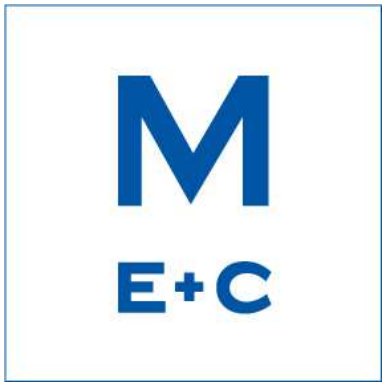


tensiledraw

AutoCAD Software Package - *REFERENCE MANUAL*

http://www.me-c.it - email: info@me-c.it

M
E+C

tensiledraw

Tensile Structure Software

The universal, adaptive and user-friendly
software package for 3D modelling
in architecture and engineering

AutoCAD software package Reference Manual

TABLE OF CONTENTS

1. PRESENTATION OF *TensileDraw*

The following paragraphs will introduce the user to the user-interface and graphic environment of *TensileDraw* FULL version.

TensileDraw is an **AutoCAD** and **Rhinoceros** fully compatible and integrated plug-in software package which can simulate well balanced force stress distribution on the membrane surface without complicating file exportation and related compatibly problems.

The application can calculate the form of complex geometry fabric structures thanks to the generation of a warp and weft beam mesh reproducing orthotropic behaviour of fabric material. Obtained 3D mesh results a powerful tool to compute the efforts transferred to the solid supports.

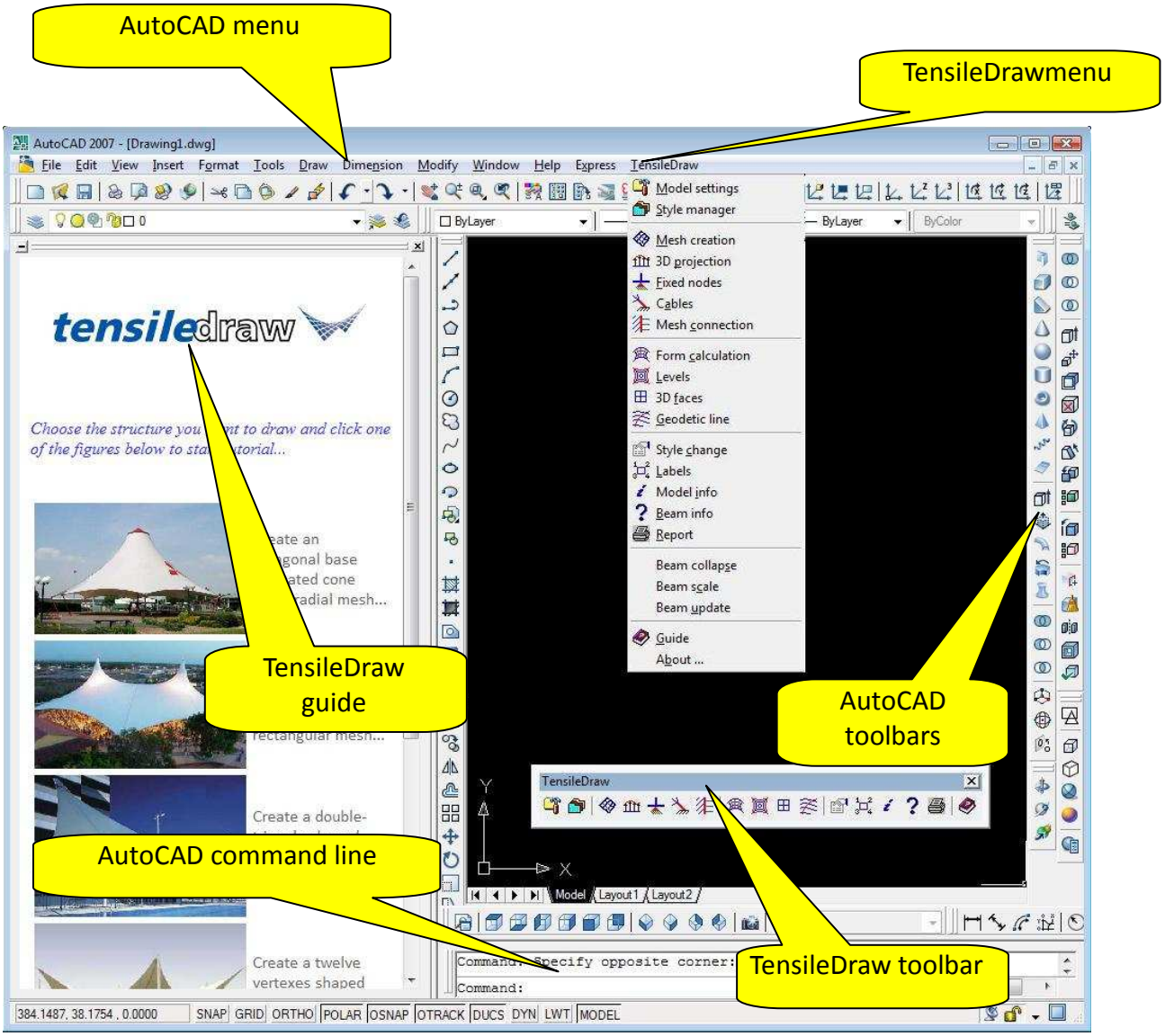
TensileDraw computational skills can be applied to various dimension structures equipped with different structural elements (i.e. fixed borders, fixed length cables, etc..).

Into a single project of *TensileDraw* an unlimited number of nodes can be generated and the user is able to compute even an huge number of elements to obtain an equilibrate fabric shape.
TensileDraw FULL is equipped with some powerful commands which can make even easier the drawing and which give further useful output elaboration data.

2. AUTOCAD SOFTWARE PACKAGE

2.1. AUTOCAD USER INTERFACE

Upon opening *TensileDraw*, the following window appears on the screen:

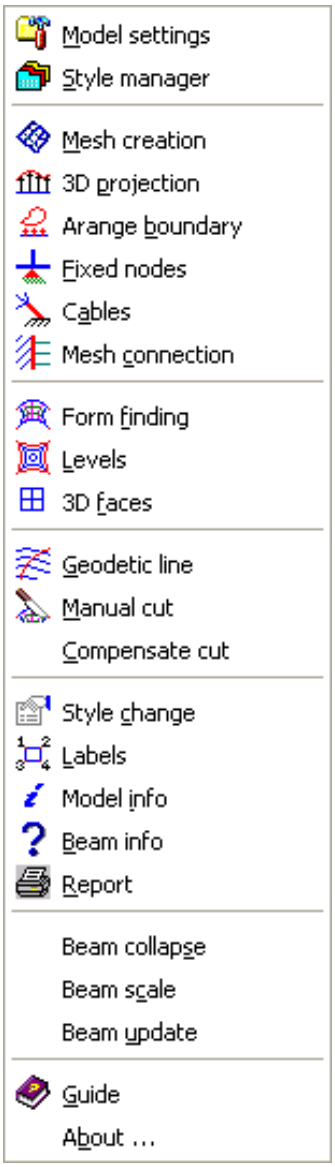


TensileDraw tools are integrated in the AutoCAD window.

The user can collocate *TensileDraw toolbar* in the easier way, either inside the screen or among AutoCAD toolbars.

2.2. TensileDraw MENU AND TOOLBAR

The user can find all commands necessary to use *TensileDraw* skills in both *Menu* and *Toolbar*, except for *Beam Collapse, Scale and Update* command which is present in *TensileDraw Menu* only (See Par. 2.2.13 *Beam Collapse, Scale, Update*).
Command functions are explained in the paragraphs below; all commands can be run typing the parenthetical expression in AutoCAD command line.



Assigns entities and properties to the project.

Sets properties of drawing entities.

Generates the warp and weft beams mesh.

Joins the 2D beams to the 3D proper polyline.

Inserts fix nodes on the drawing.

Inserts the different type of cables.

Joins single meshes into one

Runs 3D form calculation.

Draws contour lines.

Converts beams into surfaces.

Draws geodetic seam lines over the 3D surface.

Patterning.

Cut compensation

Changes element type.

Shows numbering of the points.

Lists useful general data of the model.

Shows useful properties of mesh beams.

Print a .pdf file with technical properties of the model.

Joins separated beams vertexes.

Scales beam properties.

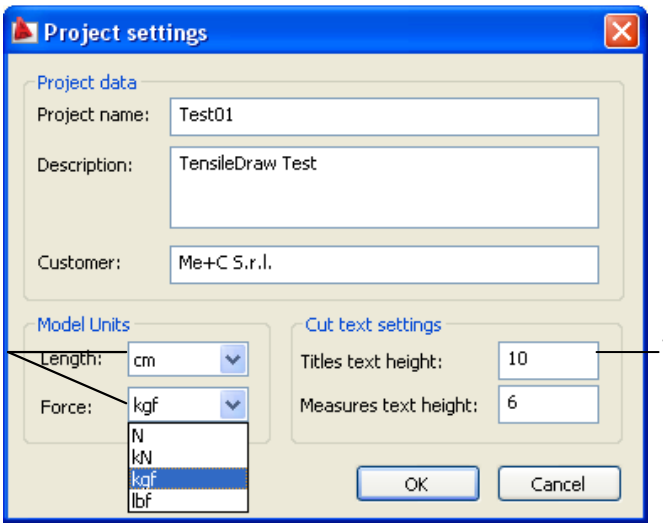
Updates beam properties.

Opens *TensileDraw* interactive guide.

2.2.1. Model settings (_TDsettings) -

In this window the user can name and sign the project and set the units which will be used to visualize element size and to print the technical report.

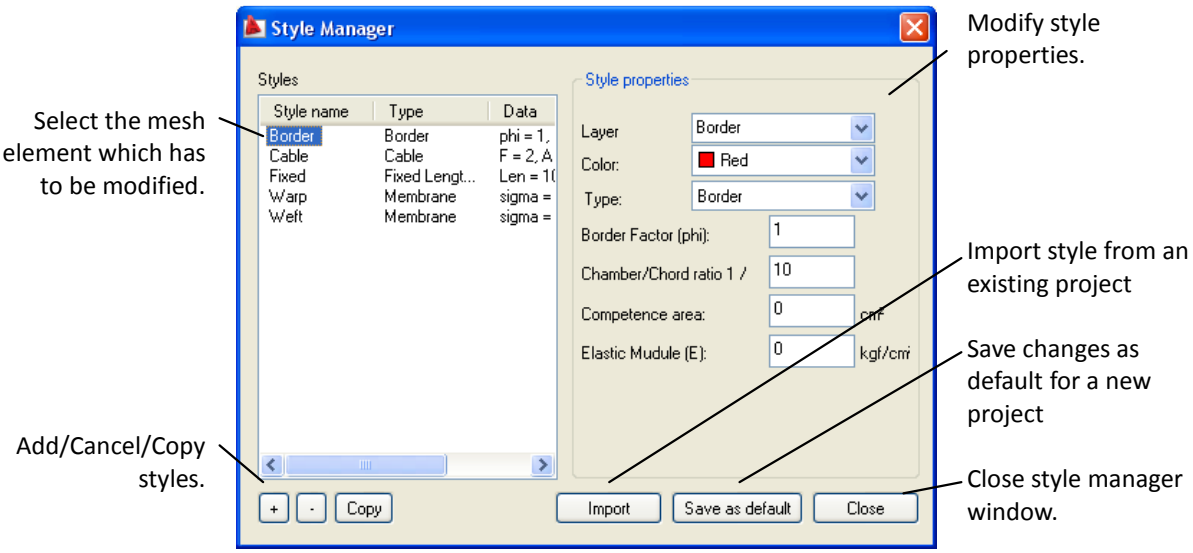
It's important to set the units which will be used to describe the project properties.



