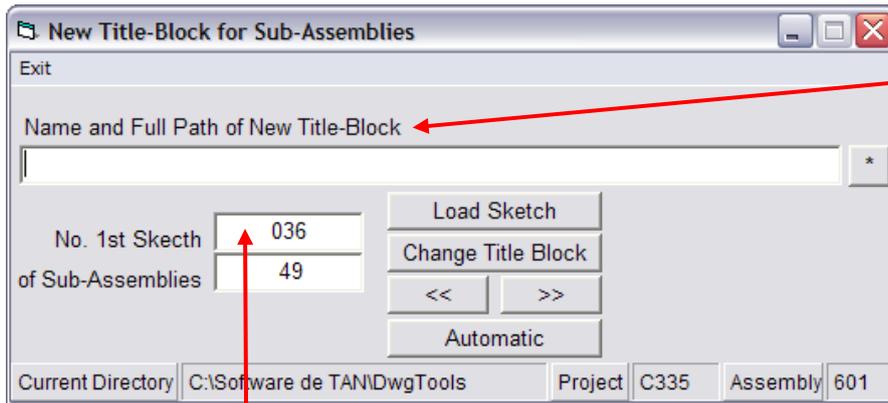
**Copy sub-Assemblies from another Assembly** : to copy previously defined sub-assemblies, in another Assembly, into the current. Using this menu, it will appear the attached window so we could indicate the source block. The target block is, of course, the current one.

Having identified the source block, press the button in **List sub-assemblies in...** This opens the relationship of these.

Note in her sub-assemblies you wish to copy. By checking each one, it will appear the image of the main piece that defines it.

The button **Copy to Assembly ...** will copy them in the Assembly's directory, changing the required data from the title block.

Insert a new TITLE-BLOCK : following window will be visible



Acting on the right button, or clicking on the label, we can specify the **Name and full path** of the directory that contains the new title block..

The application will show the total number of existing sketches and No. of the first one.

We can load the drawings one by one, with the button **Load Sketch** (will load the sketch which number is displayed in the text box), then press the button **Change Title Block**.

With buttons the << or >> we go to previous or next sketch.

The **Automatic** button will do this task for all sketches of the same type, in the current assembly.



The initial appearance of the module is as follows :



Pressing on labels **Current Directory** or **Assembly** is possible to change them.

This module is used to create nested parts on the plates and generate files for cutting machines CNC. All information produced is reduced to two files:

- tracking sheet with a picture of what is to be cut and some additional information. Helps the operator of the machine check is cut in the proper sequence.
- File with the codes for the cutting

Assbly	Part	N.P.	Length	Width	U.weight	Phase
101	53	2	1936	1198	113.0	
101	57	2	1998	1370	138.17	
101	62	2	790	1307	49.43	
101	71	2	1988	1120	112.99	
101	72	2	780	873	27.23	

Cutting Length : 72236  
 Marking Length : 7132  
 Legth Empty runs : 45401

CONST. : 335      CODE : M 101 L012      NUMBER OF PARTS IN THE PLATE 10      KPL (DATE: 07/04/03)      KPL (No. of EQUAL PLATES: 1)      No. of EQUAL PLATES (No. NUMERICAL PLATES: 0)      8000 x 2400 x 8      QUALITY (A)

Tracking sheet

Cutting file :

Is a binary file that can be edited with Windows Notepad.

Consists of a header which indicates the sheet to be cut, the date it was generated and the dimensions of the plate which is provided. If in doubt this data will be used to find out the tracking sheet that corresponds, which has the same codes.

The remaining lines in the file correspond to the instructions for marking and cutting the parts.

```

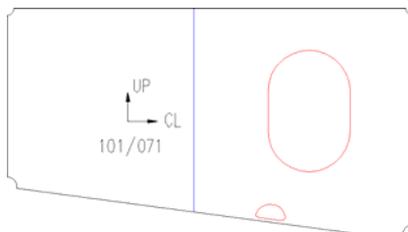
%
3
335 M101L012 07/04/2003 8000
2400 8
4
5
+12205+250
6
114
5
45
110
++12545
111
    
```

Every tracking sheet offers following information :

- Quality and thickness of the plate used.
- Dimensions of the plate.
- Gross weight of the plate
- Net weight of the cut parts. Allows to deduct the % utilization.
- Number of plates that are cut with the same cliché.
- Number plates for symmetric cut.
- Total number of parts that are obtained with it.
- Number of parts of the same mark coming out of that plate.
- cutting path length.
- Length of marking path.
- Length of empty runs.
- Each piece has a small block to indicate its mark and its relative position on board.

Also is usual to included here, hand drawing with AutoCAD, some other data such as parts to be subjected to a forming process, sketches to cut edge chamfers, bevels, etc.

Later you will see that in addition to this information, which is the one send to the workshop, the application generates lists where, for each cliché, that information is completed with the times the machine needs to perform all the process, marking time, cutting and total time of activity.



In the attached figure you can see the process used to identify the general assembly (101), the mark of the part (071) and arrows indicating the position on board of the part.

This block is automatically inserted by the application, at the time of inserting each part at the nesting. But its content is not processed. That is the cutting machine DOES NOT draw it on the cut piece. It should draw it by hand on the cutting table, at the time that each piece can be identified.

This procedure, or any similar, will avoid many mistakes when placing the parts in their true position on board.

## WORKING PROCEDURE

When creating a new nesting you have to :

- Indicate the dimensions of the plate on which to perform nesting. The list of plates reserved at all times tell us what material we have.
- Inserting the pieces, one by one, on that plate. AutoCAD utilities for loading, spin, symmetry, etc.. greatly facilitate the work. DwgTools also offers some help to simplify the task (see below for an explanation of the buttons on the toolbar module)
- Indicate the chamfers on the edges that have them.  
Process the cliché.
- Check with the emulator, that every contour is cut and in the desired order

The sequence of tasks that the application takes place automatically, which can not be altered, is as follows :

- Draw marking lines.
- Cut the lightening, drain holes, etc..
- Cutting the contours of the parts.

