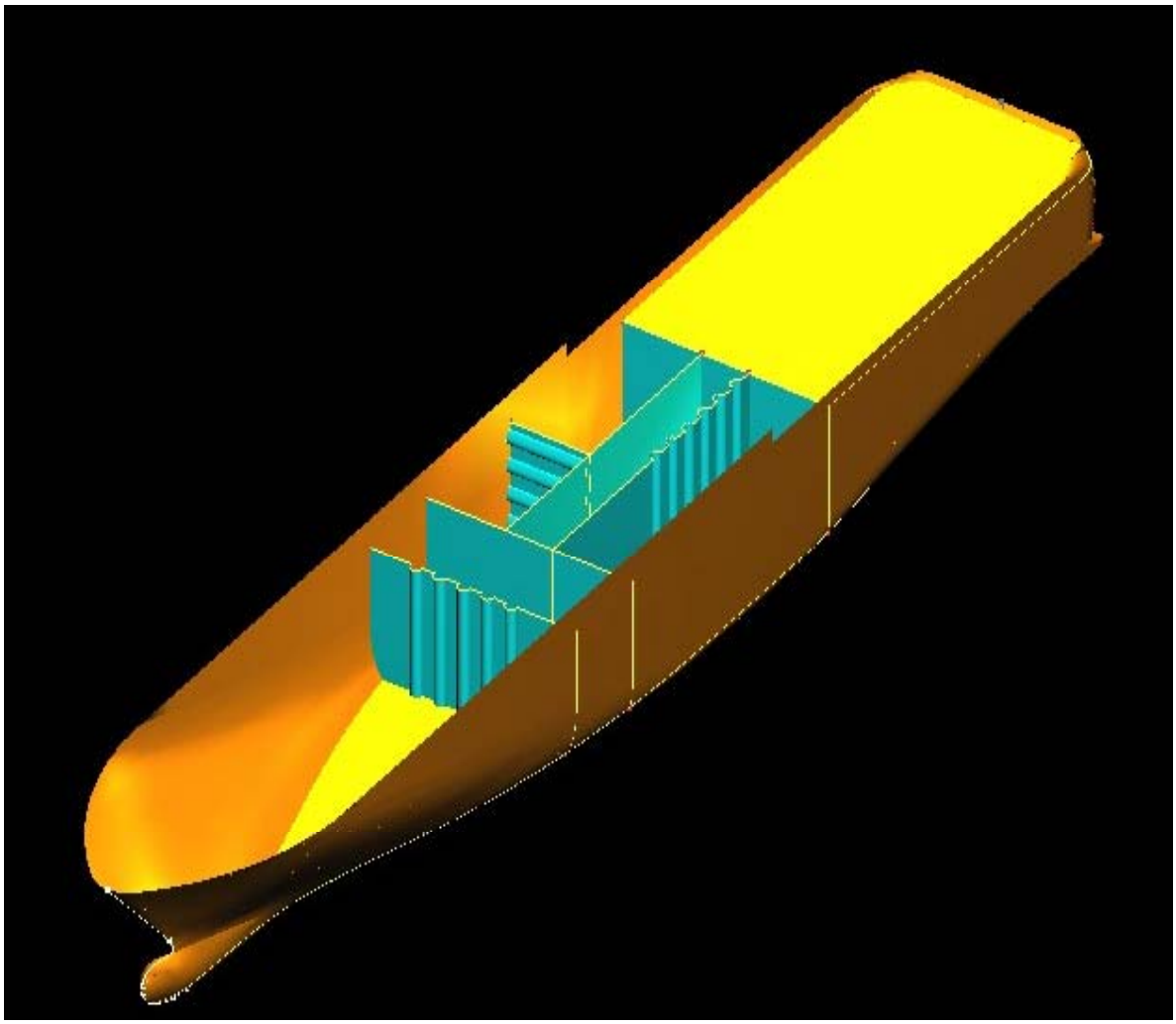
FSURF

TRAINING GUIDE



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1. Theoretic introduction

The FSURF module is based on NURBS formulation. (NON UNIFORM RATIONAL B-SPLINES).

BEZIER CURVES
The Bezier curves can be easily obtained using the Casteljau algorithm.

CASTELJAU ALGORITHM
If you have two points P1 y P2, whichever point between them can be calculated like a lineal interpolation of both points.
Whichever point is calculated as:

P1 x----------x P2

$P = (1-t) * P1 + t * P2$ where $0 \leq t \leq 1$

Let P0, P1, and P2 be any points in E3, and let t in R.

$P^0 = t .P_0 + (1 - t). P1$ for $0 \leq t \leq 1$

$P^1 = t .P_1 + (1 - t). P2$ for $0 \leq t \leq 1$

Inserting the first two equations in the next one, we obtain:

$P = t .P^0 + (1 - t). P1 = t. (t .P_0 + (1 - t). P1) + (1 - t). (t .P_1 + (1 - t).P_2)$

$P = t^2. P0 + 2t(1-t). P_1 + (1 - t)^2. P_2$ for $0 \leq t \leq 1$

This is a quadratic expression in t. the above contraction consists of repeat linear interpolation.
We also note that a parabola is a plane curve, due to the fact that is always a barycentric combination of three points.

Parabolas are plane curves. Many applications require true space curves, for those purposes, the above construction for a parabola can be generalized to generate a polynomial curve of arbitrary degree n:

$$P = \sum_{i=0,n} \binom{n}{i} t^n (1 - t)^{n-i} P_i$$

The polygon P formed by P0...Pn is called the BEZIER polygon or control polygon of the curve Bn.

1. Rational.

One extension of this formulation can be considered when the points have different weights. This weight is an additional coordinate in a homogeneous representation of our points in one space with one additional dimension.

This is the reason why, if one point has the coordinates (X, Y, Z) and the weight is W we can consider that our point has four dimensions and moved to the space 4D its coordinates are (X, Y, Z, W) where X = x.w , Y = y.w , Z = z.w and W = w. A rational Bezier curve of degree n in E3 is the projection of an n degree Bezier curve in E4 into the hiperplane W=1. X = X/W, Y = Y/W, Z = Z/W.
In the case of degree 2 a parabolic segment is projected in one plane, then the final result can be another parabolic segment, an arc of circle, ellipse or hyperbola. The result depends on the point's weight.
In non-rational, these conic sections can be approximated using parabolic segments, but in rational formulation you will obtain the required conic with mathematical precision.

2. Edition

Bezier curves provide a powerful tool in curve design. Some properties of BEZIER Curves

- Affine invariance: The Bezier curves are invariant under affine maps (repositioned, scaled and so on).
- Invariance under affine parameter transformations: Very often, one thinks of a Bezier curve as being defined over the interval [0,1]. This is done because it is convenient, not because it is necessary. The transition from the interval [0,1] to the interval [a, b] is an affine map. We can say that Bezier curves are invariant under affine parameter transformations.
- Convex hull property: For t in [0,1] Bezier lies in the convex hull of the control polygon.
- Endpoint interpolation: The Bezier curve passes through the first and the last point.
- Bezier curves provide a handy tool for the design of curves

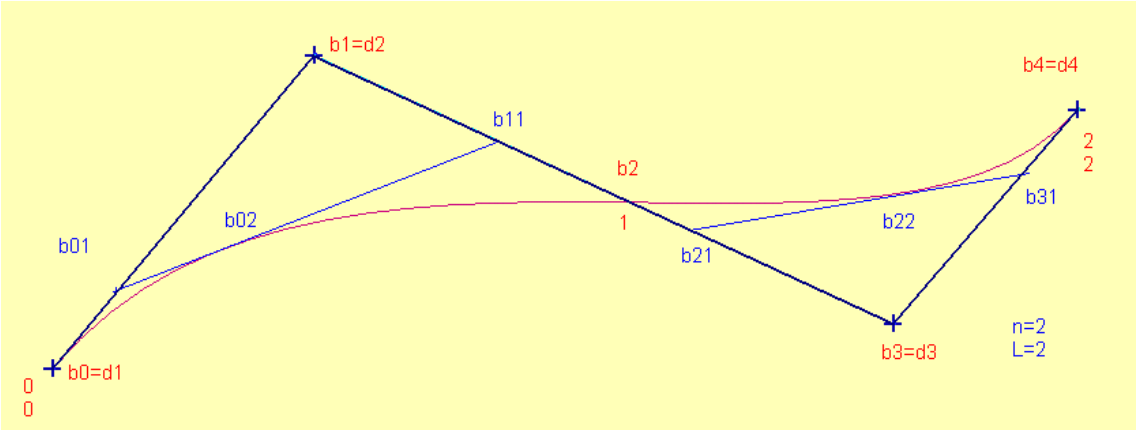
3. Disadvantages of Bezier Curves

The Bezier curves have some limitations, if the curve to be modeled has a complex shape, then its Bezier representation will have a prohibitively high degree (for practical purposes, degrees exceeding 10 are prohibitive). Such complex curves can, however, be modeled using composite Bezier curves. We shall also use the name spline curves for such piecewise polynomial curves.

4. B-Splines

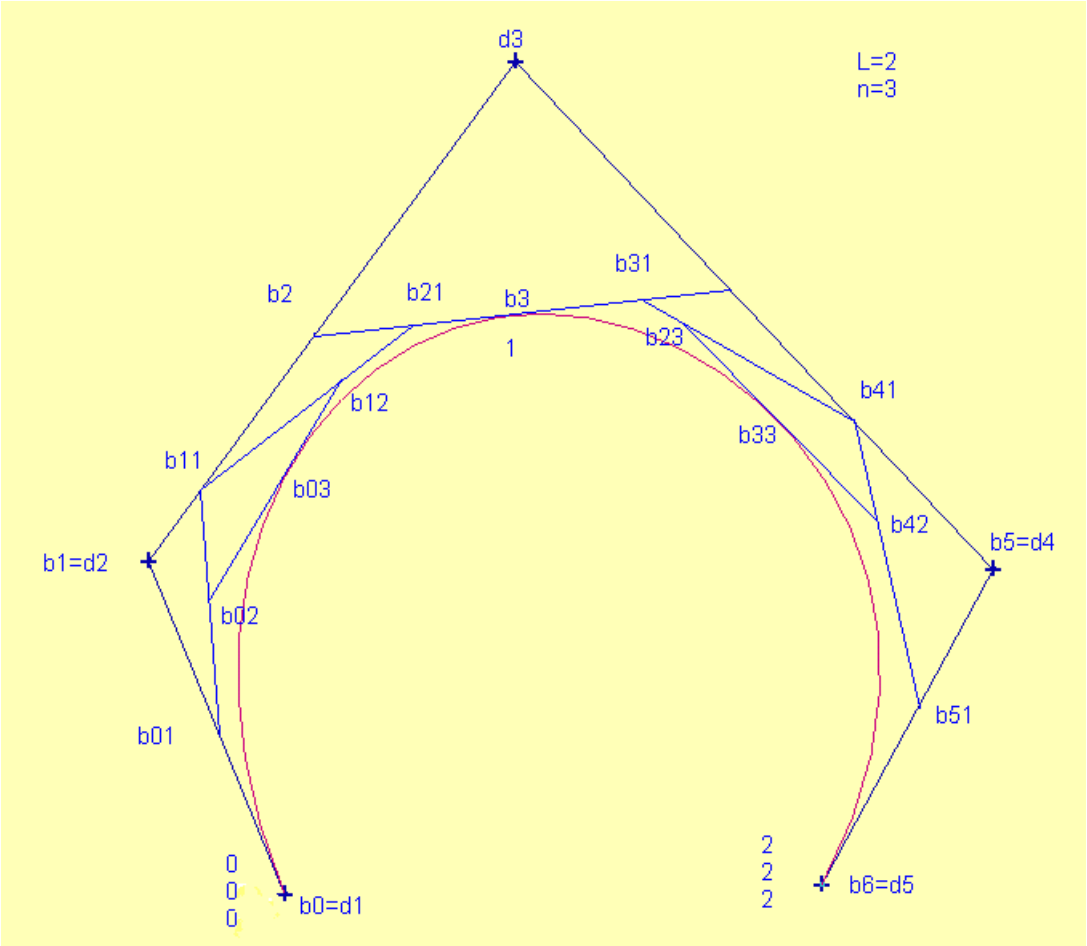
Composed by Bezier arcs, a B-SPLINE is a join of Bezier arcs with the same degree and without holes between them.
The degree of the B_SPLINE is the degree of Bezier arcs.
Every contact point between two Bezier segments is named knot.

C^1 B-Parabolic Splines



Control polygon $d_1 \ldots d_{L+2}$
L = number of segments
NPC = 4
Knots {0,0,1,2,2}

C^2 B-Cubic Splines



Control polygon $d_1 \ldots d_{L+3}$
L = number of segments
NPC = 5
Knots {0,0,0,1,2,2,2}

Then:

KNOTS = NPC + GR – 1

From the definition of a B-SPLINE polygon, we can deduce several properties, which we shall simply list since their derivation is a direct consequence of the above definitions:

- Convex hull property
- Linear precision
- Affine invariance
- Symmetry
- Endpoint interpolation

All the above properties are shared with Bezier curves.
There is on important property that singles Bezier curves do not share with B-SPLINES curves: Local control.

5. Non Uniform

If we are dealing with a single Bezier curve, we know that a change of one of the control vertices affects the whole curve, it is a global change. Changing a control vertex of a B-SPLINE curve, on the other hand, affects local curve segments. It is this local control property that made B-SPLINES curves as popular as they are. If a part of a curve is completely designed, it is highly undesirable to jeopardize this result by changing the curve in other regions. With single Bezier curves, this is unavoidable.

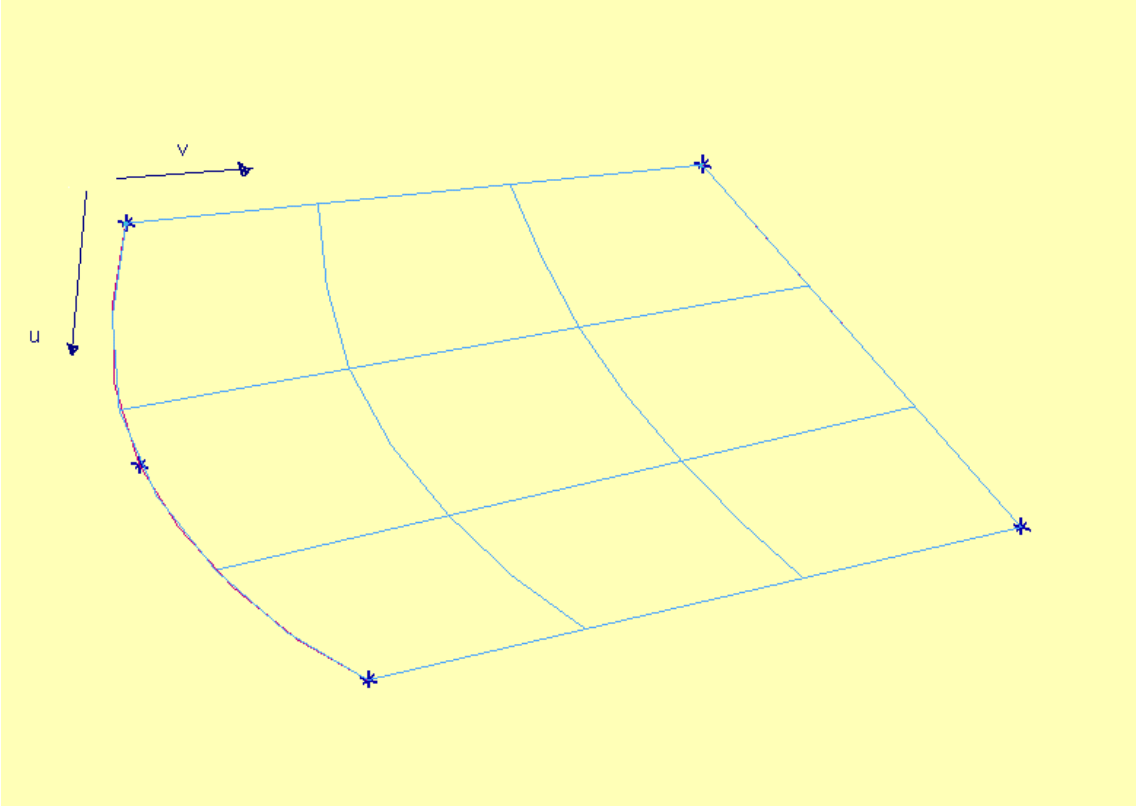
Surfaces

1. B-Spline surfaces

A surface is the locus of a curve that is moving through space and thereby changing its shape.
The surface definition indicates the use of two parameters “u” y “v”. The curve family is named isoparametrics.

The control mesh allows moving and fitting the surface. This is not the best way to work by can be used.

Regular and uniform control mesh indicates regular and uniform curvatures.



2. Type of files

2.1 Reading

- *.Fsf: reading patches and curves.
- *.Srf: reading patches.
- *.Igs: reading patches in IGES format (entity 128).
- *.Cur: reading curves
- *. Formt.fil: reading FORMF files.
- *.Pol: reading auxiliary polylines.
- *.Dxf: reading DXF format.
- *.Pnt: reading auxiliary points.
- *.db: reading a NAPA data base.

2.1 Writing

- *.Fsf: writing curves and patches.
- *.Srf: writing patches.
- *.Igs: writing patches in IGES format.
- *.Cur: writing curves.
- *.Txt: writing geometrical information of the selected patches.
- *.Pol: writing polylines in ASCII format (editable file).
- *.Pnt: writing points in ASCII format (editable file).
- *.Dxf: writing grid of polylines in DXF format.

The commands to manage files are located below FILE menu. These commands are described below:

- NEW: this command starts a new definition, deleting all the previous information.
- OPEN: this command allows reading whichever file mentioned above. This one will also delete all the previous information.
- IMPORT: this command is displayed in two:
    - IMPORT/MERGE: This one works like OPEN command, but in this case the program will maintain whichever previous definition.
    - READ FORMF SECTIONS: this command allows importing whichever WL, FR or BT from one FORMF file.

- SAVE: this command allows writing in one selected file (extension fsf) the existing curves and patches.
- SAVE AS: this command allows writing in one selected file (one of described above) whichever entity to be saved.
- SHIP – BILD SHIP: this command allows building the forms file. This one will be described with more detailed below.
- PLOT: this command allows generating drawings. This command will be explained in the next pages.
- EXIT: This command allows stopping the module. *¡BE CAREFUL!,* this command will not ask if you want to save or not the modifications you did during your last work session.

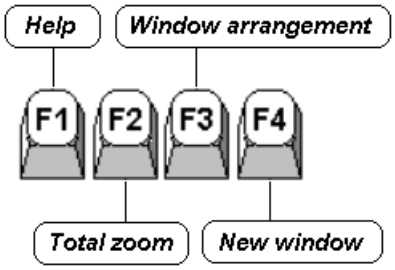
Type of menus

There are three types of menus:

- Main menu: Located in the top of the screen. They are displayed. They contain the main commands.
- POP-UP: They are displayed clicking with the right button of the mouse over the entity to be modified.
- Icon menu: The most common commands are shown.

Navigation, selection modes

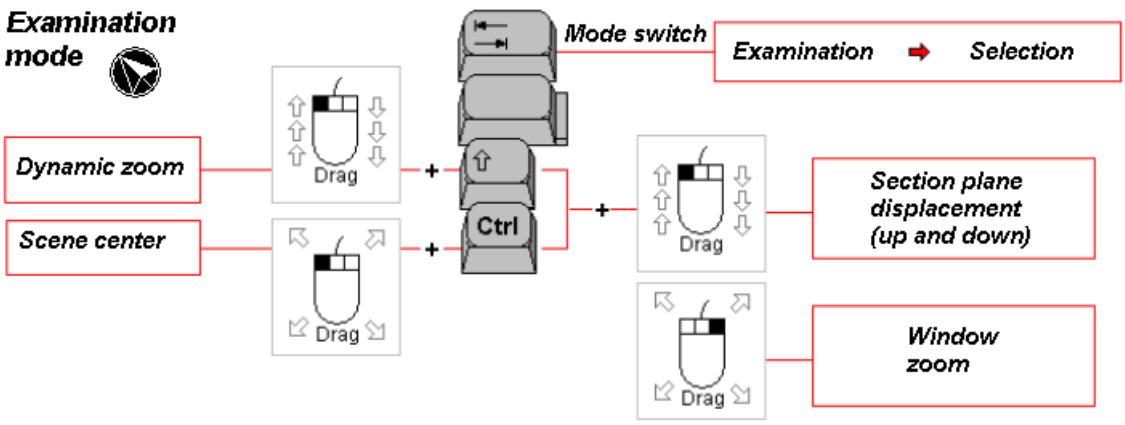
Function keys



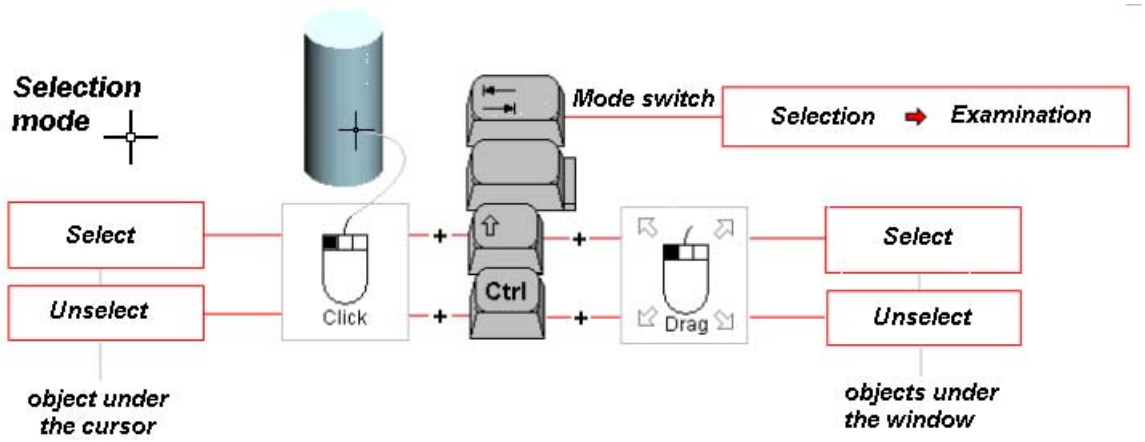
- **F1** to open the help
- **F2** to apply a total zoom, working in all the active windows
- **F3** to modify the graphic area
- **F4** to open a new window

Examination mode

Foran modules working in 3D allow the user handling the model in a very realistic way especially intended to visualize it to help in the understanding of the tridimensional arrangement.
This feature is called examination mode, being impossible to use the mouse for any key selection in the screen while the mode is activated.
Tab space key alternatively turns from selection to examination and counter.



Selection mode
The modules need to select objects from the scene for multiple reasons: to modify, to delete, to ask information,...
The default situation for the graphic area is the selection mode, showing the current cursor selector type.
Every module allow specific selections, according to the own nature of the different objects, but all of them apply the next:



Finally, clicking with the right button of the mouse in any object by the current selector type, it is opened the object popup menu, where a typical option is to select either the single object or the complete scene.

3. VIEW and EDITION menus

In this moment the user will know some tools located below VIEW and EDITION menus.
In this way, you will have the possibility to select and unselected, view and hide and so on, tools that later, working will be use a lot

3.1 Edition

- UNDO: This one recovers the scene situation as it was before the last command was executed. Undo can be used until 25 times.
- REDO: This one executes again the last undo command. Recovering the scene situation, as it was when the undo was used for the last time.
- REPEAT: This one executes again the last command in use. *¡BE CAREFUL!*, This command cannot be used when the last command was executed from the popup menu.
- SELECT ALL, UNSELECT ALL, etc.: These ones were explain above.
- DELETE: This one deletes from the scene the selected objects. By default the program will take automatically the selected entities to be deleted.
- PREFERENCES: This one edits the preferences for the module performance. The main values are the next:
    - Length: units for linear measures
    - Weight: units for weight measures. Not useful in FSURF

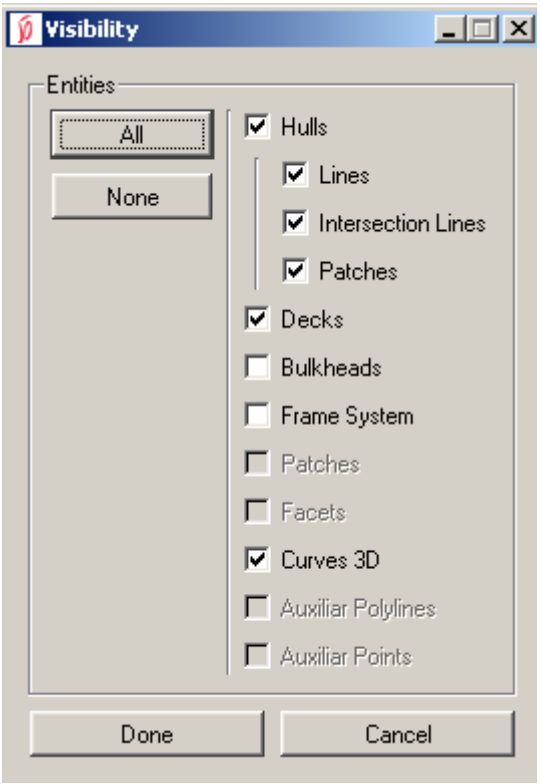
- Aprox. Tol.: approximation tolerance for distances. The default value is 1 mm, however we recommend to the user to change this value to 0,1 mm. This is because of steel program work with tolerances of 0,5 mm.
- Eps. Interaction: increment in the interactive movement. The default value will be 5 mm.
- Curves degree: By default will be 3
- Patches degree: By default will be 3.

These values can be changed in any moment of program execution.

3.2 View

- Fit visualization: This one fits the visualization of all objects in the scene, the program will redraw the objects after defining a new window.
- Zoom: This one modifies the observer distance to the scene.
- Center: This one sets the rotation center for the scene in any point, selected with the 3D-point dialog window.
- Preferences: This one is used to set by default the aspect of the background of the graphic areas. The changes made with this command are stored as the particular configuration for the model, in the current language and for the current user.
- Show and Hide: These ones allow visualizing or hiding the objects selected by the user. This new tool can be combined with selection commands.
- Visibility by type:

  This command allows the user to select which entities wants to visualizate in the current scene.
  When executing this command the next inputdata will be displayed:



If a checkbox is activated, the entities with that type will be displayed. To hide a type of entity, it is necessary to unactivated the checkbox.

It is possible to display all entities clicking on the button "All".
To hide all entites it is necessary to click on the button "None".
Please note that if you unactivate the option "hulls", all entities belonging to the hulls will be hidden.
Futher more, there are three additional buttons in the lower right corner to manage the visibility of main entities such as hulls, decks and bulkheads:

3.2.1 Hulls visibility

Clicking on the icon once, hulls are hidden.
Clicking on the icon twice, hulls are displayed.

3.2.2 Decks visibility

Clicking on the icon once, decks are hidden.
Clicking on the icon twice, decks are displayed.