Text settings for flat panels dimensions.

2.2.2. Style Manager (_TDstyles) -

Styles command opens the *Style Manager* window which permits the user to generate and edit the properties of all elements constituting the mesh (borders, cables, warp and weft beams).

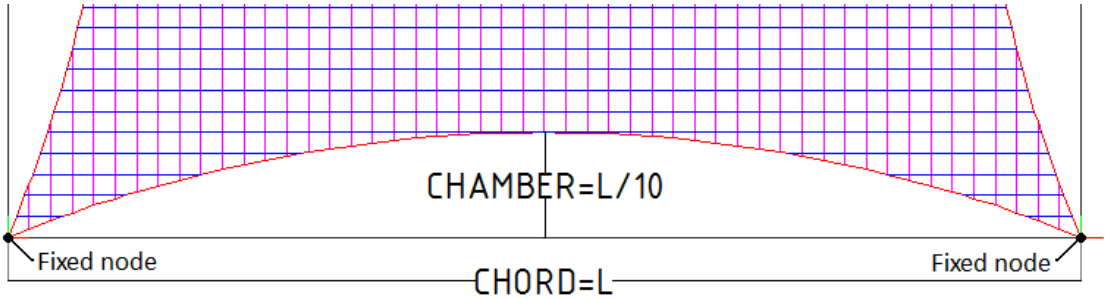


and buttons respectively add and cancel selected styles.

Close button saves changes only for the active project, while *Save as default* button makes changes valid as default properties, available for future projects.

In the *Style properties* section it's possible to assign layer, color and type to the style that is going to be modified. Different properties can be set depending on the selected type (Border, Cable, Membrane). Differences are described below.

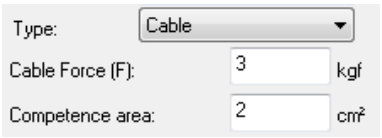
- *Border* function is to define the mesh border shape optimizing the stress distribution on the fabric. This condition is carried out when the *Chamber/Chord ratio* is equal to 1/10 (default value) as shown in the figure below.



During mesh generation, *TensileDraw* calculates the force that has to be applied to border elements in order to obtain the required shape. This force value is associated to *Border Factor (phi)* equal to 1 (default value). If the phi value is changed chamber/chord ratio could diverge from default value 1⁄10.

Setting *Chamber/Chord ratio 1/* "0" is possible to obtain a straight border simulating the fabric clamping along the border of the structure.

- *Cable* elements are characterized by the *Cable Force (F)* property: this is the force value which will be applied to the element with any geometry changing.

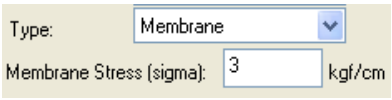


Competence area is the cross-section area of the cable.

- *Fixed Length cable* elements are defined by the *Cable length* value which will be maintained despite any changes.

Competence area is the cross-section area of the cable.

- *Membrane* is the warp and weft style type which is characterized by the *Membrane Force* (*sigma*) property. This is the starting pre-stress force density which has to be set by the user on the base of the structure characteristics: TensileDraw will re-calculate sigma value of each beam to obtain a well balanced 3D fabric geometry with the properties as close as possible to the required one.



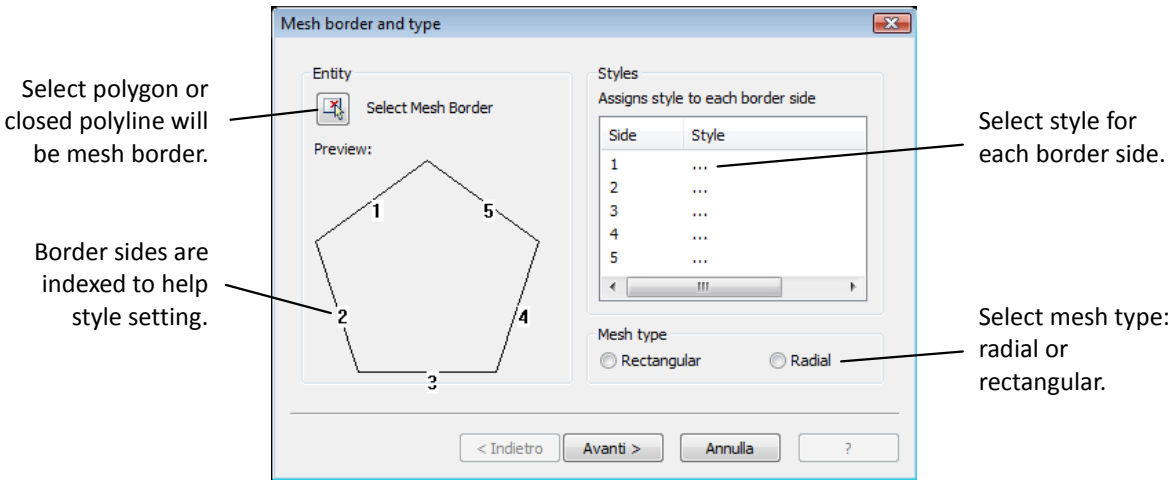
Look at the examples or join the TensileDraw on-line forum to get more help to a forceful use of membrane properties.

NB: the *Membrane Force (sigma)* is a pre-stress force density value which does not keep into account structural loads. Load and stress values given as output data by the simulation must be used strictly for form calculation and NOT for structural design or executive projects.

2.2.3. Mesh Creation (_TDmesh) -

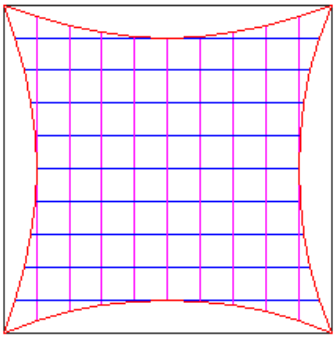
Mesh creation starts selecting the mesh border, which can be either a regular polygon or a closed polyline, both of which can be drawn by the proper AutoCAD command (see *TDGuide* for further information).

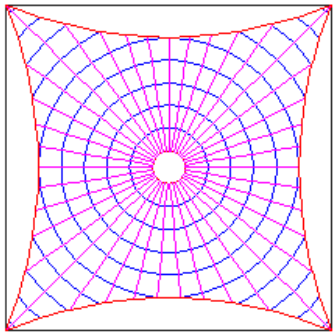
Upon executing the *Create Mesh* command the window below will open:



- *Select Mesh Border*: this button allows the selection of the border polyline in the drawing. In the figure above a regular pentagon used as border is shown.
- *Styles*: a different style can be fitted to each side. One style named *Border* is set by default. If the user needs more styles, it's possible to create them by using the *_TDstyle* command (see Par. 2.2.2. *Styles Manager*).
- *Mesh Type*: user can select between *rectangular* and *radial* mesh.

Rectangular mesh in square shape border



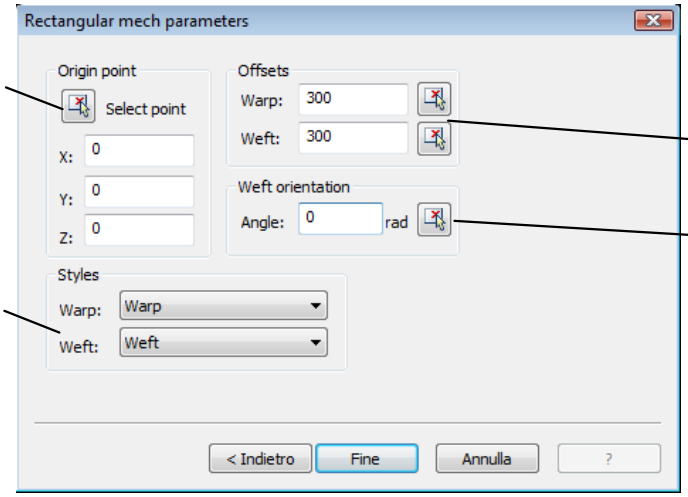


Radial mesh in square shape border

Clicking *Next* button, user will visualize a slightly different window depending on which mesh type has been chosen.

In rectangular mesh warp beams are perpendicular to weft ones in every node and both can be generated with a proper offset distance (see figure below).

Type the origin point on the screen or insert its coordinates in the cells.

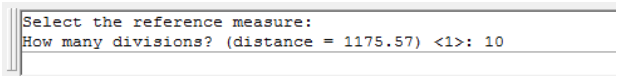


Set offsets for warp and weft beams.

Set angle relative to horizontal.

Select beam style.

- *Origin Point*: is the point from which mesh starts to be generated.
- *Offsets*: two different values can be set either by typing them directly into the cells, or clicking the reference distance and the number of sections it has to be divided into. The default divisions number is set to 1.

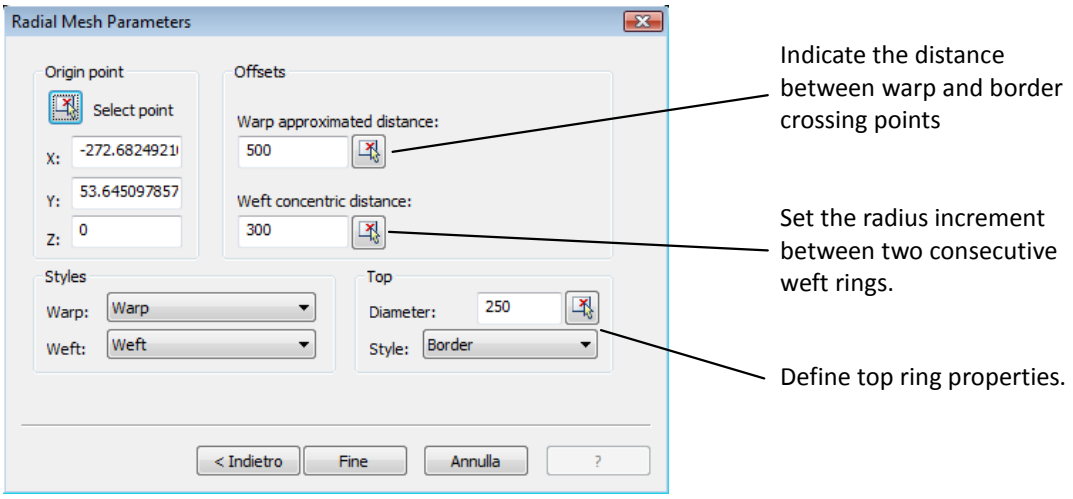


- *Beam orientation*: rectangular mesh can be rotated around the origin point of a given angle relative to horizontal.
- *Styles*: the user can create as many beam styles as needed. Two styles named *Warp* and *Weft* are set by default, but more can be added by using *_TDstyles* command (see Par. 2.2.2. *Style Manager*).

In radial mesh warp, beams all start from the origin point and end at the base border with a specified distance, while weft beams are generated concentric to the origin point each with a different radius. Differences from *Rectangular Mesh Parameters* window are described below.