The application begins with the nearest point to the origin and carries out the sequence of operations trying to minimize the empty runs of the machine. However, the cutting order the parts can be altered with an option to be discussed later. The order of marking and cutting of holes CAN NOT be changed.

## 12.1 Header Menus in module Nesting

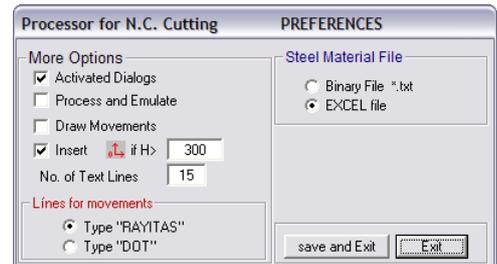


creation of this file.

**Steel Material** . You should create the file with the material, plates, reserved for the assembly. As we used plates, they will be discounted from the list. Opening the file we know the plates used and remaining at all times. You can also, once created the file, add plates, delete them, etc.

**Preferences**, with several options for drawing the tracking sheet :

- When the part is low, block with its mark and relative position, can get out of the part. Thus, for pieces of height less than that indicated here, is inserted another block.
- The number of lines that have the title block with the relationship of parts to be cut with each cliché.
- The type of lines used to draw the load movements.



**Customers**, provides access to the file to add or delete customers. Shall include the customer name and the subdirectory of the work we do for it.

**Cutting Files**, provides access to files with instructions for the CNC machine. For each customer or machine we must be created a specific file. A dialog box helps for the



**Emulate the Cut** to simulate, in a special screen, movements to be performed by the NC cutting machine and visually check that the cut will coincide with the drawing in *AutoCAD*.

You can simulate the process step by step to check in detail the trajectories of the machine.

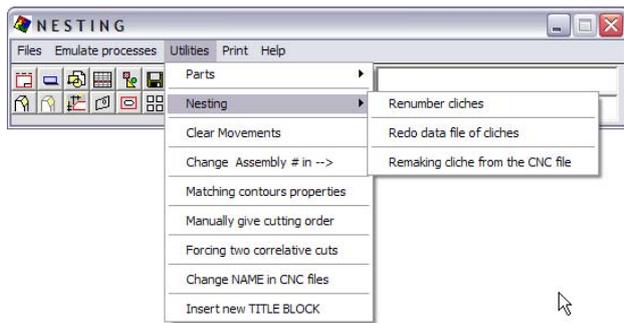
**Other Cliches** opens a dialogue box to choose any of the clichés previously processed and emulate their processes.

**Search parts in a Cliche**, locates in the current cliché, the part which marks we indicate. It is useful especially in clichés with many small parts.

**Delete parts ....** deletes part(s) we have pointed in the screen. Besides eliminating the part of the drawing, removes it from the list of parts cut with this cliché.

**Change the MARK ....** Each piece has a code that identifies it, so that the application, reading the objects in the drawing, known marks to be cut. The shape of the piece does not say anything, just the code that is assigned to it. In any case it is possible that, based on one "template" part, we have created a new one, very similar. The shape has been changed but the codes are those of the original part. Therefore, we must tell the application that is a different part and give the code for the new mark.

**Change the ASSEMBLY ....** It often happens that in different blocks are parts exactly alike. Therefore, sometimes one can use nestings of a certain assembly to cut another assembly's parts. Everything in the parts is the same except the code for assembly each part belongs to. Therefore, with this option, you must indicate the assembly to which belong the actual part.



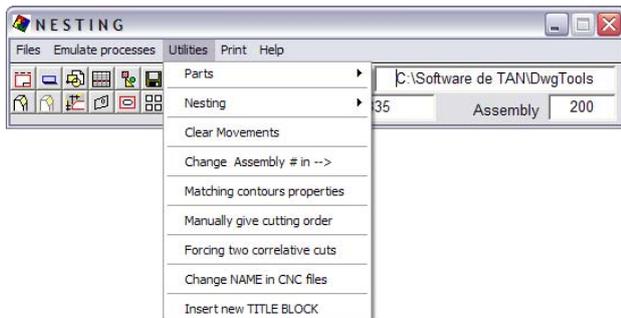
**Renumber Cliches** : When cliches are removed, added, and so on. We need to change the number assigned to each. This option is the renumbered automatically.

**Redo data file of Cliches** : *DwgTools* creates a database file of each cliché, used plate dimensions, quality, number of equal plates or symmetrical. As we create new cliches this file is generated, but because the changes will introduce at the end of the work it can be interesting this option : reads the actual situation of each cliché and rebuild the data file

**Remaking cliché from CNC file**, is the reverse process of normal. The application reads the file from the cutting machine and creates in AutoCAD tracking sheet . You may need to complete by hand the information in this drawing.

**Clear movements** deletes then in the drawing of the tracking sheet

**Change Assembly # in --->** changes assembly in the block

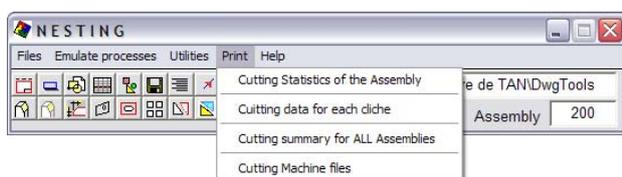


**Matching contours properties.** We can add or draw by hand a part in the nesting. We will need to indicate its mark and the assembly it belongs to. With this option, pointing to a model part in the cliché and then the new one, the latter adopted the model codes.

**Give the cutting order manually.** *DwgTools* uses a method for minimizing idle motions of the cutting machine. However it may be necessary, in some cases, the pieces are cut in a given sequence. With this option, pointing out the pieces in the desired order, the machine will cut them in that order.

**Designate two correlates cuts** to force two certain marks to be cut one after the other. The cut is made according to the path chosen by *DwgTools* but after cutting the first part marked, is cut the second, continuing after his with *DwgTools* algorithm.

**Change NAME inside CNC files** : In CNC cutting files is included the NAME of the ship. Sometimes we copy files from a ship for use in another, for example sister ships, and we need to distinguish the two vessels. This option will change, automatically, the new NAME inside the old files.