3.2.3 Bulkheads visibility

Clicking on the icon once, bulkheads are hidden.
Clicking on the icon twice, bulkheads are displayed.

4. Entities and their modifications

The main entities to be used into the FSURF module are the next:

1. Points
2. Polylines
3. Planes
4. Curves
5. Patches

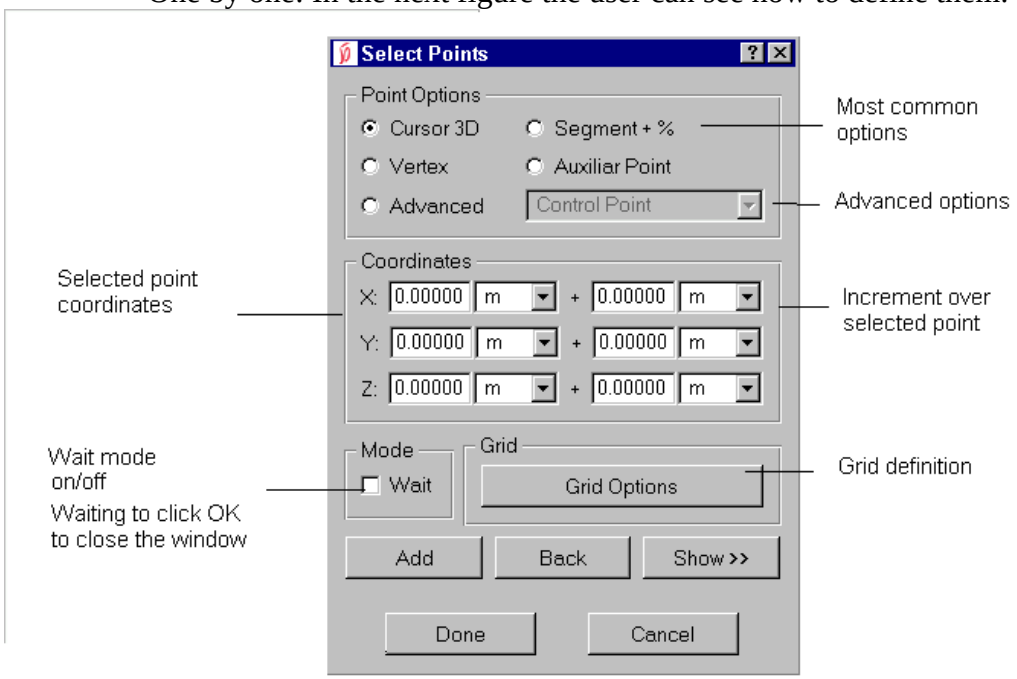
4.1 Auxiliary entities:

4.1.1 Auxiliary points:

These points will be used to define new entities or to modify some existing ones.

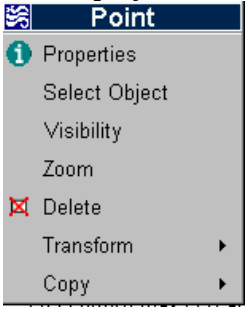
To define these points the user will select the ENTITIES menu. The definition can be made as follow:

- One by one: In the next figure the user can see how to define them:



- Intersection: There are two methods of defining points:
  1. Line-plane: The program will ask one line (point and direction) and one plane. The intersecting point is the one to be defined.
  2. Points of intersection between curves or polylines and patches or planes.
- Nearest: The program will define the nearest point (located in one existing entity) to the cursor selection.
- Pol points: The program will define the points of the selected polyline.
- By curve sections: Definition of intersection points between curves and plane.
- By curve inflexion points: Definition of the inflexion points for the selected curves.

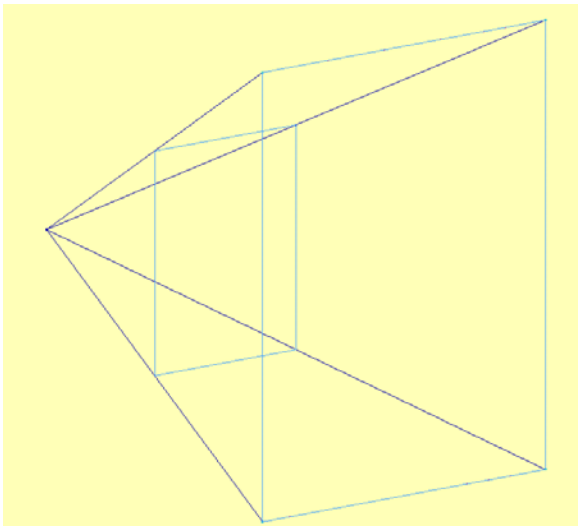
The auxiliary point's modification is only available from the popup menu. It is not existing the command in the main menu. Then, clicking with the right button of the mouse over the point the program will display the next window:



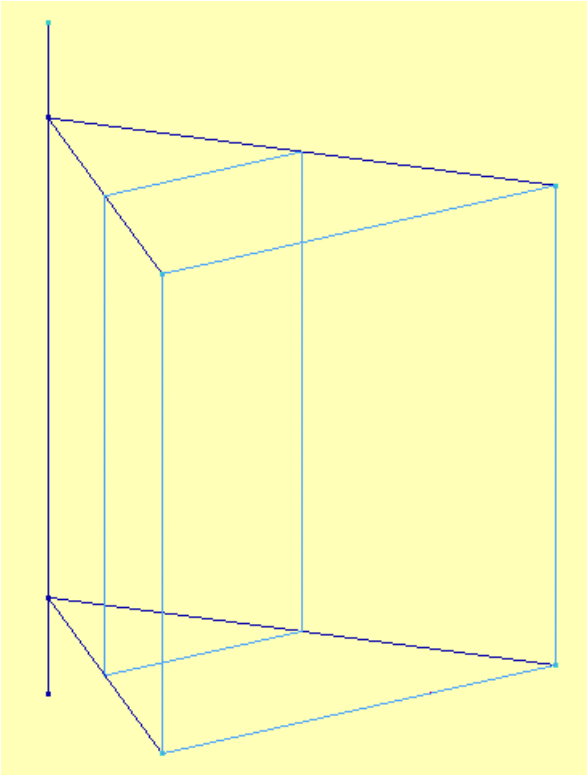
The possible modifications that you see in this window will be common for all the entities. This is the reason why it is a good moment to explain them.

- Properties: This task allows knowing the main properties of the popup-selected entity. In this way, the program will display a new screen in which the user can check the entity type, its identification, its description (editable and modifiable) and finally information about the entity. In this case, the selected entity is one point then we will see the coordinates of this one.
- Select object: This task allows selecting or unselecting the entity.
- Visibility: This task allows hiding the entity.
- Zoom: This task allows making zoom, and the center of the new window will be the selected entity.
- Delete: This task allows deleting the entity.
- Transform: This task allows transforming the selected entity. It is also located below EDITION menu and in this case the program will actuate over the selected objects. The possible transformations to be made are the next:
    - Translate: It performs the translation in whichever direction of the selected entity. The translation vector can be defined given the three coordinates, modulus and direction or two points.
    - Rotate: It performs the rotation of the popup-selected entity. To rotate entities is necessary to input as data the rotation axe (as vector), one point and finally the rotation angle (positive in counter clockwise).
    - Point scale: it performs point scaling over the popup-selected entity. For this point scaling it is necessary that the user will input the point and the scale factor. The new coordinates will be calculated as you can see in the next figure:

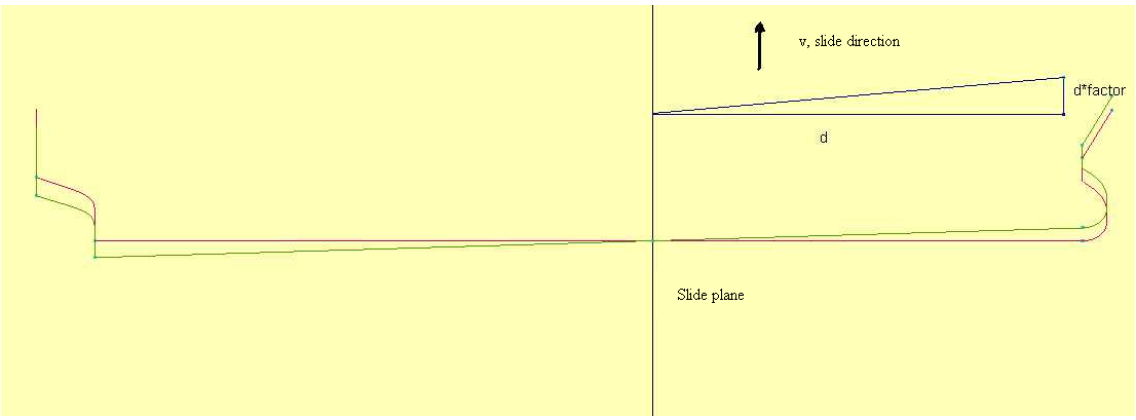
$$X_{new} = X_{old} * SCALEF$$
$$Y_{new} = Y_{old} * SCALEF$$
$$Z_{new} = Z_{old} * SCALEF$$



    - Line scale: It performs a line scaling over the popup-selected object. For this line scaling it is necessary to input the line (vector and point) and the scale factor.



- Plane scale: it performs a plane scaling over the popup-selected objects. For this plane scaling it is neccessary to input the plane and the scale factor. If the scale factor is 0 the program will perform the projection of the selected entity on the plane.
- Simetry: It performs a symetry on the popup-selected objects. The user will input the symetry plane.
- Deslizar por un plano: It performs a sliding on a plane of the popup-seleceted object. This task is very useful when the user wants to change the design trim of on ship. The user will input the sliding plane, the sliding direction and finally the sliding factor.



- Copy: The command copy has the same tasks explained above (transformations). The difference with the transformations is that in this case the program will maintain the original and the transformed entity.

4.1.2 Auxiliary polylines:

This polylines will be used to define another entities (mainly curves) or to modify some of them.
The polylines definition is located below ENTITIES menu and it can be defined as follow:

- By points: The polyline will pass through the points seleted by the user) one by one). The sorting of the polyline will be the sequence in that is defined.
- By point's cloud: The polyline will pass through the auxiliary points selected by the user. In this case the program will sort automatically the line according to the minimum distance criterion. The program also calculates the first point.
- By pol section: The polyline will be defined as intersection between a set of polylines and one plane. The user will input the polylines to be intersected and the plane. The program obtains the new one.
- By curve: There are some different types of definitions to obtain one polyline using curves. The possibilities are the next ones:
    - Equal: One (or several) polyline will be created with equally spaced points passing through the curve points.
    - Aprox: One polyline will be created passing through the curves points, but in this case, the program will decide the points it needs to fit the polyline.
    - Sections: One polyline will be created intersecting a set of curves and one plane.
    - Ctr line: One polyline will be created, and this one will be the control polygon of the curve (B-spline).
- By intersection: Definition of a polyline as a result of the intersection between patches or between patch and plane. To do that the program will request to the user that selects the patches that he wants intersect and later the intersector patch or plane.
- By GRID: Definition of a polyline as a result of transforming in polylines the visualisation of curves that the user has performed previously from a grid (FR, WL or BT), in the selected patches.
- By CLIPP: Definition of a polyline as a result of obtaining the clipping line of a patch.

The modification of the auxiliary polylines is possible from the POP-UP, or from the menu MODIFY. Firstly we will analyse the tasks located below the menu MODIFY, and later we will explain the POP-UP.
The possible polylines modifications that can be performed from the menu MODIFY - POLYLINE are the following ones:

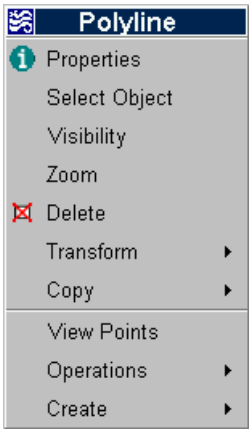
- Join: Creation of a new polyline as a result of the union of two or more polylines.
- Split: Creation of two polylines as a result of splitting an existent one in a point. This point can be calculated in the following ways:
    - On plane: The user should choose a plane to split the selected polylines. The program will check for each polyline the points on the plane and the points outside of it and it will create two new polylines,

the first one with the contained points on the plane and the second with the points outside of it. This command is very useful to look for the FOB or FOS lines in a dxf file.

- By plane: The user should choose a plane to split the selected polylines. The program will look for for each polyline the point of split of the plane with the polyline and it will perform two new polylines, one to a side of the plane and another to the other one.
- By polyline: The user should choose a polyline to split those selected. The program will look for for each polyline the nearest point to the splitter polyline and it will create two new polylines, one to a side of the polyline and another to the other one.
- By curve: The user should choose a curve to split those selected. The program will look for for each polyline the nearest point to the splitter curve and it will create two new polylines, one to a side of the curve and another to the other one.
- By patch: The user should choose a patch to split the selected polilíneas. The program will look for for each polilínea the point of split of the patch with the polyline and it will perform two new polylines, one to a side of the patch and another to the other one.
- By angle: The user should choose the angle that he wants to use to split the selected polylines. The program will look for for each polyline the adjacent segments that form superior angles to the one selected by the user, splitting the polyline in this point. This command is very useful to split polylines in knuckles in dxf files.

- Reverse: Modification of a polyline as the result of reverse its sorting. The first point will become the last one, and so on.
- Sort: Modification of a set of polylines to get the same way of sorting.
- Join and Split: it is the result of applying the tasks JOIN + SPLIT - BY – ANGLE. This command is very useful to import dxf files.

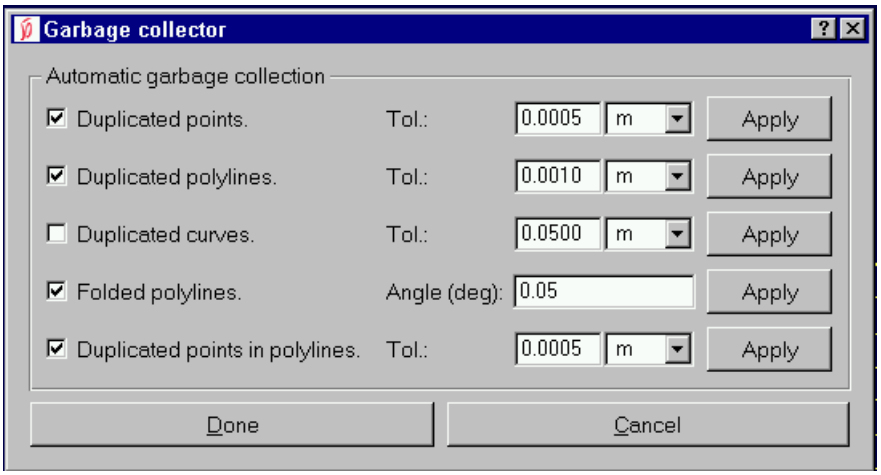
Clicking with the right button of the mouse over the polyline, the program will display the next POP-UP:



The tasks that appear on the top of this window, are those common to all the entities explained in the POP-UP of a point. The rest of tasks are explained now:

- View points: It allows the user the visualization of the points of the selected polyline.
- Operations: It allows the user to apply determined modifications on the polyline or their points:
    - Points: It allows modifications on the points of the polyline:
        - Add points
        - Erase points
        - Move the selected point selecting their new position.
        - Translate the point selected according to a vector.
        - Rotate the selected point.
        - Move the selected point interactively with the epsilon selected below the EDITION- PREFERENCES menu.
    - Reverse: explained previously below the menu modify.
    - Split: It allows the user to split the polyline in the selected point.
    - Split-up by angle: explained previously below the menu modify.
    - Trim: explained previously below the menu modify as task SPLIT BY PLANE.
        - Create: It allows the user the creation of other entities based on the polyline:
            - Points: It allows to create auxiliary points based on the polyline:
                - Polyline points: It will create the points of the polyline as auxiliary points.
                - By intersection: It allows to create auxiliary points by intersection between the polyline and other entities (another polyline, a plane or a patch).
            - Curves: It allows to create curves based on the polyline. As we will see later, these curves will be able to be defined according to the control polygon, by approximation or by interpolation.

At this time, it suits to explain another command that is located in another menu but that is used very often, overalls to import dxf files. This command of which we are talking about is located below the menu TOOLS - GARBAGE COLLECTOR. This command allows the filtrate of the entities that the user selects with a tolerance. This tolerance can be modified by the user in the window of use of this command for each one of the evaluated entities. The possibilities are the following ones:



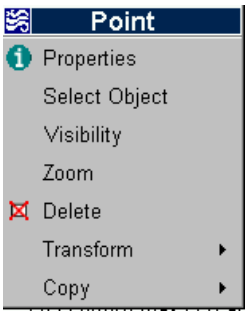
- Possibility of deleting separate points in a smaller distance that the fixed tolerance.
- Possibility of deleting separate polylines in a smaller distance that the fixed tolerance. The polylines will be checked segment by segment so that the program considers both as the same one the polyline. Command very useful that will avoid later problems in the definition of patches (duplicated curves in the construction of a patch would generate problems)
- Possibility of deleting separate curves in a smaller distance that the fixed tolerance. This command is not exact although it can be quite useful.
- Possibility of deleting backward segments in the polylines. Command very useful to import dxf files, to eliminate points that later could generate problems in the construction of the curves.
- Possibility of deleting points of separate polylines in a smaller distance that the fixed tolerance. Command very useful that will avoid later problems in the creation of the curves (Duplicated points in the construction of a curve would generate problems).

4.1.3 Auxiliary planes:

These auxiliary entities will be used for the later definition of other entities or for the modification of them.
The definition of auxiliary planes is below the menu ENTITIES and it will be performed according to:

- Plane: It allows the definition of an auxiliary plane. This can be one of the main planes, parallel to them, given by three points, etc.
- Group of planes: It can be created defining the planes one by one or giving a group of parallel planes.

The modification of the auxiliary planes is only possible from the POP-UP of plane. To do that therefore, we will click with the right button of the mouse over the plane and the program will display the corresponding menu:



The possible modifications that appear in this window, will be common to all the entities they were explained in the point menu.

4.2 Entities in surface:

4.2.1 Curves:

These entities will be used for the later definition of patches and/or lines of the ship or for the modification of them.
The definition of curves is below the menu ENTITIES and it will be perform according to the following form:

- By points: It allows the definition of a curve defined by points. This definition can be performed in three different ways:
  - Control pol.: The set of points defined by the user is the control polygon of the defined curve (it is the option fewer used).
  - Approximation: The program will create a curve that will go through all the selected points (with the tolerance defined in PREFERENCES) creating the control polygon for this curve (it is the used method in definition).
  - Interpolation: The program will create a curve that will go through all the selected points creating the control polygon for this curve (this method is also very used, but it is not advisable when one wants to use a very high number of points). In this case, the user even has the possibility to define the initial tangents and end for this curve.
- By polyline: It allows the definition of a curve defined by a polyline. This definition can be performed in three different ways:
  - Control pol.: The polyline defined by the user is the polygon of control of the defined curve (it is the option fewer used).
  - Approximation: The program will create a curve that will go through all the points of the selected polyline (with the tolerance defined in PREFERENCES) creating the control polygon for this curve (it is the used method in definition).
  - Interpolation: The program will create a curve that will go through all the points of the selected polyline creating the control polygon for this curve (this method is also very used, but it is not advisable when one wants to use a polyline with a very high number of points). In this case, the user even has the possibility to define the initial tangents and end for this curve.
- Circle Arc: The program allows the definition of circle arcs. This definition can be performed in two different ways:

- By three points: The program will create a circle arc that will go through the three selected points.
- By TG + radius: The program will create a circle arc that will be tangent to the points selected with the radius given by the user. To execute this command three points will be selected, both first ones will be the tangency points and the third one will indicate toward the radius goes. Lastly the program will request the user the radius of the arc.

- By intersection:

  This method is used to obtain the curves as intersection between diferent entities.

  The available possibilities are:

  - Patch - plane : Creation of a curve as intersection between patches and plane.
  - Patch - patch: creation of a curve as intersection between a set of patches.
  - Curves Grid: It allows to create as curves the lines drawn previously as grid.

  This command is very useful during the fairing of a patch, so that it allows to note the change in the others proyections when one of them is being modified.
  To draw the grid the user should use the following command:
    - VIEW - SHOW - CURVES GRID: This command allows to the user to intersect a selected set of curves with other curves. To do that the program will show a window in which the user will select the types of planes which he wants to intersect (X, Y or Z). Later, the point from which is wanted to begin the visualization, will be enter. Finally to introduce the number of planes to trim the set of curves and the space between these planes. This has even the possibility of checking the bend function for the lines shown as grid.
    - VIEW - HIDE - CURVES GRID: This command hides the previously visualized grid.

- Patches limits: creation of curves from patch limits.
- Curves from grid of patch: creation of curves from a grid of patches. The user has to select the set of patches and to define a grid.

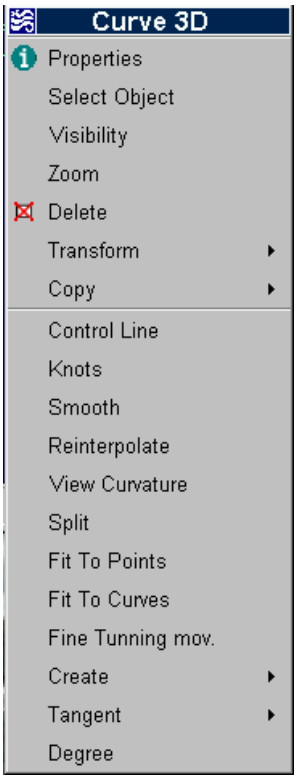
The modification of the curves is possible from the POP-UP, or from the menu MODIFY. Firstly we will analyze the tasks located below the menu MODIFY, and later we will center us in the POP-UP.
The possible modifications of curves that can be performed from the menu MODIFY - CURVES are the following ones:

- Reinterpolate: This task allows to the user recalculate a previously defined curve with a number of knots (given by the user) equally spaced. This task is very useful when patches are being defined so that its control mesh is the most uniform possible.
- Smooth: This task allows the user a process of automatic smoothing. When this task is executed, the program will evaluate on the curve a great quantity of points and using the approximation method later, the program will take a new list of points that will improve the smoothing of the line. The curve will be approximate with the smallest change of bend using the function of more homogeneous bend.

- Fit to curves: This task allows the user to approximate a set of curves to others. To do that the user has to select the curves that he wants to approximate and later the curves to which he wants to approximate the first ones.
- Join: This task allows the user the creation of a new curve as a result of the union of two or more curves. It should be realized that in the case of the existence of a knuckle like point of union of both curves this will get lost in this process.
- Split: This task allows the user the creation of two curves as a result of splitting an existent one by a point. This point can be calculated in the following ways:
  - By plane: The user should choose a plane to split the selected curves. The program will look for for each curve the point of split of the plane with the curve and it will perform two new ones, one to a side of the plane and another to the other one.
  - By polyline: The user should choose a polyline to split the selected curves. The program will look for for each curve the nearest point to the splitter polyline and it will create two new ones, one to a side of the polyline and another to the other one.
  - By curve: The user should choose a curve to split the selected curves. The program will look for for each curve the nearest point to the split curve and it will create two new, one to a side of the curve and another to the other one.
  - By patch: The user should choose a patch to split the selected curves. The program will look for for each curve the point of split of the patch with the curve and it will perform two new ones, one to a side of the patch and another to the other one.
  - By inflection points: The user should choose the curves that he wants to split. The program will look for for each curve the bend changes that take place in her and later it will split the curves by the points where these changes take place.
  - By the nearest point: The user should choose a point located on an entity to split the selected curves. The program will look for the point from the nearest curve to the selected point and it will split the curve by this point.

- Tangents: The user has the possibility to introduce tangents in the extreme points of the curve. This tangency will be able to be defined in the following ways:
  - To curve: The program will define the tangency vector of the curve parallel to the tangency vector of the curve to which one wants to make it tangent.
  - To direction: The program will define the tangency vector of the curve parallel to the direction given by the user.
  - To plane: The program will define the vector of tangency of the curve containing it in a parallel plane (or in the same plane) to the given by the user.
  - To patch soft: The program will define the vector of tangency of the curve in a tangent plane to the patch in the nearest point to the curve.
  - To patch hard: The program will define the vector of tangency of the curve in a parallel plane to the traverse direction of the patch in the nearest point. That is to say, the program will define the tangency vector so that it is parallel to the tangent vector to the patch in the nearest point to the curve.

  Clicking with the right button of the mouse over the curve, the program will display the POP-UP:



The tasks that appear on the top of this window, are common to all the entities explained in the POP-UP of point. The rest of tasks are explained now:

- Control line: It allows the user to visualize the control polygon of the selected curve. This command activates the control polygon and it disables the curve. This implies that unless the curve has been selected previously, the user won't be able to work with it until he disables the control polygon. The possible actions that the user will be able to perform with the control polygon are the following ones:
  - Disable: It disables the control polygon leaving active the curve.
  - Fine tunning move: It allows the user to perform an interactive movement of the points of the control polygon. This movement can be on a plane or following a direction given by the user.
  - Translate: It allows the translation of the points of the polygon according to a vector given by the user.
  - Rotate: It allows the rotation of the points of the polygon according to the user's approach.
  - Move: It allows the movement of the points of the polygon to points that the user selects.
- Knots: It allows the user to visualize the knots of the curve. This command the same as the previous, active the knots of the curve and it disables the curve for any other action. This command in occasions can be dangerous of using since if we act on the knots of the curve, the control polygon will be a consequence of the curve and not to the inverse one that is like it should be, however in certain occasions (smoothed) it can be useful to act directly on the curve and not on the control polygon. The possible actions that the user will be perform with the knots are the following ones:
  - Disable: It disables the knots leaving active the curve.

        - Delete: It allows the user to delete a knot of the curve.
        - Fine tunning move: It allows the user to perform an interactive movement of the knots. This movement can be on a plane or following a direction given by the user.
    - Smooth: Explained previously below the menu modify.
    - Reinterpolate: Explained previously below the menu modify.
    - View curvature: It allows the user the visualization of the bend function.
    - Split: It allows the user to split a curve by any contained point in it.
    - Fit to points: It allows the user to modify a curve locally to approximate it to a previously selected set of points.
    - Fit to curves: Explained previously below the menu modify.
    - Fine tunning move: It allows the user to perform an interactive movement of the points of the curve. This movement can be on a plane or following a direction given by the user.
    - Create: It allows the user the creation of other entities based on the curve:
        - Points: It allows to create auxiliary points based on the curve, these points will be calculated by intersection of the curve with other entities (a polyline, a curve, a plane or a patch).
        - Polylines: It allows to create polylines based on the curve. By approximation (the program will create a polyline, taking into account the tolerance fixed in PREFERENCES, so that all its points goes through the curve and in turn the curve formed by this points is the curve of which left) or control polygon so that the created polyline is the control polygon of the curve.
        - Curves comb. : It allows to the user to define a curve as a combination of two already existent curves, the one selected in the POP-UP and another. To do that the program will request a second curve and the approximation percentage to the first curve.
    - Tangent. Explained previously below the menu modify.
    - Degree: It allows the user to modify the degree of the curve.