- *Offsets*: the distance between two consecutive warp and border crossing points is an approximate one. During the mesh generation phase, *TensileDraw* will round off the given value to maintain a warp beam line in correspondence with the border polyline vertexes. Instead the distance between two consecutive weft beams rings is considered properly. Both the distances can be set either directly into the cells, or by clicking the reference distance and the number of sections it has to be divided into.

  Default divisions number is set to 1.



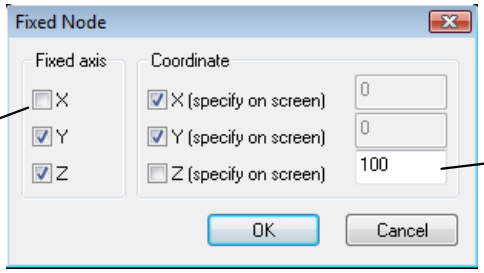
- *Top*: in this section it's possible to define the properties of 3D geometry upper part. *TensileDraw* allows the use of a circular shaped top border to calculate the fabric form (see *Calculate From*). The ring is centred in the origin point and needs to be characterized by a border style. If a diameter value is not typed warp beams will join at the origin point.

2.2.4. Set fixed nodes (_TDfixnodes) -

Fix nodes function is to block coordinates of beam vertexes which will be anchored to the solid structure. The user can choose which axis (X, Y, Z) has to be fixed or not and consequently which direction the node can move in when form is calculated.

Fixed node position can be defined in the window below by typing coordinates in their proper cells or selecting the points directly on the drawing.

Set which axes will be fixed and specify nodes on the screen (i.e. nodes will move along X direction).



Assign a fixed value to node coordinates (i.e. Z=100) .

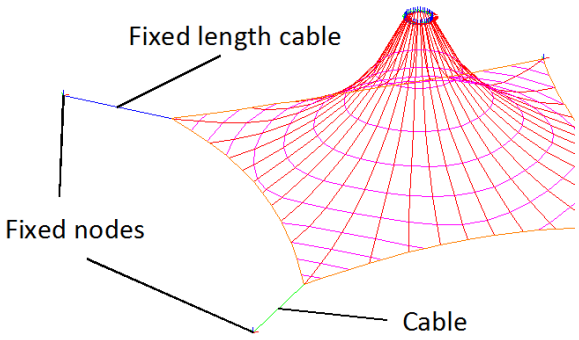
In the second case the command works in two ways:

- select beams before entering the command: all selected beam terminal points will be anchored.
- run the command and then specify fixed nodes on the screen.

2.2.5. Cables (_TDcable) -

Cables are particular kinds of beam created to simulate the behaviour of real tension elements. There are two different sorts of cable: *Cable* and *Fixed Length Cable*.

- *Cable*: the user can fix the force value which has to be apply to the element.
- *Fixed Length Cable*: the user fixes the length value the element has to maintain.

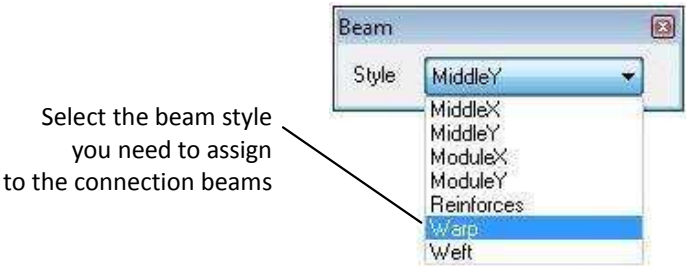


Both kinds of cable have some property values set by default, which can be modified in the *Style Manager* command window (see Par. 2.2.2. *Style Manager*).

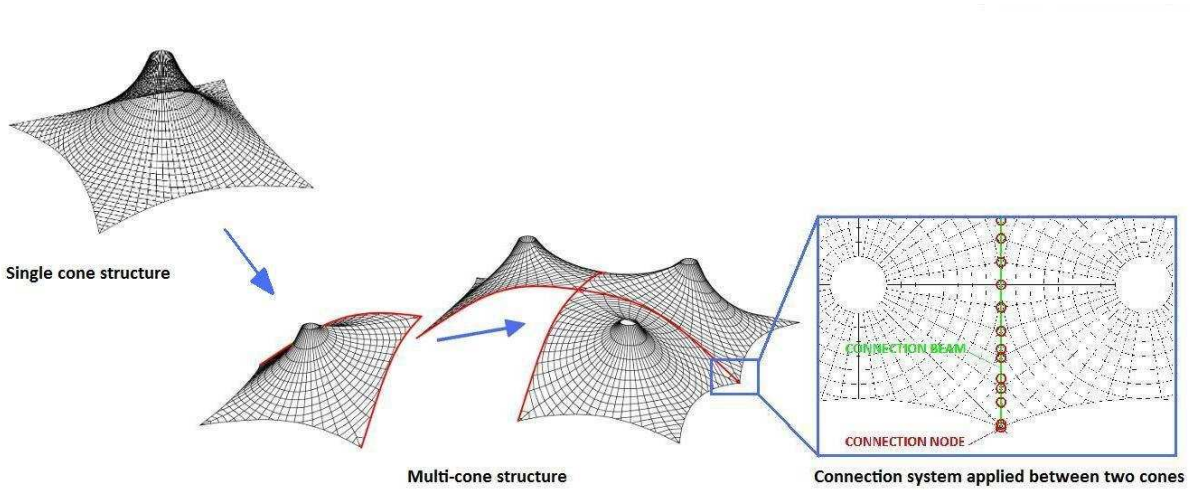
2.2.6. Mesh Connection (_TDconnection) - 

TensileDraw command *TDconnection* allows the joining of differently shaped simple structured into one complex geometry. The unlimited calculation skill enables to calculate the form of mesh composed by huge number of nodes and beams.

At least one side of the meshes you need to connect, has to be put ajar each other. Run *TDconnection* and select the beam style needed to be assigned to the connection beams.

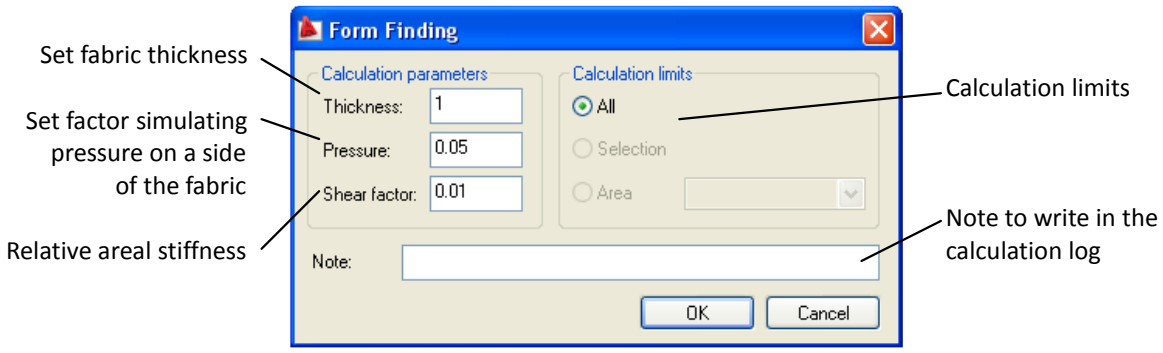


Insert the connection beams along the side in common between the meshes: every beam vertexes along this line will be “fixed” by one connection node. The connection beams are created with the proper competence area on the way to have an equilibrate form calculation; the connection nodes maintain the meshes joined but allow the movement of the connection border in order to calculate the correct 3D shape.



2.2.7. Form Finding (_TDcalc) -

TDCalc starts the form calculation: it needs at least one fixed node to run, but to obtain the expected result it's necessary that all fixed nodes are positioned in the right anchor point. After *Calculation form* properties are set, click *OK* button to start calculating.



In most of cases *Pressure*, *Thickness* and *Shear factor* default values are not necessary to be changed:

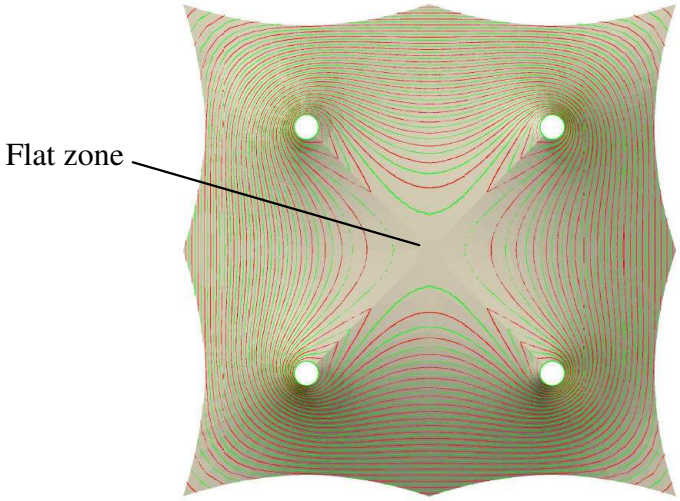
- *Thickness* can be change when the fabric material thickness could influence the form finding step.
- *Pressure: this tool allow to simulate a pressure distribution ageing on one side of the fabric. In this way it's possible to create "pillow" fabric shape structure and calculate the 3D surface and volume they take up.*
- *Shear factor:* this factor is used to reduce area deformation with pressure. This value is the proportion between warp and weft stresses and the areal stiffness. The suggested values range between 0 and 0.1.
- *Note:* after any form finding the program updates the rdf_log.txt file into the .mbr folder with the last parameter values. The Note field allows to write a free text in the log file.
- *Calculation limit* allows to calculate the *All* entities in the current drawing, the *Selected* entities only, or the entities inside a calculation *Area.*

2.2.7.1. Calculation area

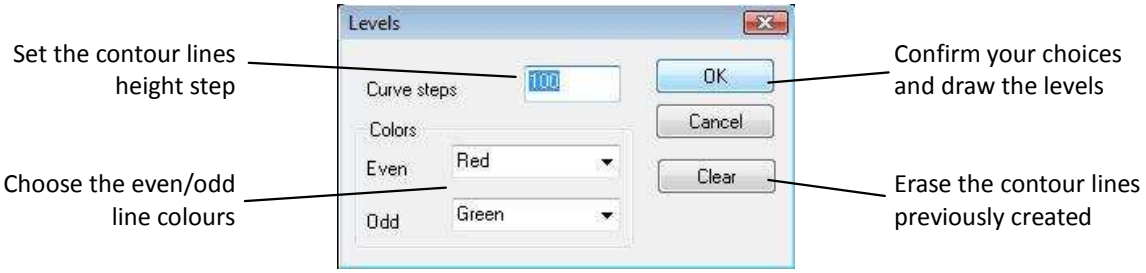
Calculation area is a polygon used to limit a form finding or pattering operation. To define a calculation area, simply assign a name and the vertices coordinate.

2.2.8. Levels (_TDlevels) - 

TDlevels command draws the contour lines on the fabric surface: this is the easiest way to visualize the surface slope and localize potential water stagnation zones.