**Cutting Statistics** allows you to print a list with the amount of plates used, total waste of the assembly, etc.

**Data for each Cliché** prints a sheet for any cliché with lengths and times used for cutting, marking and movement in empty, nested parts on each plate, wasted material, etc.

**Cut summary of all Assemblies** prints total the list of total cliches of the current vessel, total plates used, total waste, etc.

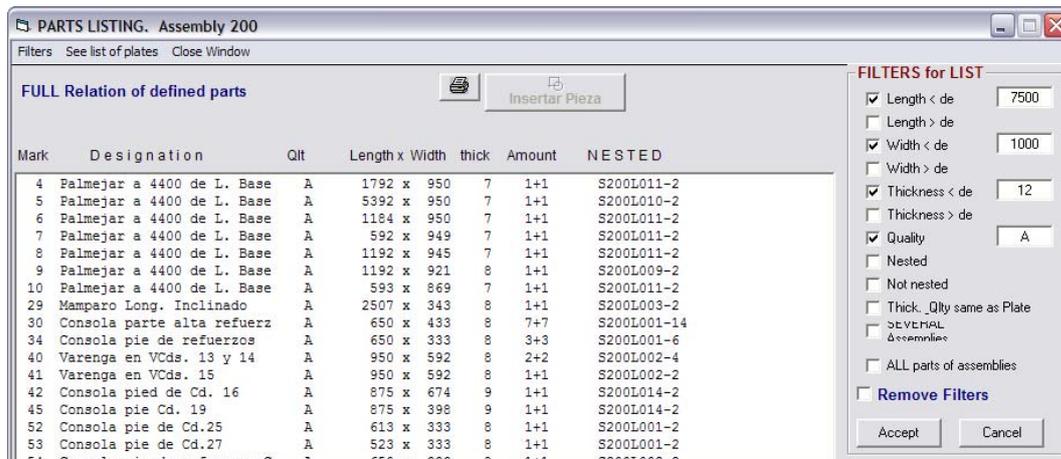
**Cutting machine Files** prints the file with cutting machine commands to cut the designated cliché.

## 12.2 Tool Bar in module Nesting

	Start a new sheet, and assigns it a number. Apply various data such as number of equal or symmetrical cliches, dimensions of the plate, material quality. Inserts the tem-plate drawing and fill in some data of the title box.
	Inserts a new plate, which replaces the existing one, if any, in the drawing
	<p>Insert a piece in the drawing. The list of pieces is made visible and marking on it, point out the desired part, that one will be inserted in the drawing <b>AS A BLOCK</b>, for easy handling. Before processing the part, the block should be exploded.</p> <p>Similarly we proceed with all the parts we want to nest in the current cliché.</p> <p>In the window with the list of parts we can choose multiple selection <b>FILTERS</b>, by pressing on the header menu <b>Options</b> (see below). With them will get various listings, as desired:</p> <ul style="list-style-type: none"> <li>• NOT nested parts,</li> <li>• Parts with thickness and quality as in the actual plate,</li> <li>• Parts Bigger or smaller than a given dimension, etc.</li> </ul>
	Open the list of existing cliches with their properties, dimensions, quality, % waste, etc.
	Start the numerical control processor. It generates files for cutting machine.
	Saves all data of the cliché in the screen. The drawing will be saved with AutoCAD's commands but all the information contained in the cliché must be saved, <i>with this command</i> , in the assembly's database.
	<p>Writes automatically texts in the cliché :</p> <ul style="list-style-type: none"> <li>• parts nested on it,</li> <li>• length for cut, marking and idle movements.</li> </ul> <p>To this end, data of the cliché must have been previously stored in the database.</p>
	Draws, on the cliché in the screen, movements of the cutting machine.
	Goes to next cliché, loading template drawing for clichés.
	To assign chamfer to the edges that have it.
	Deletes chamfer from the edge pointed in the screen
	Inserts a block that indicates the data to prepare, by hand, the bevel assigned to an edge. We will have to insert as many blocks of this type as different bevels to be performed
	Sometimes will need to draw or modify a part directly into the clichés. With this button will force <b>DwgTools</b> to consider it as a part created by the application and, thus, to process it as any other normal part.
	Holes we have added directly on the drawing must be identify as such so that <b>DwgTools</b> process them as holes and they will be cut in the appropriate direction. Pressing on this button and pointing on the screen added holes, will get this goal.
	Create a matrix of the part indicated, with its elements separated a certain distance, to be entered in the corresponding dialog box.
	Copy a part as indicated in Fig.
	Copy a part as indicated in Fig.
	Changes data in current cliché..

By pressing the icon  to insert parts, it opens a window with their relation.

On this window header there is a menu, **Filters** that displays the list of **filters**. So called because the parts list is FILTERED to select only the parts that meet the conditions imposed by filters.



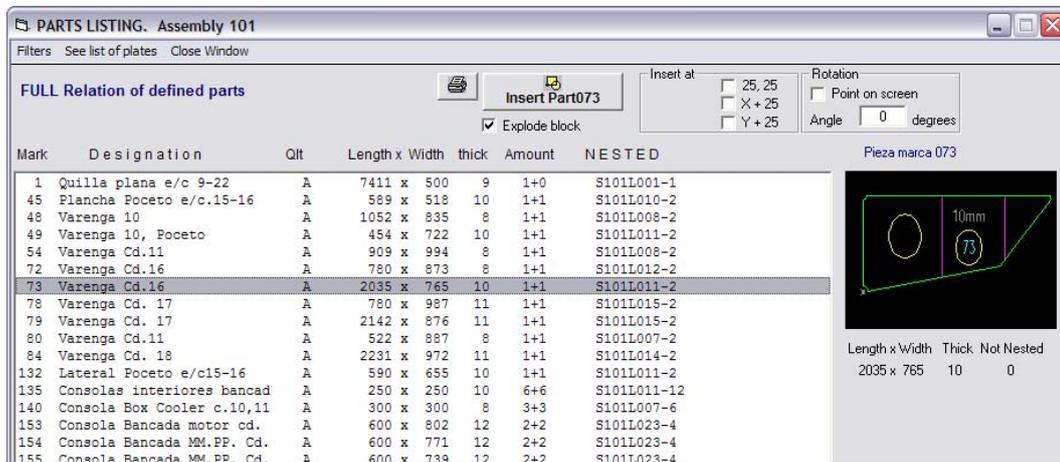
The figure represents the result obtained with the following filters :

- Parts with length less than 7500 mm
- Parts width less than 1000 mm
- Thickness equal or less than 12 mm
- Quality "A"

By checking each filter is shown a blank box to the right to write the data or value or desired.