4.2.2 Patches:

These entities are the final entities that will form the hull of the ship and any other surface (interior hulls, appendixes, etc.).
The definition of patches is below the menu ENTITIES and it will be perform according to the following form:

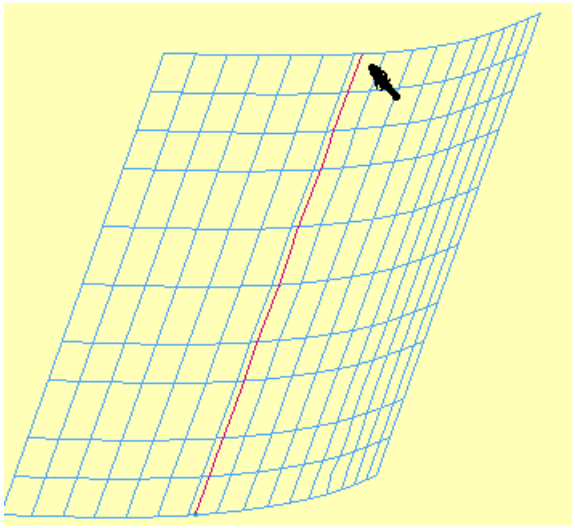
- Approximation: The program will create a patch that it will go through all the selected curves (with the tolerance defined in PREFERENCES) creating the control mesh for this patch.
- Interpolation: The program will create a patch that it will go through all the selected curves creating the control mesh for this patch (this method is the more used for the definition, but it is not advisable when one wants to use a number of curved very high).
- By limits: The program will create a patch whose limits will be the curves selected by the user. For the definition of this type of patches the program will display a window in which the user will introduce the limits of the

patch. The user should enter the limits in pairs, that is to say, to select the opposed curves that will define the first limits first and without closing the window, he will select the other ones two limits. The program will calculate the patch taking into account a lineal interpolation of the tangents of an extreme from the patch to the other one. This type of patches is advisable in plane areas or in small areas with bend. The user even has the possibility to define triangular patches (although these they are not very advisable). Finally there is the possibility of defining only two limits, in this case the program will join these limits with a straight line.

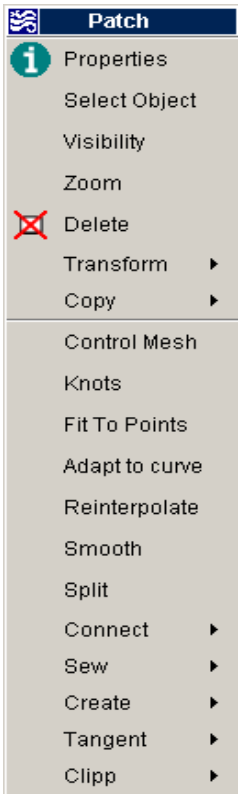
- By trimming limits: The program will create a patch whose limits will be curves that they trim themselves. To do that the program will look for the points of intersection of the curves and later it will create a patch taking into account the limits curves that limit it.

The modification of the patches is possible from the POP-UP, or from the menu MODIFY. Firstly we will analyze the tasks located below the menu MODIFY, later we will center us in the POP-UP.
The possible modifications of patches that can be performed from the menu MODIFY - PATCH are the following ones:

- Reinterpolate: This task allows to the user to recalculate a previously defined patch with a number of knots equally spaced in the directions “u” and " v " given by the user. This task is very useful to make standard a patch, but CAREFUL! The patch doesn't maintain the limits when this command is executed, that could give place to enter holes between patches.
- Smooth: This task allows the user a process of automatic smoothing. When this task is executed, the program will request the user the number of knots in “u” and " v " with you wants to perform this action. The program will evaluate on the patch a great quantity of points and using the approximation method later the program will take a new list of curves (reparametered with the selected number of points) that improves the smooth of the patch. The patch will be approximate with the smallest change of possible gaussian bend using the function of more homogeneous bend.
- Split: This task will allow the user to split the patch by an isoparametric curve.To do that, the program will request the user the patch that he wants to split and later a point. The program will split the patch by the isoparametric in the opposite direction to the limit nearest to the selected point. In the explanatory figure one can see that selected the patch in question, if the selected point is the pointed with the cursor, the program will split the patch by the isoparametric curve in red color, since, the nearest limit to the position of the cursor goes in the other direction.



Clicking with the right button of the mouse over the curve, the program will display the POP-UP:



The tasks that appear on the top of this window, are those common to all the entities explained in the POP-UP of a point. The rest of tasks are explained now:

- Control Mesh: It allows the user to visualize the control mesh of the selected patch. This command activates the control mesh and disables the patch. This implies that unless the patch has been selected previously, the user won't be able to work with it until it disables the control mesh. The possible actions that the user will be able to perform with the control mesh are the following ones:
  - Disable: It disables the control mesh leaving active the patch.

    - Inter move: It allows the user to perform an interactive movement of the points of the control mesh. This movement can be on a plane or following a direction given by the user.
    - Translate: It allows the translation of the points of the mesh according to a vector given by the user.
    - Rotate: It allows the rotation of the points of the mesh according to the user's approximation.
    - Move: It allows the movement of the points of the mesh to points that the user wants.
    - Knots: It allows the user to visualize the knots of the patch. This command the same as the previous, active the knots of the patch and disables the patch for any other action. The only possible action that the user will be perform with the knots is the following one:
        - Disable: It disables the knots leaving active the patch.
- Fit to points: It allows the user to modify a patch locally to approximate it to a set of previously selected points.
- Adapt to curve: It allows the user to modify a patch locally to approximate it to a curve. This command will be used as a smooth command not as an adjustment command. The program will deform the patch to adapt it to the curve. For their use the program will request the user the curve and the rigidity. This last one will be a value between 0 and infinite. The standard value will be 1. As much as minor this value is, the bigger it will be the adaptation from the patch to the curve. RECOMMENDATION! not to introduce the value 0, this could give place to vagueness in the algorithms of mathematical calculation.
- Smooth: Explained previously below the menu modify.
- Reinterpolate: Explained previously below the menu modify.
- Split: Explained previously below the menu modify.
- Connect: Moves the control points of one side of the patch to make them equal to the ones at the target entity, which may be another patch side or a curve. Both connecting entities (path side or curve) must have the same degree and number of control points.
- Sew: It allows the user to sew patches or to sew a patch to a curve. Depending on the performed definition of patches, it can be possible that two adjacent patches have different limit curve. It is for this reason that this command exists. The user should click with the mouse near the border that he wants to sew and later to select the patch (curve) that he wants to sew this patch to. The program will reinterpolate the patch to sew so that the limit of this patch it is similar to the limit of the other one (or to the curve) taking into account the tolerance defined by defect. Whenever one wants to sew patches or patch to curve, the curve limit of the patch to sew, should be smaller or the same as the curve limit to which you want to sew. It is at this time that it would be good to explain some new commands:
    - TOOLS - SURFACE ANALYZER: It allows the user to know the holes between adjacent patches taking into account the tolerance defined by default. The holes will be shown in yellow color and they won't be editable.
    - VIEW - HIDE - HOLES BETWEEN PATCHES: It allows the user to hide the holes between patches. In case you want to see the holes, you have to obtain them again.
- Create: It allows the user the creation of curves and polylines based on the patch:

POLYLINE:

- Control polyline: It allows to create auxiliary polylines based on the patch, these polylines will be those corresponding to the isoparametric curves that define the patch.
- Intersect with patch: It allows to create polylines as a result of the intersection between patches. The program will intersect the patch POP-UP with the patches selected by the user.
- Intersect with plane: It allows to create a polyline as a result of the intersection of the patch with a plane.
- Grid polylines: It allows to create as polylines the lines drawn previously as grid. To draw the grid the user should use the following command:
    - VIEW - SHOW - PATCHES GRID: This command allows to the user to intersect the selected patches with faces of parallel planes to the projection planes. To do that the program will show a window in which the user will select the types of planes which he wants to intersect (X, Y or Z). Later, the point from which is wanted to begin the visualization, will be enter. Finally to introduce the number of planes to trim the surface and the space between these planes. The has even the possibility of checking the bend function for the lines shown as grid.
    - VIEW - HIDE - PATCHES GRID: This command hides the previously visualized grid.

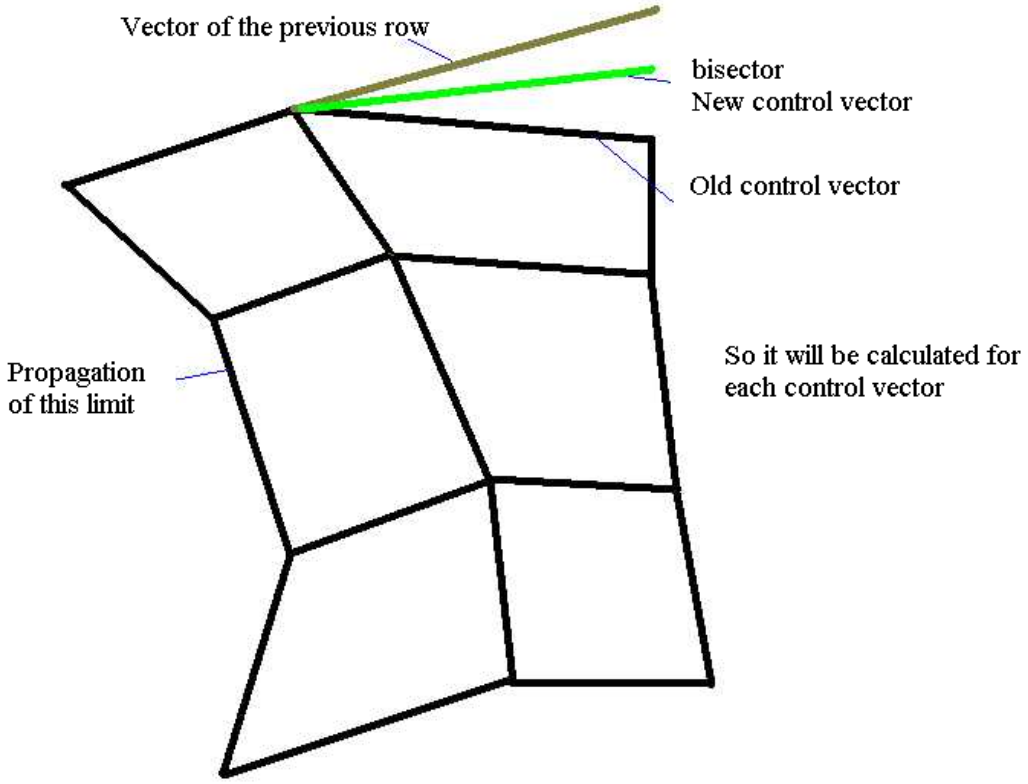
- Clipp polylines: It allows the user to create a set of polylines the clipping lines of a patch.

CURVE:

- Isocurve: the command ISO - CURVE allows the definition of a isoparametric line in the point selected by the user, giving the possibility to create the curve according to the direction “u” or, " v " or both.
- N- Isocurve: the command N - ISO - CURVE allows the definition of a set of isoparametric lines in the direction “u” or, " v " or both. If in this command the user decides to define only two curves, the program will define like curves the limits of the patch.
- Intersection patch – patch: to obtain a curve as intersection between patches.
- Intersection patch – plane: to obtain a curve as intersection between patch and plane.

- Clipping: The user has the posibility of clipping a patch. To perform the clipping in a righ way, it is necessary to click with the mouse and display de POP-UP, on the side of the patch that it wants to keep. When this action is donek the other side will be removed. The clipping it can be done in several ways:
    - By plane: The program will request to enter a plane which it wants to clipp the patch with.
    - By patch: The program will request the selection of the intersector patch for to perform the clipping.
    - By polyline: The program will request the selection of a polyline on a patch (from a patch limit to other patch limit) which the clipping wants to perform with.
    - Remove clipping: If this option is selected, the clipping performed on the patch will be removed, and the patch will keep its original form.

- Tangents: The user has the possibility to enter tangents in the limits of the patch. This tangency will be able to be defined in the following ways:
    - To direction: The program will define the tangent vectors from the parallel patch to the direction given by the user.
    - To plane: The program will define the vectors of tangency of the patch containing them in a parallel plane (or in the same plane) to the given one by the user.
    - To patch soft: The program will define the vectors of tangency of the patch in a tangent plane to the reference patch in the nearest points to this patch.
    - To patch hard: The program will define the vectors of tangency of the patch in a parallel plane to the traverse direction of the reference patch in the nearest point. That is to say, the program will define the tangency vectors so that they are parallel to the tangent vectors to the reference patch in the nearest points to the patch.
    - Propagation: The program will allow the propagation of the tangent in the limits of the patch. To do that the user will select the limit from which wants to propagate the tangent and the number of propagation rows. For each row the new control vector will be calculated as the bisector of the angle formed by the vector of the previous row and the vector to modify (to see fig.).



4.2.3 Facets:

These entities will make up the external hull and whichever surface (internal hulls, appendages, etc)

Facets definition is below the menu ENTITIES and it is treated as a piece of plane limites by a contour.

Its definition can be done with:

- Entities->Facet->By contour: The definition is the same as patch by limits, but in this case the user selects the contour defined by polylines and curves.
- When defining a planar patch by limits, it is posible to choose between defining a patch or a facet, selecting in Edition->Preferentes, the option: "Use facets as planar patches". If this option is activated, the program will generate automatically a facet. If the option is not activated, the program will request if the user wants to create a facet or a patch.

Clicking on the right button of the mouse on the facet, the program will display the following POP-UP:

The tasks in the up part of this window, are commons to all entities (already explained in POP-UP of an auxiliary point). The rest of tasks are explained then:

- Create:
  - Intersection with patch: a polyline is generated as intersection between the facet with a set of patches of facets
  - Intersection with a plane: a polyline is generated as intersection between a facet and a plane.
  - Plane of facet: it is obtained a plane where the facet is defined.
  - Clipp polylines: it is obtained the contour and the clips polylines of the facet.

- Clipping:
  - To clip a facet by a polyline: the facet is clipped by a polyline.
  - To clip a facet by a plane: the facet is clipped by a plane.
  - To clip a facet by a pathc: the facet is clipped by a patch.
  - Remove clipping: the clipping of the facet is removed.

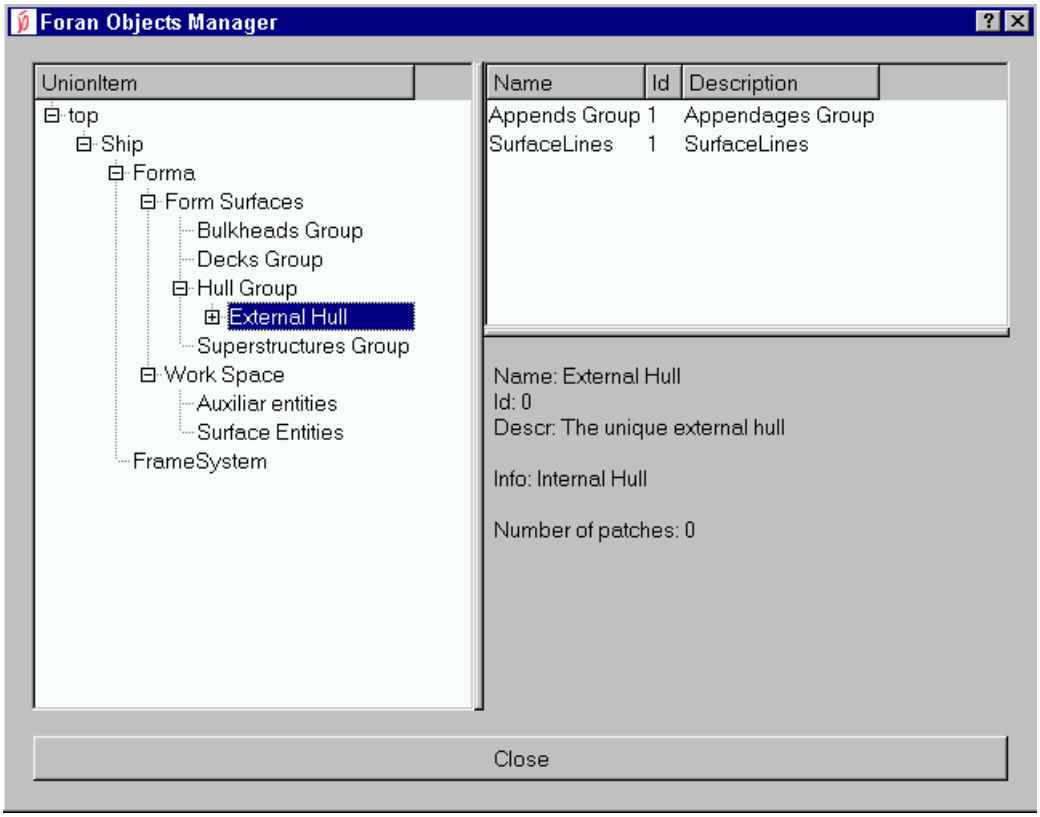
5. Revision of commands:

The module has a serie of commands that can be important of use and that we have not seen along this guide. We will make in this moment a summary of the most important and that they are often used:

- Método: It allows the user to visualise the surfaces in two different ways. The first one as wire model and the second as softened. This command is below VIEW.
- Light focus: It allows the user the placement of focuses for a better visualisation of the surface in softened way.
- Patches Gaussian Mesh: It allows the user the check-up of the gaussian curvatures of one or more patches.
- Information: It allows the user the check-up of points, angles and distances. It also allows the user to know the elements that compose a scene. This command is located below TOOLS.

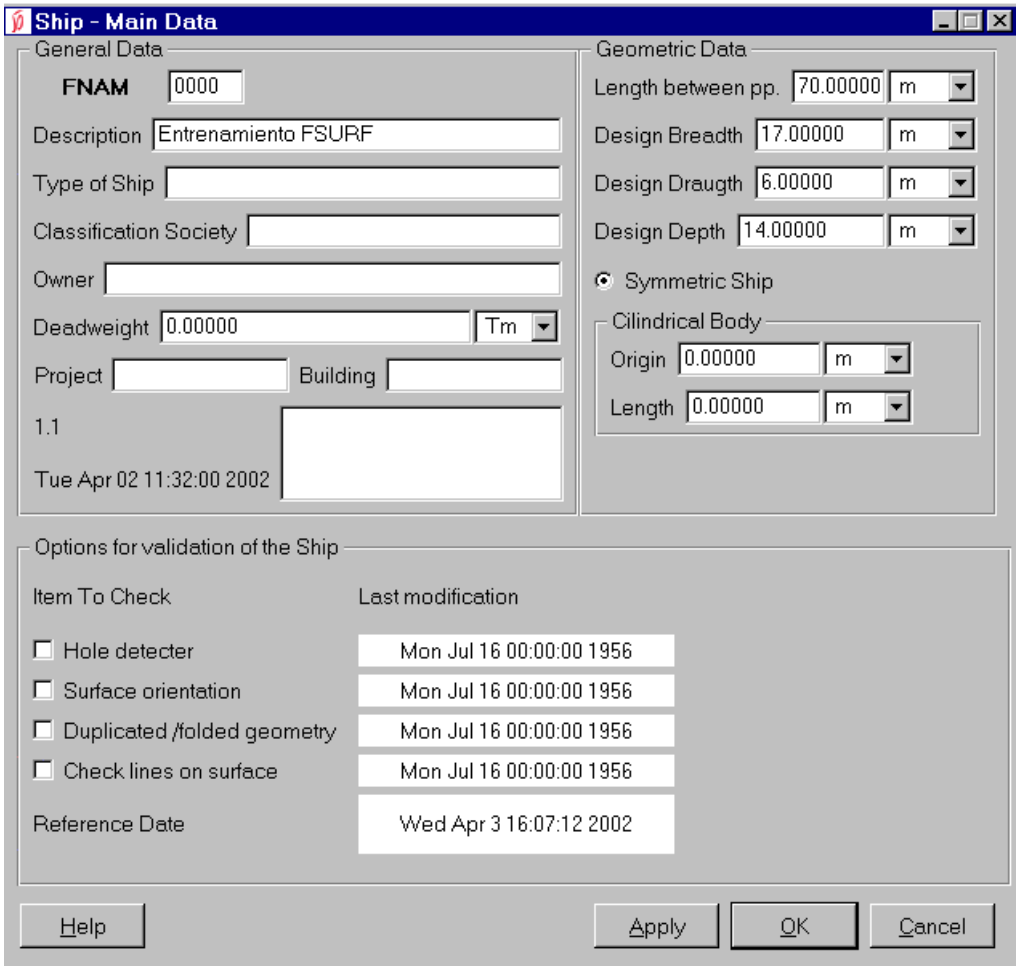
6. Definition of the forms file:

To create the form file of a ship, the user has to perform diverse tasks that will be explained next.
Firstly the user will define, at least, the line of profile of the ship. To do that firstly will be necessary to select (or to define) the curves that will form this line. Once those curves have been defined, below the command STRUCTURE - LINE the user will select the creation of a new line. Selected this command, the program will show the user the window to select curves, so that they are selected those that formed the line. Once executed this task, the program will show the user a new window in which can define the type of line (compulsory), an identifier, a name and a description for this line.
The following step will be the association from the patches to the external hull. To do that it exists the command TOOLS - MANAGER. This command (MANAGER) allows the user the handling of entities and their association. In this way one will be able to select, to visualize, etc., entities. It exists also the possibility to associate patches to different surfaces of the ship, as well as the association of different types of lines to the external hull. To perform this task, it is enough with selecting on the entity we want to act and dragging it with the mouse will associate it to any surface or clicking in the right button of the mouse the program will show the POP-UP of the entity and we will be able to act on her. The window that manages all these possibilities is the following one:



In the case that occupies us that it is the creation of the ship, the first thing it will be to associate the profile to the group of lines of the external helmet. To do that we will display the directory Forma\Form Surfaces\Hull Group\External Hull". Subsequently we will display the directory Work Space and we will click with the mouse on the menu “Surface Entities". In the right part of the window the program will show all the curves, patches and lines that the user has defined during the execution of the program. We will click on the profile line and without stopping to click, we will drag it with the mouse to the directory " SurfaceLines " located below “External Hull". Subsequently we will select all the patches that will form the external hull of the ship. To do that we can use the keys “control” and “shift". Again, without stopping to click with the mouse we will drag this patches to the directory “External Hull". If we wanted to add more definition lines (knuckles, sustentation, etc.) we would act the same as in the association of the profile. In the same way, if we want to associate any other surface like appendix, it would be included below the directory “Appends Group". ATTENTION!, If a patch already existent is modified in any other session of the module, the user won't need to associate this patch again to the external hull (or to any other surface), automatically the program will know the localization of the patch. But if the user deletes a patch and/or defines a new one, if one wants that this patch is part of one of the surfaces of the ship, it has to be assigned as such in the MANAGER. To create a ship with the FSURF the user should take into account that he should define both sides it is or not a symmetrical ship. Therefore, if the user only worked with one of the sides, he should make the symmetry regarding the plane Y=0. This would be the best moment, since, as the patches of the side have been assigned to the external hull, the symmetrical patches will also be associated to the external hull without necessity of using the MANAGER again.

Lastly the user will make use of the command FILE - SHIP - BILD SHIP. This command will display the following window:



It is compulsory to fill the field FNAM as well as the length between perpendiculars, the breadth, the draft and the depth. The rest of values can be filled or not, since these will be merely informative.
Although it is necessary to define with patches both sides of the ship, the user has to set the key for symmetry if this one is symmetric or not.
This key has to be set only when every patch belong strictly (all the control points) to one side of the ship (port or starboard). If some patch belongs to both sides, this key has not to be set.
Once filled the data that the user wants, the following step will be to execute the command OK. The program will create a new file named FNAMFSURF.FSF. This new file will be the forms file of the new ship. The command APPLY will store the data entered by the user in this window while the open session is executed, in the moment that the user closes session, the module will lose all the entered data.
The command CANCEL will escape from the window without saving any entered data.
If the user wants to perform any modification in the patches and to assemble the ship again, he will only have to execute again the command OK of this window, since the data are imported automatically from the forms file and the program will update this file.

From this point the user will be able to work with the rest of subsystems, placing the forms file in the corresponding MARHL directory.

Check- up of surfaces

There is the possibility of checking the surfaces before building the ship in a more or less automatically way. To do this action, the user has to select in the window different options:

- Hole Detector : check-up of holes between patches
- Surface Orientation: check-up of the surfaces orientation
- Duplicated/Folded geometry: check-up of duplicated geometry and folds. The program will remove automatically these geometries.
- Check lines on surface: check-up of the distance between the special lines and the container surface. A file named *FNAM_date(hour)VER.log* will be generated in the working directory. In this file will be the curves whose distance to the surface is greater as the allowed tolerance.

If one of the two first options is selected, the program will display a window with the defined surfaces, their states, the possibility of analyse them, to go back the previous option, to go to the next option or to cancel the process.
If during the analysis, it wants to modify a patch, it can be modified and then follow with the check-up process.

If the third or fourth option is selected, moreover the previous options, there is the possibility of checking all the surfaces simultaneously.

When all these actions has been performed, the last modification date will appear as an information data.

Additionally, it will appear the check-up reference date, it says, the date in which the user click on the OK button.

7. Analysis of the surface orientation:

Once the forms file FNAMfsurf.fsf is generated, suits to make an analysis of the orientation of the surfaces that we have until this moment. Especially the external hull, since a non appropriate orientation can generate problems when using the production modules.
The commands that allow this analysis are:
· SHOW - SURFACES ORIENTATION: it allows to select a hull or a deck. They are drawn in green colour the patches of the surface that have a positive orientation (normal vector toward the exterior) and in red those that have a negative orientation (normal vector toward the interior).
· HIDE - SURFACES ORIENTATION: hidden the red and green colours of the patches, visualised previously with the previous command.
· SHOW - NORMAL VECTORS: it shows the normal vectors of a group of patches.
· HIDE - NORMAL VECTORS: hidden the normal vectors previously visualised with the previous command.

· TOOLS - VECTORIAL NORMAL CHANGE ORIENTATION: it allows to carry out two actions. If a hull is selected, the program looks for its positive orientation. If a group of patches is selected, the program will only change the orientation of the normal vectors of this group.

8. Generation of drawing files:

Drawing files of the type * .d can be generated by means of the command FILE - PLOT. To do that it is necessary that the user selects previously the grid of polylines to draw. By means of the command PLOT it will be displayed a window in which the user will be able to choose the view in which wants the drawing, the objects to draw, the elementary drawings and the file * .d.

It is also possible to export this grid of polylines to a file * .dxf. To do that it is necessary that the user reads a hull or a set of patches, selects the grid of wanted polylines, and that he stores it with the command FILE - SAVE AS like * .dxf.
In the same way, it is possible to save as a *dxf file the special lines of a hull.

9. Hydrostatic calculations:

As complement to the forms generation, it is also possible to carry out hydrostatic calculations. To do that it is necessary to have generated the file FNAMfsurf.fsf and to execute the command FILE - SHIP - HYDROSTATIC CHARS.