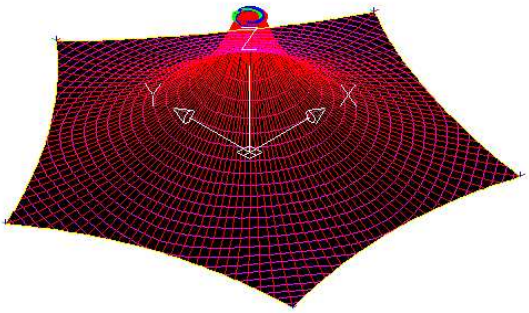


Run the *TDlevels* command: in the appeared command window the user has to choose the height step of the contour lines and the colours to assign them to obtain the best visualization of the surface slope.

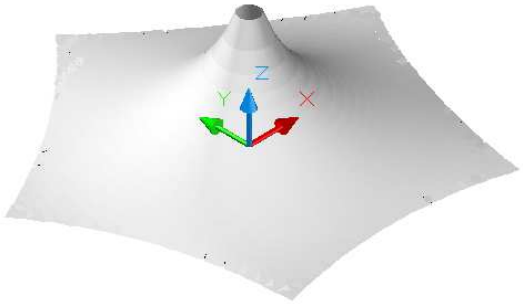


2.2.9. 3D Faces (_TDFaces) -

This command draws 3D faces on the mesh: this is an useful tool to visualize more clearly the calculated form with AutoCAD skills and to export geometry in different files format.



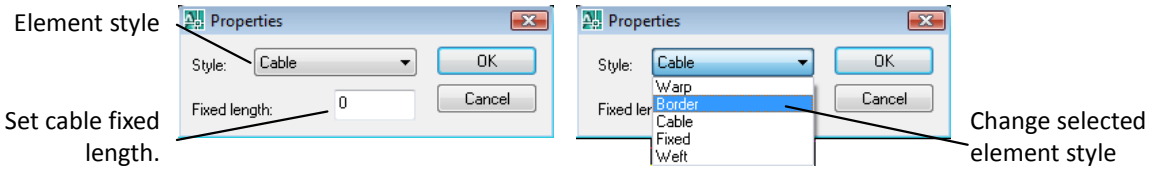
Warp/Weft Mesh.



3D Faces.

2.2.10. Style Change (_TDchange) -

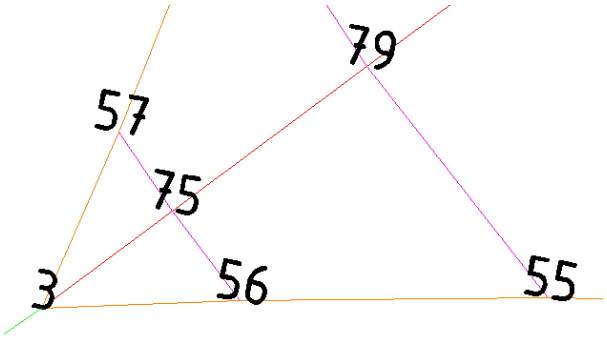
This command permits the user to select one or more elements of the mesh and visualize their style. It's also possible to change the element style selecting between all the styles defined by the user.



If a Fixed *Length Cable* is selected, the user can modify its length directly in the window above.

2.2.11. Labels (_TDlabels) -

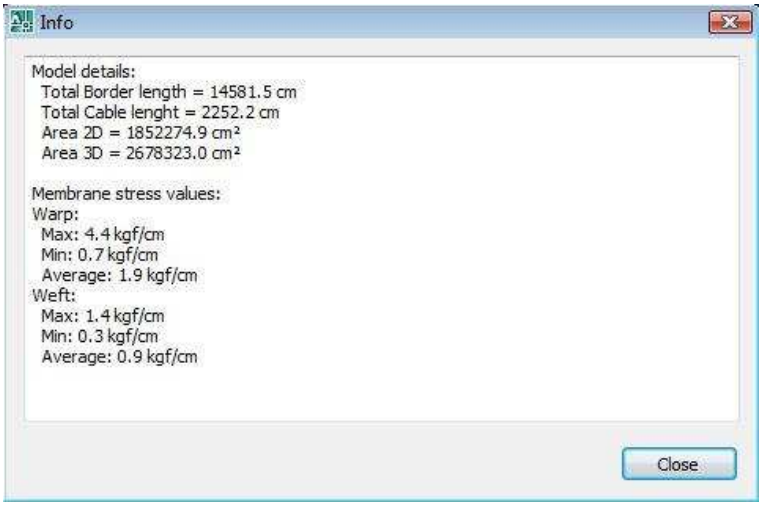
Labels command assigns a progressive number to the mesh nodes and visualizes them on the drawing.



2.2.12. Model Info (_TDmodelinfo) -

This command gives some useful information about the project:

- *Model details:* border and cable style total length, 2D and 3D surface area are functional informations to have a preliminary idea of the structure dimension.

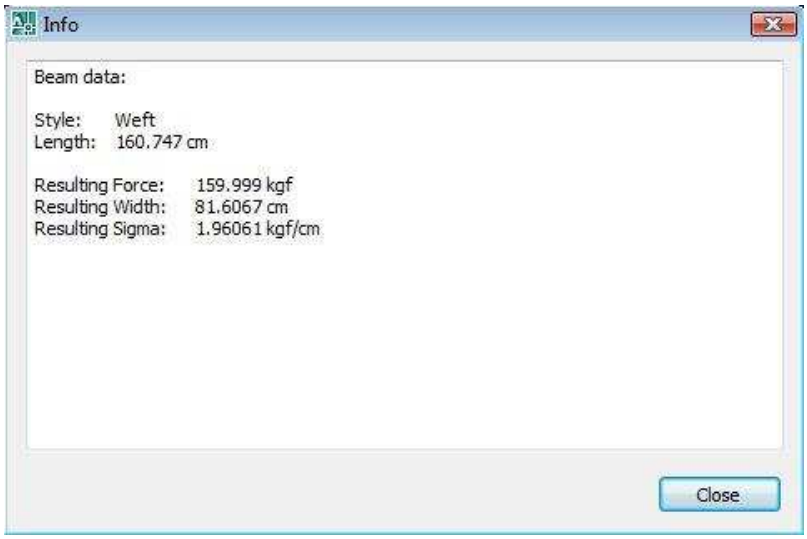


- *Membrane stress values*: the user can immediately know the warp and weft pre-stress range and understand if the starting sigma value could be correct or not.

_TDmodelinfo works strictly after form has been calculated (see *2.2.6. Form Calculation*).

2.2.13. Beam Info - 

TDbeaminfo gives useful technical informations about the properties of the selected element: beam dimension and competence area, cable stress value, fixed node reactions, pre-tension membrane value. The user has just to run the command and click on the element that has to be examined (beam, cable, node...).



2.2.14. Beam Collapse, Scale, Update

The commands of this group should not be used as often as the other, because their function is to correct or repair potential errors occurred during the mesh generation phase or caused by geometry changes during the form finding. These commands are not present in the *TensileDraw* toolbar, but can be run from *TensileDraw* menu or typing them into the command line.

- *Beam collapse* does three actions at the same time: it joins the beam vertexes whose reciprocal distance is smaller than the specified tolerance value, it cancels duplicated beams and null length beams. This is a very powerful command and it's important to use it in a proper way to avoid damages to the mesh structure.
- *Beam scale* has to be used when the model is scaled using AutoCAD command *scale*. In fact the AutoCAD command changes only the geometric dimension of the elements, while *Beam scale* works on the competence area dimension and stress value.
- *Beam update* is necessary to update the beam properties when the structure geometry is changed.

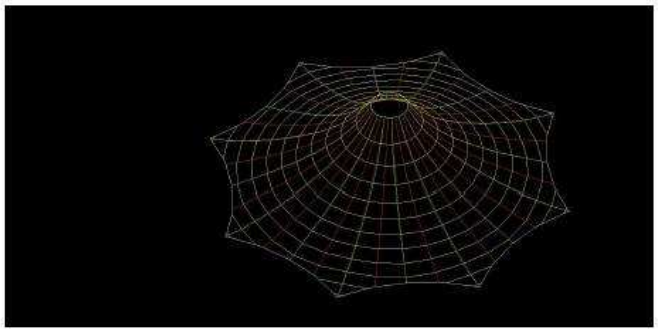
2.2.15. Report (_TDreport) - 

TDreport command prints a .pdf format file with a picture of the drawing, the project properties set by the user and some useful informations about the structure (membrane surface, pre-stress value, cable and border length..).

tensiledraw

PROJECT SHEET OVERVIEW

Project title:
Description:
Customer:



Project relevant informations:

- Membrane surface : 2813851.4 cm²
- Total Border length : 6906.7 cm

2.2.16. Guide (_TDguide) - 

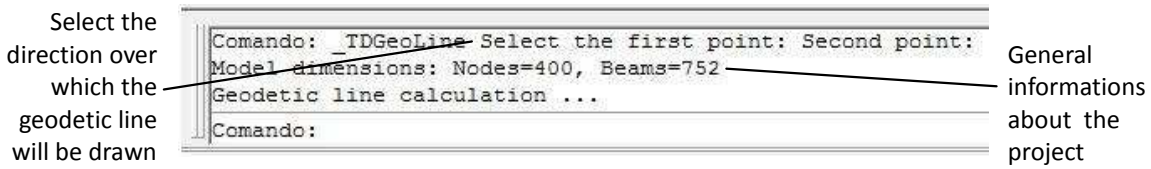
This command visualise the *TensileDraw Interactive User Interface.* This is visualized by default at *TensileDraw* starting, but it can be optionally disabled in *Tools/Options/TensileDraw* AutoCAD menu.

2.3.PATTERNING

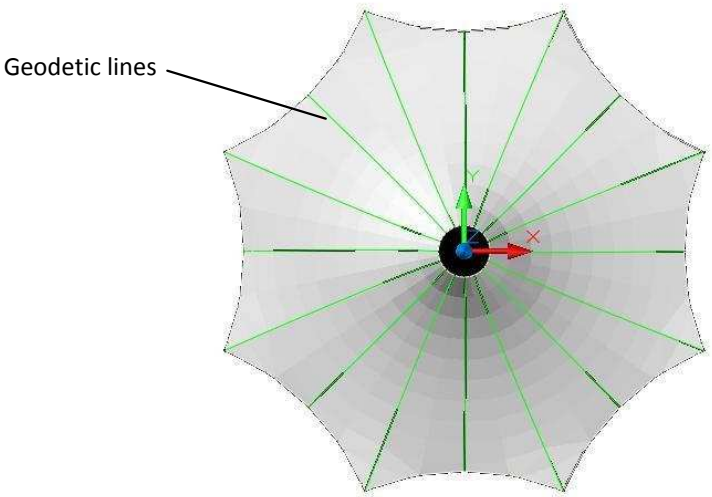
Patterning is the process of defining two-dimensional pieces of fabric, which can be spiced together to form a desired three-dimensional shape.

2.2.10. Geodetic lines (_TDgeoline) - 

TDgeoline command draws geodetic lines over the fabric surface in correspondence to user defined direction lines.