A very useful filter is "Thickness and Quality = Plate": only the parts which are the quality and thickness of the plate shown in the drawing on the screen will appear in the list. This avoids parts to be fit incorrectly on the plates

Another feature is what can be seen in the figure, part 10 mark. There are only two parts with the same mark, symmetrical. But the list tells us that we have nested 2 pieces on the cliché 021 and two more at the cliché 029.



Pointing the piece on the list, the application will notify the mistake.

At the top right of the window there are some boxes that will help us to insert the part into the desired point and position.



Inserts part directly in the point with coordinates 25, 25

Inserts the part 25 mm to the right of the point indicated by the cursor on the screen.

Inserts the part 25 mm above of the point indicated by the cursor on the screen.

The latter two options can be marked at a time.

By clicking on them with the right turn into **X-25** or **Y-25**

The header menu **See Plates List** opens the file in Excel with steel material for the assembly, if previously created.

## 12.3 EMULATION of cutting and marking process

If we press **Other Cliches** a list the existing ones will appear, so that we can select any of them. The emulation is done on a screen like this:



Letters for the Name of the shops and its port of registry.

We can emulate the entire process continuously or **step by step**, allowing us to verify with precision cutting of each piece and lightening, where it begins, the entry radius and in what sense is carried out, ie, if the cutter compensation has been applied correctly

The sliders, along with the right mouse button, help us to zoom in to see the minute details of court initiated, fillet, cutting direction, etc.

We can edit, on another screen, the file with the commands of the cutting machine and, although it is never necessary (nor desirable), we can change the content or add any comments to it.

Once the change is done, it will be activated a button on the toolbar to save the file.

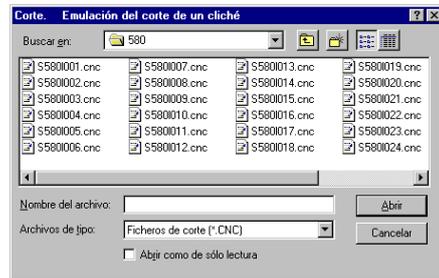
Another option on the toolbar allows us to send the list to the printer. Listings module (see below) allows other options.

```
COMANDOS DE CORTE DEL CLICHE : D:\tan\Ejemplo 1\580\S580L012.cnc
D:\tan\Ejemplo 1\580\S580L012.cnc
%
3
Ejemplo 1S580L012 19/9/99 8000 2400 7
4
5
+9938+250
110
++9430
111
5
-2538+4640
110
++9430
111
5
+7000-9430
110
++9430
111
5
+7000-9430
110
++9430
111
5
+11200-9430
110
++9430
```

## Toolbar

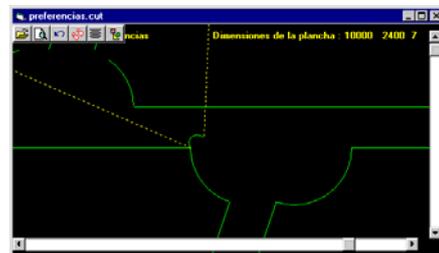


-  Open to emulate any file.  
This opens attached dialogbox in which we can select the cliché we wish to emulate.



## Zoom

-  Zoom to check the start point of a part cut :  
Radius of the curve ent ry = 5mm  
(it is possible to change it)  
cutting sense = clock wise



-  Original view : maximizes the size of the plate in the current window

-  Emulation step by step.



-  Edit cutting machine file

(see above)

-  Back to module Nesting

With this module we can get listings, in screen or by a printer, of any file created or used by other modules of DwgTools.

You can edit, to modify, all files even it is NOT recommended if you are not very familiar with their structure. All previous modules, if used correctly, modify their files automatically as they are going to work with them. The manual modification thereof should not be necessary in any case.

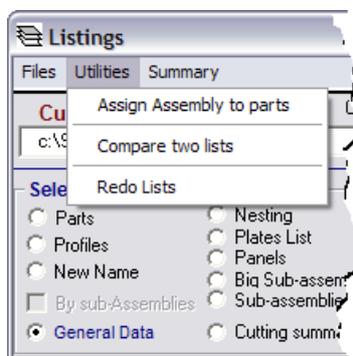
Changes throughout the work, changes in the blocks, error correction, etc. can create some confusion. So there is provided an option to redo the parts lists and profiles. To achieve this, **DwgTools** reads the drawings, opening them in **AutoCAD**, and extracts from them the information needed to generate some new listings. So the lists correspond EXACTLY with the final drawings.

**Listings** is an active module, ie it allows us to work with the listings to make certain actions on files. In general, we repeat, it is best to use the various options available to change things than to edit the files and change them by hand

The appearance of the window, is as follows :



As before, pressing on the labels **Curret Directory**, **Assembly** o **Ship's NAME** we can change these parameters.



The header menu **Utilities** opens several possibilities:

**Assign assembly to parts** : it is actually to differentiate in the parts listing of the various sub assemblies that may exist within the same assembly.

Although, when defining a piece, you must indicate the assembly and sub-assembly in which it is, sometimes may be useful to be able to do this assignment later.