This calculation does not take into account decks or any other type of surface, only it is applied on the external hull.

The program selects automatically a number of sections along the hull form in order to assure that the longitudinal integration will be performed with enough accuracy. In these sections the halfbreadths, areas, moments and girths are calculated at the required draughts, then a longitudinal integration is performed.

It is also possible to define appendages in this task, such as the rudder and the propeller, but only to be taken into account in hydrostatic calculations.
The calculations of the abscissa of the buoyancy centre with appendages (XCBA) takes also into account the rudder and propellers defined in this module.

This calculation gives the listed below characteristics of the ship:

- Design Breadth (maximum breadth of the hull form for the design draught).

- Design displacement for draught T (DISV).

- Block coefficient (for the design draught and the design breadth).
- Abscissa of the Centre of Buoyancy (XCB) (positive aft from M.S)
- Abscissa of the section with maximum transversal area (XAX) (positive aft from M.S).
- Maximum section coefficient (AX). Corresponding to the transversal section with biggest area to the design draught.

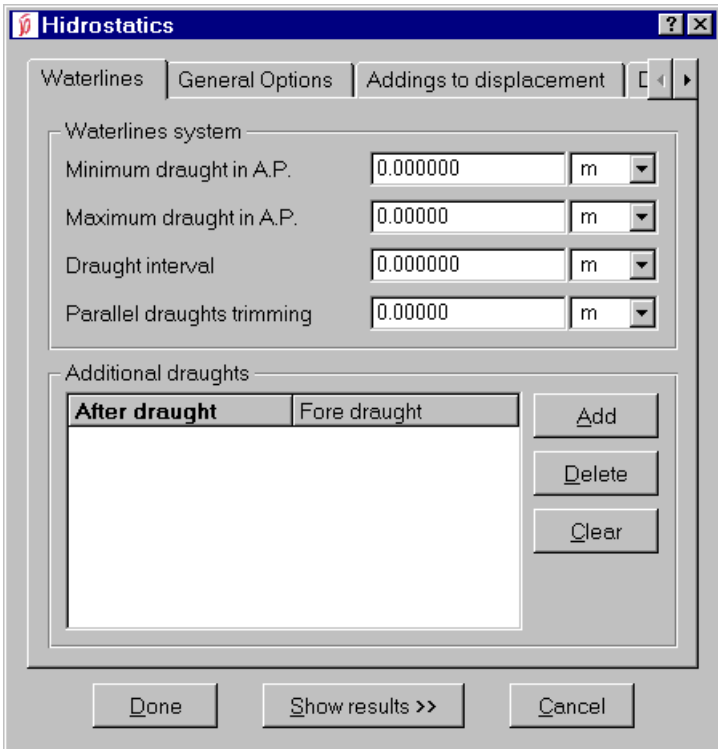
- Percentile area of the transversal section at the fore perpendicular (ABT).

- Waterline coefficient (CW).

- Longitudinal location of c. of g. of the waterline (LCW) (positive aft from M.S)

When this command is selected, the program will display a window to input data. This window has four parts:

9.1 Waterlines:



The user has to enter the minimum draught in A.P, the maximum draught in A.P, the draught interval, and the parallel draughts trimming if it is considered.

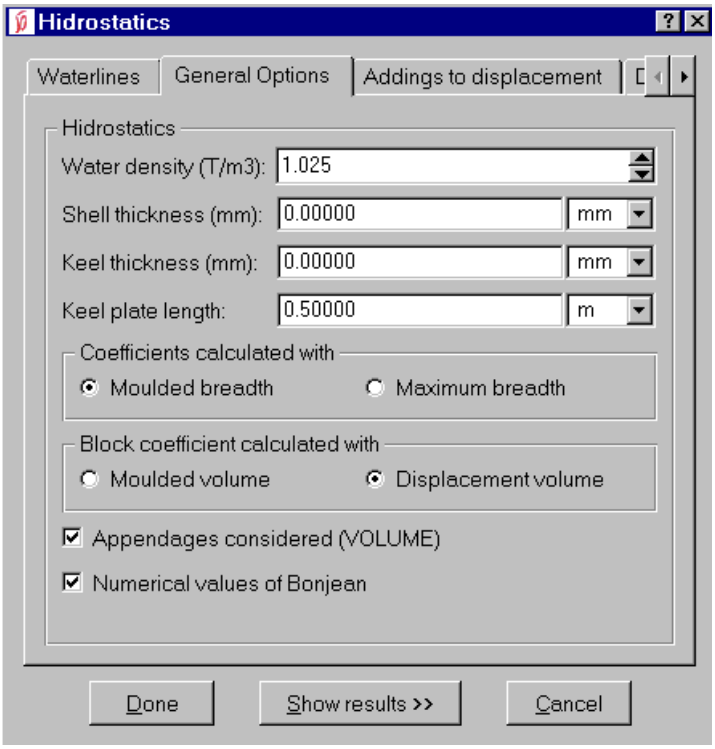
It is not necessary to enter all data. Only the minimum, maximum draughts in A.P, and an interval between them are compulsory. The program estimates automatically some values for these data and show them by default , but the user can change them when it is necessary.

If the Done button is clicked, the values will be saved, and the window will be closed. If the Show results button is clicked, the program will run the hydrostatic calculations and will show the results in a list. The results can be saved in a *.txt file with the button

"Save as...".

The list of calculated values can be sorted in different ways (just dragging with the mouse the chosen column) or evermore they can be hidden. (just clicking on a value of the chosen column with the right button of the mouse and selecting the option "hide" on the menu).

9.2 General options:



The available options are:

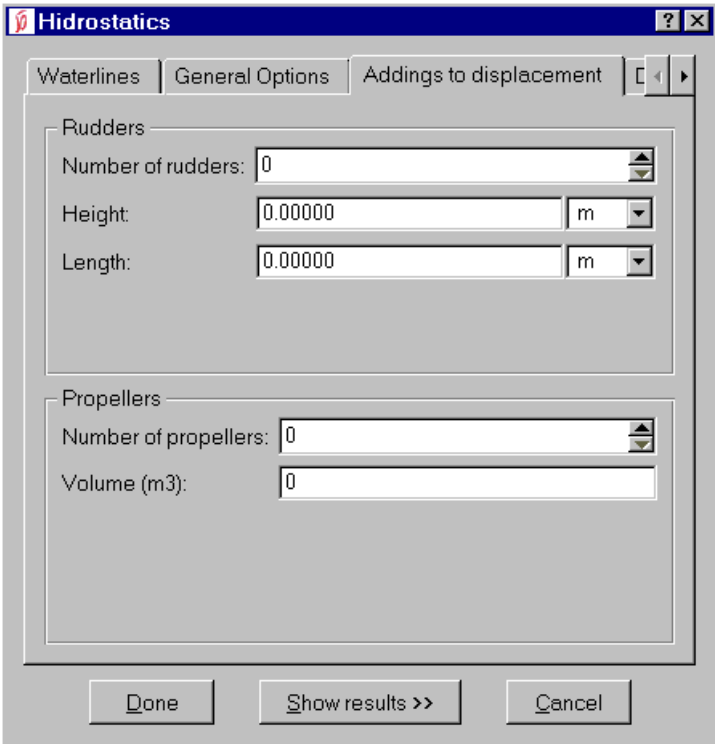
- **Moulded Breadth**: design breadth defined by the user with the command Build Ship.
- **Maximum Breadth**: maximum breadth calculated for every waterline selected in the calculations.
- **Displacement Volume**: it takes into account the possible appendages.
- **Moulded Volume**: it does not takes into account any appendages.

The main hydrostatics coefficients can be calculated as regards the MOULDED BREADTH or the MAXIMUM BREADTH in each waterline. The user can change the option just clicking on the button with the option indicated in. The maximum section coefficient will be calculated with the breadth of maximum section corresponding to each waterline.

The block coefficient can be calculated with the DISPLACEMENT VOLUME or the MOULDED VOLUME . The user can change the option just clicking on the button

with the option indicated in.

9.3 Adding to displacement:

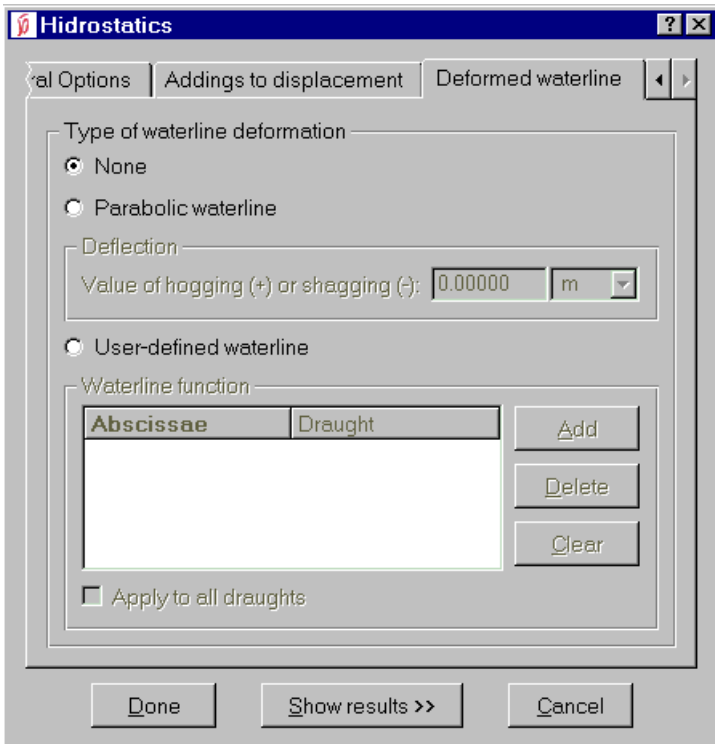


There are two options:

- Rudders: the user have to enter the number of rudders, height and length .

- Propellers: the user have to enter the number of propellers and volume.

9.4 Deformed waterline:



With the data it is possible to simulate deformed situations of the ship.

There are different options to enter this deformation:

- None: none waterline deformation is considered.
- Parabolic waterline: a parabolic deformation is considered. When this option is selected, the user has to input the value of hogging or shagging in midship.
- User defined waterline: the deformation is calculated with a distribution given by the user. Some abcissae and draughts must be entered to define this distribution. In this case it is possible to calculate this flotation or to apply this distribution to all the set of draughts.

10. Hydrostatic coefficient transformations:

10.1 LCB transformation:

This command will be used to modify the length of the center of buoyancy.

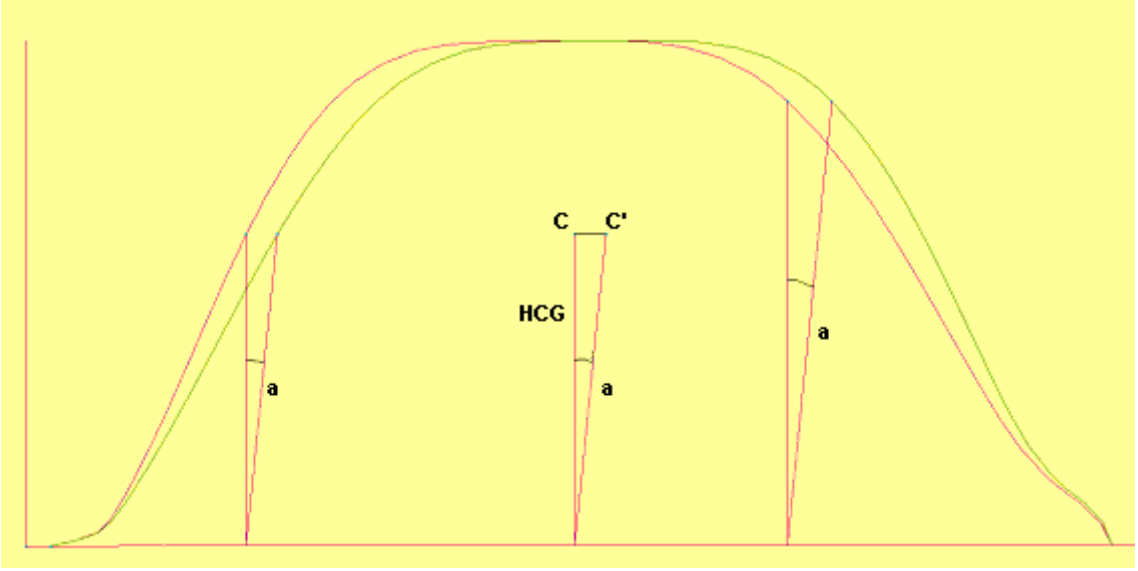
Formulation

The modification will be performed translating the transversal sections of the ship in the X direction. The coefficient of the translation will be calculated as follow:

$$\Delta x = area(x) \cdot \tan(a)$$

$$a = \arctan\left(\frac{CC'}{HCG}\right)$$

Where,
Area (x) is the area of the section in the abscissa 'x'
CC' is the movement of the center of buoyancy
HCG is the height of the center of gravity of the area curve



LCB transformation

Process

When this command is executed the program will display a new screen in which the user will input the new position of the center of buoyancy. This one will be referenced to the midship section and positive means aft wards.
In the same screen the program will show the current position of the center of buoyancy and the height of the center of gravity of the area curve.
Therefore, if the ship contains internal hulls these ones can be transformed in the same way that the external one. Selecting the ones to be transformed the program will apply the transformation to all of these ones and the external hull just in one action.
The screen displayed by the program will also show the current area curve of the ship. The Y-axis represents the percentage of the area referred to the maximum area. The X-axis represents the abscissa of the section. The screen will show the design sections.
When the user press the “preview” command the program will display in red color the new shape of the area curve according with the new position of the center of buoyancy.
When “accept” command is pressed the program will start an interactive process to get the new position of the center of buoyancy. If the program is not able to get the value input by the user in 20 iterations, it will appear a message indicating the last value achieved and it will give to the user the possibility to validate this one or to restore the original one.
The “cancel” bottom will stop the process without any transformation. During the iterations the program will show a new screen in which the user can check the intermediate result that the program fit before getting the value input by the user.

10.2 Block coefficient transformation:

This command will perform modifications in the block coefficient of the ship. This one is a very useful command when the objective is to increase the block coefficient, but it has to be managed carefully when the objective is to decrease it (in this case will recommend to use the quadratic transformation.

Formulation:

The modification is performed translating the transversal sections of the ship in the X direction following the next steps:

$$\begin{cases} 0 < X < \frac{L}{2} & X_N = X_0 \frac{CM - CB_N}{CM - CB_0} \\ X = \frac{L}{2} & X_N = X_0 \\ \frac{L}{2} < X < L & X_N = X_0 \frac{CM - CB_N}{CM - CB_0} + L \frac{CB_N - CB_0}{CM - CB_0} \end{cases}$$

Where,
CM is the midship coefficient
CBN is the new block coefficient
CBO is the current block coefficient
L is the length between perpendiculars
XN is the new abscissa of the section
XO is the current abscissa of the section

Process

When this command is executed, the program will display a new screen in which the new block coefficient has to be input. In this window will also appear the midship coefficient (it will not change) and the current block coefficient.

Therefore, if the ship contains internal hulls these ones can be transformed in the same way that the external one. Selecting the ones to be transformed the program will apply the transformation to all of these ones and the external hull just in one action.
When “accept” command is pressed the program will start an interactive process to get the new block coefficient. If the program is not able to get the value input by the user in 20 iterations, it will appear a message indicating the last value achieved and it will give to the user the possibility to validate this one or to restore the original one.
The “cancel” bottom will stop the process without any transformation. During the iterations the program will show a new screen in which the user can check the intermediate result that the program fit before getting the value input by the user.

10.3. Quadratic transformation:

This command will perform modifications in the block coefficient of the ship.

Formulation

The modification is performed translating the transversal sections of the ship in the X direction. The aft and fore body are independents, this is the reason why in some cases the length of the center of buoyancy can change at same time.

Process

When this command is executed the program will display a new screen in which the new block coefficient and the abscissas to be maintained (in aft and fore bodies) will be input.
In this new screen will appear the midship coefficient, the current block coefficient and the LCB. The new block coefficient has to be input.
Therefore, if the ship contains internal hulls these ones can be transformed in the same way that the external one. Selecting the ones to be transformed the program will apply the transformation to all of these ones and the external hull just in one action.
The abscissas of the aft and fore bodies that the user will maintain must be defined. By default the program will maintain the perpendiculars (aft and fore one) and the limits of the parallel body. If this last one does not exist the program will fit the midship section.
If “accept” bottom is pushed the data input by the user are checked and the program will give error messages in the case that any defined date is wrong. If all the data are right the program will start an interactive process to get the new block coefficient. If the program is not able to get the value input by the user in 20 iterations, it will appear a message indicating the last value achieved and it will give to the user the possibility to validate this one or to restore the original one.
The “cancel” bottom will stop the process without any transformation. During the iterations the program will show a new screen in which the user can check the intermediate result that the program fit before getting the value input by the user.

10.4 Area curve transformation:

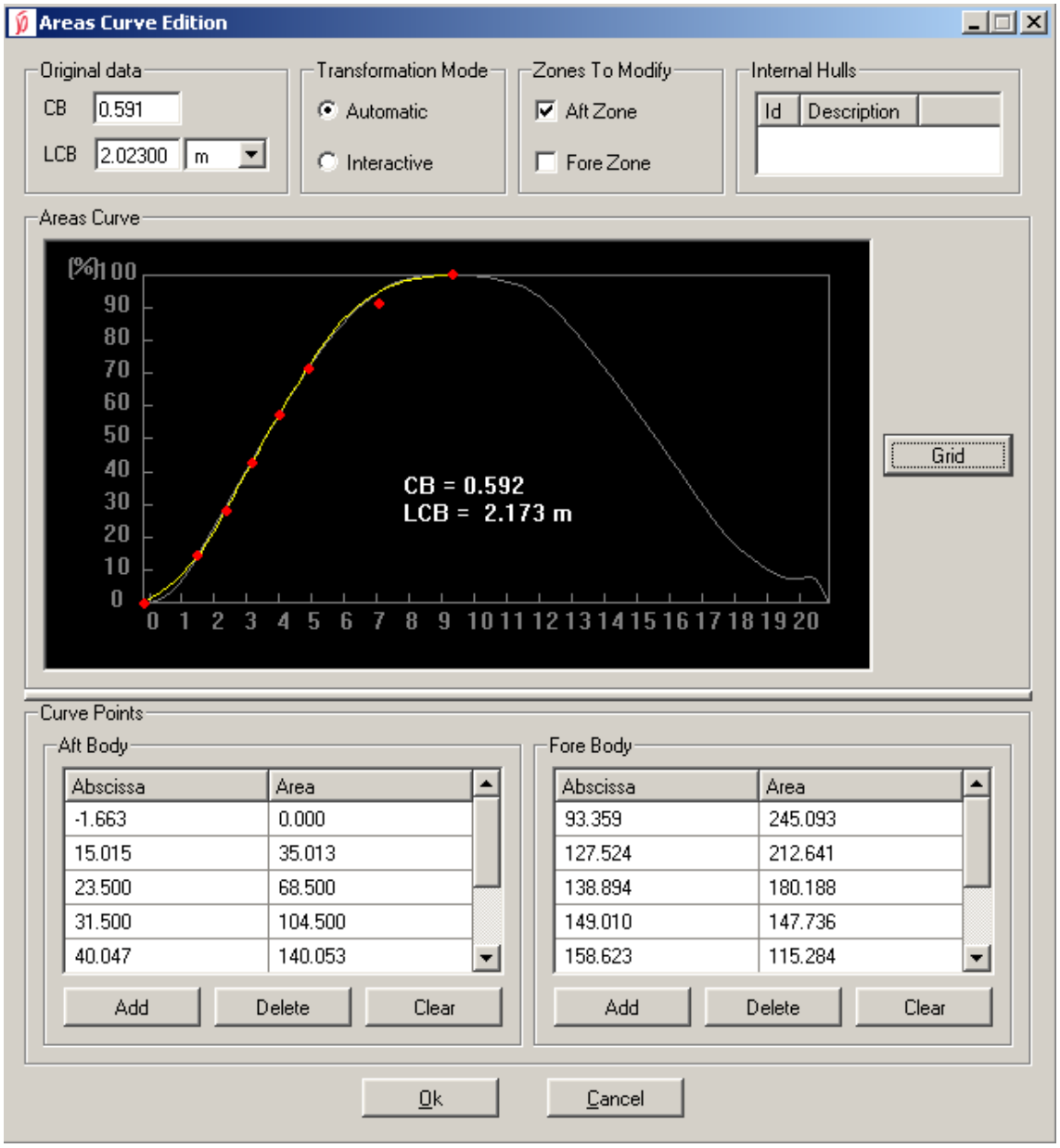
This command will perform modifications in the area curve of the ship.

Formulation

The modification is performed translating the transversal sections of the ship in the X direction, taking into account the differences (in abscissa) between the current area curve and the new one modified by the user. The current area curve and the new one should be similar due to perform big modifications can result wrong lines. Local extremes are not permitted in the area curve. For example, if the area curve has a minimum due to the fore bulbous, the area forward to the minimum cannot be modified.

Process

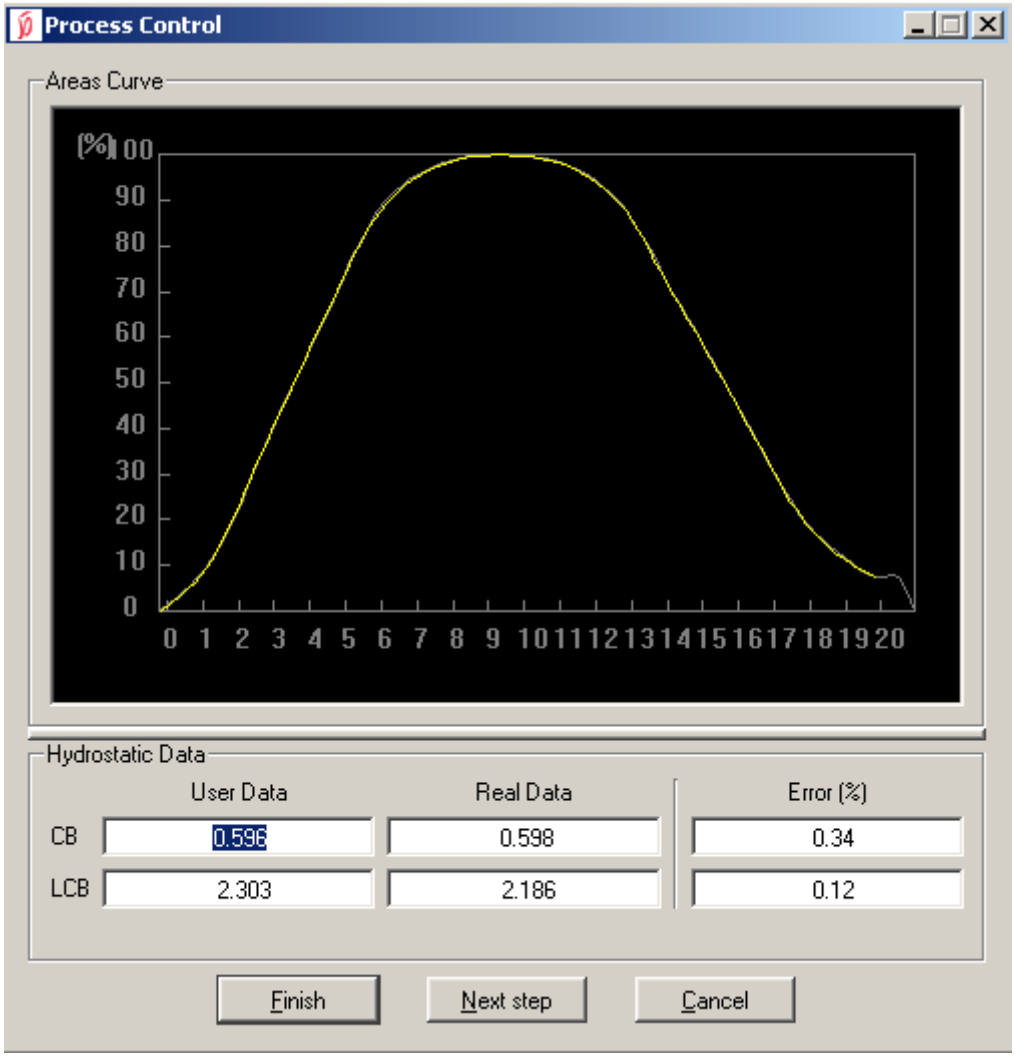
When this command is executed, the program will display the next window:



In this window the current block coefficient and the length of the center of buoyancy are shown.
The current original curve is drawn in grey and the new one in yellow colors. The Y-axis is the percentage of area referred to the maximum one. The X-axis is the abscissa of the section. The area curve is represented using the area of the design sections.
The curve can be modified moving its knots. The movement of these knots can be where:
Graphically, clicking with the left bottom of the mouse and dragging the point.
Numerically, modifying the value shown in the table under the drawing.
Knots can be added in any moment to the area curve. In the same way the knots can be deleted at any moment.
The first and the last point for each curve are not modified, and both curves will be tangents to the parallel body.

When the curve is modified, the program will calculate automatically the new block coefficient and the position of the center of buoyancy taking into account the new shape of the area curve.
The way of transformation indicates the method to control the transformation process: If the method selected is the **automatic** one, the process will finish when the length of the center of buoyancy and the block coefficient are the same ones input by the user. If the method selected is the **iterative** one, the process will stop every step, the program will show the result of the transformation and will ask to the user the possibility to follow with the calculation or to stop the calculation in the current step.
The aft and fore bodies are handled independently. The user has the possibility to maintain one of the bodies without transformation.
Therefore, if the ship contains internal hulls these ones can be transformed in the same way that the external one. Selecting the ones to be transformed the program will apply the transformation to all of these ones and the external hull just in one action.
The “cancel” bottom will stop the process without any transformation. During the iterations the program will show a new screen in which the user can check the intermediate result that the program fit before getting the value input by the user.
If “accept” bottom is pushed the program will start an iterative process to get the new block coefficient. If the selected method is automatic, the process will finish when the program get the new block coefficient and the length of the center of buoyancy. If the program is not able to get the value input by the user in 20 iterations, it will appear a message indicating the last value achieved and it will give to the user the possibility to validate this one or to restore the original one.

If the selected method is interactive, the process will be stopped for each step showing the next screen:



Window showed working manually:
The grey curve is the current area curve; the yellow one will be the new one in the current step defined by the user.
In the lower part of the screen the block coefficient and the length of the center of buoyancy are shown. The program will show the current value the new one (taking into account the new shape of the area curve) and the discrepancy between them. The user has the possibility to finish the transformation in this step or to perform another one obtaining new values for the hydrostatic coefficients. The position os the center of buoyancy is referred to the midship and positive means aft wards.

10.5 Pseudoparalelism:

This command will create new patches pseudoparallel to the original ones.
The first step will be to select the patches to be transformed. These ones can be patches or facets. After this, the program will ask about the number of knots to be used to reinterpolate the patches after the transformation.
Finally the program will ask about the distance between the original and the copy. For each point of the patch, the program will measure the distance taking into account the

normal vector to the patches in the correspondence point. If this command is executed from the command copy the program will maintain both of them.
Note: If the result is unsatisfactory the user can try to increase the number of knots for the original patches and repeat the process. This command is very useful when we try to define surfaces parallels to the external hull.