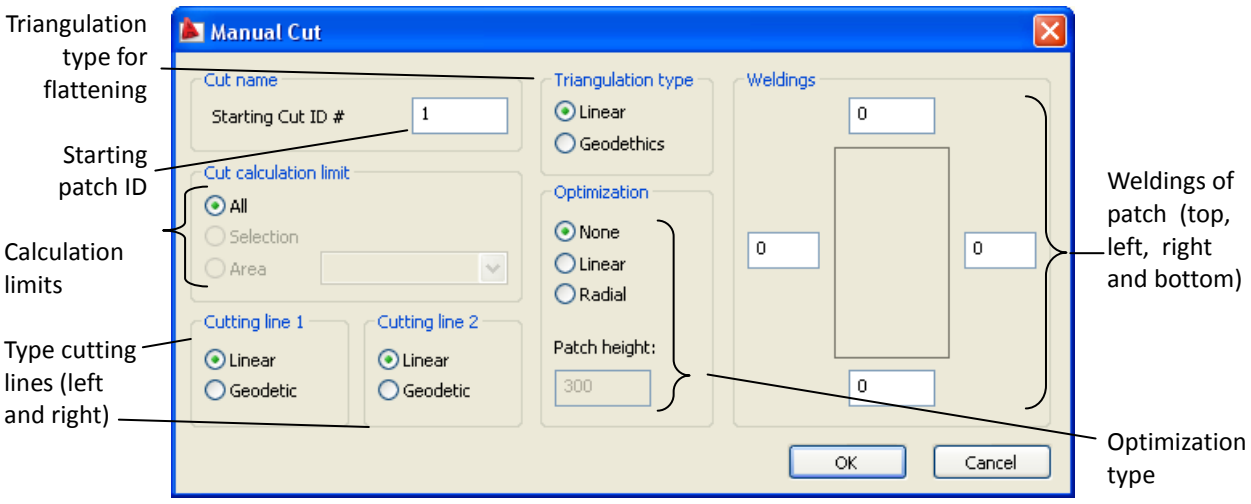


Geodetic lines are a useful tool to visualized the position of pattern cutting lines over the 3D surface.



2.3.2.Cut (_TDcut)

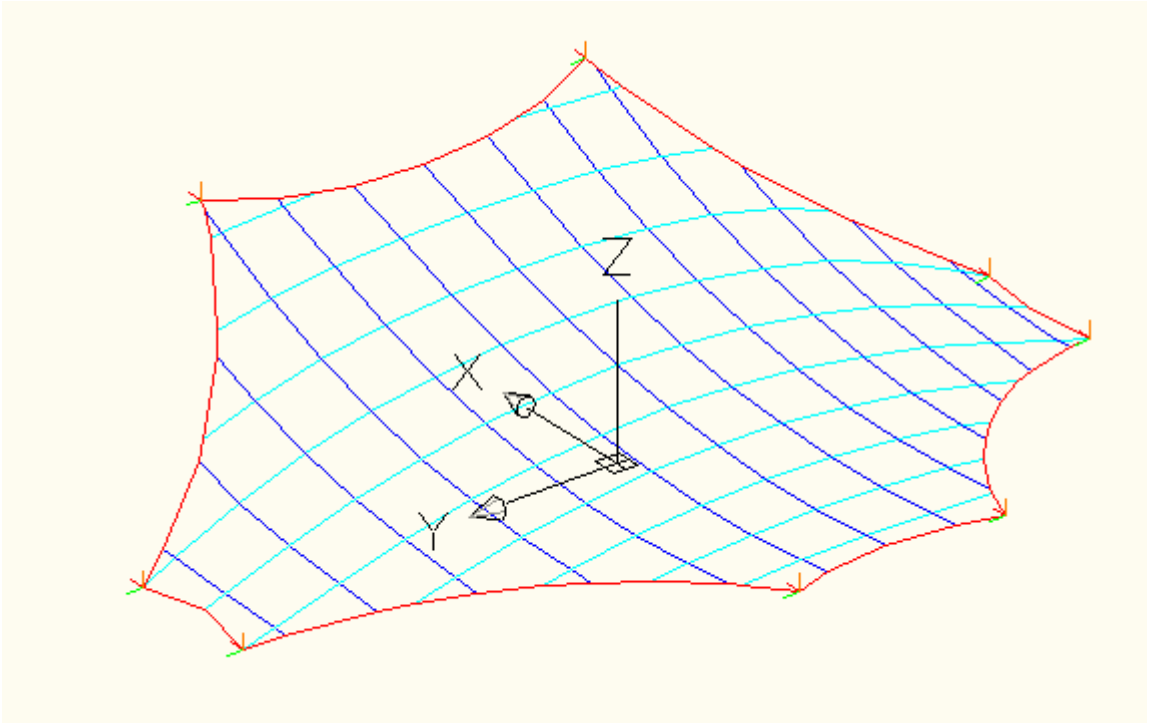
This command creates two-dimensional pieces of fabric (cut or patch) starting by the user defined cutting lines.



- *Starting Cut ID:* the unique ID number of the first flat panel to be generate by the command
- *Calculation limit:* allows to use a portion of structure to calculate the patch.
- *Cutting line 1/2:* the type of cutting line to generate: linear or geodetic.
- *Triangulation type:* the type of triangulation that the program use to calculate the flat panels.
- *Optimization:* this option allows to force the height of the patch by moving the second cutting line until to obtain the specified value.
- *Weldings:* draw the weldings at the specified distance offset.

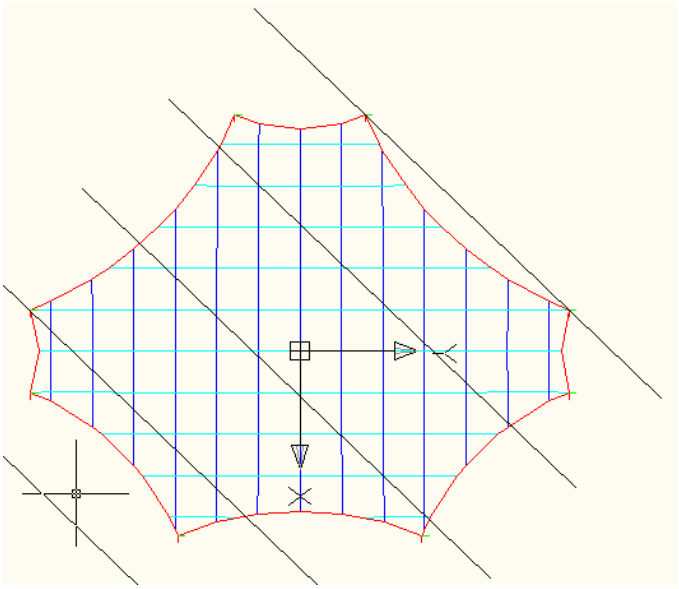
How to proceed:

1. Open the model file



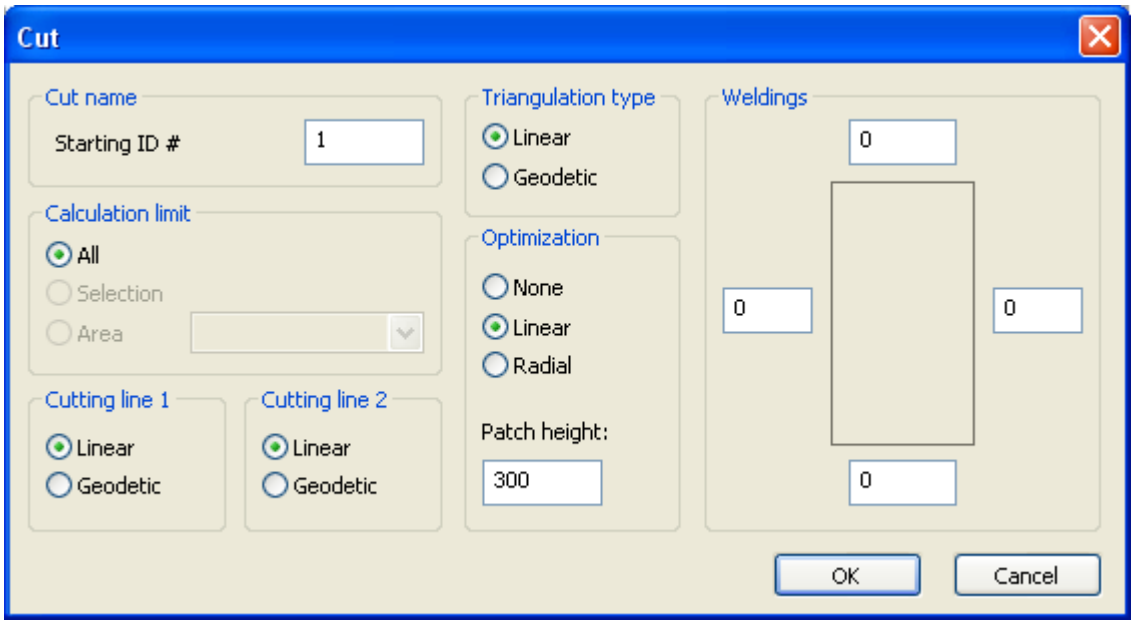
Model perspective view

2. Draw the cutting lines using polyline tool. You can use the offset tool to create the polylines at the patch height distance.

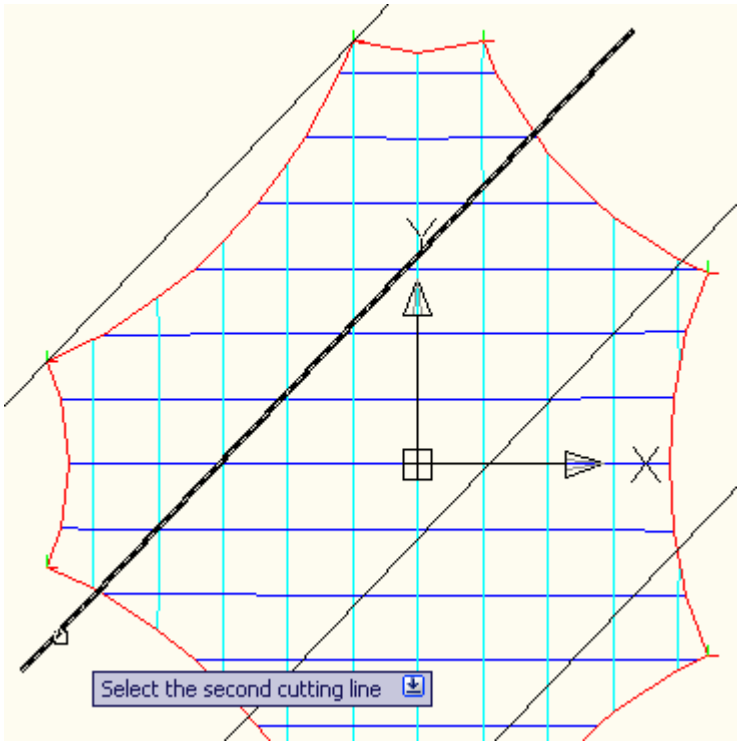


Model top view: polylines used as patch limits

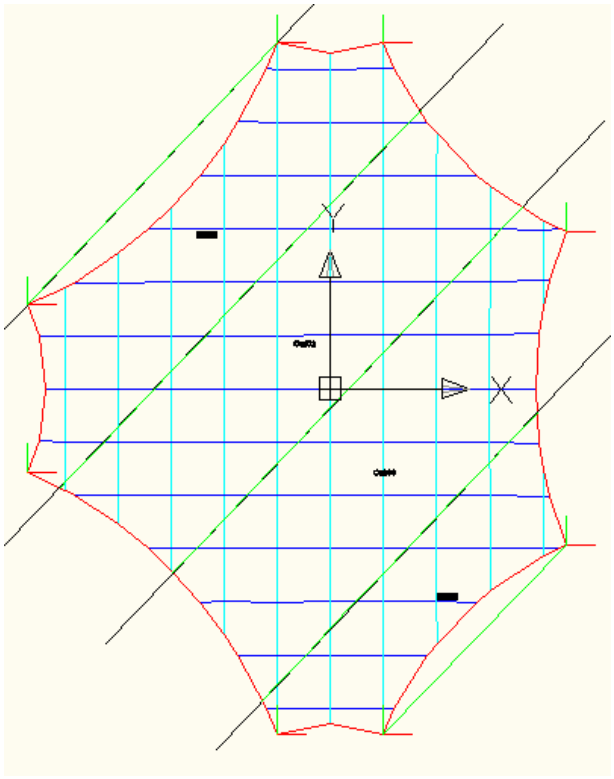
3. Run _TDcut command and set Triangulation type = Linear, Optimization = linear and the path height at the desired value.