By choosing this option it is displayed the window to assign the sub-assemblies

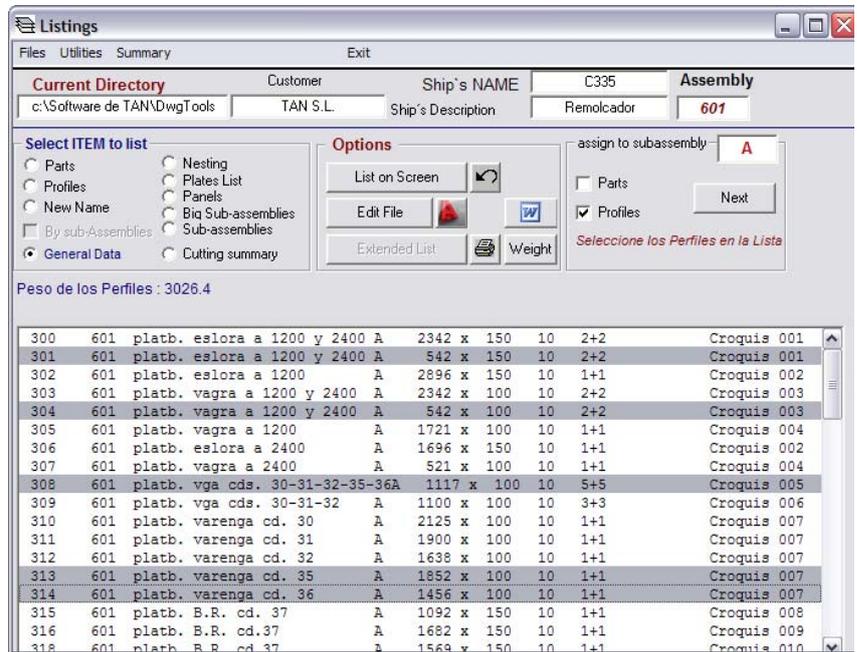
Marking parts or profiles, displays the corresponding list.

In the example we choose Profiles.

Select with the cursor the profiles we want to change, type in the box the sub-assembly and click **Next**.

The profiles can be selected one at a time or multiple selection, as in Fig.

Profiles marked will be assigned to the specified sub-assembly. (A in the example)



**Compare two lists** : it sometimes happens that, once released to the workshop the assembly information, we need to make corrections, new parts have been used, modified or deleted some, change the number of pieces of the same brand, etc.. This option compares the lists of old and new parts and prints a relationship with the differences between them. Thus the workshop known with precision the changes.

**DwgTools** generates two files, one called **piezas.new** and another **perfiles.new**, that are stored in the assembly's Directory.

When you make changes, before carrying out any, copy these files with another name for the record of old listings. (See below, **Copy a File**). We can call, for example, **piezas.old**, etc

Al ir modificando las piezas, **DwgTools** irá corrigiendo los ficheros correspondientes y al final tendremos en **piezas.new** y **perfiles.new** los listados actualizados.

If we now compare the files **NEW** with files **OLD**, will have the desired result.

The option **Compare two Lists** is also much more generic:



Pressing the option from the **Utilities** menu, the following window, in which will fill the data of the two files we want to compare.

Files can be analyzed in the same ship and assembly (have different extensions), different assemblies of the same ship or files from different ship and assemblies.

It is possible to compare just parts, files or both at the same time.

The results will be obtained by printer, **Normal Listing**, generated by **DwgTools**, or in a Microsoft Word file  that, later, we can modify or send to a printer.

Simply fill in the boxes and press either of the buttons.

The result with the comparison will be something similar to :

Sistema CamTan2000  
 Cliente : TAN s.l.

Fecha : 18/9/99

#### DIFERENCIAS EN LOS FICHEROS DE PIEZAS

Marca	Datos en Fichero Ejemplo1\518.new	Datos en Fichero Ejemplo2\518.new
23	Cons longit. escotilla	
	Total Piezas	4
	Lamina	518L007-4
		24
		518L040-24
57	Consola sobre mo. Central	
	Peso Unitario	174.5
		174.4
63	Presilla Eslora Inclinada	
		NO EXISTE
	Bloque	518-B
	Tipo Pieza	CHAPA-R
	Calidad	E
	Largo	300
	Ancho	200
	Espesor	12
	Total Piezas	2
	Peso Unitario	5.6
	Lamina	518L011-2
72	Consola final eslora inclin.	
	Total Piezas	2
	Peso Unitario	36.9
	Lamina	518L017-2
		3
		36.5
		518L017-3

**TAN s.l.**

**Redo Lists**: Listings, in **DwgTools**, are always an easy thing to achieve and AUTOMATIC.

During the preparation of the plans for the block, we can produce so many changes, errors, additions, etc.. it would be very time consuming having to redo the lists by hand. **DwgTools** therefore planned an automatic way to update the files for parts and profiles.

At any time you can choose the option **Redo Lists** to have them updated. **DwgTools** does the following :

- Opens in AutoCAD drawings of all parts in the directory of the current assembly to read their characteristics, number of parts, etc.
- Open in AutoCAD drawings of the nesting to check and note which pieces are nested in each cliché.
- Open in AutoCAD sketches to construct profiles to read the properties of each, trimmings, chamfers, etc.. and in which drawings are included.
- Generates new files for parts and profiles

Pressing the **Redo Lists** option, attached window appears :

Each button performs the action indicated.

**Parts List** reads all parts and all clichés.

**Parts in eac cliché** just reads the clichés to update nesting's data

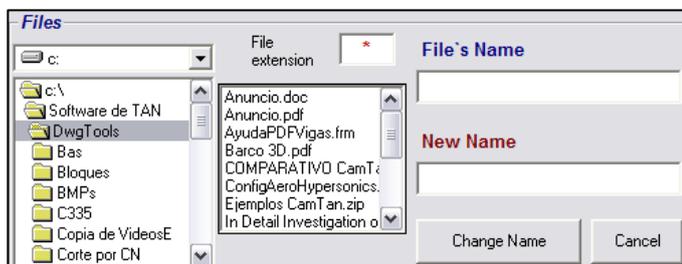


**NOTE** : There are several other files that can be updated with the utilities menu in corresponding module. For example, in the module **Nesting** there is an option redoes the file "laminas.dat" in which are existing clichés, dimensions and quality of each plate, waste material, cutting meters, length of markings, etc.

**Change a File's Name and Copy File** offer very similar options :

- in the first one, the old file disappears, but
- the second one keeps the old file and creates a new file, similar, just under another name.

The dialogue box is the same in both cases.



The directory tree helps and simplifies the search for the original file. By clicking on subdirectories and file names it will be filled the box of original file name.