11. Importing decks and bulkheads from version v50r1.3.

This command should be used to import in module Fsurf the information available in old file *fdkbh.fil of version R1.3, containing the information of decks and bulkheads.

11.1 Decks:

The command allows to import all decks of any type defined in module Deckb.

The command does not transfer those decks not defined in Foran system, coming from a external software and imported previously in module Deckb. These decks can be visualized in module Deckb but they does not save any information in the file *fdkbh.fil.

The command does NOT transfer the bulwark.

11.2 Corrugation laws:

The command allows to import all corrugation laws defined in module Deckb.

11.3 Corrugated bulkheads:

The command allows to import all corrugated bulkheads, transversal and longitudinal, with a variable corrugation law (bulkheads type 6 and 7), and with the fabrication code of “manufactured” type. The command does NOT import those corrugated bulkheads with the fabrication code of “comercial” type.

The command does NOT import those old corrugated bulkheads type 4 and 5 with a corrugation law not “variable”.

11.4 Entities not transferred:

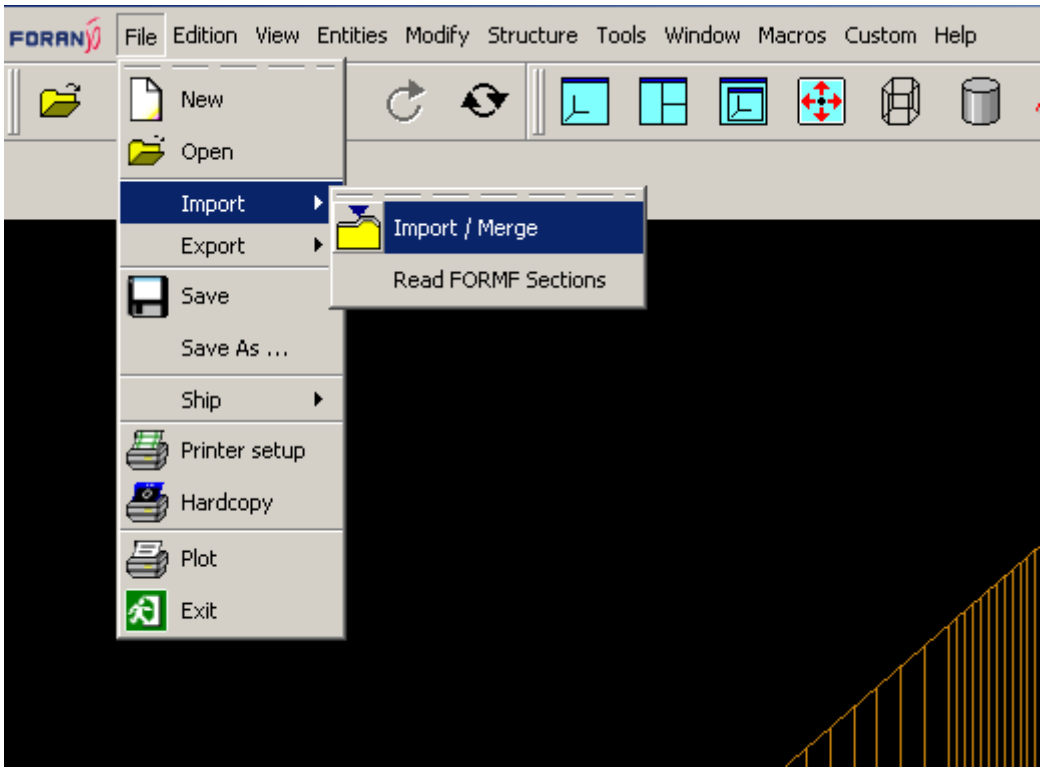
The command does NOT import the definition of the frame system, which should be transferred when importing the information in the form file *formt.fil.

The command does not transfer the planar transversal bulkheads.

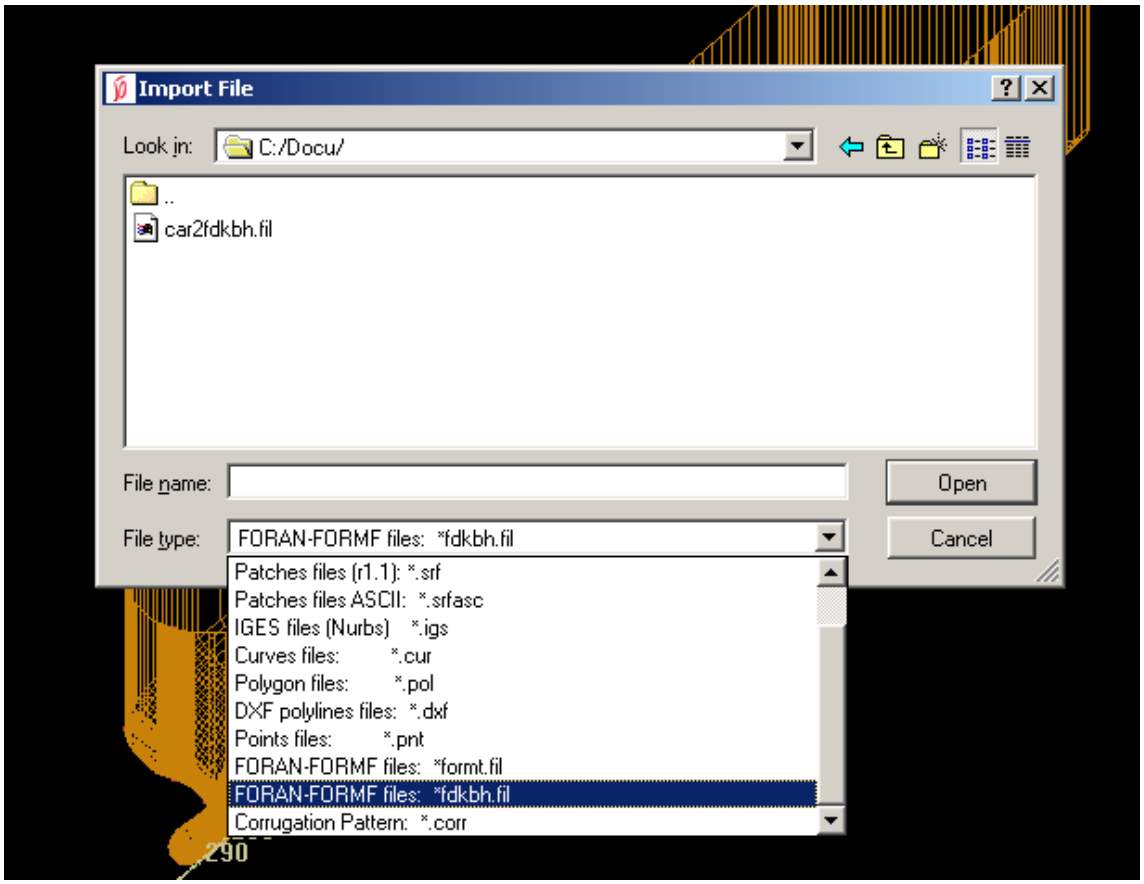
The command does NOT import the longitudinal bulkheads defined by traces.

To use this command it is compulsory to read previously the form file of the project. Once the ship in on the screen, and previous to the import process, user should check that there are not any deck, corrugation law or corrugated bulkhead already existing in the project.

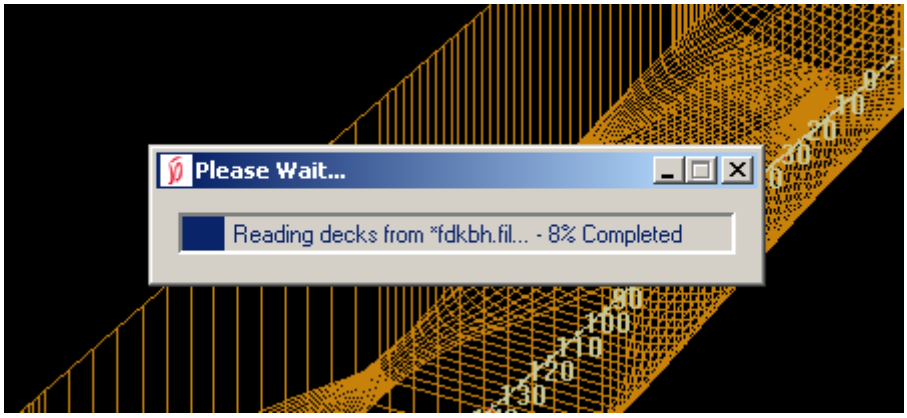
Then user has to select the Import command.



The program shows the available extensions to be selected. User has to select the file *fdkbh.fil to import.



Program will start with the import process of the decks, corrugation laws and corrugated bulkheads existing in that file. It will display a progress bar for each one of the groups of entities to load.



Once this process has finished, Fsurf will have all the imported elements, which will be able to edit, modify or save it, in the same way as if it were created in the Fsurf module.

12. Recalculation of objects:

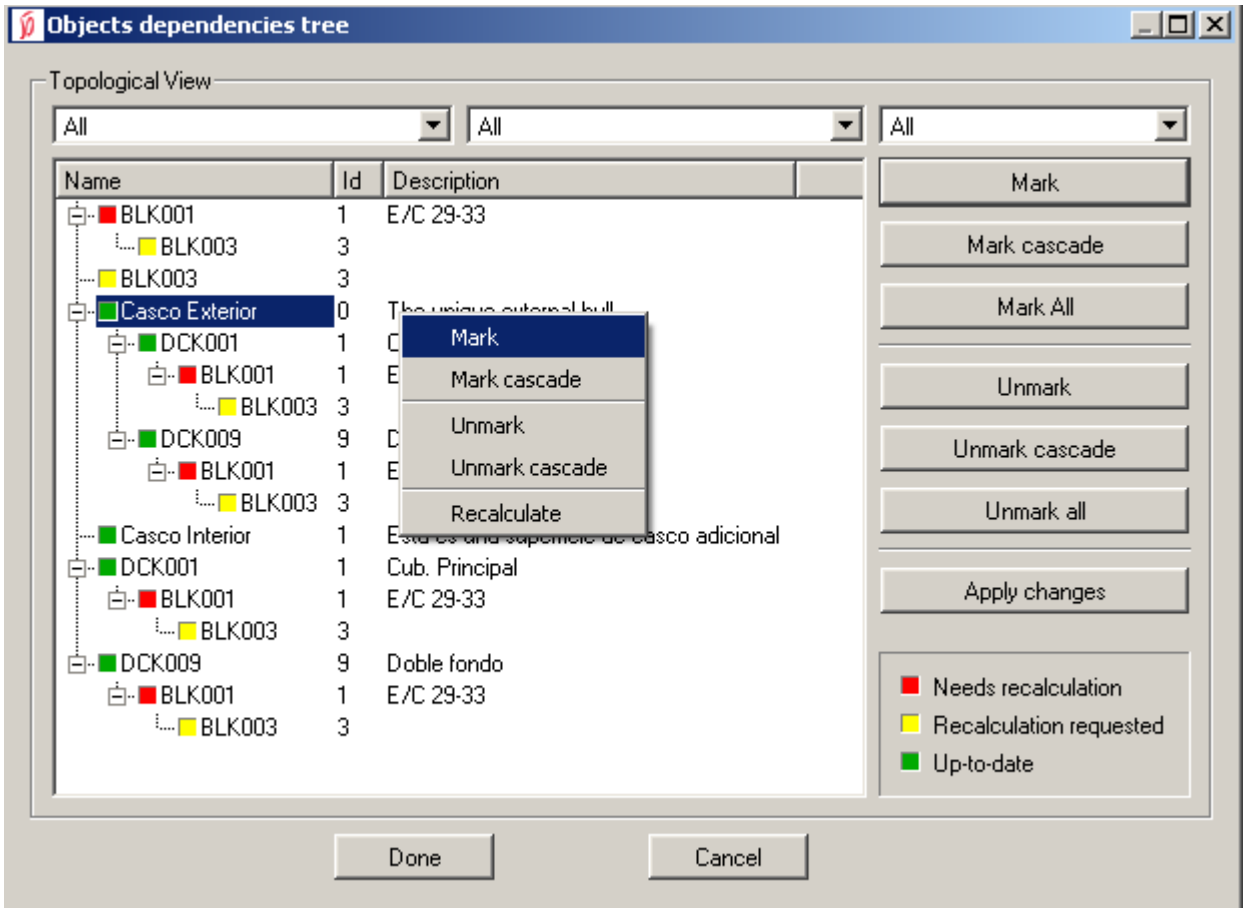
Recalculation of objects allows to redefine elements depending on others, when these objects are modified.
This task can be done with three commands below the “Tools” menu:

12.1 Switch recalculation mode:

This command allows to change the mode of recalculation from manual to automatic and vice versa.
When changing the mode, a text will appear in the left lower corner of the screen.

12.2 Topologic manager:

It is represented with a hand handling objects. This command shows a window with the calculated objects and the required calculation objects. It is not possible to undo this command.
This command is also accessible from the object pop-up.
The status of an object is identified through a colour code:



Green

The object does not need any recalculation.

Red

The object needs a recalculation. An object is marked as "need recalculation", when it depends on another modified object and the system of topologic relations is desactivated. Thus, the recalculation is not carried out, but the object remains marked in red.

Yellow

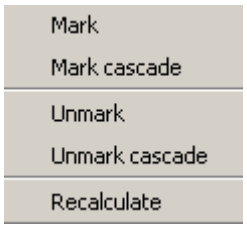
The object needs to be recalculated and the user marks it for recalculation. The requestion of recalculations is made through the window of selectiv recalculations.

Working with the window:

The comboboxes are used to select the objects. It is inappropiate to represent as a tee the ship structure, due to both,the structure and the dependence relation would be represent in the same way.
Therefore, the **comboboxes** select **structure** and the **tree** represent exclusively **topologic relations**.
So, the first unfoldable select the type of objects to show (hulls, decks, bulkheads, bulwark and all).
The second combobox varies with the first one. It is used to select the subtype.
For example, if you can select the bulkheads, the unfoldable allows to show just the transversals, longitudinals, etc.. The last combobox shows all the objects in function of the previous comoboxes.
When selecting one of the comboboxes, this is set up as unique root of the represented tree. So, it is obtained the same result as executing the comand from the pop-up of the selected object. The unique difference is that the pop-up cut off the comboboxes, allowing see the type, subtype and name of the object but with allowing its handling.
The lateral buttons are used to controle the recalculation. Specifically, "Mark" and "Unmark" work on the selected element currently in the tree. "Mark cascade" and "Unmark cascade" work on the current element and its topologic dependences. "Mark all" and "Unmark all" work on all elements.
The recalculation of marked objects is made when OK button is clicked, but it can be obtained the same result if the user clicks on "Aplly changes" without closing the window.
After recalculation, the objects are shown in the tree with theirs new colour code which could be green or red.
The central view shoes the selected elements throug the filters of the comboboxes. From each element, topologic dependences are shown.
Thus an element can depend on several ones, it is possible that some repeated sub-trees appear in the main tree. The objects are marked with colours which legend is shown in the right lower corner. Name, identifier and description are shown from each element.
Therefore, when clicking on an element, it is marked as working- object in the scene to make easy its visual identification.

In the central view, it is possible to access to the pop-up of the current element clicking

on the right button of the mouse. It represents an alternative mode of doing the same actions that the buttons placed in the right part, but working only on the selected element:



The first two options produce a mark in the object, single or in cascade, while the two following actions produce the unmarked. The last action consist of recalculating the current element.

12.3 Recalculating elements:

When executing this commands all elements of the sep will be recalculated. This task can take some time if there are too many elements.

13. Decks definition:

The aim of this task is to define and store decks. This information will be used in the Basic Design and Hull Structure subsystems of FORAN system.

For a better definition and due to its characteristics, decks are grouped in two types:

13.1 Main decks:

The geometrical characteristics for main decks are to have a camber along the length of the ship.

The definition is carried out in two levels, first of all to define the main characteristics of the deck and then the characteristics for each discontinuity.

Characteristics

The reference line is a line in the central plane that will be used in the definition of the discontinuities to fix the depth of those in the central plane with respect to it. When there is not parabolic sheer this reference line is a horizontal line to constant height (RDP). When there is parabolic sheer, this will be defined typing the sheer in the aft perpendicular (RSA) and the abscissa at which it starts (RHA), as will as the sheer in the fore perpendicular (RSF) and the length in which it starts (RHF).

In relation with the camber, eleven types of decks can be considered:

- Plane
- Platform

- Tropical
- Tropical with two knuckles
- Variable in angle
- Plane – parabolic
- Parabolic – plane
- Circular – plane
- Double bottom
- Upper tanks
- Upper tanks with double bottom

13.2 Special decks:

- Main deck (identifier 1)
- Castle deck (identifier 11)
- Top garage deck(identifier 12)
- Poop deck(identifier 13)
- Double bottom (identifier 9)
- Upper tanks (identifier 10)

Decks can be defined with plane and/or parabolic shear, with plane camber, circular or parabolic, with knuckles or with jumps.

Upper tanks can be defined with horizontal plane bottom,etc.

Main deck, identifier 1, will be defined from aft to fore, and it will be taken into account for the rest of modules as the compartment deck, as well as castle deck (11), bridge deck(12) and poop deck(13), although these decks are optionals.

These one have to be defined over main deck (1) and taking into account that the deck 12 has to be placed between 11 (aft) and 13 (fore).

Decks can be define in two different ways:

13.3 Parametric definition:

The aim of the parametric definiton is to determine the parameters of the surface which compose the deck. The surface of each deck is a unlimited surface in breadth which intersecting with the hull originates the deck.

In general the decks can have parabolic sheer and trapezoidal, parabolic or circular camber, as well as knuckles and breaks along the ship length.

Along the length the deck surface is considered divided in zones. The zone is characterised because along of its length, the depth at the central plane on the reference line, the halfbreadth of the horizontal zone and the angle or coefficient of the camber

change lineally. The reference line is a horizontal or parabolic horizontal line situated in the central plane that is useful to define the depths of each section with respect to it.

The limits of the zone are called discontinuities. It will have a discontinuity when there is a change in the variation pattern of the depth at the central plane on the reference line, of the halfbreadth of the horizontal zone or of the angle or of the coefficient of the camber.

The user must define the reference line of the deck at the central plane and the deck characteristics (height, halfbreadth of the horizontal zone and camber) in each of the discontinuities. The program will calculate for each discontinuity the parameters derived of the defined characteristics by the user and will interpolate to obtain the characteristic of the deck at any section.

13.4 Non parametric definition:

The aim of the non-parametric definition is to assign to the current deck, the entities which are going to form it.

It is also possible to trim the deck with other surface.

Decks definition is placed in the main menu below the submenu:

STRUCTURE->DECKS->NEW DECK

To define a deck it is necessary to enter the following data:

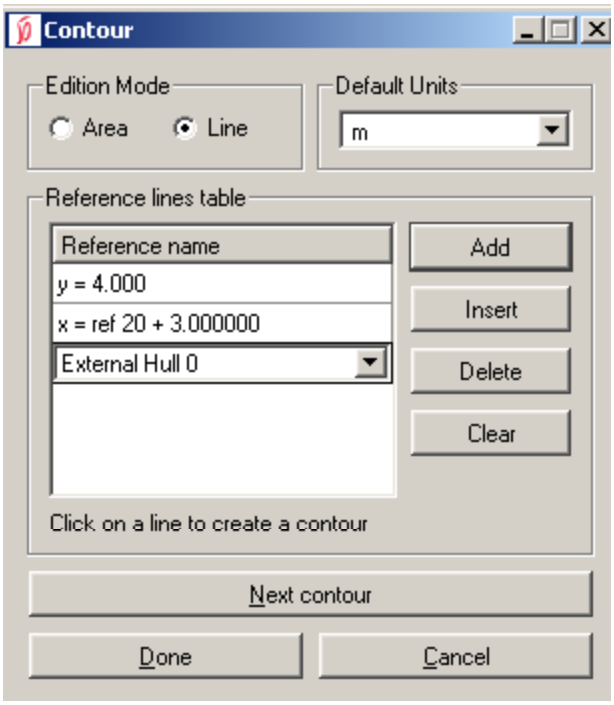
- **Identifier**: some identifiers are specifics for special decks.

- Main deck (1)
- Double bottom (9)
- Upper tanks(10)
- Castle(11)
- Bridge (12)
- Poop deck(13)

- **Name**
- **Type of deck**
- **Description**
- **Limits of the deck:**

- No trimming
- External hull

- Internal hull
- Edit manually:



When defining the limits of a deck manually, the user has to select its contour selecting an area or a set of lines.

The selected items will appear listed in the inputadata.

There is the possibility of selecting more than a contour clicking on the button “next contour” and selecting a new one.

To select lines, it is possible to select them from the list or to create new ones clicking on the button “Add” and editing the new field clicking twice on it. It can be created:

- **Straight line**

There are two possibilities:

- To create a straight line using a constant coordinate: the user has to write:

x = abcissa, **y = ordinate** o **z = height**.

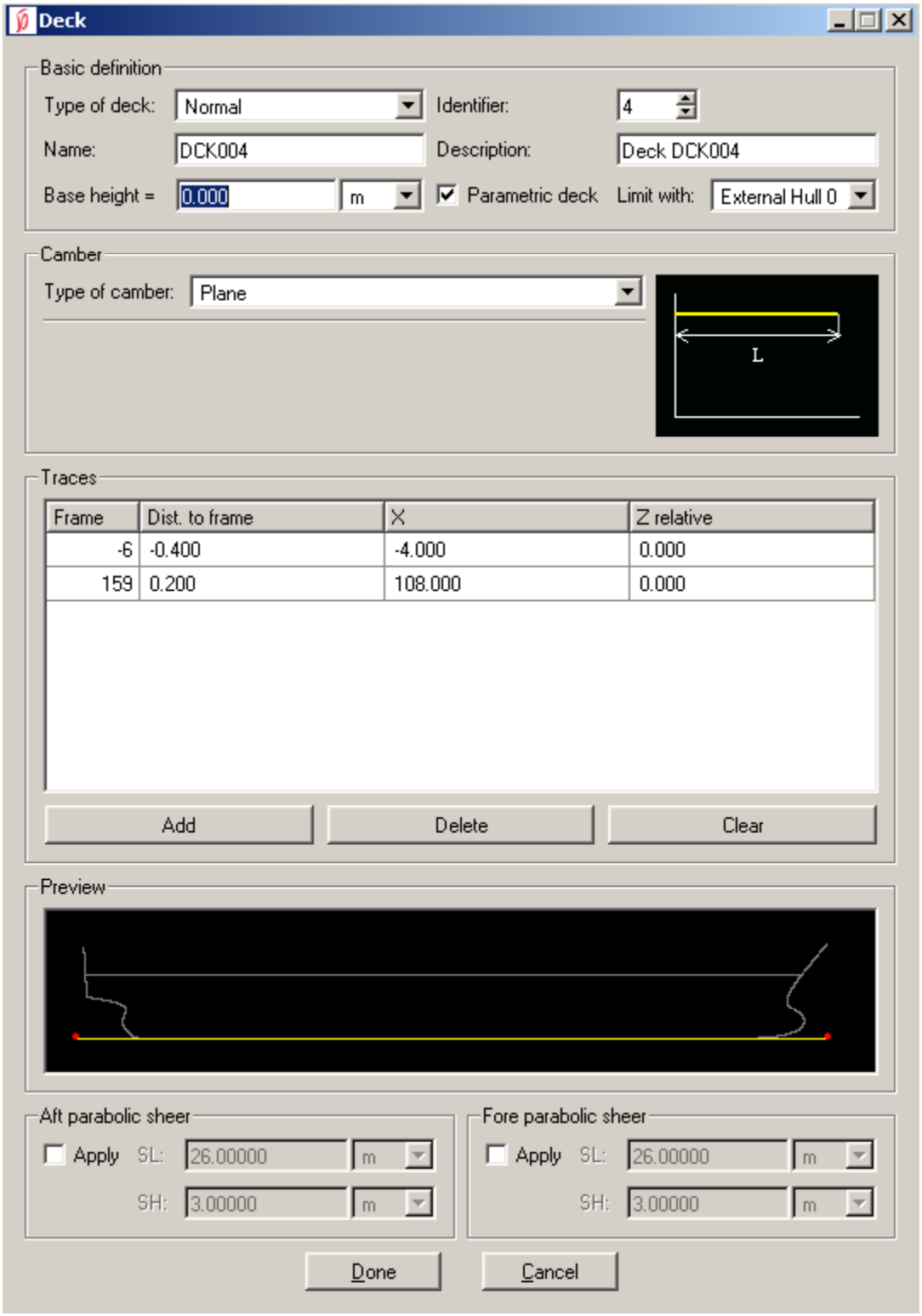
- To create a straight line using frame + distance: the user has to write:

x = num. ref. frame + distance o **x = num. ref. frame – distance.**

- **Pseudo – parallel line** to an existent one.

The user has to write: **name of line + distance** or **name of line – distance.**

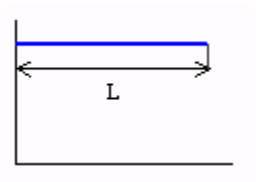
When defining a PARAMETRIC deck will appear the following inputdata:



For parametric definition the user has to enter the following data:
- **Reference height**
- **Type of camber**

Cambers are measured in the maximum breadth.

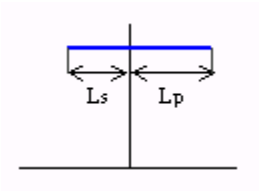
1. Planar:



The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.
- Height over reference line

1. Platform:

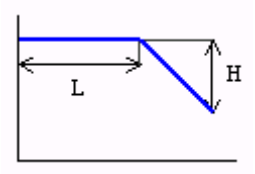


The user has to enter for each discontinuity:

- Abscissa from aft perpendicular
- Height over reference line
- Starboard length(Ls)
- Port length (Lp)

It is compulsory Lp > Ls.

2. Trapezoidal:



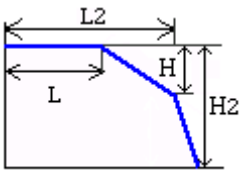
To define the angle of the camber, length (L) and height (H) have to be entered.

The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.

- Height over reference line
- Halfbreadth of horizontal part.

3. Trapezoidal with two knuckles:



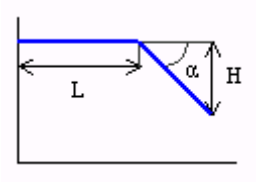
To define the angle of the camber, length (L) and height (H) have to be entered.