4. Click the OK button and select the first and the second polyline: if optimization is active, the command moves the second polyline in accord with the optimisation parameters.

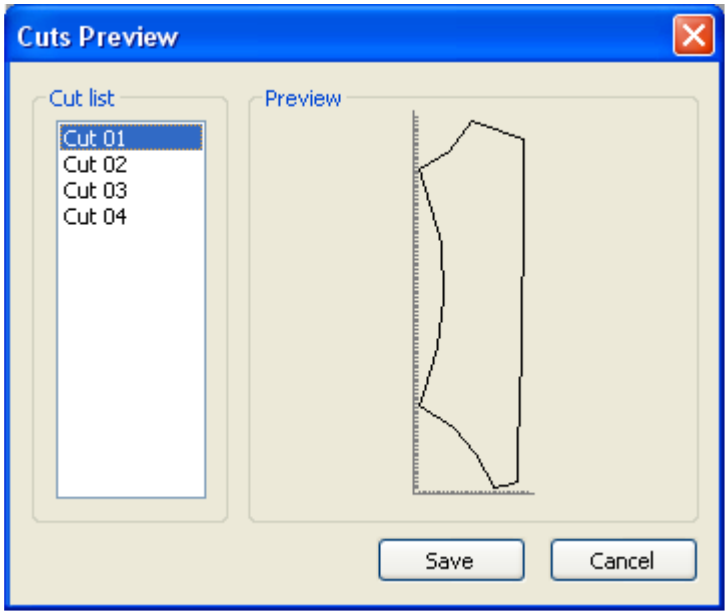


5. Carry on selecting the other polylines: if optimisation is active the polylines are moved automatically to the patch height distance.

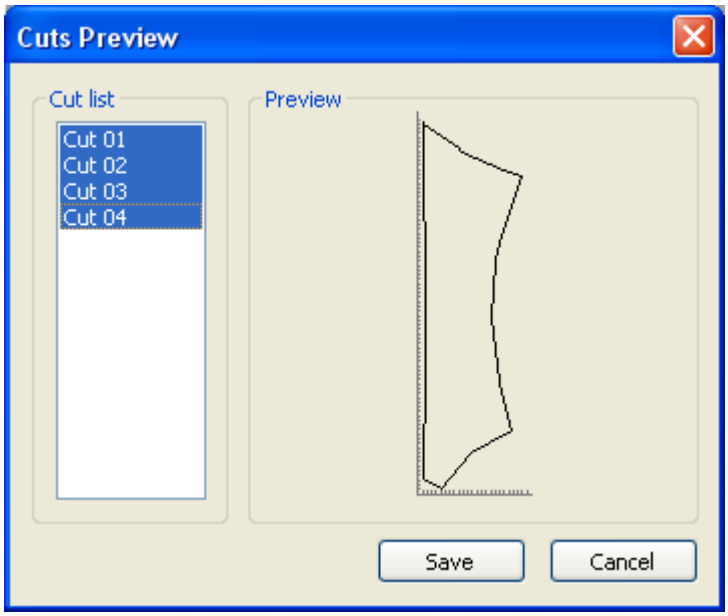


The polyline has been moved to the defined patch heignt

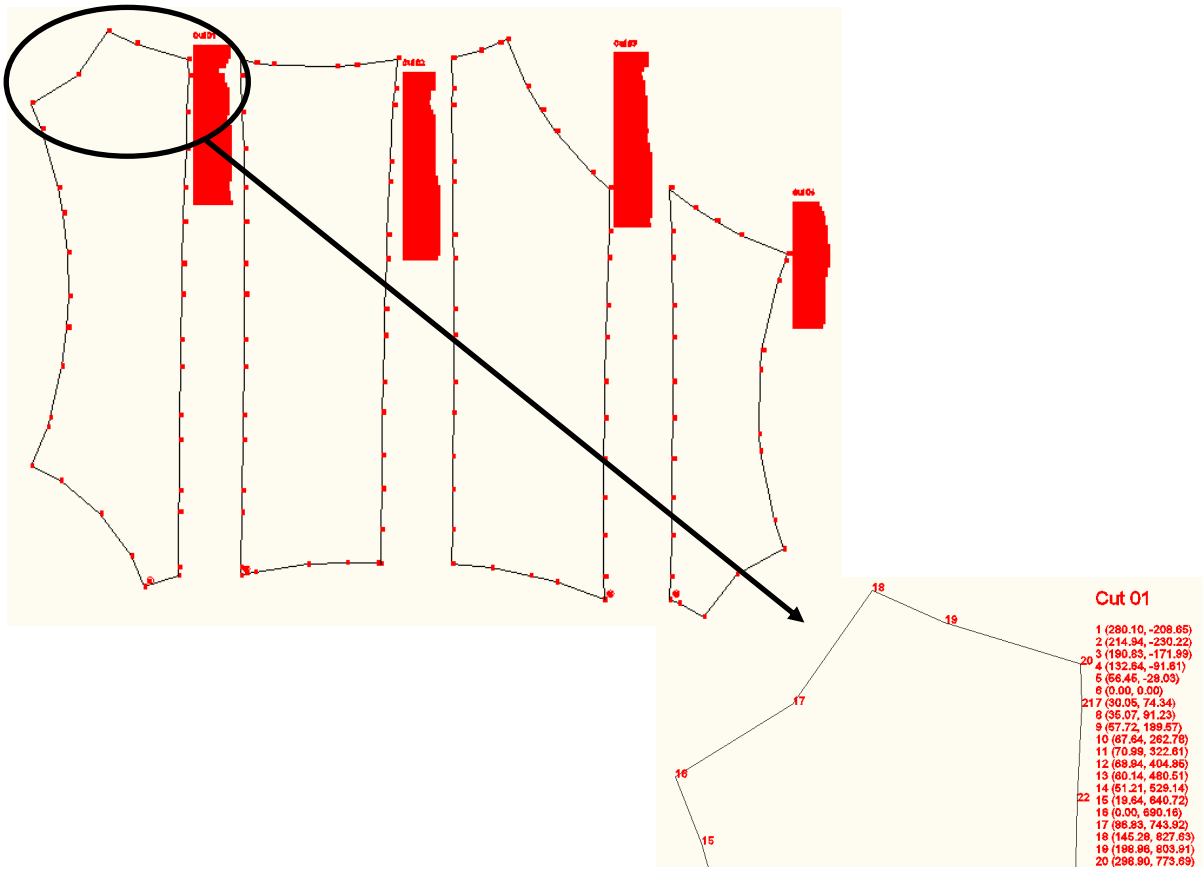
6. Press [Enter] to finish. Now the program shows the 2D panels into the preview window. You can see the patch by clicking the proper name in the cut list.



7. Use shift key to select more panels at the same time. To save the selected panels, press Save button.

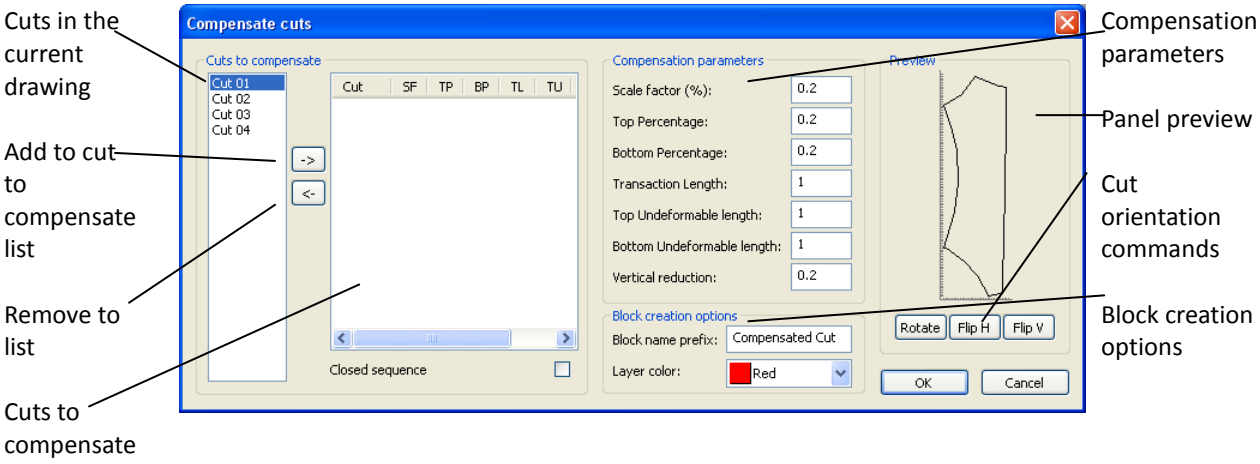


8. The program creates a block for each patch. You can place the blocks in the drawing using the insert tool.



2.3.3 Compensation (_TDCompensateCut)

Tensile structure 3D final shape is obtained giving to the membrane the proper pre-stress value. This is obtained by the installation of 3D membrane that in relax condition has a smaller surface area than in stressed condition. Compensation command allows to reduce the dimensions ofthe panels generated by _TDCut command (see 2.3.2) to take into account the proper pre-stress values.



- *Cuts to compensate:* this section is composed by two list and two buttons. The list on the left (L) contains all the generated patches in the current drawing. The button '->' move the selected cuts from the L list to the list on the right (R) that contains the cuts to be compensated. When a cut is added to the R list the program assign the current compensation parameter values.
- *Closed sequence:* this option tells to the program that the last cut in the R list is attached to the first one.
- *Compensation parameters:* this section contains the compensation parameters which are applied to every cut in the R list.
- *Block creation options:* you can set the prefix for the compensated patches block name and the layer color.
- *Preview*: this area shows a preview of the selected pattern. Rotate, Flip H and Flip V buttons change the orientation of the pattern selected from in the R list.