Clicking on it, it is copied the name in the box for the new file. Be enough, usually by changing the file extension in the latter

Then press the button with the desired command, copy or rename.

Must give FULL NAME, directories and subdirectories, for both files.

## FILES LISTING

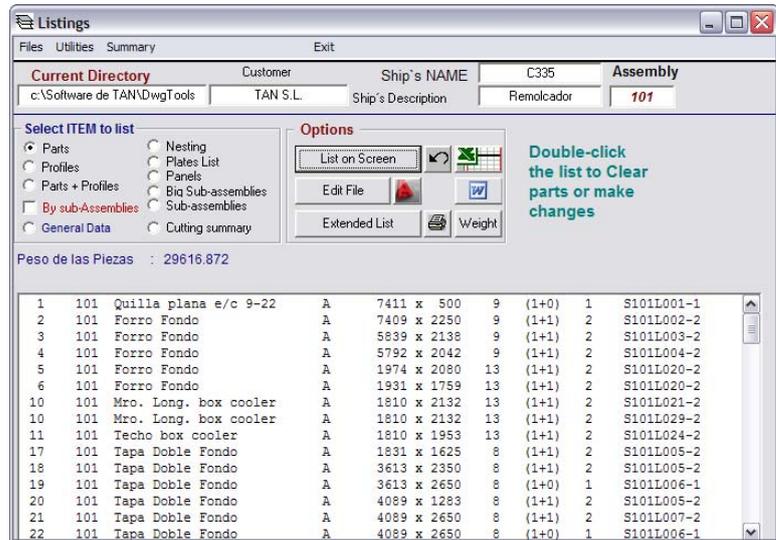
Must be selected then ITEM to list and then select an option .

**List on Screen :**  
shows the list of selected items in the bottom of the module window.

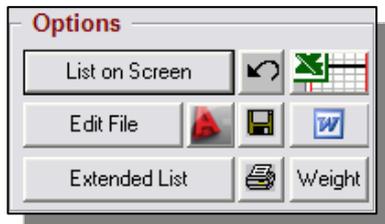
By choosing **Parts** appears somewhat similar to the figure.

The **Options** box is completed according to the selected item.

We will discuss later, the listings of the various items and variants that each generates.



The full **Options**, are :



**List on Screen** opens the list of "items" in the low area of the module's window.

**Edit File** open then ITEM's file in the same window. Then it is possible to modify and save the file but it is NOT RECOMMENDED modify files in this way.

**Extended List** is a list of parts, in A3 format, simulating FORAN listings.



, Closes the active.



Saves changes.



Opens MS Word and writes in it listing in the current screen.



Prints current list as it is.

**Weight** : if selected several items in the list, this option calculates the total weight of all them.

## LISTINGS AND ITS VARIANTS

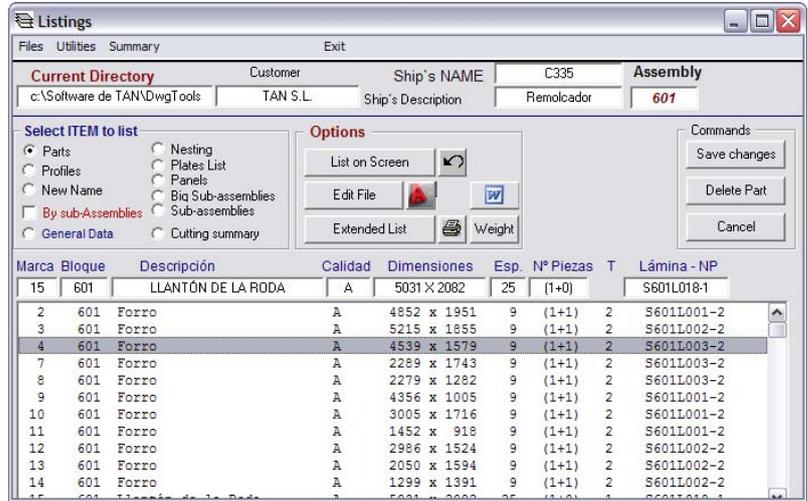
<b>ITEM</b>	Parts
<b>Option</b>	List on Screen

Opens one window as the 1st figure of the previous page.

Double clicking on any element of the list gives us the ability to change things. Open a few boxes with information on the item labeled, we can modify.

After making changes, we will save them so they are recorded in the data base:  
**Save Changes**

**Delete Part** will delete selected part. Its drawing will NOT be deleted.



This way of modifying data is reliable and does not result in errors like that can occur when EDITING FILES.

<b>ITEM</b>	Profless
<b>Option</b>	List on Screen
	As with item Parts

<b>ITEM</b>	Parts + profiles
<b>Option</b>	List on Screen
	Complete list of parts and profiles of the assembly. <b>Edit File</b> option and double click to <b>CHANGE</b> things are not available.

**NOTE** : in the three previous items will be listed all parts of the assembly. If we want a list of sub-assemblies, we will mark the corresponding Option.  **By sub-Assemblies** This opens a window for us to choose the desired sub-Assembly

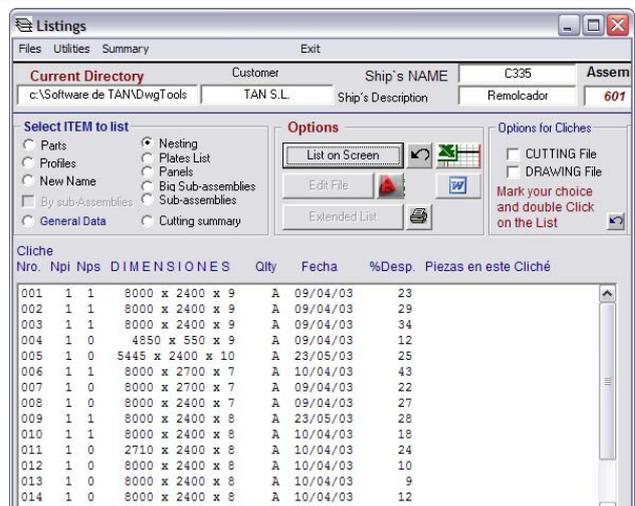
<b>ITEM</b>	Nestings
<b>Option</b>	List on Screen

Edit File button is disabled, as it has no interest in being able to do, nay, MUST NOT be modified by hand.