To define the second angle of the camber, length (L2) and height (H2) have to be entered

The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.
- Height over reference line
- Halfbreadth of the first horizontal part
- Halfbreadth of the second horizontal part

4. Variable in angle:



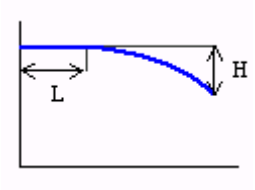
The user has to enter for each discontinuity:

- Abscissa from aft perpendicular
- Height over reference line

To calculate the angle of the camber in each discontinuity, it is necessary to enter:

- Length (L)
- Height(H)

5. Planar- Parabolic:

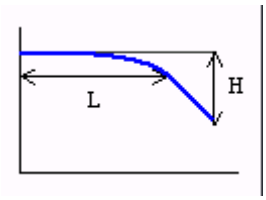


To define the coefficient of the parabola, it is necessary to enter the length (L) and the height (H).

The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.
- Height over the reference line.
- Halfbreadth of the horizontal part.

6. Parabolic- planar:

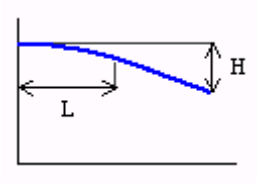


To define the coefficient of the parabola, it is necessary to enter the length (L) and the height (H).

The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.
- Height over the reference line.

7. Circular – Planar:



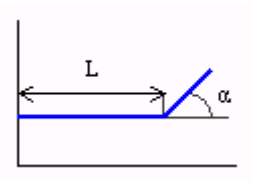
To define the radius of the circle, it is necessary to enter length (L) and height (H).

The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.

- Height over the reference line.

8. Double bottom:

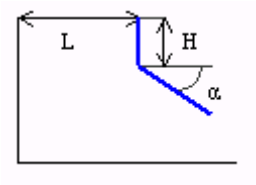


The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.
- Height over the reference line.
- Halfbreadth of the horizontal part.

Angle of camber (Alpha is positive clockwise sense)

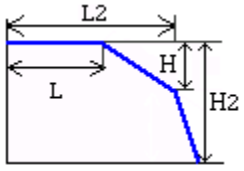
9. Upper tanks:



The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.
- Height of the inner bulkhead until the intersection with main deck (H).
- Halfbreadth of inner bulkhead (L).
- Angle of wing tank in degrees, positive in clockwise sense (ALPHA)

10. Upper tanks with planar bottom:



The camber of upper tanks with planar bottom is measured in the maximum halfbreadth.

The user has to enter for each discontinuity:

- Abscissa from aft perpendicular.

- Height of inner bulkhead until the intersection with main deck (H).
- Halfbreadth of inner bulkhead(L)
- Height of planar bottom (H2)
- Breadth of planar bottom (L2)

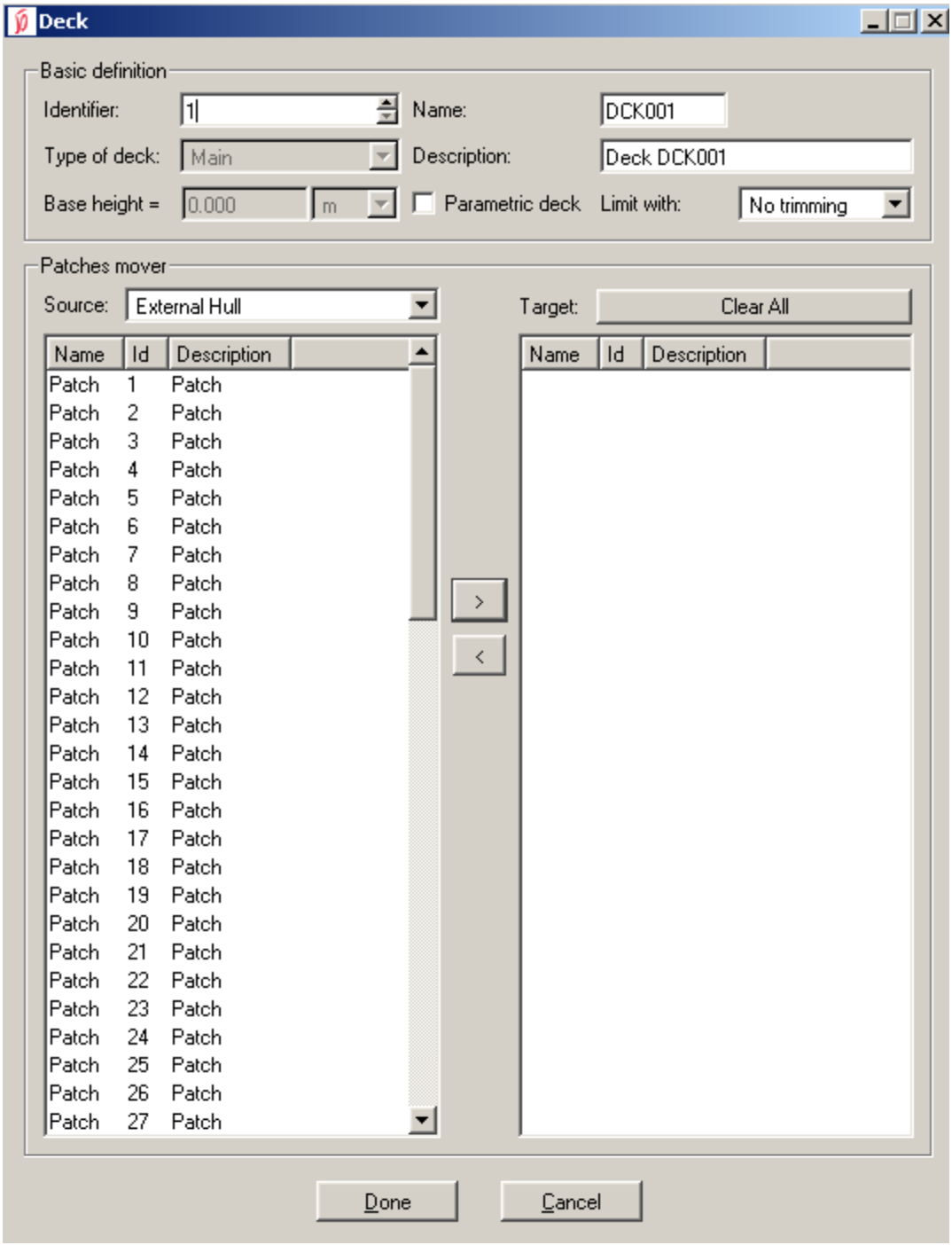
- **Traces of deck**:

- Numerically: position of the trace in X axis (frame + distance or abscissa) and height over the reference line.
- Graphically: definition of the traces with the red points over the limits of the yellow line. When adding a new trace, it will appear a new red point.
- It is possible to delete a discontinuity clicking on the button "delete".

- **Fore and aft parabolic sheer:**

- Height (SH)
- Position of the vertex of the parabola (SL) measured from aft perpendicular.

When defining a deck NON- PARAMETRIC, it will appear the following inputdata:



To create a non-parametric deck, the user has to select the patches and facets to be used in the definition of the deck.

The entities have to be translated from the source structure (external hull, internal hull or workspace) to the deck.

The entities have to be selected in the left menu, one by one, or by multiselection and then translated to the right menu clicking over the “arrows” buttons.

14. Bulwark:

The bulwark is defined in function of the intersection of the main deck, the forecastle, the bridge and the poop with the hull forms, and has to be extended from the after profile to the fore profile.

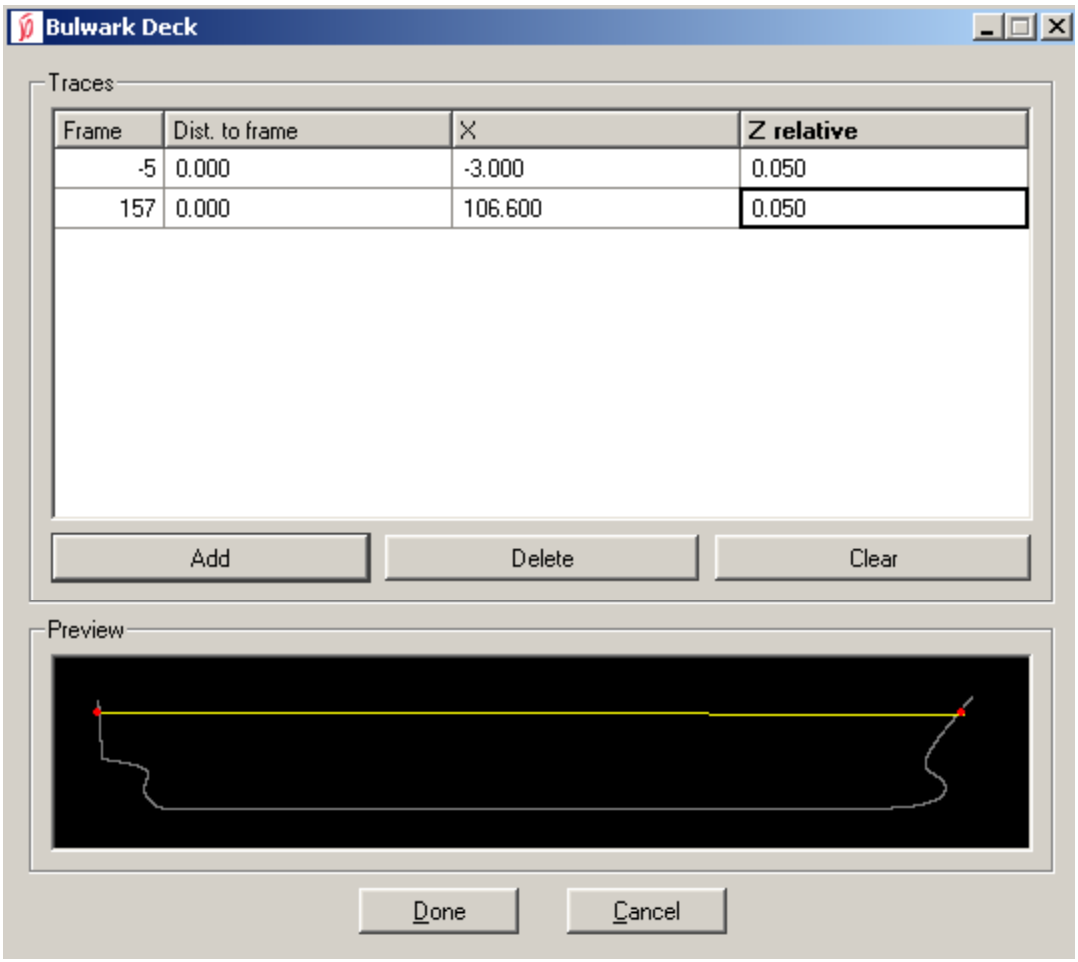
It can be defined by straight segments over the closest lower deck.

To define the bulwark it is necessary to enter:

- Abscissa (frame + distance)
- Relative height over the lower deck

The definition of the bulwark is placed below the submenu: STRUCTURE->BULWARK.

When running the command “Bulwark”, the program will display the following inputdata.



Using the buttons “Add”, “Delete” and “Clean”, it is possible to add a new discontinuity, to delete it, or to clean the list of discontinuities already defined.

15. Bulkheads definition:

The aim of this task is to define and store the bulkheads.
This information will be used in Basic Design and Hull Structure modules of FORAN system.

For a better definition and due to its characteristics, bulkheads are divided in four groups:

Defined in a plane, by extrusion in a direction, corrugated and by traces.
Whichever definition, it is always possible to define the symmetrical one.

15.1 Defined in a plane:

A bulkhead is defined in a plane when all its geometry is contained in a support plane. It is possible to define three types of bulkheads depending on the type of plane:

- Transversal (X = k)
- Longitudinal (Y = k)
- General (**A** x + **B** y + **C** z + **D** = 0)

15.2 By extrusion in a direction:

The definition of a bulkhead by extrusion consist of:

- Extrusion polyline
- Extrusion length
- Extrusion direction: X, Y, Z, or defined by the user.

15.3 Corrugated:

The definition of a corrugated bulkhead consist of:

- Support plane, which is the basis for bulkhead definition.
- **Corrugation pattern**:

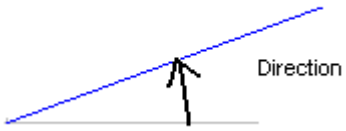
  Bended:

  In this geometry it is possible to define straight and circular segments.

  A straight segment consist of:

  - Direction: referred to the previous segment.
  - Length: distance between the first and the last point of the segment.

- Rounding: to obtain the markins in NEST module (Hull structure subsystem), it is necessary to enter a value in this step.



An arc of circle consist of:

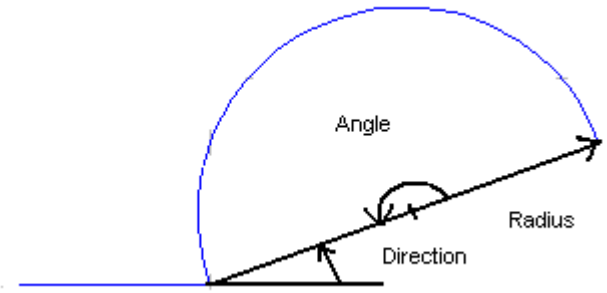
- Direction: referred to the previous segment.
- Radius of circle.
- Angle: referred to the portion of circle to draw.

There are two possibilities to define an arc of circle:

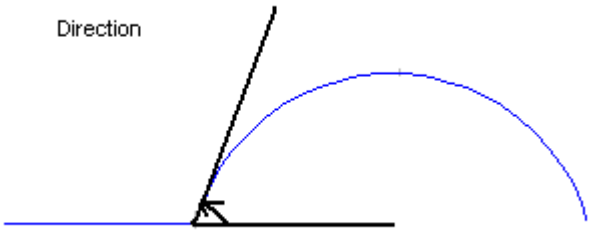
- By centre
- By tangent

Depending on the definition, the direction has different meanings:

By centre:



By tangent:



15.4 By traces:

Trace:

The trace is defined as a planar polyline. In this case, the intersection of the bulkhead is defined with the plane.

The bulkhead will consist of a group of facets connected by traces, being the points of the traces the corners of the facet.

The fact, the elements connected by traces are facets, imposes an additional condition to the points definition.

Therefore, in general, the user has to enter just one condition when defining a point of a trace, except in the first trace where it is necessary to enter two conditions.

Conditions:

- First condition: to belong to the facet connecting the current trace with the previous one.
- Second condition: defined by the user.

In the first trace and in the first point of each trace, the second condition is not defined, and the user has to set it.

Note: Traces can be container just in transversal planes. (X= constant)

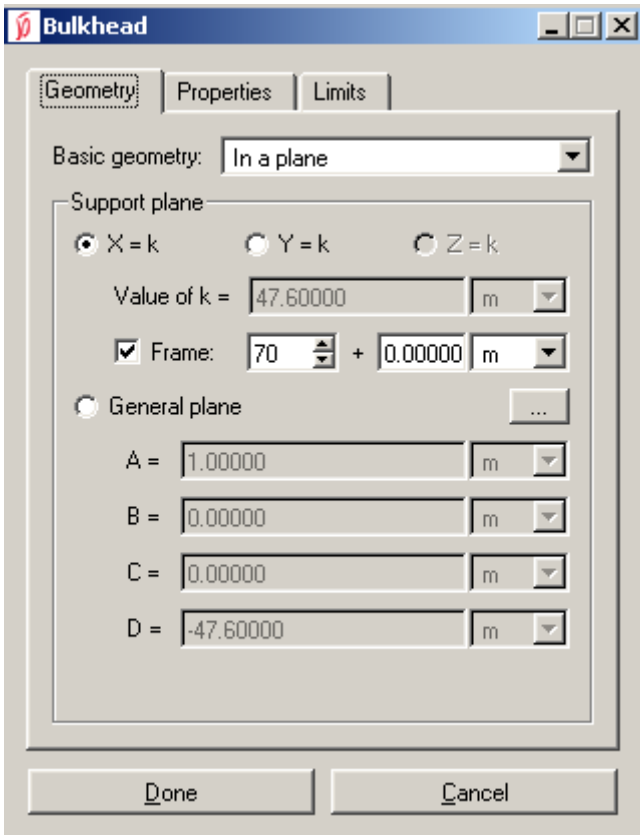
Bulkheads definition is in the main menu, below the submenu:

STRUCTURE->BULKHEADS->NEW BULKHEAD

To define a bulkhead it is necessary to enter the following data:

- **Identifier**
- **Name**
- **Type**: Longitudinal, Transversal or General.
- **Description**
- **Symmetry**

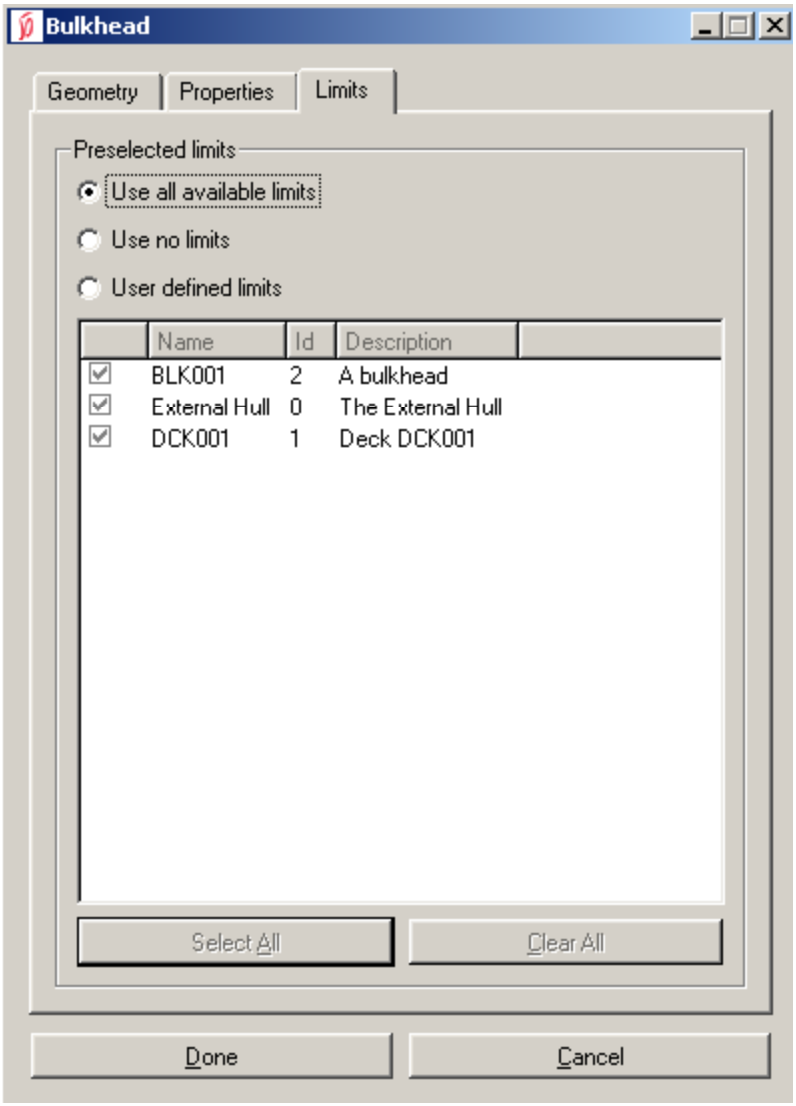
When defining a **PLANAR** bulkhead, the next inputata will be displayed:



There are several possibilities:

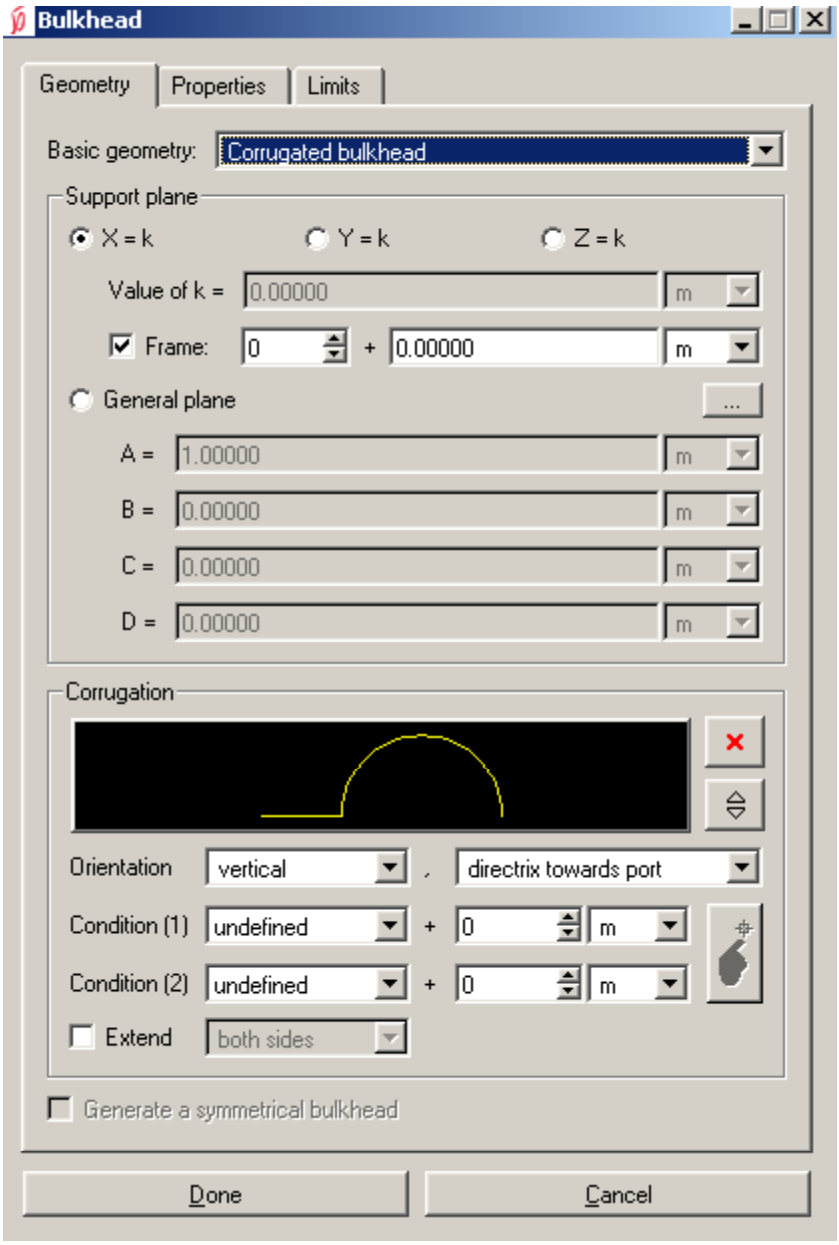
- **Transversal bulkhead**: the user has to enter the abscissa or frame + distance.
- **Longitudinal bulkhead**: the user has to select the ordinate.
- **General bulkhead**: the user has to enter the four coefficients of plane (A B C D) or to use the 3D option clicking on he right button.

After bulkhead definition, the user has the possibility of selecting its contour, depending on the chosen option in the jacket “limits”.



- Use all available limits: the program uses all the defined structures until that moment.
- Use no limits: the bulkhead is not trimmed with any surface.
- Use defined limits: the user selects the structures which are going to trim the bulkhead.

When defining a **CORRRUGATED** bulkhead, the next inputdata will be displayed:



The definition of a corrugated bulkhead is based on the support plane and the corrugation pattern.

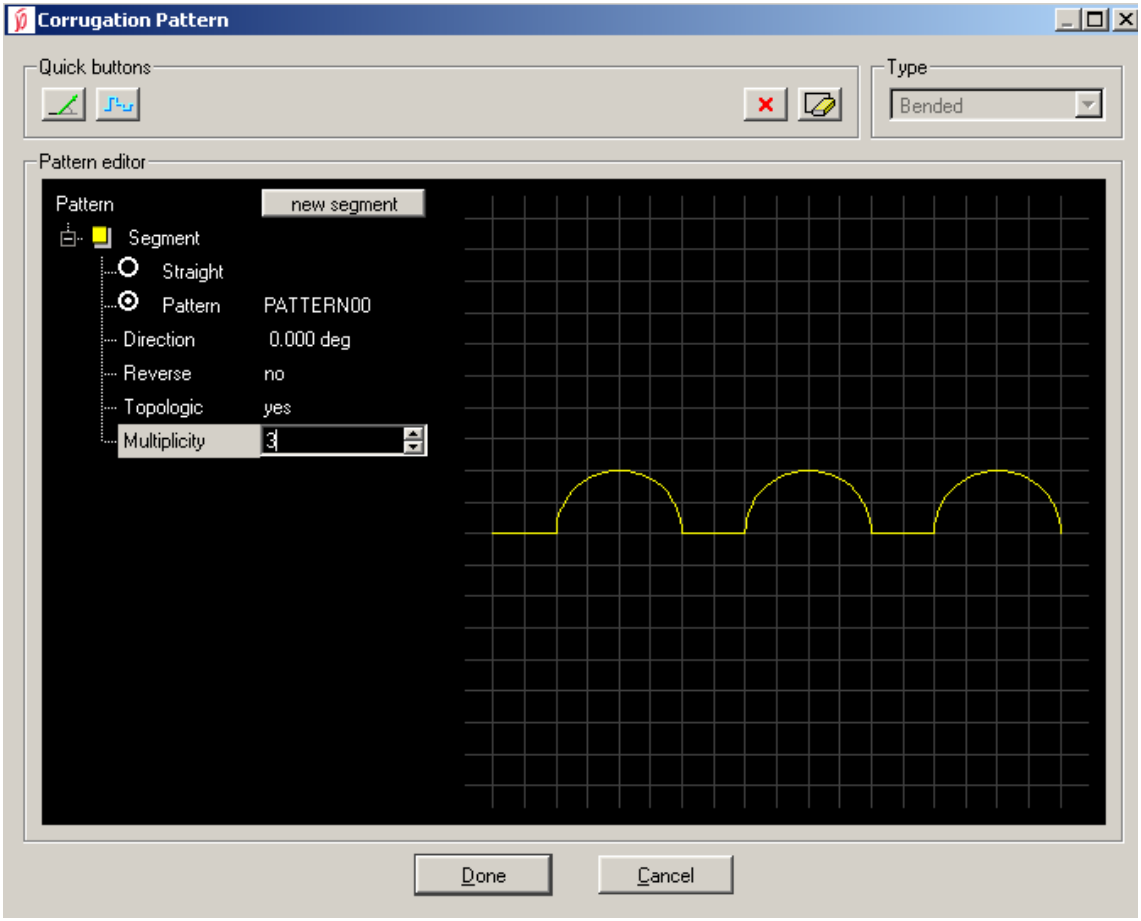
To define the support plane there are several possibilities:

- **Transversal bulkhead**: the user has to enter the abscissa or the frame + distance.
- **Longitudinal bulkhead**: the user has to select the ordinate.
- **General bulkhead**: the user has to enter the four coefficients of the plane (A B C D) or to use the 3D option clicking on the right button.

To choose the corrugation pattern, the user has to click on the black window and select a corrugation.

This corrugation has to be defined previously with the command **Structure-> Corrugation pattern manager.**

After clicking on the corrugation, it will appear the following inputdata:



It is possible to add a new segment or an existing pattern clicking on the fast buttons placed in the upper left corner.