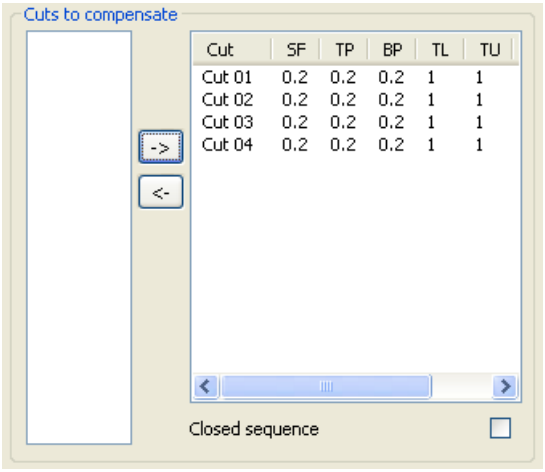
How to proceed:

1. Create the 2D panels using _TDcut command as described in the 2.3.2 section.

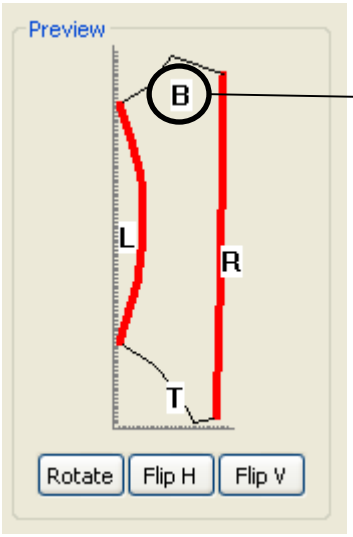
2. Run _TDcompensatecut command.

3. Change the 'Compensation parameters' values in according to the fabric material.

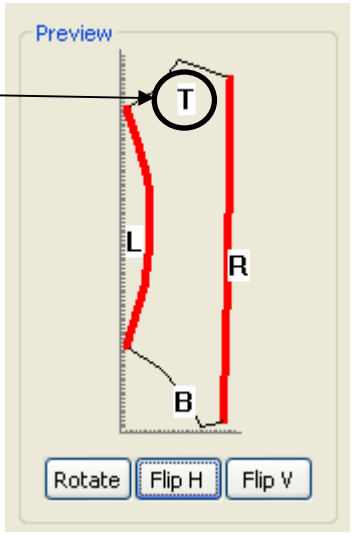
4. Select the panels in the L list and press the '->' button to put them in the R list. The order of the elements in the R list, must be the same of the ones in the L list: each panel side must match the side of its contigous panel. If the right side of the last panel matches the left side of the first one (e.g. a cone structure) tick the 'Closed sequence' option.



5. Select R list items one by one to see the preview. Check if the panel is properly oriented so that the top side is facing up the screen. You can change the orientation using the buttons in the preview area. In the following sample the orientation has been changed from image 1 to image 2.



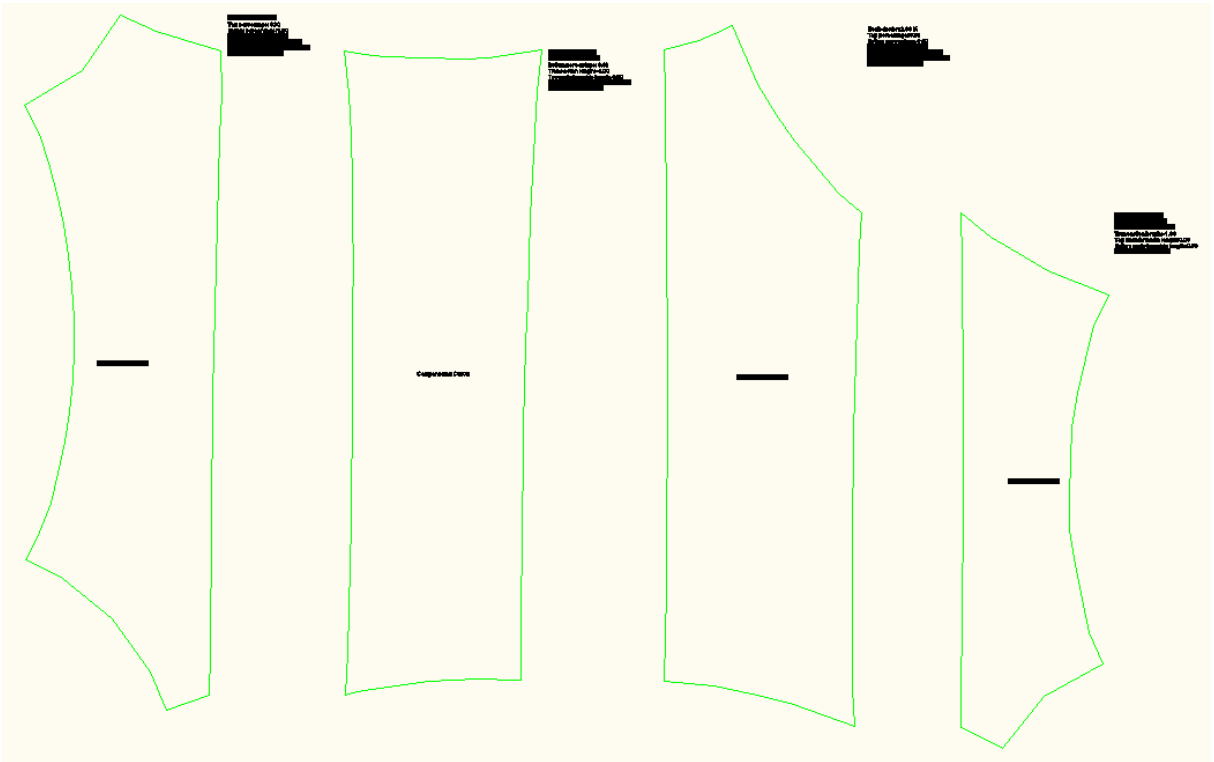
1



2

6. Set the block name prefix and the desired color.

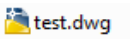
7. The program creates a block for any compensated 2D panel. It possible to place these blocks by using the insert tool.



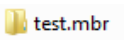
2.3. TENSILEDRAW OUTPUT FILES

When a *TensileDraw* project is saved two elements with the same name are generated:

- *.dwg* extension file: this is the typical AutoCAD output file which describes shape and geometric dimensions of the structure.



- *.mbr* folder: in this folder *TensileDraw* saves all project informations, mesh elements settings and styles properties.



Before calculating the form, *TensileDraw* requires to save the project, because the name of the project it's necessary to create the . *mbr* folder.

In order to allow the application working properly and maintain the properties previously created, both the elements have to be in the same folder and named in the same way.

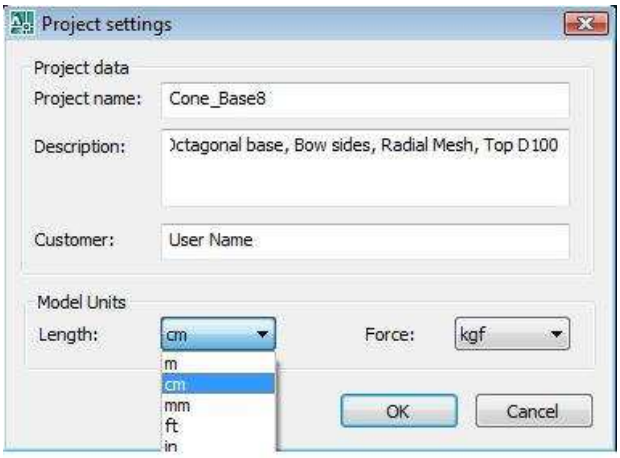
2.4. EXAMPLES

2.4.1. EXAMPLE 1: Create an octagonal base truncated cone using radial mesh

Open AutoCAD with *TensileDraw* software package: a new AutoCAD file (.dwg) is created automatically. Save it in *C:\Program Files\Me+C\TensileDraw\Samples* as *Cone_base8.dwg.* If it's not possible save the file now, remember that it will be required before run the form calculation.

- *Set project properties*

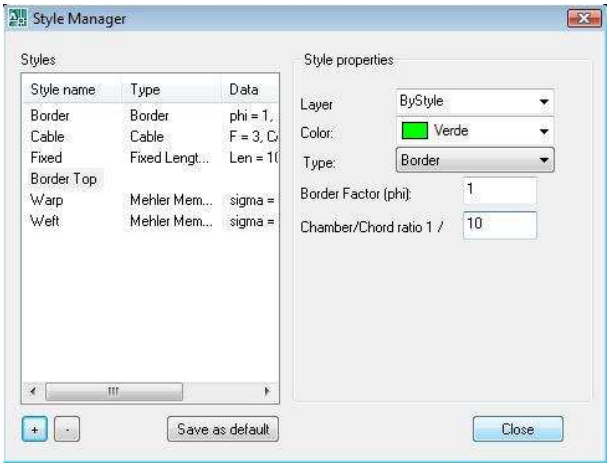
Run ***_TDSettings*** command and fill the dialog box cells as shown below.



Optionally modify *Text settings* but maintain default *Model units*, then click Ok to confirm or Cancel to annul changes.

- *Set styles properties*

Open the ***_TDStyles*** command window and create a new style named *Top*. Fill the window cells as shown below.



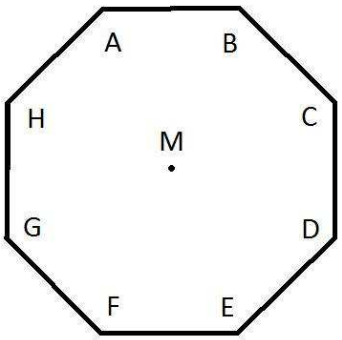
Then click *Close* to save the changes and leave the command.

- *Draw the base border polyline*

Using the proper Autocad command ***_polygon*** draw the base border polyline with the following properties:

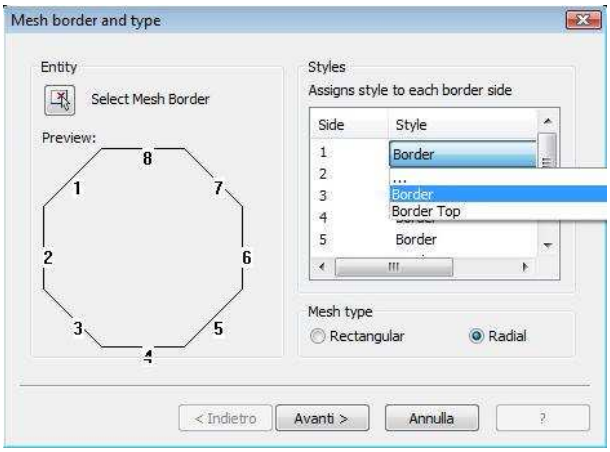
- number of sides: 8;
- centre of the polygon: 0,0,0;
- inscribed in the circle: <I>;
- radius length [cm]: 1000.

The obtained polygon is shown on the right.