There is a list of the cliches with their most important features.

Pointing to a picture in the list, we can choose to open its drawing in **AutoCAD** or by editing the file with the cutting instructions.

Both the list and the Cutting file can be printed directly or build a version of it in MS Word.



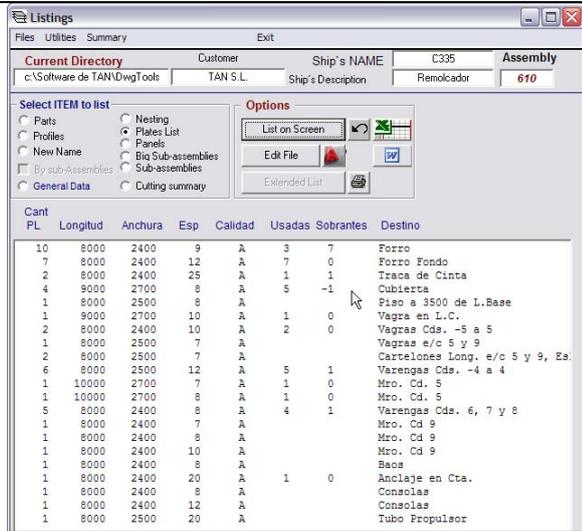
<b>ITEM</b>	Plates List
<b>Option</b>	List on Screen

Steel material relationship will appear, number of plates with its size and quality, with the plates used and the remaining plates.

With the **Edit File** button opens the file of plates, and can be modified.



When making any change button will be activated, not visible until then. We use it to save the changes.



<b>ITEM</b>	Standards
<b>Option</b>	List on Screen

Construction standards used on the boat. This file can only be created or modified from the parts definition or profile definition modules. Here we can view and print.

<b>ITEM</b>	General Data
<b>Option</b>	List on Screen

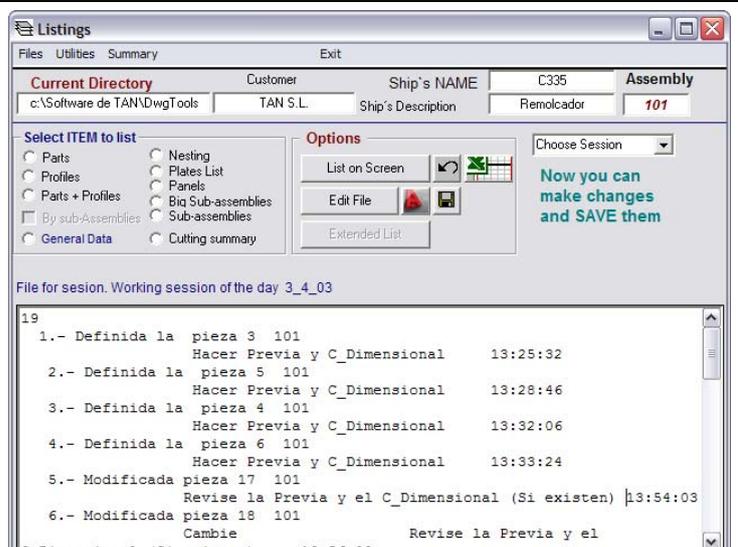
Customer data appears, the ship and the last directory we have been working in. Acting on **Edit File** we can change the data and then save them.

<b>ITEM</b>	Work Session
<b>Option</b>	

This ITEM opens a pop-up list with existing work sessions.

Marking one of them will list the actions taken during the same with, in some cases, suggestions to complete the action. For example, if we changed a part, we are advised to amend the sheet on which it is nested

These notes can be erased as no longer keep them interested and save the file to the session with only the actions they want to remember.





Icon in *AutoCAD* Tool Bar



The window of this module is as follows :

**Header menu Change Title Block** : Although the application has a box prepared for each of its drawings or sketches, you can create a new one customized. With this option you can change the existing box in a drawing with the new one. This may look like you want but its attributes must be the same as the original. Otherwise the application could not fill it automatically. The easiest way to modify a box is to exploit the existing, changing lines, colors, etc., and, without modification, placing attributes in the desired locations.

**Current Directory** : Clicking on this label opens a directory tree in which we point out that we want to become the current directory.

**List of Parts in Drawing**. Generates a list with all plate parts found in the current drawing in AutoCAD's screen.

**Locate part**. Enter the tag number of the part and press the OK button. The application shall draw a line from the origin to the part located, or will tell us that is not found, if the mark entered does not exist.

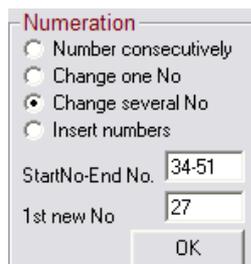
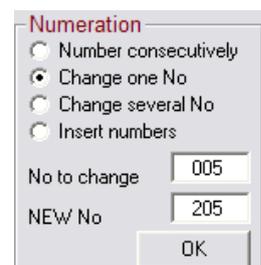
**Assign Assembly to parts**. Enter the desired block, press the button **Next** and select on the screen the parts to be updated. The application will change the assembly-code for such parts, assigning them to the new assembly.

**Numeration**. Allows several changes to the marks used to number the parts.

**Number Consecutively**. When defining parts, we assign marks, sometimes without a proper order, or by leaving unassigned numbers. Although not very important, it may be interesting, in some cases, make sure all parts are correlative marks without "jumps" in the numbering.

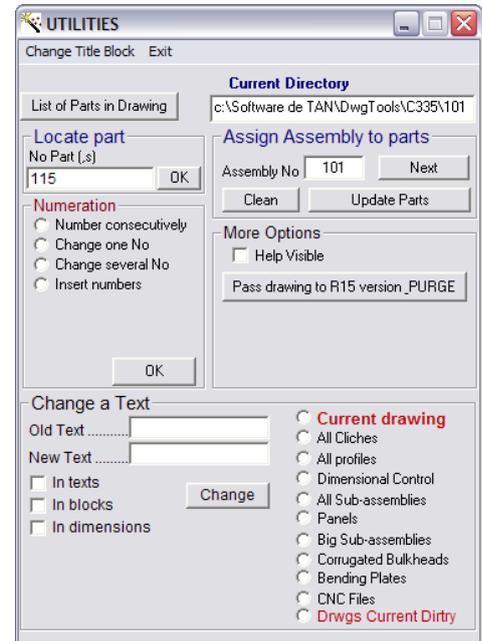
This option will perform the ordination from an initial number which must indicate, in a box that is opened for that purpose.