When the user select the option pattern, it is possible to chose:

- Corrugation pattern
- Multiplicity: number of corrugations to draw.
- Direction: referred to X axis.
- Reverse: obtains the mirror corrugation.
- Topologic: if this option is activated and the user changes the type of corrugation or its parameters, the bulkhead will be automatically calculated.

After selecting a corrugation pattern, the user has to place the corrugation in the plane. To get this aim, it is necessary to enter

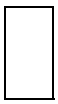
Orientation:

- Vertical: direction towards port or starboard.

- Horizontal: direction towards top or bottom.
- Inclinated: the user has to select the angle of inclination.

Conditions

With this action, the user selects the point where the first corrugation starts. It is necessary to enter both conditions: (1) and (2), so that the point will be defined as intersection between the selected elements in both conditions. It is also possible to change this point from one side to the other clicking on the button:



Extend: it is possible to define the bulkhead with the corrugation length or to enlarge it on:

- Both sides
- Begin side
- End side

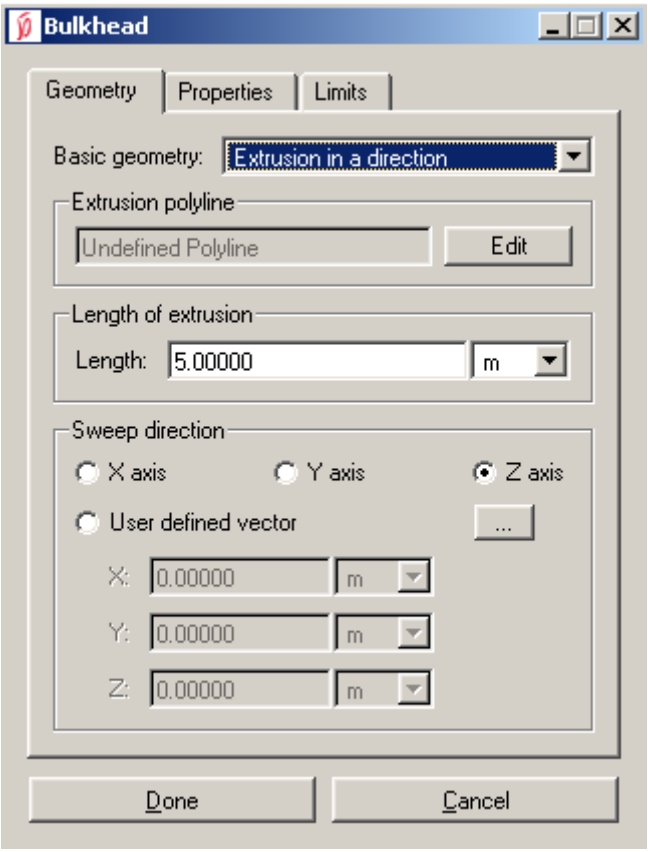
Symmetry:

- None
- Both sides
- Begin side
- End side

Clicking on the arrows towards top and bottom, it is possible to obtain the symmetrical corrugation.

After bulkhead definition, the user has the possibility of selecting its contour, depending on the option chosen in the jacket “limits”.

When defining a bulkhead **BY EXTRUSION**, it will appear the next inputdata:



The definition of a bulkhead by extrusion consist of:

- **Extrusion polyline**
- **Extrusion length**
- **Direction**

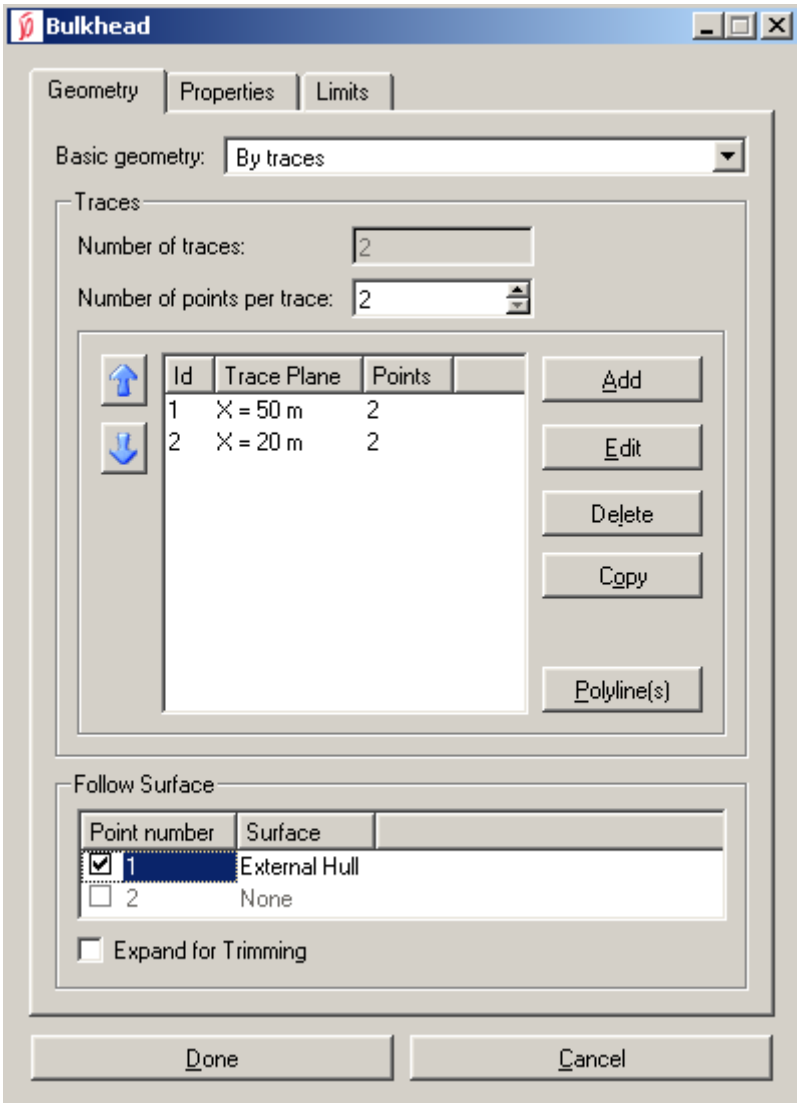
To define the **extrusion polyline**, the user has to click on the button “edit”.

It is possible to define the polyline with the 3D Point option.
There are several possibilities to define the **direction**:

- X axis
- Y axis
- Z axis
- Vector defined by user (X, Y , Z)
- 3D direction

After the bulkhead definition, the user has the possibility of selecting its contour, depending of the chosen option in the jacket “limits”.

When a bulkhead is defined **BY TRACES**, it will appear the next inputdata:



The definition of a bulkhead by traces consist of:

- **Number of traces:** number of defined traces.
- **Number of points in each trace:** number of points defined in each trace.
- **List of traces:** list of defined traces showing the number of trace, the plane and the number of points composing the trace.

Buttons to use with this list:

- **Add:** to define a trace and to add it to the bulkhead.
- **Edit:** to edit an existing trace. The trace has to be selected.
- **Delete:** to delete the selected traces.
- **Copy:** to create a copy of the selected traces.

- **Polyline(s):** to use polylines as traces. A new inputdata is displayed to allow the user select them.

The blue arrows to the left allows the user to move the selected trace towards top and bottom in the list changing, by the way the order of the traces.

A pop-up is displayed if the user click on the right button of the mouse over one of the traces of the list.

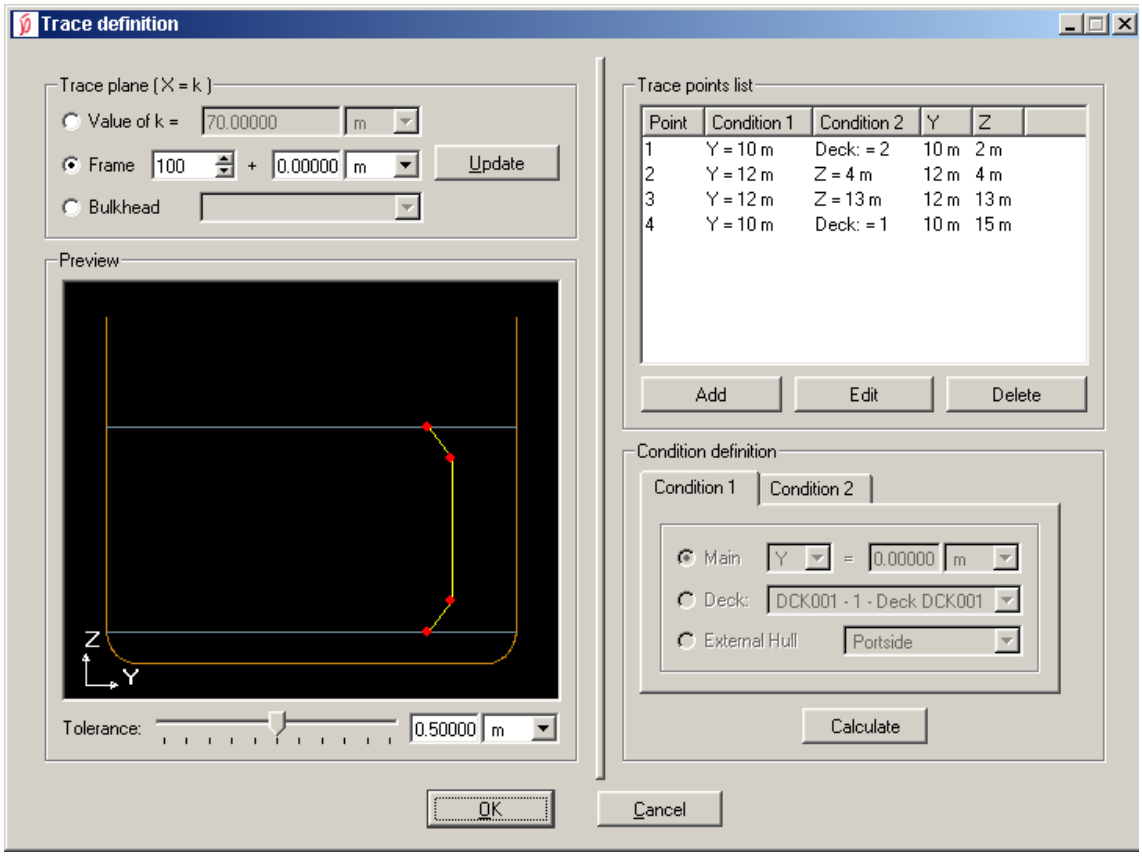
Clicking twice with the left button over one of the traces, it will be possible to edit it. Once the traces have been defined, the user has the possibility of selecting its contour, depending on the chosen option in the jacket “limits”.

The user can select the possibility of the bulkhead following a surface.
For example, if in each trace the first point is associated to the external hull and the user selects “follow surface”, the limit of the bulkhead will be the surface hull. In other words, the limit of the bulkhead will be the intersection between the bulkhead and the external hull.

Other option is to expand the surface to define the trimming contour later on.

Points of trace

The points of the trace are defined using the inputdata “point of trace”:



The inputdata **trace edition** is used for (re)defining a trace of a bulkhead.

The inputdata has four main components:

Plane of trace

To define the plane of trace the user can use:

- **Transversal plane (X constant):** the trace will be always in the same abscissa.
- **A frame and a distance:** the trace will be always referred to the same frame and to the same distance of this one.
- **A transversal bulkhead:** the trace will agree always with the bulkhead.

The user can select the frame or the bulkhead directly from the scene just clicking on it.

Once the plane of the trace has been defined, the user has to click on the button **update** to recalculate the existing points (if there are some one) and redraw the section.

List of points of trace

A list with the points of the trace is shown in this section. The columns of the list are the following:

- **Point:** Position of the point of the trace
- **Condition 1:** Type of the first condition which define the position of the point.
- **Condition 2:** Type of the second condition which define the position of the point.
- **Y:** Breadth of the point, after calculation
- **Z:** Height of the point, after calculation

The user can add, edit or delete using the respective button.

An element of the list can be edit clicking twice on the left button of the mouse on it.

Definition of the point

In this section the user has to set the conditions which define the position of the point. It is possible to set four conditions:

- **Y constant:** the point should have the given breadth.
- **Z constant:** the point should have the given height.
- **Deck:** the point should belong to the selected deck.
- **External hull:** the point has to belong to the external hull.

After defining the conditions, the user has to click on the button **Calculate**. If all the conditions define an only point, the input data are accepted.

Previous view

A transversal section of the ship is shown, as well as the points and the traces defined. The user can move the points dragging them with the mouse.

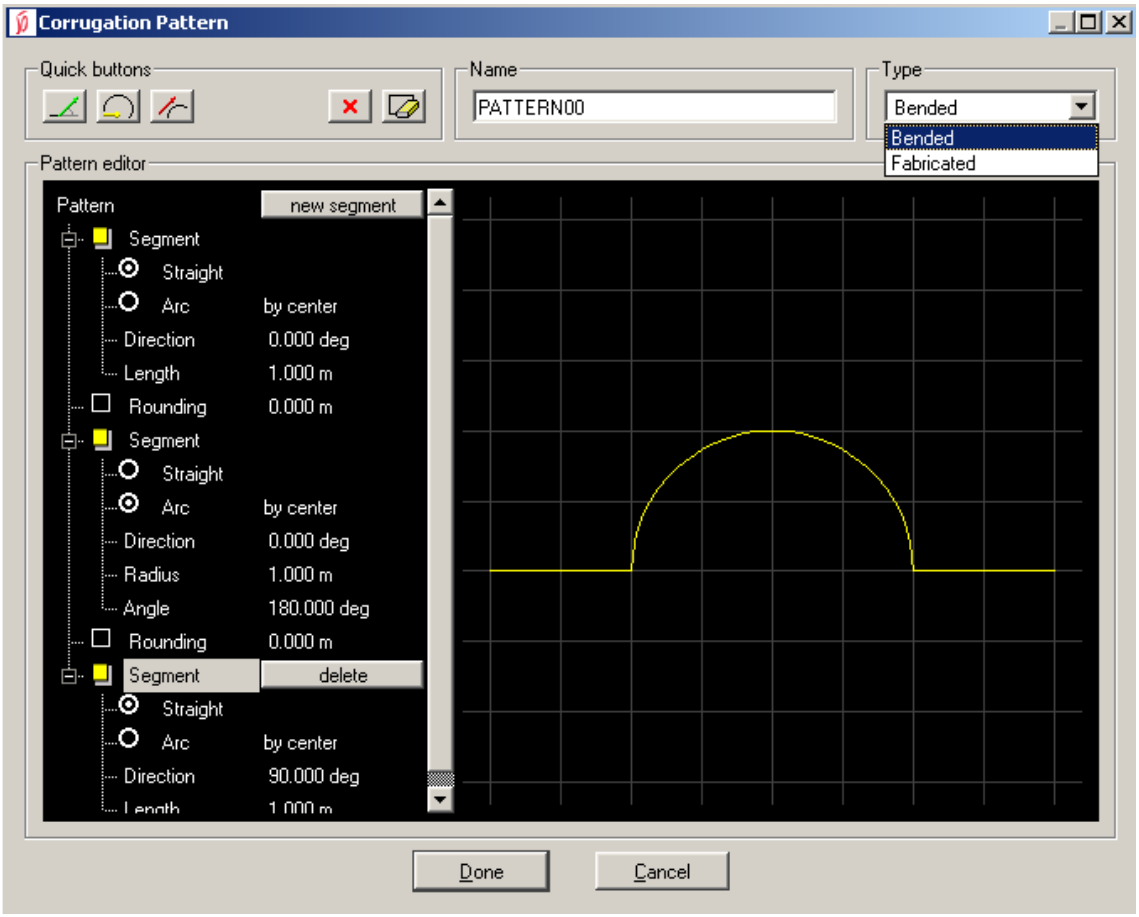
It is possible to modify the tolerance to select a surface. This distance is the limit which the point is adhered to the closest surface.

16. Corrugation patterns manager:

When executing this command the user has the possibility of :

- Adding to the list a new pattern
- Deleting an existing pattern
- Cleaning the list of patterns
- Copying a pattern
- Editing a pattern
- Importing from a file (*.corr) a list of patterns. This action can be also executed from the command Import/Merge.
- Exporting to a file (*.corr) a list of patterns.

If the user selects “ Add a new pattern”, the next inputata will be displayed:



To define a corrugation it is possible to use the fast buttons:

- Add a new segment.
- Add an arc of circle with a centre given by the user.
- Add an arc of circle with a tangent given by the user.
- Delete the current segment.
- Clean all the segments

When defining a pattern, the user can change the name by default and select the type: Bended or fabricated.

When the user changes the type of pattern, all the definition of the current pattern will be deleted to create the new one.

With the editor window, the user can create the geometry of the pattern adding it straight or circular segments.

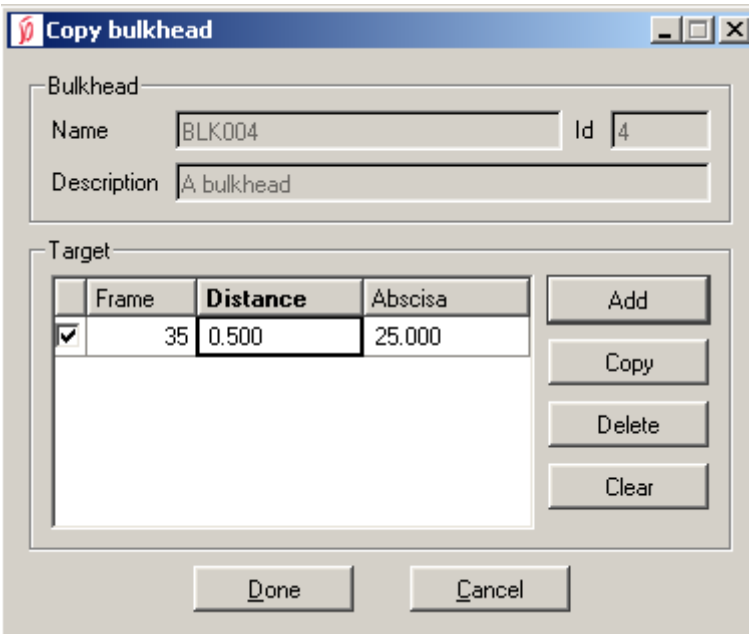
17. **Copy of bulkheads:**

This task allows the user to create topological copies of planar or corrugated bulkheads. It is not possible to create copies of bulkheads generated by traces or by extrusion.

There are two posibilities of performing this command depending on the support plane of the bulkhead:

17.1 Transversal bulkheads:

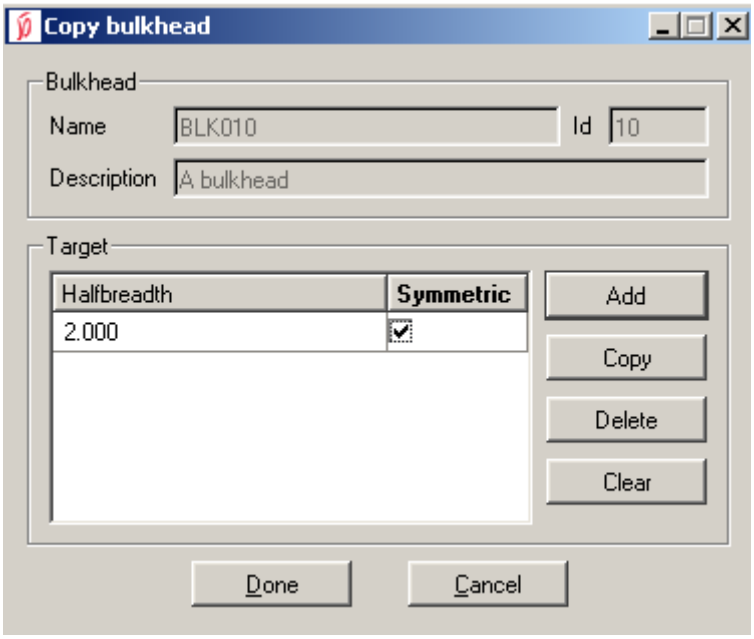
When creating a copy of a transversal bulkhead, it will display the next inputdata:



The user has to enter the location of the copies entering frame plus distance or an absolute abscissa.
If the checkbox of the first column is activated, it means that the copy will be topological, if a frame changes its position, the bulkhead associated to that frame, also changes its position.

17.2 Longitudinal bulkheads:

When creating a copy of a longitudinal bulkhead, it will display the next inputdata:



The user has to enter the location of the copies entering the halfbreadth.
If the checkbox of the second column is activated, it means that the copy will be generating a symmetrical bulkhead.
In both cases, it is possible to manage the list of bulkheads with the buttons "Add a new element", "Copy an element", "Delete an element" or "Clear the list".

18. Reading a NAPA data base:

To import forms from NAPA it is necessary to open a NAPA database. This operation can be done with the command FILE->OPEN and by selecting a NAPA database which is a file with extension “db”. When a database has been selected the program FSURF shows a dialog in which the user can select the different elements that can be imported.

The dialog window offers several possibilities located in five different tabs:

- **General**. It contains useful information about the ship selected in NAPA database. This corresponds to the version selected by the user.
- **Surfaces**. This tab offers the management for reading surfaces. The surfaces shown here are the elements corresponding to the concept “surfaces” of NAPA. The external hull and additional hulls are normally located in this area.
- **Surface Objects**. This tab offers the management for reading the elements that correspond to the concept “surface objects” of NAPA. Please, refer to the User Manual of NAPA for more information.
- **Curves**. This tab offers the management for reading curves defined in NAPA.
- **Miscellaneous**. This tab offers a window list with all the information read by the interface. There are buttons to obtain a report of all this information.

This input data allows the user to select from the different versions existing in a NAPA database. This selection can be performed by selecting in the version combobox. The program will read the elements existing in that version and it will update the information shown in the different tabs.

How to proceed?

1. Select the version to be read.
2. Select the surfaces that compounds the external hull.
3. Select the surfaces or surfaces objects corresponding to decks
4. Select the relevant curves, such as profile, flat of botom, flat of side, knuckles or bulwark line.
5. Click on the **EXECUTE** button. By clicking on the button **CANCEL**, the process will be cancel.

The surfaces that compound the external HULL will be duplicated with a symmetry. The frame system will be automatically transferred. The program will calculate the intersection between decks and the external hull in order to assign deck intersection lines on the hull. The deck intersection lines are necessary for the production modules of FORAN and actually is not available in NAPA.

The complete process can be logged in a MACRO. This macro collects all the information defined by the user to perform the transference and it can be executed further for updates on the forms. Even more the macro file can be edited and updated by the user with the proper commands, identifications in a very comprehensive language.

All the information appearing in the input data, depend on the existence of the NAPA API dynamic library and the corresponding NAPA license. The information obtained when a NAPA database read is provided by the NAPA API functionality. FSURF module only reads the available information provided by the NAPA API.

NAPA is property of NAPA Oy (ltd).

18.1 Napa Surfaces:

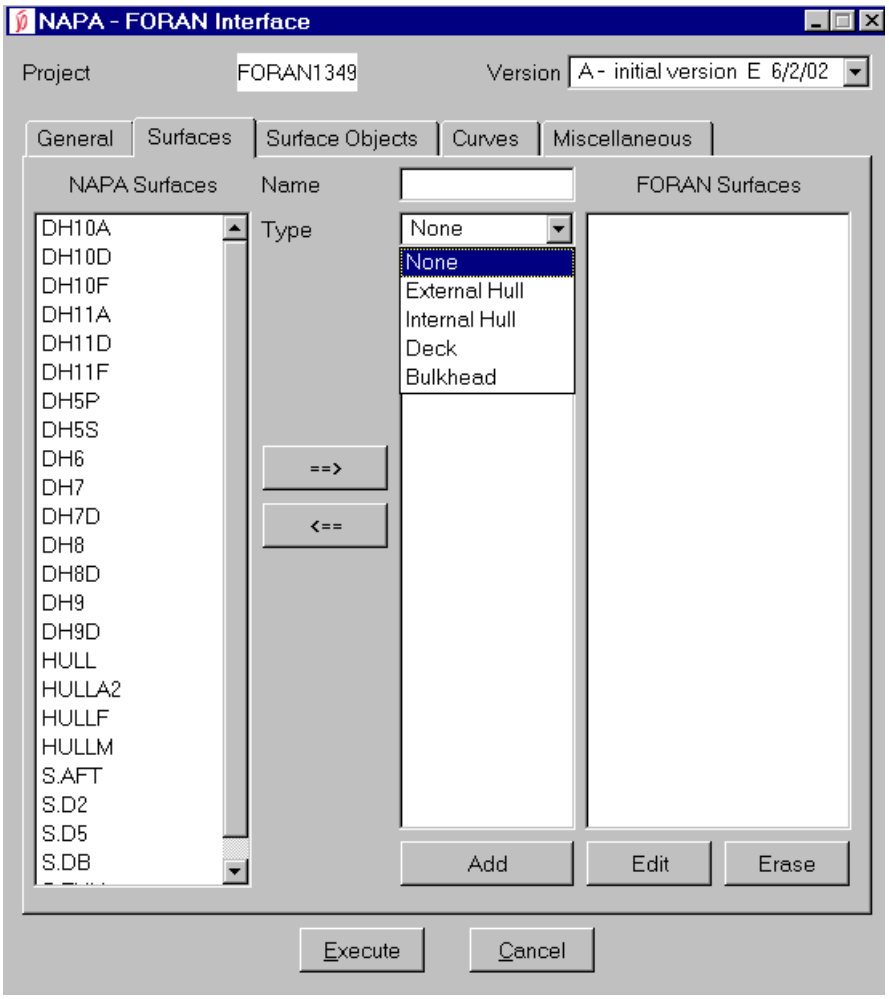
To transform a **NAPA** surface into a **FORAN** one, it is necessary to follow the next steps:

1.- In the first left column it will appear the NAPA surfaces. It is necessary to select the NAPA surface or surfaces and locate them in the center column, which can be done one by one or like a multiselection command. After the surface is selected, this has to be located in the center column of the window. This movement can be performed selecting the surface and clicking over the right arrow or doing double-click over the surface directly.

2.- The second step is to give the name (not compulsory) and select the type that will have the surface in the FORAN system. There are three posibilities:

- *None*: the surface will be contained in the **WORKSPACE** directory in the FSURF module.
- *External hull*: the surface will be considered as the external hull in the FSURF module. (please note that the external hull in FSURF module is unique but with this link more than one NAPA surface can be added to the external hull).
- *Internal hull*: the surface will be considered as an additional hull.
- *Deck*: The complete surface will be considered as a DECK. In this case the program will give a numerical identification, overriding the name given by the user.

3.- The third step is to add the surfaces positioned in the central column to the right one. To do this, it is necessary to click on the button **ADD**. In order to make changes in the definition, it is possible to edit the surface with the button **EDIT**, the surface will be moved in the central column and it will be possible to modify it. To eliminate a surface from the FORAN column, it is necessary to click on the button **ERASE**, and the surface will go to the NAPA column again. The surfaces located in this column will be built in FORAN with the geometrical definition that the surfaces have in NAPA and with the attributes given by the user.