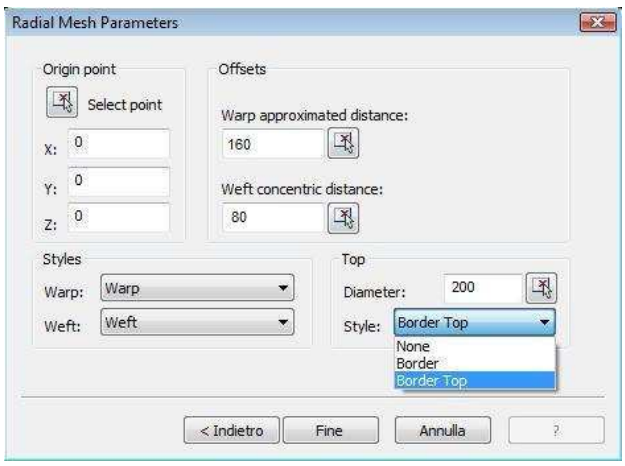


- *Mesh generation*

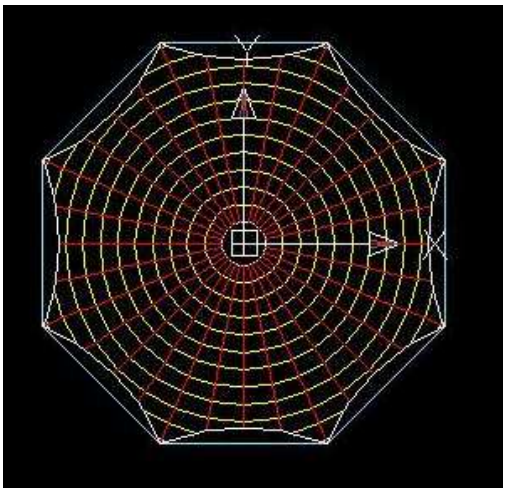
Start mesh generation procedure typing ***_TDMesh*** in the command line or clicking the proper button in *TensileDraw* toolbar. After the window below is open, select the polygon previously created as mesh border, assign *Border* style to every side and choose *Radial* mesh type. Click *Next* button.



In the *Radial Mesh Parameters* window set the warp and weft offsets and the top ring properties as shown below. Default *Origin point* and beam *Styles* can be maintained.

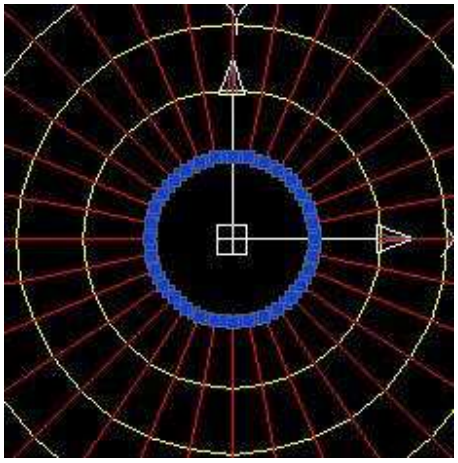


Once mesh generation is finished the following 2D mesh will appear:



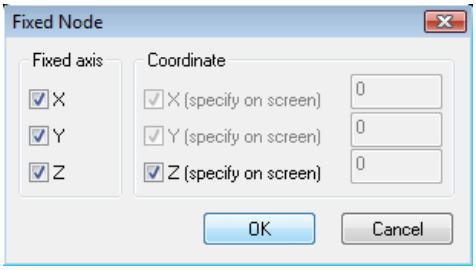
- *Insert fixed points*

Select the top ring beams.



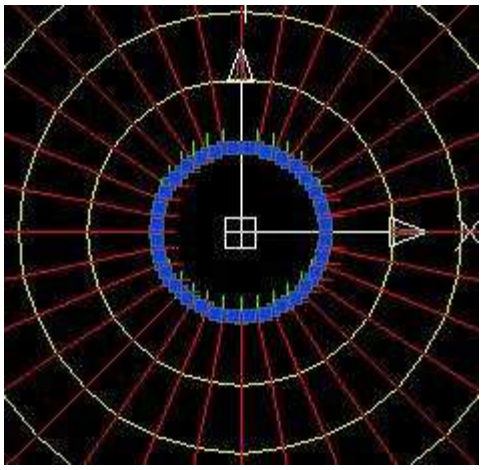
Run ***_TDFixNodes*** command:
all coordinates are fixed by default.

Click *Ok* button.

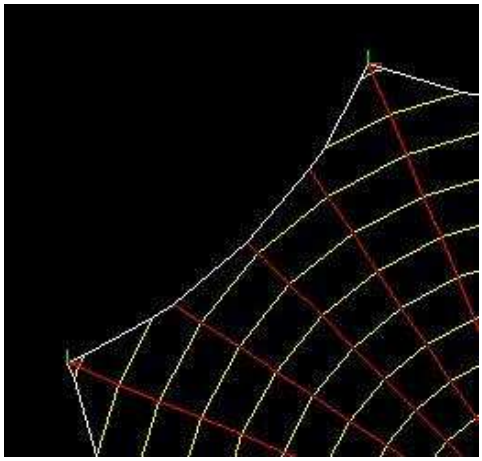


The fixed point are visualized as in the

figure on the right.



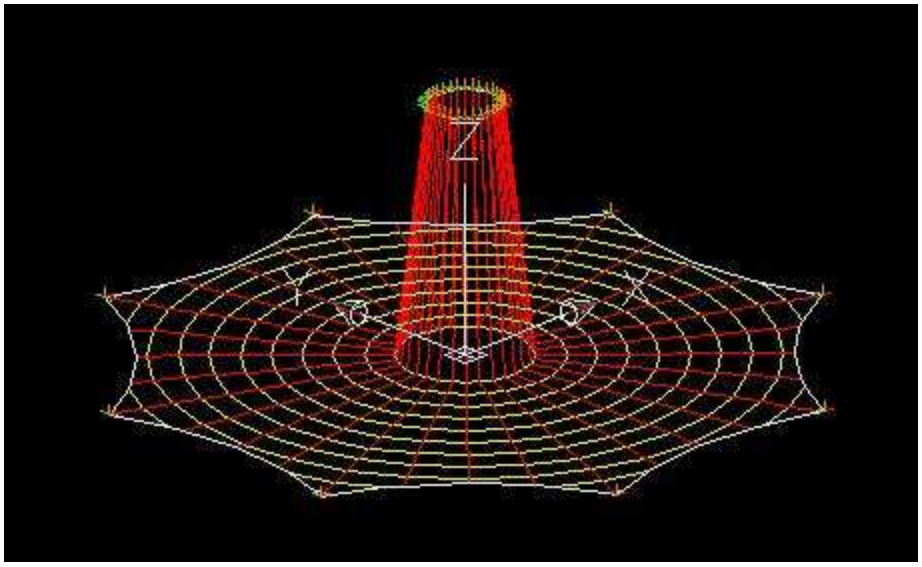
Run ***_TDFixNodes*** command again and insert a Fixed point for each vertex of the base border.



- *Stretch fixed points*

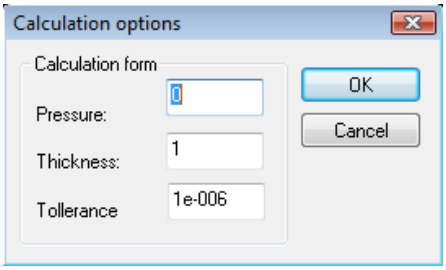
Select the top ring beams and the warp beam vertexes where the fixed nodes have been positioned.

Then using ***_stretch*** AutoCAD command move selected elements of 700 along Z axis: click any point on the screen as reference; type *@0,0,700* in the command line to give a relative stretch distance.

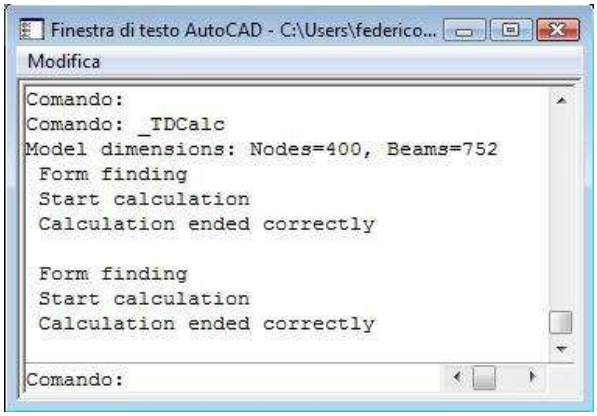


- *Calculate form*

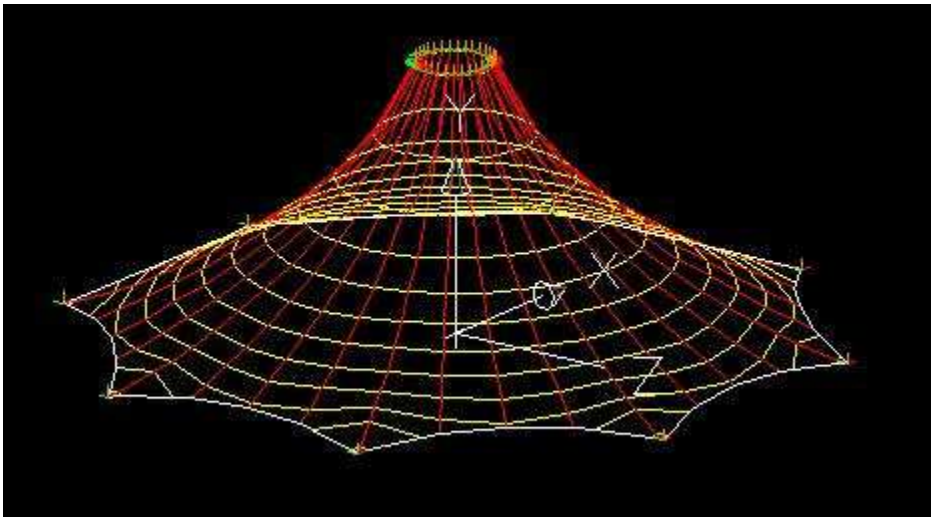
Run ***_TDCalc*** command and click *Ok* button in the *Calculation options* window.

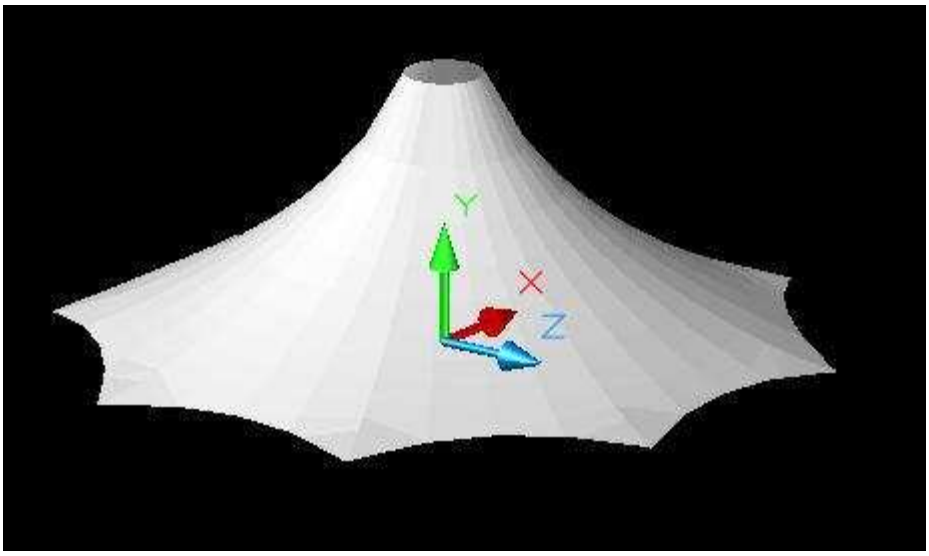


In the command line you can get informations about the amount of elements which form the geometry.



The figures below show the result of calculation and the 3D render visualization obtained with ***_TDFaces*** command that allows an easier analysis of the shape.




Y
X
Z