**Change one No.** We indicate the current number of the part and the new number for it. The application searches all existing parts in the plane with that mark and assigns them the new number.



**Change several No**. Similar to the above for various brands. Indicate the initial and final number of the parts we want to change, separated by a hyphen, and the first issue of the new numbers and press **OK**.

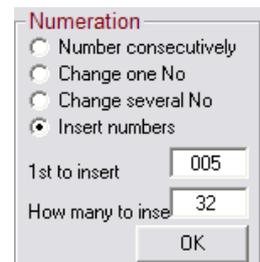
They will be numbered sequentially from the first issue. In the example, the mark 34 is assigned the number 27 and correspondingly with all marks to locate between the current No 34 to 51.



**Insert Numbers** among existing brands. It indicates the first number to be interleaving and the number of new brands you want. The application will change the existing brands in the drawing, renumbering the necessary markings, with numbers from the last one to two.

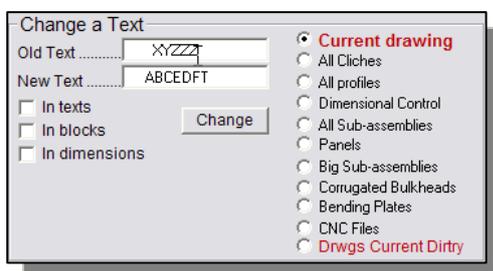
Example :

There are defined with the marks 5, 6, 7, 8, 9, and we want to insert 3 new brands from the 7. The application switches marks 7, 8 and 9, assigning them the number 10, 11 and 12 respectively, so that the marks 7, 8 and 9 are now available for new parts



## Change a Text

It should be noted the current text, we need to replace, and new text.



The change can be made :

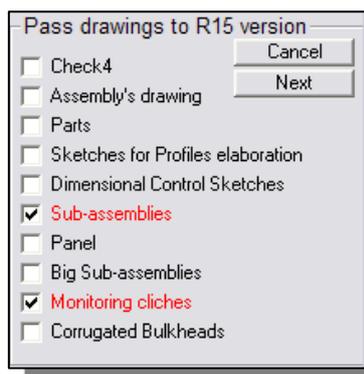
- In the texts of the drawing.
- The attributes of the blocks.
- In the texts of the dimension.

Activating one or more boxes, you tell the application which objects it must change.

We can also choose whether to do so only in current plane, in one of the sketches, or in all drawings in the current directory.

After selecting all the options, press the **Change** button. The application will open each and every one of the listed plans or sketches to perform the desired changes.

## More Options



Pressing the button **Pass drawing to R15 version\_PURGE** a new window is opened.

It shows the whole list of drawings and sketches that DwgTools handles.

You can select just the items you want, or all existing drawings in the current directory.

The selected items are highlighted in red.

Pressing **Next**, the application loads, one by one, all selected drawings, and saves them as AutoCAD 2000 drawing because, as we know, the drawings generated by AutoCAD 2012 can not be opened in previous versions if not saved as compatible therewith.



Icon in *AutoCAD* Toolbar



Module window appears with a two entries table :

- Header row indicates all sketches or lists that can be printed.
- The first column contains the list of existing block in the current directory.
- The inner boxes are used to indicate the number of copies to print everything.

	Assembly's Drawing	Dimens. Copntrol.	Profiles Sketches	Sub-Assbl's Drawings	Corrugated Bulkheads	Tracking Cliches	Plates Bending templates	Prof. Bending templates	Cutting Summary	Cutting data	Extended List	Prefabr. Beds
101	1	1	1	1	1	1	1	1	1	1	1	1
102												
103												
200												
201												
500												
507												
530	1	1	1	1	1	1	1	1	1	1	1	1
530A												
540												
560												
601												
602												
603												
607												
610	1	2			3	3	1	1	1	2	1	1
611												
700												
702												
800												
Crimenea Inicial												
Datos												
Disparos												
Escantillones												

Clicking on the first column in a row will activate all cells in the row. Shows the number 1, one copy.

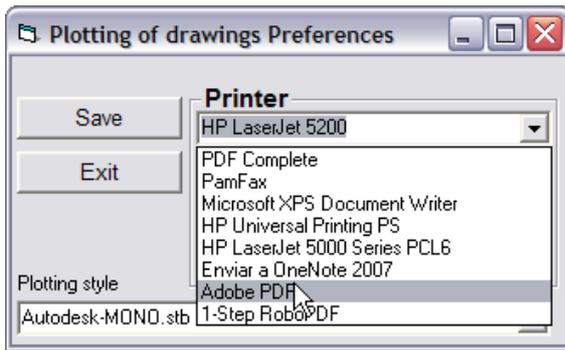
Clicking on one of the boxes internal :

- Left button increases the number of copies in one unit.
- Right button, reduces the number of copies in one unit..

Having listed the desired copies of everything, press **Print**. The application automatically prints all of the above.

If you also checked the box  **Print Indexes** they will be printed indexes, that for each listing the application creates in MS Word.

You can create a drawing in AutoCAD to serve as a cover for all listings. This drawing, which must be saved in the directory of any assembly, with the name "Portada.dwg" shall be printed now, if we select the box  **Print Cover Page**



The header menu **Preferences** opens a window that should aid in selecting the printer and plotting style we want to assign to the current set of drawings.

Plot styles are those that we have defined in AutoCAD for its drawings.

If you do not need to print all the drawings but only some of them :

1. Will mark the box for the assembly and the type of sketch to print.
2. Will mark the box  **Select from a list**
3. The list with the existing sketch.
4. In that list pointed out the desired sketch.
5. Press the button **Print**.