18.2 NAPA surfaces objects:

The NAPA surface objects are master surfaces delimited by some restriction conditions, like intersections, etc. Normally this elements are used to define decks and bulkheads. The tab corresponding to surface objects shows the elements with this attribute. The way of proceed is the same as SURFACES with some considerations:

- DECKS. In FORAN decks have a numerical identification from 1 to 75 and with some restrictions:
  - Number 1 is reserved for the main continuous upper deck that intersect the hull.
  - Number 9 is reserved for the double bottom.
  - Number 10 is reserved for the upper tanks.
  - Number 11 is reserved for the fore castle deck
  - Number 12 is reserved for the bridge deck
  - Number 13 is reserved for the poop deck
  - Number 14 is reserved for the bulwark that is a virtual deck

  Due to the difficulties to obtain these concepts form the NAPA information, the link works in the following way:

  - Numbers 10 and 14 are not accessible.
  - The bulwark should be read as a curve or it has to be defined in module DECKB
  - It is recommendable that the user sets the rest of the identification numbers with their corresponding decks, i.e. to select number 9 to the double bottom deck, and the same for deck1, etc...

- BULKHEADS. The bulkheads are completely treated in the FORAN with this attribute, so the best option is to add these elements as internal hulls.

18.3 NAPA curves:

The curves are used in FORAN system with several purposes. The main purpose is to be reference for locating hull structure elements, but also the lines are used to get information about limits of the decks or even the hull.
The lines used in FORAN have different types; these types are: KNUCKLE, PROFILE, SIDE, BOTTOM, HELD, SEGMENT, CONTOUR, BOUNDARY, HOLE, BULWARK,NONE. Some of these types exist for compatibility purposes but they are not used in any case. The FORAN entity to support lines is the LINE. The lines in FORAN have to be hosted in a container known as SURFACE LINES, and it is attached to each existing surface. The interface allow the user to select what NAPA curves are going to be a FORAN Line. After this operation the user may need to move one line from a SURFACE LINES object to another, but in general the interface will locate the LINES in the proper container. The following conventions exists:

- There must exist one unique line of type PROFILE.
- The profile will be located in the external hull
- The lines of type KNUCKLE, BOUNDARY,SIDE, BOTTOM, HELD, SEGMENT, are located in the external HULL. The user can move these lines to different internal hulls in the case that additional hulls were defined. This operation must be done with the structure manager.
- Lines of type CONTOUR, HOLE are not treated.

- Line of type BULWARK must exist. This line can be read from NAPA database if it exists or defined in module DECKB as a virtual DECK.
- Lines with type NONE are located in the WORKSPACE.

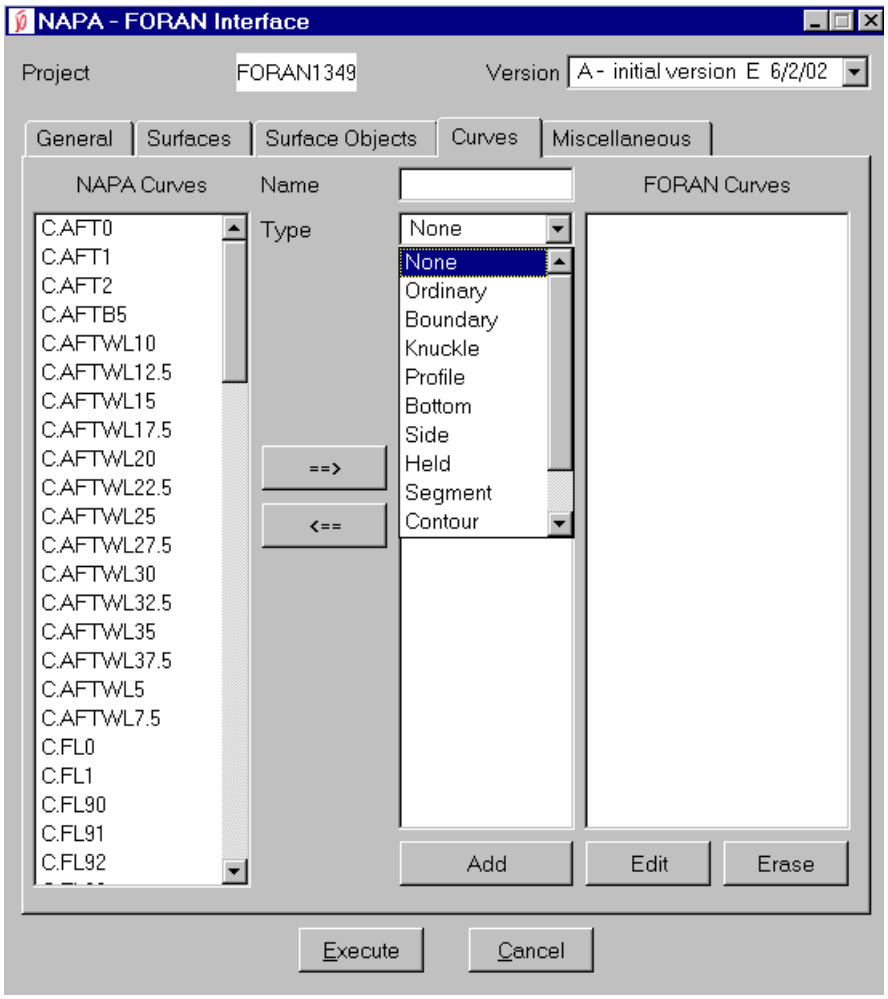
To transform a **NAPA** curve into a **FORAN** one, it is necessary to follow the next steps:

1.- In the first left column it will appear the NAPA curves. It is necessary to select the NAPA curve or curves and locate them in the center column. That can be done one by one or like a multiselection command. After the curve is selected, this has to be located in the center column of the window. This movement can be performed selecting the curve and clicking over the right arrow or doing double-click over the curve directly.

2.- The second step is to give the name (not compulsory) and select the type that will have the curve in the FORAN system. The following types are available:

None, ordinary, boundary, knuckle, profile, bottom, side, held, segment, contour and hole.

3.- The third step is to add the curves positioned in the center column to the right one. To do this, it is necessary to click on the button **ADD**. In order to make changes in the definition, it is possible to edit the curve with the button **EDIT**, the curve will be moved in the center column and it will be possible to modify it. To eliminate a curve from the FORAN column, it is necessary to click on the button **ERASE**, and the curve will go to the NAPA column again.



18.4 NAPA miscellaneus:

Here can be found some information about the objects existing in the NAPA data base.

- Name
- Description
- Geom type
- Code type
- Items: number of items that forms this entity. It can be null items, one item or if there are more of one item, it will appear the number 9999.
- Date

ANEX

POINTS AND POLYLINES

1. Open the file “test.pol”
2. Practice navigation and selection modes. (Pop-up and menu)

3. Deleting entities
4. Creating points from Polylines
5. Saving the points in the file “name.pnt”
6. Deleting the fore body (Polylines)
7. Obtaining Polylines using the command “By points cloud”
8. Transforming the LBP of the ship. Command: “Plane scale”
9. Changing the design trim. Command: “Plane sliding”
10. Splitting the plane zones. Command: “Split on plane”
11. Obtaining the knuckles. Command: “Split by angle”
12. Join Polylines
13. Making the symmetry of Polylines
14. Saving the Polylines in the file “name.pol”
15. Creating a plane passing through one Polyline point

CURVES

16. Creating curves by interpolation method
17. Deleting the previous ones
18. Creating curves by approximation method
19. Interpolating curves with the adequate number of points
20. Visualizing the curvature function
21. View the control polygon of this curve
22. Moving interactively the points of the polygon
23. Moving interactively the knots of the curve
24. Moving the points of the curve
25. Visualizing the grid of curves. Move the curves checking the grid curves
26. Splitting the curves by plane
27. Defining curves by points

PATCHES

28. Creating patch by interpolation
29. Creating patch by approximation
30. Creating patch by limits
31. Interpolating patches
32. Visualizing the control mesh
33. Moving the control mesh
34. Visualizing the grid of the patches checking its modification when moving the control mesh
35. Sew patch to patch
36. Sew patch to curve
37. Defining tangent between patches
38. Trim patches
39. Defining the patches of the ship from the file “test.pol” (define first curves)

CREATING AND CHECKING THE FILE FNAMfsurf.fsf

1. Associating patches using the manager
2. Creatimg the main lines of the ship

3. Checking the surface (holes between patches)
4. View/Changing the patches orientation
5. Checking the entities that will take part in the file: curves in lines, patches in surfaces, etc...
6. Creating the file testfsurf.fsf
7. Modification of the forms and create again the file testfsurf.fsf

HOW TO BUILD A SHIP WITH FSURF

This annex will be useful to show to the user how to start with the FSURF module. We will see how to define one ship starting from the minimum information you can receive. The final result will be the ship. We will go step by step, however it is too important that previously the user has read the user guide or at least knows how to handle the selection and visual modes of the module. We will split the document in different points, which shows the different steps we will follow to obtain the final result (the ship).

STARTING FROM ASCII FILE

We will start the definition of one ship. To perform this task we can select one DXF file. The program will read the DXF file as polylines.
Another possibility to start the FSURF working can be inputting points or polylines using ASCII files or by directly definition into the program. Using ASCII files the format will be the next one:

- Points: X Y Z
  ...
- Polylines: No. of points
  X Y Z
  ...
  No. of points
  X Y Z
  ...

In this way we can input whichever offset table we receive from.... Instead of using the DXF file we have the possibility to read the file “test.pol”, using this one, we will obtain the same polylines that using the DXF file.
This kind of entity is an auxiliary one (check type of entities in the user guide) and will be the starting point for our example.
The first thing we have to do is thinking about the best way in which we will define our patches. It is so important to spend some time in this moment. A good definition of patches will save a lot of time and problems.

PREPARING THE POLYLINES FOR CURVES DEFINITION

We will delete all the information we do not need or the information that is easy to be defined. In this way we will delete the FOS, the FOB and the parallel body. In this particular case we will also delete the information above the fore knuckle, because the surface it is easy to be defined.

PREPARING THE CURVES FOR PATCHES DEFINITION

We will create the curves using the approximation method. For this task we will select all the polylines and we will create the curve corresponding to these ones.
Once the curves have been created we can delete the polylines. From this moment we will work all the time using the curves. For the definition, in this ship we will perform 11 patches without the easy patches we mentioned previously. The first step will be to prepare the curves. To obtain homogeneous definition in patches we will interpolate all the curves to perform one patch with the same number of points. In this way to perform the two aft patches we will interpolate the curves using 16 points and we will maintain this definition in the fore body up to the fore peak, in which we will change this definition. The fore peak zone we will use 5 different patches to be performed. Then we will prepare the curves for this definition.

PATCHES DEFINITION

Working with the curves previously prepared we will perform the patches definition. After the patches definition we have to check that we did not input any hole in the ship. Below tools we have to check the surface. In the moment we finish to check the surface and we do not have any hole we have to finish the patches definition with the easy surface we deleted in the first step. We will perform this task by means of definition the patches by limits.
When we finish to define the port side we can make the symmetry of the patches to create the complete ship. Now in this moment, you can move the patches positioned in the workspace to the corresponding hull (internal or external). For this task we will use the structure manager.
Finally, the last task we have to perform should be to check the patch orientation. This task is very important to be done if you want to follow working later on with structure definition.

MAIN CURVES DEFINITION

At least we have to define the profile of the ship. May be we can define more lines if we want (FOS, FOB, Knuckles, etc). To define these lines we have previously to create the 3D curves, which will compose the lines. After the lines definition, again working with the structure manager, we can move these ones to the corresponding hull definition.

BUILD SHIP

This is the last task to be performed. We will input the main dimensions of the ship and the FNAM of the new project. Finally we have to create the ship and the program will create the file “FNAMFSURF.FSF” that we use for the rest of definitions (decks, etc)

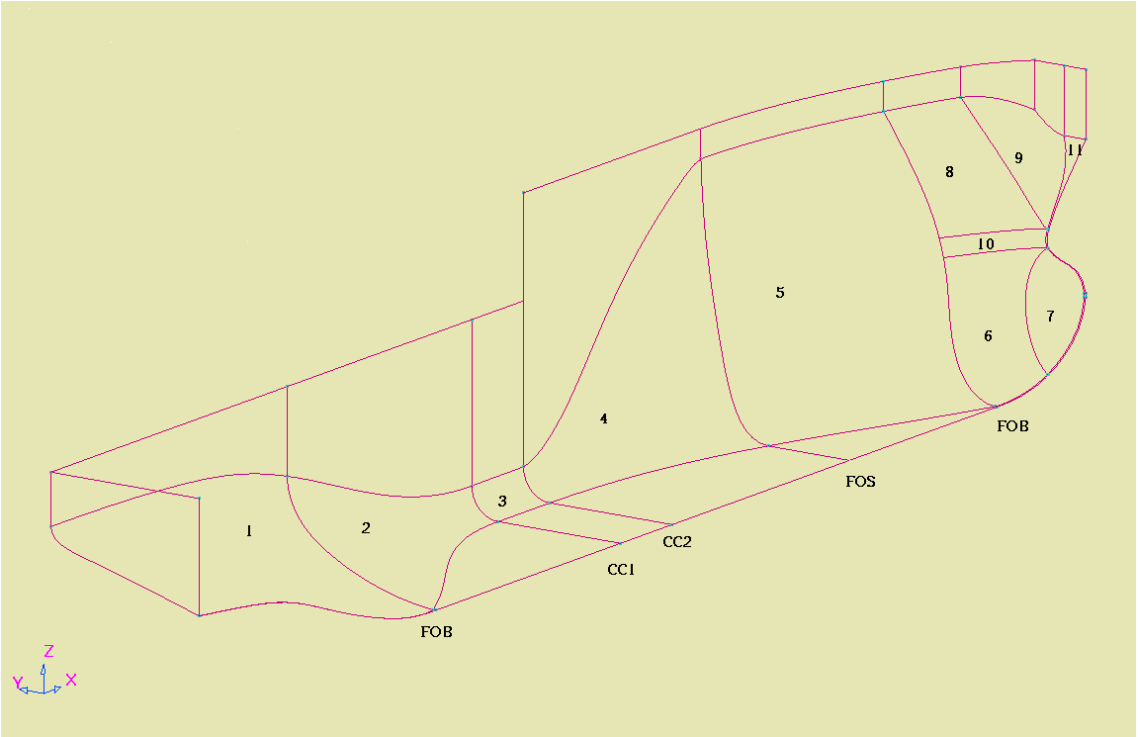
PRACTICAL EXAMPLE

The example will be performed with the polylines file " test.pol".
In first place, the polylines from flat of bottom, flat of side and transom will be removed, due to they can be generated easily at the end of the example.

The approximation tolerance will be adjust to 0.0001 m to get a better precision in calculations (EDITION →PREFERENCES).

The polylines will be approximated to curves(ENTITIES →CURVE →BY POLYLINE→ APROX) and it will proceed to generate the following patches taking into account the main limits of the ship such as FOB, FOS, CC (parallel body), knuckles, etc...(ENTITIES →PATCH →INTERPOLATE).

As a consequence of this, the most appropriate subdivision it could be the following:



Patch 1 → 16 x 15 knots (sections)
Patch 2 → 16 x 20 knots (sections)
Patch 3 → 16 x 2 knots(limits)

Patch 4 → 16 x 20 knots (sections)
Patch 5 → 16 x 15 knots (sections)
Patch 6 → 15 x 8 knots (sections)
Patch 7 → 15 x 8 knots (sections)
Patch 8 → 15 x 4 knots (sections)
Patch 9 → 15 x 4 knots(sections)
Patch 10 → 8 x 4 knots (limits)
Patch 11 → 4 x 2 (limits)

It is suitable not to "charge of knots" the patches, since if could be necessary to sew in the future and this will involve the increase of these ones. This is the reason why, perhaps it would be suitable to carry out some reinterpolation to adjust the patches to the previous ones.

It is necessary to check holes between patches with tolerance 0.0005 (TOOLS →SURFACE ANALYZER) to avoid a new redefinition of patches when the ship is enough advanced in design. To follow working, the tolerance has to be changed to 0.0001 m.

The majority of the patches can be performed by sections and the patches 10 and 11 can be obtained by limits when all the previous ones have been defined.

The "flat" patches can be defined with the limit isoparametric curves of the already generated patches, obtaining a copy of them as a projection on an adequate plane, and generating the new patches by limits.

When all the patches have been generated, holes has to be checked again to guarantee that there is not any gap between patches greater than the prefixed tolerance (0.0005 m).

This is a good moment to check the fairing of the patches. To do that, it is possible to display the grid curves of all patches to verify tangency, bad faired zones, etc...

In the case of it would be necessary to fair some zone, it could use the commands of adjust tangency, curves or points, or in particular cases, to edit the control mesh of the patch and to perform small movements until obtaining the desired shape.

Following it will check the orientation of every surface by means of the orientation viewfinder or the normal vectors.

The next step will be to assign the patches to the suitable surfaces (usually the external hull), creating new hulls if it would be necessary, through the structure manager.

A symmetry of all patches will be performed, checking that they are assigned to the appropriate surface automatically.

Following holes will be checked, this time for all ship, keeping the precision 0.0005, and orientating all the ship properly.

If everything is ok, it will be obtained the special lines of the ship (profile is always compulsory) extracting the limits of the patches.

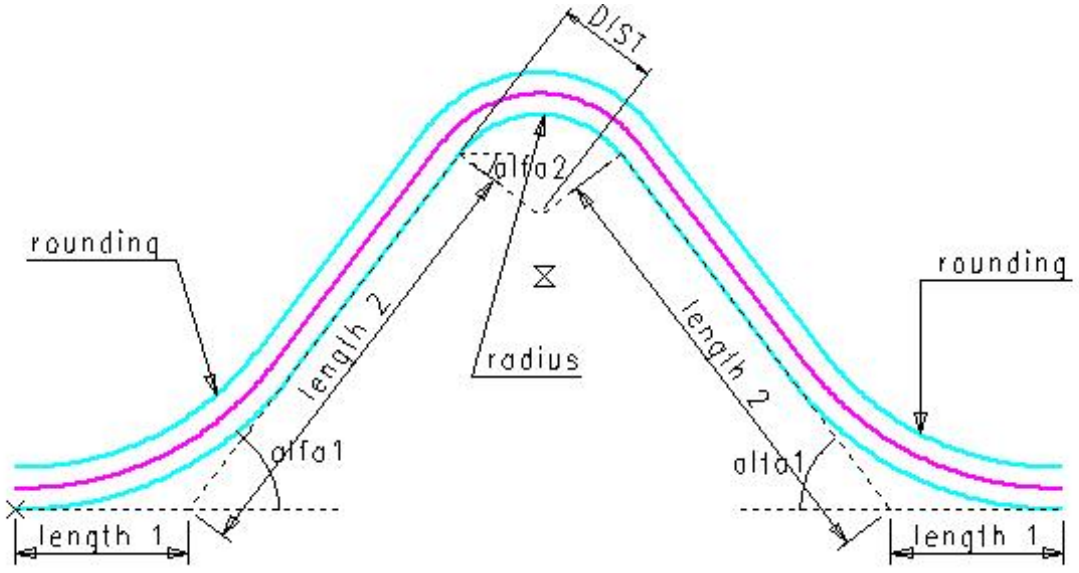
New lines will be created and the curves will be assigned to these ones (lines in surfaces) using for this action the structure manager.

Once this has been performed, the ship can be created to follow working with the rest of FORAN system.
At time of creating, it is suitable to select all the options of checking surfaces to guarantee that all the process has been correctly executed.

PRACTICAL EXAMPLE OF SYMMETRICAL CORRUGATED BULKHEADS

We have two longitudinal corrugated bulkheads in port side. Both of them are symmetrical respect to their corrugation law. The first one (A) is positioned at 1 m of the CL the other one (B) is positioned at 2 m of the CL. The corrugation to be used is the same in both bulkheads (we use in the workshop the same tool to build both of them). The thickness and the corrugation of bulkhead A are looking at side. The thickness and the corrugation of bulkhead B are looking at centre line. The plate's thickness for both is 4 mm.

When defining the corrugation law in FSURF module, we have to note in the sketch, that the thickness of the piece is always looking upwards. It is also important to note that roundings have to be defined on the side with lower radius and the circular segments are also measured in the same way. Measurements are represented in the following drawing:

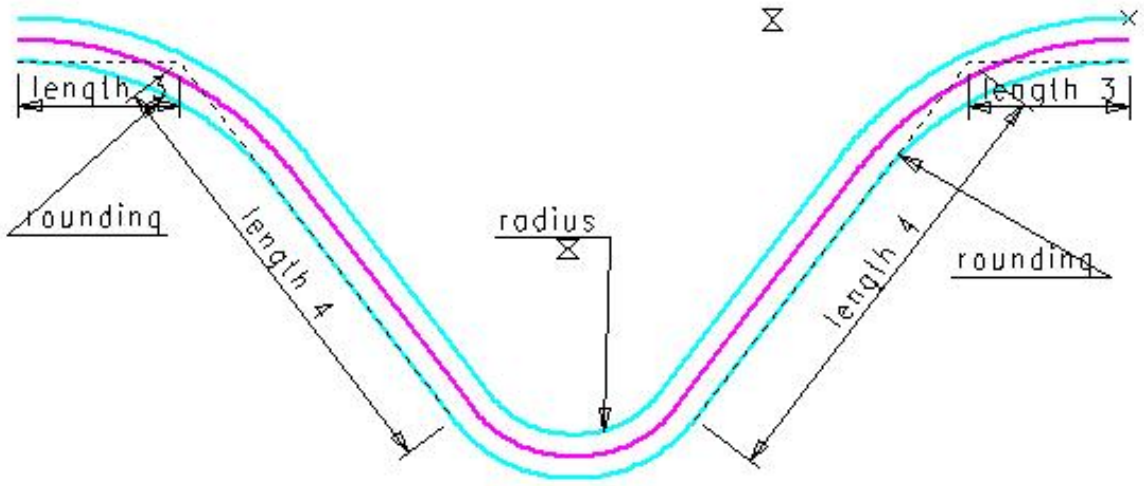


The first corrugation we are going to define is corrugation for bulkhead A, and it is represented in the above sketch. The data to enter are:

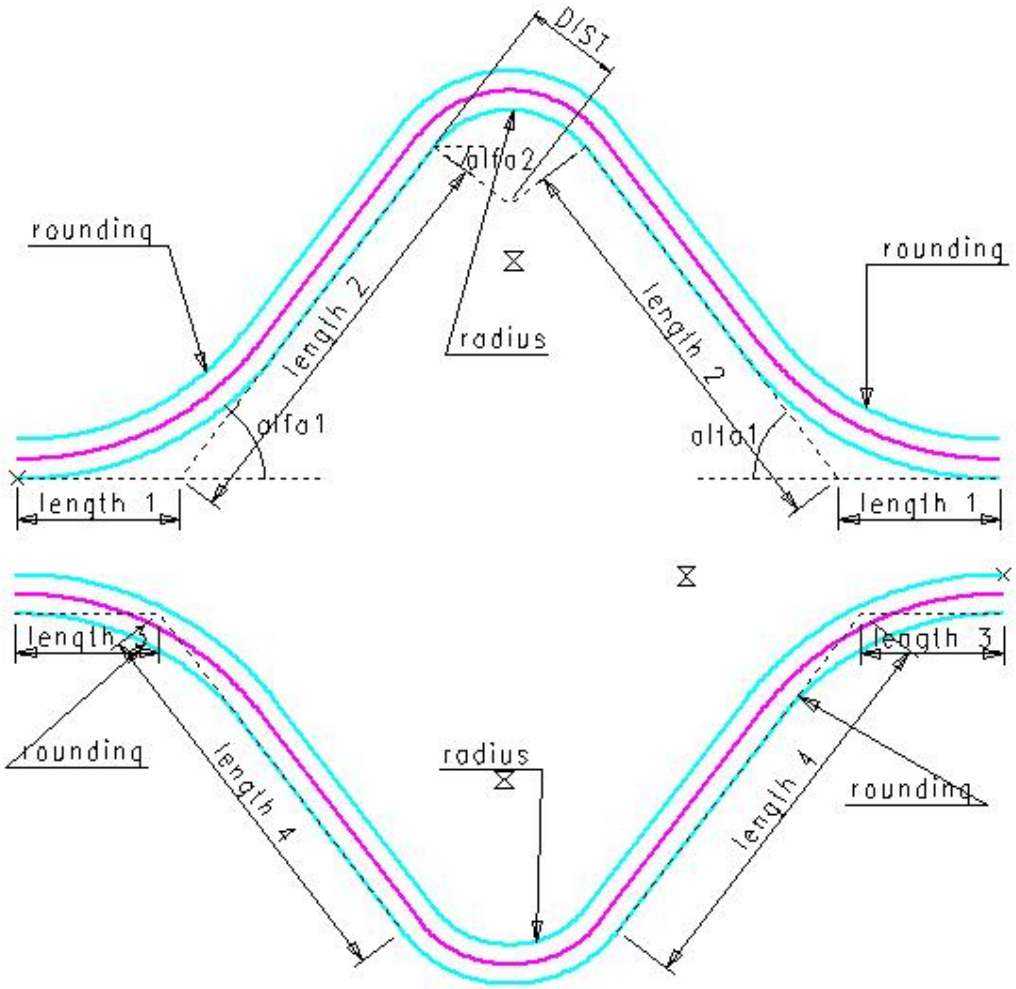
- Length 1	0	0
- Length 2	rounding	alfa1
- Dist	0	-alfa2
- Length 2	0	-alfa1

- Length 1 rounding 0

The corrugated bulkhead is defined in FSURF defining as halfbreadth the moulded line. For the bulkhead A the halfbreadth is 1 m. In IPANEL module, the corrugation is built with the values entered in DECKB and taking into account the thickness assigned to the panel (4mm).
With these data, moulded and thickness lines are built. The thickness orientation is independent of all this, whether it is external (looking at side) or internal (looking at CL), the program moves the built zone in one or other side.
If we look at the above sketch, we can see that this definition is independent of the plate thickness used in IPANEL.
To define the bulkhead B, it is necessary other corrugation definition, due to there is not possibility of decide where the corrugation is looking towards.
We could think that the definition to do is symmetrical to the other one and that changing values of angles we have the corrugation; but this is not true. In the following sketch we have the definition of corrugation:



Roundings and circular segments, as in the previous case, always are measured in the zone with lower radius. Now lengths are not the same as before, since we resemble this sketch to the above one, in the first case we have to measure in the thickness side to obtain lengths 3 and 4. In the following sketch are both definitions and it is easy to note that lengths are different.



The definition we have to do is:

- Length 3	0	0
- Length 4	rounding	-alfa1
- Dist	0	alfa2
- Length 4	0	alfa1
- Length 3	rounding	0

Now lengths 3 and 4 depend on the thickness of the pieces to use, namely if we need to use plates of different thickness for the same corrugation, this requires us to do a definition of corrugation for each thickness.

Defining corrugations in such way, the developed plates for bulkhead A and B will be the same in development length.